

AERODYNAMIC DESIGN AND CHARACTERIZATION OF NOVEL WOUND
COMPOSITE MULTISTAGE COUNTER-ROTATING AXIAL COMPRESSORS

By

Blake Ernest Gower

A DISSERTATION

Submitted to
Michigan State University
in partial fulfillment of the requirements
for the degree of

Mechanical Engineering – Doctor of Philosophy

2018

ABSTRACT

AERODYNAMIC DESIGN AND CHARACTERIZATION OF NOVEL WOUND COMPOSITE MULTISTAGE COUNTER-ROTATING AXIAL COMPRESSORS

By

Blake Ernest Gower

This thesis examines two different generations of axial compressor developed within the framework of the patented wound composite impeller technology created at Michigan State University. The technology itself allows for a departure from both the construction and operation of traditional single and multistage axial machines. Rather than using casting and machining methods to produce the impellers, they are wound from carbon fiber or other fiber/matrix material on a mandrel with curved slots. Winding layer-by-layer in the axial direction builds the blades while simultaneously creating the outer (and inner) shroud(s). The winding technique ensures that the fibers are closely aligned with the forces associated with high speed rotation, thereby yielding a high strength, light weight composite rotor capable of operating in chemically aggressive environments once cured. Traditional multistage axial compressors typically have a single drive shaft and hence require unidirectional rotation at a single operating speed. Non-rotating stators are utilized between rotors to impose the appropriate velocity distribution at the subsequent rotor inlet. The stators however do not perform useful work in terms of boosting the total pressure, and they contribute substantially to the overall footprint of a multistage machine. The employment of counter-rotating stages serves to eliminate the need for all intermediate stators as they themselves impose the necessary velocity distribution for the subsequent rotor while simultaneously performing useful work. Counter-rotation can be achieved by integrating a permanent magnet motor with each rotor. Rotors can be mounted on a non-rotating shaft and can therefore be driven in opposite directions through the use of variable frequency drives.

Initially developed for strength and ease of construction, a full geometric characterization of the first-generation “star pattern” impeller is performed and it is found that it operates under the forced-vortex flow regime. Reductions in terms of polytropic efficiency, mass flow rate, and total pressure ratio are seen from analytical prediction to numerical simulation, and again from simulation to experimental measurement. These reductions have led to the development of the second-generation impeller, which operates under the free-vortex flow regime.

Enhanced performance of single stage second-generation impellers in numerical simulation has led to a vast investigation matching geometrical parameters, rotational speeds, and flow velocities to best-point operating conditions for up to seven counter-rotating stages compressing initially saturated water vapor under vacuum pressure for 22 different inlet temperatures. Numerical simulations of select cases agree well with analytical predictions.

For achieving maximum specific work transfer from the rotors to the working fluid, it is determined that the critical relative Mach number at each rotor tip should always be maximized. Hub/tip ratio at the first rotor inlet, aspect ratio, critical absolute Mach number, and turning angle are all temperature-dependent. The number of stages employed also has a large effect on how each rotor behaves (e.g. the second stage of a three stage machine looks and behaves differently from the second stage in a six stage machine), however utilizing an odd vs. an even number of total stages will have a much larger effect on inlet flow angle and the dimensionless flow coefficient, blade loading coefficient, and specific speed of each rotor.

Seven other gas mixtures have been investigated in similar fashion and exhibit similar behavior. Overall, billions of designs have been evaluated and the best operating conditions are determined for each individual set of inlet conditions and number of stages used. This research lays the necessary ground work for multistage counter-rotating axial compressor construction.

Copyright by
BLAKE ERNEST GOWER
2018

This one's for Mom and Dad

ACKNOWLEDGEMENTS

I would first and foremost like to thank my advisor Dr. Norbert Müller for making this work possible. Without his guidance and encouragement over the years both professionally and personally, I would certainly not be at the point where I am today.

I would also like to thank Dr. Abraham Engeda, Dr. Thomas Pence, and Dr. Elias Strangas for serving as my committee members and providing the crucial advice necessary to improve this work.

To my friends and colleagues TJ Qualman, Younis Najim, Zack Hoyle, and Johannes Pohl, I would not have made it this far without the camaraderie and opportunity to float ideas.

Special thanks to Anna, Dave, and Colin Gower, my parents and brother, and Amanda Farhat my incredible girlfriend. I don't know what I would have done without all your love and moral support.

TABLE OF CONTENTS

LIST OF TABLES	x
LIST OF FIGURES	xii
KEY TO SYMBOLS AND ABBREVIATIONS	xxix
Chapter 1 Introduction and Applications.....	1
1.1 Background	1
1.2 Vapor Compression Refrigeration Cycle	2
1.3 Utilizing Water (R718) as a Refrigerant	5
1.3.1 Comparative Case Study: R718 vs. R134-a.....	6
1.4 Woven Wheel Development	11
1.4.1 Counter-Rotating Axial Compressors.....	12
1.4.2 Original Woven Impeller	14
1.5 Geothermal Power Plant Application.....	18
1.5.1 Geothermal Power Plants.....	18
1.5.2 Non-Condensable Gases	20
1.5.3 Current NCG Removal Techniques	21
1.6 Desalination Application.....	25
1.6.1 Reverse Osmosis	25
1.6.2 Multistage Flash Distillation.....	26
1.6.3 Vapor Compression Distillation	26
1.7 Wastewater Aeration Application	27
1.8 Mechanical Vapor Compression Application	28
1.9 Advantages of Woven Impeller Technology	29
1.10 Research Objective and Dissertation Outline.....	31
Chapter 2 Modeling and Design of Axial Compressors.....	35
2.1 Thermodynamics and Gas Dynamics of Axial Compressors	35
2.1.1 Continuity and 1-Dimensional Compressible Flow.....	35
2.1.2 Mixture Composition	38
2.1.3 Thermodynamic Laws and Isentropic Relations.....	39
2.1.4 Critical Speed of Sound	42
2.1.5 Flow Property Ratios	46
2.2 Euler's Equation of Turbomachinery	48
2.2.1 Moment of Momentum	48
2.2.2 Thermodynamic Work Transfer	51
2.3 Circular Arc Blade Description.....	55
2.3.1 Rotor Blades.....	55
2.3.2 Inlet Guide Vane Blades	59
2.4 Velocity Diagrams and Reference Frames	60
2.4.1 Inertial Reference Frame.....	60
2.4.2 Rotating Reference Frame	61

2.5	0- and 1-Dimensional Design Procedure	68
2.5.1	Inlet Conditions and Fluid Properties	69
2.5.2	Geometric and Kinematic Considerations	70
2.5.3	Variable Constraints.....	75
Chapter 3	Simulation of Axial Compressors.....	79
3.1	Computational Fluid Dynamics – Governing Equations	79
3.2	RANS Equations	83
3.3	Turbulence Modeling	85
3.3.1	Turbulent Viscosity Hypothesis.....	86
3.3.2	Zero and One Equation Turbulent Viscosity Models	87
3.3.3	Two Equation Turbulent Viscosity Model – $k-\varepsilon$	88
3.3.4	Two Equation Turbulent Viscosity Model – $k-\omega$	90
3.3.5	Two Equation Turbulent Viscosity Model – Shear Stress Transport	91
3.4	Meshing and Discretization.....	95
3.5	Boundary Conditions.....	101
Chapter 4	Star Pattern Impeller Performance.....	102
4.1	Circular Arc Blade Description at Shroud in Global Coordinates	102
4.2	Circular Arc Blade Description at Interior Points.....	106
4.3	Cross Sectional Area of Star Pattern Impeller	112
4.4	Star Pattern Impeller Flow Calculations	115
4.4.1	A Note on Blade Stagger, Solidity, and Diffusion Factor	123
4.4.2	Boundary Layer Growth Prediction.....	125
4.5	Star Pattern Impeller 3-Dimensional Simulation	128
4.5.1	Mesh Independence	133
4.5.2	Simulation Results	135
4.6	Experimental Apparatus and Results	144
4.7	Chapter Summary and Conclusions	149
Chapter 5	Free-Vortex Blade Impeller Performance.....	153
5.1	Theory of Radial Equilibrium	154
5.2	Free-Vortex Flow	158
5.3	2- and 3-Dimensional Circular Arc Blade Description.....	161
5.3.1	Geometry Selection.....	161
5.4	Free-Vortex Blade Impeller 3-Dimensional Simulation	164
5.5	Comparison of Free-Vortex Blade Impeller to Star Pattern Impeller	176
5.6	Chapter Summary and Conclusions	183
Chapter 6	Optimal Performance of Multistage Counter-Rotating Axial Compressors.....	185
6.1	Design Procedure	187
6.1.1	Design Objective.....	188
6.1.2	Design Variables	188
6.1.3	Design Constraints	190
6.2	Initial Design Exploration	191
6.2.1	Design Variable Reduction	195
6.2.2	Design Constraint Evolution.....	198

6.3	Design Exploration.....	200
6.3.1	Scope of Design Exploration	203
6.3.2	Objective and Constraint Sensitivity	204
6.4	Best Design Conditions.....	216
6.4.1	Comparison to Traditional Rotor-Stator Case	230
6.5	Numerical Simulation of Select Cases	238
6.6	Chapter Summary and Conclusions	251
Chapter 7	Conclusions.....	253
7.1	Summary of Conclusions	253
7.1.1	Chapter 4.....	253
7.1.2	Chapter 5.....	254
7.1.3	Chapter 6.....	255
7.2	Contribution	256
7.3	Recommendations for Future Work.....	258
APPENDICES		261
APPENDIX 1	Mesh Independence Study for Star Pattern Impeller	262
APPENDIX 2	Polytropic Efficiency	272
APPENDIX 3	Supplemental Objective and Constraint Sensitivity Plots	279
APPENDIX 4	Simulation Results for Best-Point Design of Single Stage with IGV Compressing Air Under Vacuum Pressure	288
APPENDIX 5	Multistage Simulation Results	294
APPENDIX 6	Best Design Points for Other Gases.....	317
REFERENCES		388

LIST OF TABLES

Table 1.1. Ideal Vapor Compression Refrigeration Cycle Assumptions.....	6
Table 1.2. Ideal Refrigeration Cycle Properties for R134-a in Case Study [2]	6
Table 1.3 Ideal Refrigeration Cycle Properties for R718 in Case Study [2]	7
Table 1.4. Performance Comparison of R134-a and R718 in Case Study.....	9
Table 1.5. Room-Area-Specific Flow and Compressor Properties in Ideal Case Study	10
Table 1.6. Winding Patterns for Continuous Fiber Impellers.....	16
Table 1.7. Performance Summary of Selected Impellers	17
Table 2.1. 0D and 1D Design Considerations.....	69
Table 2.2. Compressor Inlet Conditions	69
Table 2.3. Compressor Geometry and Flow Kinematics – Design Selections	74
Table 2.4. Compressor Geometry and Flow Kinematics – Determined Values.....	74
Table 2.5. Summary of Constraints	77
Table 3.1. k - ε Model Constants	90
Table 3.2. k - ω Model Constants	91
Table 3.3. SST Model Constants	95
Table 3.4. Tri-Linear Element Shape Functions.....	100
Table 4.1. Star Pattern Impeller Inlet Conditions and Geometry	116
Table 4.2. Forced Vortex Constants from Inlet to Outlet	120
Table 4.3. 2D Performance and Geometry Summary.....	123
Table 4.4. Region-Dependent Blade Stagger, Solidity, and Diffusion Factor.....	125
Table 4.5. Boundary Layer Thickness at Rotor Outlet	126
Table 4.6. Near-Tip Boundary Layer Thickness vs. Normalized Stream-wise Location.....	127
Table 4.7. Main Simulation Results.....	135

Table 4.8. Simulated de Haller and Diffusion Factor Values	142
Table 4.9. Experimental Results and Predicted Performance Comparison	147
Table 5.1. Geometrical Parameter Comparison of FVB and Star Pattern Impellers	162
Table 5.2. Performance of FVB Impeller	166
Table 5.3. FVB de Haller Numbers and Diffusion Factors	176
Table 5.4. FVB and Star Pattern Performance Comparison	177
Table 6.1. Summary of Initial Design Variables	189
Table 6.2. Number of Initial Design Variables by Number of Stages Considered.....	189
Table 6.3. Summary of Design Constraints	190
Table 6.4. Blade Number by Stage	195
Table 6.5. Reduced Design Variable Set	198
Table 6.6. Variable Range and Resolution for Global Design Space Search	200
Table 6.7. Variable Range and Resolution for Local Design Space Search.....	201
Table 6.8. Design Evaluations for One Set of Inlet Conditions.....	202
Table 6.9. Designs Evaluated in Full Scale Exploration	204
Table 6.10. Case Study Comparing Counter-Rotor to Rotor Stator at Same Inlet Conditions..	231
Table A1.1. Mesh Independence Study Parameters	262
Table A6.2.1. NCG Mixture Components	327

LIST OF FIGURES

Figure 1.1. Vapor Compression Refrigeration Cycle Schematic.....	2
Figure 1.2. T - s Diagram of Ideal Generic Vapor Compression Refrigeration Cycle	3
Figure 1.3. p - h Diagram of Ideal Generic Vapor Compression Refrigeration Cycle	4
Figure 1.4. T - s Diagram for R134-a in Case Study	7
Figure 1.5. p - h Diagram for R134-a in Case Study	8
Figure 1.6. T - s Diagram for R718 in Case Study	8
Figure 1.7. p - h Diagram for R718 in Case Study	9
Figure 1.8. Comparison of COP for R718 and R134-a for different Compressor Efficiencies ...	11
Figure 1.9. Comparison of Power Density for Axial Compressors	13
Figure 1.10. Comparison of Diameter for Equal Capacity Compressors	13
Figure 1.11. An 8-Slotted Mandrel.....	15
Figure 1.12. Mandrel with a Partially-Wound Impeller	15
Figure 1.13. Geometries 8B and 8C with Differing Hub/Tip Ratios.....	17
Figure 1.14. Single-Flash Geothermal Power Plant Schematic.....	19
Figure 1.15. Steam Jet Ejector Schematic	22
Figure 1.16. Counter-Clockwise Rotating Liquid Ring Vacuum Pump Schematic [21].....	23
Figure 1.17. GE SRL 903 3-Stage Centrifugal Compressor [Source: GE Power Systems].....	24
Figure 1.18. Vapor Compression Distillation for Desalination Schematic	27
Figure 1.19. Activated Sludge Wastewater Treatment Schematic	28
Figure 1.20. Mollier Diagram of MVC Process	29
Figure 2.1. Fixed Finite Control Volume.....	35
Figure 2.2. Quasi-One-Dimensional Control Volume	37
Figure 2.3. Circular Arc Blade Segment.....	55

Figure 2.4. Detail of Circular Arc Blade Segment.....	57
Figure 2.5. Circular Arc Blade Segment for IGV.....	60
Figure 2.6. IGV Velocity Diagrams.....	61
Figure 2.7. Rotor Velocity Diagrams.....	62
Figure 2.8. Counter-Rotor Velocity Diagrams	64
Figure 2.9. Schematic of Unidirectional Rotation and Singular Speed without Intermediate Stators	65
Figure 2.10. Schematic of Unidirectional Rotation and Speed with Intermediate Stators	66
Figure 2.11. Schematic of Counter Rotation without Intermediate Stators	66
Figure 3.1. Normal Stress Acting on Fluid Element.....	81
Figure 3.2. Shear Stress Acting on Fluid Element.....	81
Figure 3.3. 2D Finite Volume Mesh Representation [46]	96
Figure 3.4. 2D Mesh Element [46]	97
Figure 3.5. Hexahedral Volume Element [46].....	99
Figure 3.6. Tetrahedral Volume Element [46].....	99
Figure 4.1. Fiber Pattern of Star Impeller	103
Figure 4.2. Frontal View of Star Impeller Showing One Blade	104
Figure 4.3. Axial View of One Blade at Outer Shroud.....	104
Figure 4.4. Blade Profile at the Outer Shroud for Location A.....	108
Figure 4.5. Blade Profile at the Outer Shroud for Location B.....	108
Figure 4.6. Individual Fiber Paths Crossing a Single Set of Mandrel Slots	110
Figure 4.7. Individual Fiber Paths Crossing All Mandrel Slots.....	111
Figure 4.8. Fiber Intersections, Inflections, and End Points at Different Axial Locations.....	112
Figure 4.9. Full-Wheel View of Fibers Intersecting a Single Blade.....	113
Figure 4.10. Single Blade View with Intersecting Fibers.....	114
Figure 4.11. Representative Star Pattern Impeller Inlet Velocity Diagram.....	115

Figure 4.12. Mass Flow Rate Contours (a) vs. Global Coordinates (b) vs. Percent Stream and Span.....	117
Figure 4.13. Total Pressure Contours (a) vs. Global Coordinates (b) vs. Percent Stream and Span	117
Figure 4.14. Absolute Velocity Contours vs. Global Coordinates	118
Figure 4.15. Absolute Velocity Contours vs. Percent Stream and Span.....	118
Figure 4.16. Static Density Contours (a) vs. Global Coordinates (b) vs. Percent Stream and Span	119
Figure 4.17. Star Pattern Swirl Distribution	120
Figure 4.18. Relative Velocity Contours vs. Global Coordinates.....	121
Figure 4.19. Relative Velocity Contours vs. Percent Stream and Span.....	121
Figure 4.20. Absolute and Relative Critical Mach Number Contours vs. Global Coordinates..	122
Figure 4.21. Absolute and Relative Critical Mach Number Contours vs. Percent Stream and Span	122
Figure 4.22. Differing Blade Stagger at Single Span-wise Locations	124
Figure 4.23. Linear Distance l Between Blades Emanating from One Mandrel Slot.....	127
Figure 4.24. Two Blades Emanating from One Mandrel Slot	129
Figure 4.25. Single 3D Blade.....	130
Figure 4.26. Four 3D Blades.....	130
Figure 4.27. Four 3D Blades with Periodic Sections.....	131
Figure 4.28. Enclosed Volume with Inlet and Outlet Regions	131
Figure 4.29. Fluid Region for Simulation.....	132
Figure 4.30. Very Coarse (Left), Medium (Center), and Very Fine (Right) Meshes	133
Figure 4.31. Efficiency Variation with Mesh Refinement.....	134
Figure 4.32. Medium-Density Mesh.....	134
Figure 4.33. Meridional View of Total Pressure Contours (a) Simulation (b) Calculation.....	136
Figure 4.34. Meridional of Axial Velocity Contours (a) Simulation (b) Calculation.....	137

Figure 4.35. Meridional View of Radial Velocity Contours	137
Figure 4.36. Meridional View of Entropy Contours.....	138
Figure 4.37. Meridional View of Absolute Tangential Velocity Contours (a) Simulation (b) Calculation	139
Figure 4.38. Meridional View of Relative Tangential Velocity Contours (a) Simulation (b) Calculation	140
Figure 4.39. Blade-to-Blade View of Axial Velocity Contours	141
Figure 4.40. Full-Span View of Axial Velocity Contours	142
Figure 4.41. Meridional View of Absolute Mach Number Contours	143
Figure 4.42. Meridional View of Relative Mach Number Contours	144
Figure 4.42. Star Pattern Impeller Prototype	145
Figure 4.43. A Mandrel with 8 Circular Arc Slots [13].....	145
Figure 4.44. Open Test Loop Showing Installed Impeller.....	146
Figure 4.45. Closed Test Loop Showing Pressure Measurement Probes	147
Figure 5.1. A Second Generation Mandrel	153
Figure 5.2. Radial Equilibrium Flow Across a Blade Row	155
Figure 5.3. Fluid Element in Radial Equilibrium	155
Figure 5.4. Fluid Element Side-Wall Pressure Force Decomposition	156
Figure 5.5. Circulation about a Fluid Element.....	159
Figure 5.6. Blade-to-Blade View of FVB Circular Arcs at Left: Tip, Right: Mid-Span.....	163
Figure 5.7. Blade-to-Blade View at Hub of Left: 90°-Limited Case, Right: Full Free-Vortex Turning.....	163
Figure 5.8. A single Free-Vortex Blade for Left: 90°-Limited Case, Right: Full Free-Vortex Turning.....	164
Figure 5.9. Full FVB Rotor View for Left: 90°-Limited Case, Right: Full Free-Vortex Turning	164
Figure 5.10. Mesh Surfaces for FVB Simulation	165

Figure 5.11. Meridional View of Total Pressure Contours. Upper: Simulation, Lower: Calculation	167
Figure 5.12. Meridional View of Simulated Axial Velocity Contours	168
Figure 5.13. Meridional View of Simulated Radial Velocity Contours	169
Figure 5.14. Meridional View of Simulated Entropy Contours.....	170
Figure 5.15. Meridional View of Absolute Tangential Velocity Contours. Upper: Simulation, Lower: Calculation.....	171
Figure 5.16. Meridional View of Relative Tangential Velocity Contours. Upper: Simulation, Lower: Calculation.....	172
Figure 5.17. Blade-to-Blade View of Axial Velocity Contours	173
Figure 5.18. Full Span View of Axial Velocity Contours	173
Figure 5.19. Meridional View of Absolute Mach Number Contours. Upper: Simulation, Lower: Calculation	174
Figure 5.20. Meridional View of Relative Mach Number Contours. Upper: Simulation, Lower: Calculation	175
Figure 5.21. Incidence Angle vs. Span for FVB and Star Pattern Impellers	178
Figure 5.22. Star Pattern Velocity Vectors at Span 0.945	179
Figure 5.23. FVB Velocity Vectors at Span 0.9	179
Figure 5.24. Star Pattern Velocity Vectors at Span 0.5	181
Figure 5.25. FVB Velocity Vectors at Span 0.5	181
Figure 5.26. Deviation Angle vs. Span for FVB and Star Pattern Impellers	182
Figure 6.1. Design Process Diagram for a Single Rotor.....	187
Figure 6.2. Rotor 1 Work Transfer vs. Inlet Flow Angle and Mach Number	192
Figure 6.3. Rotor 2 Work Transfer vs. Inlet Flow Angle and Mach Number	192
Figure 6.4. Total Two Stage Work Transfer vs. Inlet Flow Angle and Mach Number	193
Figure 6.5. Total Four Stage Feasible Work Transfer vs. Inlet Flow Angle and Mach Number	194
Figure 6.6. Work Transfer vs. Inlet Mach Number of First Rotor around Best-Point.....	196

Figure 6.7. Work Transfer vs. Inlet Mach Number of Second Rotor around Best-Point	197
Figure 6.8. Work Transfer vs. Inlet Mach Number of Third Rotor around Best-Point	197
Figure 6.9. Specific Work Transfer vs. First Rotor Blade Turning	205
Figure 6.10. Specific Work Transfer vs. Second Rotor Blade Turning.....	206
Figure 6.11. Specific Work Transfer vs. Third Rotor Blade Turning.....	206
Figure 6.12. de Haller Numbers vs. First Rotor Blade Turning	208
Figure 6.13. Diffusion Factors vs. First Rotor Blade Turning.....	208
Figure 6.14. de Haller Numbers vs. Second Rotor Blade Turning	209
Figure 6.15. Diffusion Factors vs. Second Rotor Blade Turning	209
Figure 6.16. de Haller Numbers vs. Second Rotor Blade Turning	210
Figure 6.17. Diffusion Factors vs. Second Rotor Blade Turning	210
Figure 6.18. Absolute Mach Number vs. Blade Turning in the First Rotor	211
Figure 6.19. Absolute Mach Number vs. Blade Turning in the Second Rotor	212
Figure 6.20. Absolute Mach Number vs. Blade Turning in the Third Rotor.....	212
Figure 6.21. Work Transfer vs. Absolute Inlet Flow Angle	213
Figure 6.22. Work Transfer vs. Absolute Inlet Critical Mach Number	214
Figure 6.23. Work Transfer vs. Inlet Hub/Tip Ratio	214
Figure 6.24. de Haller Number vs. Inlet Hub/Tip Ratio	215
Figure 6.25. Diffusion Factor vs. Inlet Hub/Tip Ratio	215
Figure 6.26. Stage Work Transfer vs. Saturation Temperature at First Rotor Inlet	217
Figure 6.27. Normalized Stage Work Transfer vs. Saturation Temperature at First Rotor Inlet	218
Figure 6.28. Tip Speed Ratio vs. Saturation Temperature at First Rotor Inlet	219
Figure 6.29. Absolute First Rotor Inlet Flow Angle vs. Saturation Temperature at First Rotor Inlet	220
Figure 6.30. First Rotor Absolute Critical Mach Number vs. Saturation Temperature at First Rotor Inlet	220

Figure 6.31. Hub/Tip Ratio at First Rotor Inlet vs. Saturation Temperature at First Rotor Inlet	221
Figure 6.32. Blade Turning Angle of Each Rotor vs. Saturation Temperature at First Rotor Inlet	221
Figure 6.33. Saturation Pressure vs. Saturation Temperature.....	222
Figure 6.34. de Haller Number for Each Rotor vs. Saturation Temperature at First Rotor Inlet	223
Figure 6.35. Diffusion Factor for Each Rotor vs. Saturation Temperature at First Rotor Inlet..	224
Figure 6.36. Critical Absolute Mach Number at the Hub Inlet for Each Rotor vs. Saturation Temperature at First Rotor Inlet	225
Figure 6.37. Critical Absolute Mach Number at the Hub Outlet for Each Rotor vs. Saturation Temperature at First Rotor Inlet	226
Figure 6.38. Flow Coefficient vs. Saturation Temperature at First Rotor Inlet.....	228
Figure 6.39. Blade Loading Coefficient vs. Saturation Temperature at First Rotor Inlet	229
Figure 6.40. Counter-Rotating Reaction Pairs vs. Saturation Temperature at First Rotor Inlet.	233
Figure 6.41. Rotor Work Fraction of Combined Work Total vs. Saturation Temperature at First Rotor Inlet	234
Figure 6.42. Work Contribution Fraction of Even-Numbered Stages in Machines with Even Number Total Stages vs. Saturation Temperature at First Rotor Inlet	235
Figure 6.43. Work Contribution Fraction of Odd-Numbered Stages in Machines with Odd Number Total Stages vs. Saturation Temperature at First Rotor Inlet	235
Figure 6.44. Specific Speed vs. Saturation Temperature at First Rotor Inlet	237
Figure 6.45. Total Pressure Ratio vs. Saturation Temperature at First Rotor Inlet	238
Figure 6.46. Performance Charts for 2, 3, and 4 Stages Used at 60°C.....	240
Figure 6.47. Normalized Combined Work Transfer for Highest Polytropic Efficiency and Largest Combined Work Transfer vs. Saturation Temperature at First Rotor Inlet	241
Figure 6.48. Rotor Work Fraction of Combined Work Total vs. Saturation Temperature at First Rotor Inlet for Cases of Highest Overall Polytropic Efficiency in Simulation Compared to Calculated Work Fraction	242
Figure 6.49. Rotor Work Fraction of Combined Work Total vs. Saturation Temperature at First Rotor Inlet for Cases of Largest Combined Work Transfer in Simulation Compared to Calculated Work Fraction	243

Figure 6.50. Polytropic Efficiency vs. Saturation Temperature at First Rotor Inlet.....	244
Figure 6.51. Incidence and Deviation Angles vs. Axial Location for Highest Polytropic Efficiency Cases.....	245
Figure 6.52. Absolute and Relative Flow Angles vs. Axial Location for Highest Polytropic Efficiency Cases.....	246
Figure 6.53. Incidence and Deviation Angles vs. Axial Location for Largest Combined Work Transfer Cases.....	247
Figure 6.54. Absolute and Relative Flow Angles vs. Axial Location for Largest Combined Work Transfer Cases.....	248
Figure 6.55. Meridional View of Relative Mach Number Contours for 4 Counter-Rotating Stages with Saturated Vapor Inlet at 60°C.....	249
Figure 6.56. Blade-to-Blade View of Relative Mach Number Contours for 4 Counter-Rotating Stages with Saturated Vapor Inlet at 60°C	250
Figure 7.1. Test Loop Schematic Including Vacuum Pump and Water Reservoir for Low Pressure Evaporation [53].....	259
Figure 7.2. Compressor Section Schematic [53]	260
Figure A1.1. Very Coarse Mesh	262
Figure A1.2. Very Coarse Mesh RMS Residuals	263
Figure A1.3. Very Coarse Mesh Efficiency Monitor History	263
Figure A1.4. Coarse Mesh	264
Figure A1.5. Coarse Mesh Section View.....	264
Figure A1.6. Coarse Mesh RMS Residuals	265
Figure A1.7. Coarse Mesh Efficiency Monitor History	265
Figure A1.8. Medium Mesh.....	266
Figure A1.9. Medium Mesh RMS Residuals.....	267
Figure A1.10. Medium Mesh Efficiency Monitor History	267
Figure A1.11. Fine Mesh	268
Figure A1.12. Fine Mesh RMS Residuals	269

Figure A1.13. Fine Mesh Efficiency Monitor History	269
Figure A1.14. Very Fine Mesh	270
Figure A1.15. Very Fine Mesh RMS Residuals	271
Figure A1.16. Very Fine Mesh Efficiency Monitor History	271
Figure A2.1. Mollier Diagram of Small-Stage Compression Processes. Adapted from [54].....	273
Figure A2.2. A Single Small Stage Compression Process	275
Figure A2.3. Isentropic Efficiency vs. Pressure Ratio.....	278
Figure A3.1. First Rotor Relative Mach Number vs. Hub/Tip Ratio	279
Figure A3.2. Second Rotor Relative Mach Number vs. Hub/Tip Ratio	279
Figure A3.3. Third Rotor Relative Mach Number vs. Hub/Tip Ratio.....	280
Figure A3.4. First Rotor Absolute Mach Number vs. Hub/Tip Ratio	280
Figure A3.5. Second Rotor Absolute Mach Number vs. Hub/Tip Ratio.....	280
Figure A3.6. Third Rotor Absolute Mach Number vs. Hub/Tip Ratio.....	281
Figure A3.7. Tip Speed and Tip Speed Ratio vs. Absolute Mach Number at First Rotor Inlet .	281
Figure A3.8. Total Pressure Ratio vs. Absolute Mach Number at First Rotor Inlet.....	281
Figure A3.9. de Haller Number vs. Absolute Mach Number at First Rotor Inlet	282
Figure A3.10. Diffusion Factor vs. Absolute Mach Number at First Rotor Inlet.....	282
Figure A3.11. First Rotor Relative Mach Number vs. Absolute Mach Number at First Rotor Inlet	282
Figure A3.12. Second Rotor Relative Mach Number vs. Absolute Mach Number at First Rotor Inlet	283
Figure A3.13. Third Rotor Relative Mach Number vs. Absolute Mach Number at First Rotor Inlet	283
Figure A3.14. First Rotor Absolute Mach Number vs. Absolute Mach Number at First Rotor Inlet	283
Figure A3.15. Second Rotor Absolute Mach Number vs. Absolute Mach Number at First Rotor Inlet	284

Figure A3.16. Third Rotor Absolute Mach Number vs. Absolute Mach Number at First Rotor Inlet	284
Figure A3.17. Tip Speed and Tip Speed Ratio vs. Absolute Flow Angle at First Rotor Inlet ...	284
Figure A3.18. Total Pressure Ratio vs. Absolute Flow Angle at First Rotor Inlet.....	285
Figure A3.19. de Haller Number vs. Absolute Flow Angle at First Rotor Inlet.....	285
Figure A3.20. Diffusion Factor vs. Absolute Flow Angle at First Rotor Inlet	285
Figure A3.21. First Rotor Relative Mach Number vs. Absolute Flow Angle at First Rotor Inlet	286
Figure A3.22. Second Rotor Relative Mach Number vs. Absolute Flow Angle at First Rotor Inlet	286
Figure A3.23. Third Rotor Relative Mach Number vs. Absolute Flow Angle at First Rotor Inlet	286
Figure A3.24. First Rotor Absolute Mach Number vs. Absolute Flow Angle at First Rotor Inlet	287
Figure A3.25. Second Rotor Absolute Mach Number vs. Absolute Flow Angle at First Rotor Inlet	287
Figure A3.26. Third Rotor Absolute Mach Number vs. Absolute Flow Angle at First Rotor Inlet	287
Figure A4.1. Contours of Total Pressure in Meridional Plane	288
Figure A4.2. Contours of Axial Velocity in Meridional Plane.....	289
Figure A4.3. Contours of Radial Velocity in Meridional Plane	289
Figure A4.4. Contours of Absolute Tangential Velocity in Meridional Plane	289
Figure A4.5. Contours of Relative Tangential Velocity in Meridional Plane	290
Figure A4.6. Contours of Absolute Mach Number in Meridional Plane.....	290
Figure A4.7. Contours of Relative Mach Number in Meridional Plane.....	290
Figure A4.8. Contours of Entropy in Meridional Plane.....	291
Figure A4.9. Blade-to-Blade View of Axial Velocity Contours	291
Figure A4.10. Axial Velocity Contours at Rotor Inlet.....	292

Figure A4.11. Axial Velocity Contours at Rotor Mid-Stream.....	292
Figure A4.12. Axial Velocity Contours at Rotor Outlet.....	293
Figure A5.1. 2, 3, and 4 Stage Performance at 5°C	294
Figure A5.2. 2, 3, and 4 Stage Performance at 20°C	295
Figure A5.3. 2, 3, and 4 Stage Performance at 40°C	296
Figure A5.4. 2, 3, and 4 Stage Performance at 80°C	297
Figure A5.5. 2, 3, and 4 Stage Performance at 95°C	298
Figure A5.6. Total Pressure Ratio vs. Saturation Temperature at First Rotor Inlet for Cases of Highest Overall Polytropic Efficiency and Largest Combined Work Transfer.....	299
Figure A5.7. Meridional View of Relative Mach Number Contours for 2 Counter-Rotating Stages with Saturated Vapor Inlet at 5°C	300
Figure A5.8. Blade-to-Blade View of Relative Mach Number Contours for 2 Counter-Rotating Stages with Saturated Vapor Inlet at 5°C	300
Figure A5.9. Meridional View of Relative Mach Number Contours for 2 Counter-Rotating Stages with Saturated Vapor Inlet at 20°C	301
Figure A5.10. Blade-to-Blade View of Relative Mach Number Contours for 2 Counter-Rotating Stages with Saturated Vapor Inlet at 20°C	301
Figure A5.11. Meridional View of Relative Mach Number Contours for 2 Counter-Rotating Stages with Saturated Vapor Inlet at 40°C	302
Figure A5.12. Blade-to-Blade View of Relative Mach Number Contours for 2 Counter-Rotating Stages with Saturated Vapor Inlet at 40°C	302
Figure A5.13. Meridional View of Relative Mach Number Contours for 2 Counter-Rotating Stages with Saturated Vapor Inlet at 60°C	303
Figure A5.14. Blade-to-Blade View of Relative Mach Number Contours for 2 Counter-Rotating Stages with Saturated Vapor Inlet at 60°C	303
Figure A5.15. Meridional View of Relative Mach Number Contours for 2 Counter-Rotating Stages with Saturated Vapor Inlet at 80°C	304
Figure A5.16. Blade-to-Blade View of Relative Mach Number Contours for 2 Counter-Rotating Stages with Saturated Vapor Inlet at 80°C	304
Figure A5.17. Meridional View of Relative Mach Number Contours for 2 Counter-Rotating Stages with Saturated Vapor Inlet at 95°C	305

Figure A5.18. Blade-to-Blade View of Relative Mach Number Contours for 2 Counter-Rotating Stages with Saturated Vapor Inlet at 95°C	305
Figure A5.19. Meridional View of Relative Mach Number Contours for 3 Counter-Rotating Stages with Saturated Vapor Inlet at 5°C	306
Figure A5.20. Blade-to-Blade View of Relative Mach Number Contours for 3 Counter-Rotating Stages with Saturated Vapor Inlet at 5°C	306
Figure A5.21. Meridional View of Relative Mach Number Contours for 3 Counter-Rotating Stages with Saturated Vapor Inlet at 20°C	307
Figure A5.22. Blade-to-Blade View of Relative Mach Number Contours for 3 Counter-Rotating Stages with Saturated Vapor Inlet at 20°C	307
Figure A5.23. Meridional View of Relative Mach Number Contours for 3 Counter-Rotating Stages with Saturated Vapor Inlet at 40°C	308
Figure A5.24. Blade-to-Blade View of Relative Mach Number Contours for 3 Counter-Rotating Stages with Saturated Vapor Inlet at 40°C	308
Figure A5.25. Meridional View of Relative Mach Number Contours for 3 Counter-Rotating Stages with Saturated Vapor Inlet at 60°C	309
Figure A5.26. Blade-to-Blade View of Relative Mach Number Contours for 3 Counter-Rotating Stages with Saturated Vapor Inlet at 60°C	309
Figure A5.27. Meridional View of Relative Mach Number Contours for 3 Counter-Rotating Stages with Saturated Vapor Inlet at 80°C	310
Figure A5.28. Blade-to-Blade View of Relative Mach Number Contours for 3 Counter-Rotating Stages with Saturated Vapor Inlet at 80°C	310
Figure A5.29. Meridional View of Relative Mach Number Contours for 3 Counter-Rotating Stages with Saturated Vapor Inlet at 95°C	311
Figure A5.30. Blade-to-Blade View of Relative Mach Number Contours for 3 Counter-Rotating Stages with Saturated Vapor Inlet at 95°C	311
Figure A5.31. Meridional View of Relative Mach Number Contours for 4 Counter-Rotating Stages with Saturated Vapor Inlet at 5°C	312
Figure A5.32. Blade-to-Blade View of Relative Mach Number Contours for 4 Counter-Rotating Stages with Saturated Vapor Inlet at 5°C	312
Figure A5.33. Meridional View of Relative Mach Number Contours for 4 Counter-Rotating Stages with Saturated Vapor Inlet at 20°C	313

Figure A5.34. Blade-to-Blade View of Relative Mach Number Contours for 4 Counter-Rotating Stages with Saturated Vapor Inlet at 20°C	313
Figure A5.35. Meridional View of Relative Mach Number Contours for 4 Counter-Rotating Stages with Saturated Vapor Inlet at 40°C	314
Figure A5.36. Blade-to-Blade View of Relative Mach Number Contours for 4 Counter-Rotating Stages with Saturated Vapor Inlet at 40°C	314
Figure A5.37. Meridional View of Relative Mach Number Contours for 4 Counter-Rotating Stages with Saturated Vapor Inlet at 80°C	315
Figure A5.38. Blade-to-Blade View of Relative Mach Number Contours for 4 Counter-Rotating Stages with Saturated Vapor Inlet at 80°C	315
Figure A5.39. Meridional View of Relative Mach Number Contours for 4 Counter-Rotating Stages with Saturated Vapor Inlet at 95°C	316
Figure A5.40. Blade-to-Blade View of Relative Mach Number Contours for 4 Counter-Rotating Stages with Saturated Vapor Inlet at 95°C	316
Figure A6.1.1. Specific Work vs. Temperature at First Rotor Inlet (Air)	317
Figure A6.1.2. Normalized Specific Work vs. Temperature at First Rotor Inlet (Air)	318
Figure A6.1.3. Rotor Work Fraction of Combined Work Total vs. Temperature at First Rotor Inlet (Air)	319
Figure A6.1.4. Absolute Flow Angle at First Rotor Inlet vs. Temperature at First Rotor Inlet (Air).....	320
Figure A6.1.5. Absolute Critical Mach Number at First Rotor Inlet vs. Temperature at First Rotor Inlet (Air)	320
Figure A6.1.6. Hub/Tip Ratio at First Rotor Inlet vs. Temperature at First Rotor Inlet (Air)....	320
Figure A6.1.7. Blade Turning Angle vs. Temperature at First Rotor Inlet (Air).....	321
Figure A6.1.8. Tip Speed Ratio vs. Temperature at First Rotor Inlet (Air).....	322
Figure A6.1.9. Flow Coefficient vs. Temperature at First Rotor Inlet (Air)	323
Figure A6.1.10. Blade Loading Coefficient vs. Temperature at First Rotor Inlet (Air)	324
Figure A6.1.11 Specific Speed vs. Temperature at First Rotor Inlet (Air).....	325
Figure A6.1.12 Total Pressure Ratio vs. Temperature at First Rotor Inlet (Air).....	326
Figure A6.2.1. Specific Work vs. Temperature at First Rotor Inlet (NCG Mix).....	328

Figure A6.2.2. Normalized Specific Work vs. Temperature at First Rotor Inlet (NCG Mix)....	329
Figure A6.2.3. Rotor Work Fraction of Combined Work Total vs. Temperature at First Rotor Inlet (NCG Mix)	330
Figure A6.2.4. Absolute Flow Angle at First Rotor Inlet vs. Temperature at First Rotor Inlet (NCG Mix).....	331
Figure A6.2.5. Absolute Critical Mach Number at First Rotor Inlet vs. Temperature at First Rotor Inlet (NCG Mix)	331
Figure A6.2.6. Hub/Tip Ratio at First Rotor Inlet vs. Temperature at First Rotor Inlet (NCG Mix)	331
Figure A6.2.7. Blade Turning vs. Temperature at First Rotor Inlet (NCG Mix).....	332
Figure A6.2.8. Tip Speed Ratio vs. Temperature at First Rotor Inlet (NCG Mix).....	333
Figure A6.2.9. Flow Coefficient vs. Temperature at First Rotor Inlet (NCG Mix).....	334
Figure A6.2.10. Blade Loading Coefficient vs. Temperature at First Rotor Inlet (NCG Mix) ..	335
Figure A6.2.11. Specific Speed vs. Temperature at First Rotor Inlet (NCG Mix).....	336
Figure A6.2.12. Total Pressure vs. Temperature at First Rotor Inlet (NCG Mix).....	337
Figure A6.3.1. Specific Work vs. Temperature at First Rotor Inlet (CH ₄).....	338
Figure A6.3.2. Normalized Specific Work vs. Temperature at First Rotor Inlet (CH ₄).....	339
Figure A6.3.3. Rotor Work Fraction of Combined Work Total vs. Temperature at First Rotor Inlet (CH ₄).....	340
Figure A6.3.4. Absolute Flow Angle at First Rotor Inlet vs. Temperature at First Rotor Inlet (CH ₄).....	341
Figure A6.3.5. Absolute Critical Mach Number at First Rotor Inlet vs. Temperature at First Rotor Inlet (CH ₄).....	341
Figure A6.3.6. Hub/Tip Ratio at First Rotor Inlet vs. Temperature at First Rotor Inlet (CH ₄) ..	341
Figure A6.3.7. Blade Turning vs. Temperature at First Rotor Inlet (CH ₄).....	342
Figure A6.3.8. Tip Speed Ratio vs. Temperature at First Rotor Inlet (CH ₄)	343
Figure A6.3.9. Flow Coefficient vs. Temperature at First Rotor Inlet (CH ₄).....	344
Figure A6.3.10. Blade Loading Coefficient vs. Temperature at First Rotor Inlet (CH ₄)	345

Figure A6.3.11. Specific Speed vs. Temperature at First Rotor Inlet (CH_4)	346
Figure A6.3.12. Total Pressure Ratio vs. Temperature at First Rotor Inlet (CH_4)	347
Figure A6.4.1. Specific Work vs. Temperature at First Rotor Inlet (CO_2).....	348
Figure A6.4.2. Normalized Specific Work vs. Temperature at First Rotor Inlet (CO_2).....	349
Figure A6.4.3. Rotor Work Fraction of Combined Work Total vs. Temperature at First Rotor Inlet (CO_2).....	350
Figure A6.4.4. Absolute Flow Angle at First Rotor Inlet vs. Temperature at First Rotor Inlet (CO_2).....	351
Figure A6.4.5. Absolute Critical Mach Number at First Rotor Inlet vs. Temperature at First Rotor Inlet (CO_2).....	351
Figure A6.4.6. Hub/Tip Ratio at First Rotor Inlet vs. Temperature at First Rotor Inlet (CO_2) ..	351
Figure A6.4.7. Blade Turning vs. Temperature at First Rotor Inlet (CO_2).....	352
Figure A6.4.8. Tip Speed Ratio vs. Temperature at First Rotor Inlet (CO_2).....	353
Figure A6.4.9. Flow Coefficient vs. Temperature at First Rotor Inlet (CO_2).....	354
Figure A6.4.10. Blade Loading Coefficient vs. Temperature at First Rotor Inlet (CO_2)	355
Figure A6.4.11. Specific Speed vs. Temperature at First Rotor Inlet (CO_2)	356
Figure A6.4.12. Total Pressure Ratio vs. Temperature at First Rotor Inlet (CO_2)	357
Figure A6.5.1. Specific Work vs. Temperature at First Rotor Inlet (H_2)	358
Figure A6.5.2. Normalized Specific Work vs. Temperature at First Rotor Inlet (H_2).....	359
Figure A6.5.3. Rotor Work Fraction of Combined Work Total vs. Temperature at First Rotor Inlet (H_2)	360
Figure A6.5.4. Absolute Flow Angle at First Rotor Inlet vs. Temperature at First Rotor Inlet (H_2)	361
Figure A6.5.5. Absolute Critical Mach Number at First Rotor Inlet vs. Temperature at First Rotor Inlet (H_2)	361
Figure A6.5.6. Hub/Tip Ratio at First Rotor Inlet vs. Temperature at First Rotor Inlet (H_2).....	361
Figure A6.5.7. Blade Turning vs. Temperature at First Rotor Inlet (H_2)	362
Figure A6.5.8. Tip Speed Ratio vs. Temperature at First Rotor Inlet (H_2).....	363

Figure A6.5.9. Flow Coefficient vs. Temperature at First Rotor Inlet (H_2)	364
Figure A6.5.10. Blade Loading Coefficient vs. Temperature at First Rotor Inlet (H_2)	365
Figure A6.5.11. Specific Speed vs. Temperature at First Rotor Inlet (H_2)	366
Figure A6.5.12. Total Pressure Ratio vs. Temperature at First Rotor Inlet (H_2)	367
Figure A6.6.1. Specific Work vs. Temperature at First Rotor Inlet (H_2O/CO_2)	368
Figure A6.6.2. Normalized Specific Work vs. Temperature at First Rotor Inlet (H_2O/CO_2)	369
Figure A6.6.3. Rotor Work Fraction of Combined Work Total vs. Temperature at First Rotor Inlet (H_2O/CO_2)	370
Figure A6.6.4. Absolute Flow Angle at First Rotor Inlet vs. Temperature at First Rotor Inlet (H_2O/CO_2)	371
Figure A6.6.5. Absolute Critical Mach Number at First Rotor Inlet vs. Temperature at First Rotor Inlet (H_2O/CO_2)	371
Figure A6.6.6. Hub/Tip Ratio at First Rotor Inlet vs. Temperature at First Rotor Inlet (H_2O/CO_2)	371
Figure A6.6.7. Blade Turning vs. Temperature at First Rotor Inlet (H_2O/CO_2)	372
Figure A6.6.8. Tip Speed Ratio vs. Temperature at First Rotor Inlet (H_2O/CO_2)	373
Figure A6.6.9. Flow Coefficient vs. Temperature at First Rotor Inlet (H_2O/CO_2)	374
Figure A6.6.10. Blade Loading Coefficient vs. Temperature at First Rotor Inlet (H_2O/CO_2) ...	375
Figure A6.6.11. Specific Speed vs. Temperature at First Rotor Inlet (H_2O/CO_2)	376
Figure A6.6.12. Total Pressure Ratio vs. Temperature at First Rotor Inlet (H_2O/CO_2)	377
Figure A6.7.1. Specific Work vs. Temperature at First Rotor Inlet (He)	378
Figure A6.7.2. Normalized Specific Work vs. Temperature at First Rotor Inlet (He)	379
Figure A6.7.3. Rotor Work Fraction of Combined Work Total vs. Temperature at First Rotor Inlet (He)	380
Figure A6.7.4. Absolute Flow Angle at First Rotor Inlet vs. Temperature at First Rotor Inlet (He)	381
Figure A6.7.5. Absolute Critical Mach Number at First Rotor Inlet vs. Temperature at First Rotor Inlet (He)	381

Figure A6.7.6. Hub/Tip Ratio at First Rotor Inlet vs. Temperature at First Rotor Inlet (He)	381
Figure A6.7.7. Blade Turning vs. Temperature at First Rotor Inlet (He)	382
Figure A6.7.8. Tip Speed Ratio vs. Temperature at First Rotor Inlet (He)	383
Figure A6.7.9. Flow Coefficient vs. Temperature at First Rotor Inlet (He)	384
Figure A6.7.10. Blade Loading Coefficient vs. Temperature at First Rotor Inlet (He).....	385
Figure A6.7.11. Specific Speed vs. Temperature at First Rotor Inlet (He)	386
Figure A6.7.12. Total Pressure Ratio vs. Temperature at First Rotor Inlet (He).....	387

KEY TO SYMBOLS AND ABBREVIATIONS

Due to the difficulty of avoiding more than one use for many of the symbols employed herein, a detailed description of each symbol is also provided throughout this document when it is first used and with clarified context.

ABBREVIATIONS

0/1/2/3D	Zero-/one-/two-/three-dimensional
ABS	Acrylonitrile Butadiene Styrene
ASHRAE	American Society of Heating, Refrigerating, and Air-Conditioning Engineers
ATM	Automatic Topology and Meshing
CAD	Computer-Aided Design
CFC	Chlorofluorocarbon
CFD	Computational Fluid Dynamics
CNC	Computerized Numerical Controlled
<i>COP</i>	Coefficient of Performance
CVRC	Composite Vehicle Research Center
DO	Dissolved Oxygen
FVB	Free-Vortex Blade
FVM	Finite Volume Method
GPP	Geothermal Power Plant
GWP	Global Warming Potential
HCFC	Hydrochlorofluorocarbon
HFF	Hollow Fine Fiber

HIPS	High Impact Polystyrene
IGV	Inlet Guide Vane
LRVP	Liquid Ring Vacuum Pump
MSF	Multistage Flash (distillation)
MSU	Michigan State University
MVC	Mechanical Vapor Compression
NCG	Non-Condensable Gases
ODP	Ozone Depletion Potential
PLA	Polylactic Acid
PVC	Polyvinyl Chloride
SJE	Steam Jet Ejector
SP	Star Pattern
SST	Shear Stress Transport
R134-a	ASHRAE designation for Tetrafluoroethane
R718	ASHRAE designation for water
RANS	Reynolds Averaged Navier-Stokes (equations)
RMS	Root Mean Square
RO	Reverse Osmosis
VCD	Vapor Compression Distillation
VFD	Variable Frequency Drive

LATIN SYMBOLS

A	Area, cross-sectional area, mandrel slot designation
$\tilde{A}$	General square matrix
a	Speed of sound

a_i^{nb}	Coefficient (scalar or matrix) in discrete set of conservation equations
AR	Aspect ratio
B	Mandrel slot designation
$\bar{B}$	General unknown vector
C	Mandrel slot designation
$\bar{C}$	General known vector
$\mathbf{c}$	Absolute fluid velocity vector
c	Absolute fluid speed, turbulence model constant
ch	Chord length
c_p	Specific heat capacity at constant pressure
c_v	Specific heat capacity at constant volume
$CD_{k\omega}$	Cross diffusion term in SST
$C_{\varepsilon 1}$	k - ε model constant
$C_{\varepsilon 2}$	k - ε model constant
C_μ	k - ε model constant
D	Mandrel slot designation
DF	Diffusion factor
dH	de Haller number
e	Energy, specific internal energy
$\tilde{e}$	Specific shaft work
$\hat{e}$	Orthogonal coordinate unit vector
$\mathbf{F}$	General field quantity
$\mathbf{F}$	Force vector
$\mathbf{f}$	Mass specific force vector
F_1	SST blending function

f_μ	Proportionality constant
$\mathbf{H}$	Moment of momentum vector
h	Enthalpy
I	Turbulence intensity
k	Turbulent kinetic energy
L	Characteristic length scale
l_m	Mixing length
l_t	Turbulent length scale
$\mathbf{M}$	Moment vector
M	Mach number, magnitude of moment vector
m	Line slope (of fiber)
M'	Normalized meridional coordinate
$\dot{m}$	Mass flow rate
mf_i	Mass fraction of mixture component i
MW	Molecular weight
$\mathbf{n}$	Outward pointing unit normal vector
n	Rotational speed in revolutions per second
N_b	Number of blades
N_i	Shape function at node i
N_s	Number of stages
$\tilde{P}$	Shaft power
p	Pressure
p'	Modified pressure
P_k	Turbulence production
q	Heat

$\dot{q}$	Heat generation
$\dot{Q}_{in}$	Heat absorbed, cooling capacity
$\dot{Q}_{out}$	Heat rejection, heat loss
R	Gas constant, rotor-stator stage reaction
$\mathbf{r}$	Radial position vector
r	Radius
r_b	Blade radius
Re	Reynolds number
r_m	Area-mean radius
r^n	Numerical residual of FVM
$\bar{R}_u$	Universal gas constant
$\mathbf{S}$	Surface vector
S	Surface area, strain rate
s	Entropy, local coordinate in tri-linear shape function
S_M	Momentum source
T	Temperature
t	Time, blade stagger, local coordinate in tri-linear shape function
t_b	Blade thickness
u	Tangential velocity, velocity vector component, local coordinate in tri-linear shape function
U_t	Turbulent velocity scale
$\mathbf{V}$	Velocity vector
V	Velocity
$\mathcal{V}$	Volume
$\dot{\mathcal{V}}$	Volume flow rate

$\mathbf{v}$	Velocity vector component
$\mathbf{w}$	Relative velocity vector
w	work, specific work, relative fluid speed, velocity vector magnitude
w_1	Geometric length used to determine blade radius
X	Global Cartesian coordinate
x	Cartesian coordinate
x_1	Geometric length used to determine blade radius
$\dot{W}_c$	Work input, compressor work
Y	Global Cartesian coordinate
y	Geometric length used to determine blade radius, Cartesian coordinate
y_i	Mole fraction of mixture component i
Z	Global Cartesian coordinate
z	Axial direction, axial length, Cartesian coordinate

GREEK SYMBOLS

α	Specific heat polynomial coefficient, IGV blade angle, absolute flow angle, $k-\omega$ model constant
β	Specific heat polynomial coefficient, blade angle, relative flow angle. $k-\omega$ model constant
β'	$k-\omega$ turbulence model constant
Γ	Circulation
γ	Specific heat polynomial coefficient, specific heat ratio
δ	Specific heat polynomial coefficient, identity matrix, boundary layer thickness, deviation angle
δ_{ij}	Kronecker delta function
ε	Specific heat polynomial coefficient, turbulent energy dissipation rate

η	Efficiency
θ	Angle between velocity and surface normal vectors, global cylindrical coordinate
ι	Incidence angle
$\mathcal{A}$	Linear distance between star-pattern impeller blades
λ	Stokes hypothesis variable
μ	Molecular (dynamic) viscosity
μ_{eff}	Effective viscosity
μ_t	Turbulent viscosity
ν	Specific volume
Π	Pressure ratio
ρ	Density
σ	Blade solidity, specific speed
σ_k	k - ε turbulence model constant
σ_ε	k - ε turbulence model constant
σ_ω	k - ω turbulence model constant
τ	Stress
φ	General variable
ϕ	Camber angle, flow coefficient
ϕ_1	Represents k - ω turbulence model in SST
ϕ_2	Represents k - ε turbulence model in SST
ϕ_3	SST equations
Φ_b	Angular space between blade slots
ψ	Planar angle of mandrel slot width measured from rotor center referencing global coordinates, blade loading coefficient
ω	Vorticity vector
ω	Angular velocity, turbulence frequency, vorticity magnitude

SUBSCRIPTS AND SUPERSSCRIPTS

*	Reference to critical (sonic) condition
$\cdot$	Denotes time rate of change
$-$	Denotes time-average quantity
$\wedge$	Denotes unit vector
$'$	Denotes modified or corrected variable
$\sim$	References shaft quantity
0	Refers to previous time level
1	Denotes property at state point 1, property in rotor 1
2	Denotes property at state point 2, property in rotor 2
<i>adj</i>	Denotes adjusted quantity
<i>avg</i>	Denotes average quantity
<i>ax</i>	Denotes axial direction component
<i>b</i>	References blade
<i>c</i>	Denotes property corresponding to compressor, inertial reference frame
<i>h</i>	Denotes hub location
<i>in</i>	References component inlet
<i>ip</i>	References integration point
<i>is</i>	Denotes isentropic process or efficiency
<i>k</i>	Denotes quantity related to turbulent kinetic energy
<i>m</i>	Denotes meridional direction component
<i>mix</i>	Denotes property for fluid mixture
<i>nb</i>	References neighboring element or node
<i>norm</i>	Denotes normalized quantity
<i>out</i>	References component outlet

pt	Denotes polytropic process or efficiency
r	Denotes radial direction component
$ring$	Indicates discretized ring segment
rot	Reference to rotor
$crot$	Reference to counter-rotor
s	Denotes isentropic process
sat	Denotes saturated vapor condition
t	Denotes tip location, indicates total (stagnation) property, denotes turbulent quantity
u	Denotes tangential direction component
w	With respect to relative reference frame
z	Denotes axial direction component
ε	Denotes quantity related to turbulence dissipation rate
θ	Denotes tangential direction component
ω	Denotes quantity related to turbulence frequency

CHAPTER 1 INTRODUCTION AND APPLICATIONS

1.1 Background

The original impetus leading to the development of the novel patented woven impeller technology was using water (R718) as a refrigerant in a vapor compression refrigeration cycle [1]. The selection of a refrigerant for a given application is typically based on its performance, safety, and environmental impact. From a safety standpoint, the flammability and toxicity of refrigerants used in the early 20th century lead to the development and widespread use of highly molecularly stable chlorofluorocarbon (CFC) and hydrochlorofluorocarbon (HCFC) refrigerants, largely known by their trade name: “Freons”. After decades of increased use as a result of worldwide economical and technical development, the global scientific community realized that these chlorine-containing refrigerants were having extremely detrimental effects on the ozone layer and contributing to climate change. CFCs and HCFCs, initially highly sought after for their stability, were also causing long term harmful environmental impact; this has resulted in their gradual out-phasing [2].

This then lead to the creation and implementation of refrigerants lacking the damaging chlorine molecule (HFCs, one of the most common being R134-a), which do not contribute to ozone depletion, but still contribute to global climate change [1], [2], [3]. The search for cost-effective and completely environmentally benign refrigerants has lead to increased attention and investigation being directed toward natural refrigerants.

Water is an environmentally friendly and abundantly available substance that is stable, non-flammable, non-toxic, and has zero stratospheric ozone depletion potential (ODP) and zero global warming potential (GWP) [1], [3]. It is also has no storage or disposal issues. These characteristics, combined with superior heat transfer properties [1], [3], make it an ideal

candidate for widespread use in refrigeration applications. However the initial stumbling block for using water as a refrigerant is the fact that water vapor at atmospheric pressure is not what is generally considered to be cool.

1.2 Vapor Compression Refrigeration Cycle

The refrigeration of an enclosed area is achieved by the continuous cycling of a refrigerant through an evaporator, compressor, condenser, and expansion valve as shown in Figure 1.1.

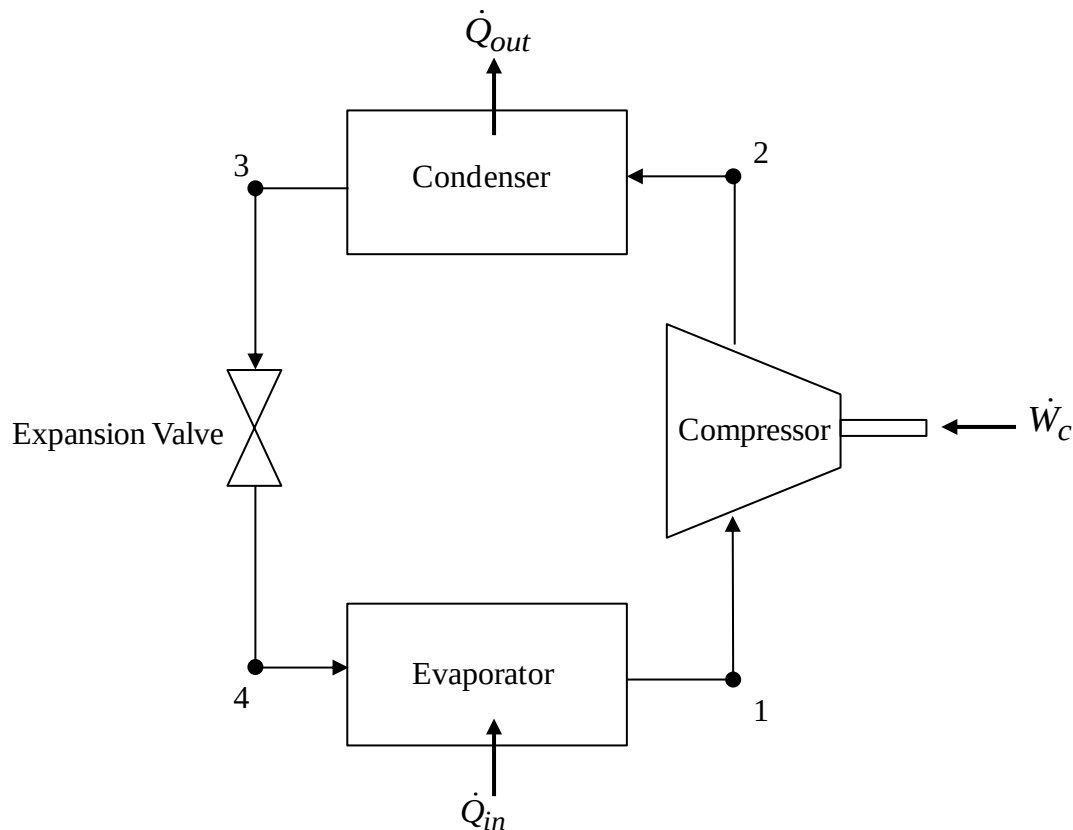


Figure 1.1. Vapor Compression Refrigeration Cycle Schematic

In the ideal cycle, mixed phase (liquid/vapor) refrigerant is passed through the evaporator (4 to 1) in a heat exchange process that draws heat away from the space to be cooled as it evaporates at constant temperature and pressure. This heat added to the refrigerant serves to evaporate it to its saturation point at the evaporator operating pressure. The saturated vapor then

undergoes an isentropic compression process (1 to 2s) requiring a work input to bring it up to the condenser pressure while simultaneously increasing the temperature. The superheated vapor then passes through the condenser (2 to 3) which rejects heat to the warm, generally ambient, region, bringing the refrigerant to a saturated liquid at constant temperature and pressure. The refrigerant then passes through an expansion valve in a throttling process (3 to 4) that reduces the temperature and pressure of the refrigerant bringing it back to the two-phase mixture, at which point the cycle repeats itself.

The preceding paragraph describes the ideal cycle; in reality both the evaporator and condenser will incur pressure losses, and the compression process will not be isentropic (1 to 2). Additionally, the evaporator temperature will be lower than the resulting temperature of the enclosed cold space, and the condenser will operate at a temperature higher than the warm space. The ideal generic cycle, with the exception of the compression process, is shown in Figure 1.2 on a temperature-entropy (T - s) diagram and in Figure 1.3 on a pressure-enthalpy (p - h) diagram.

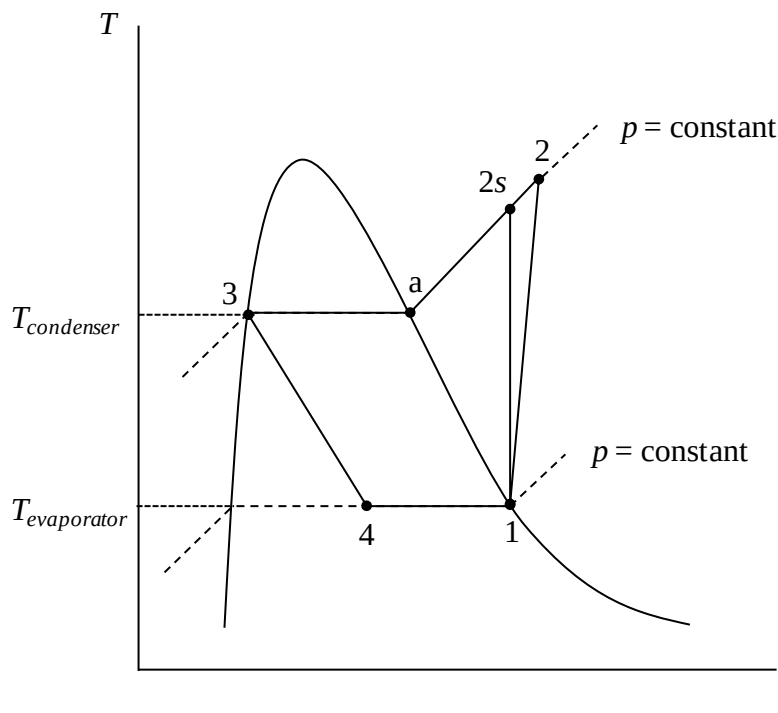


Figure 1.2. T - s Diagram of Ideal Generic Vapor Compression Refrigeration Cycle

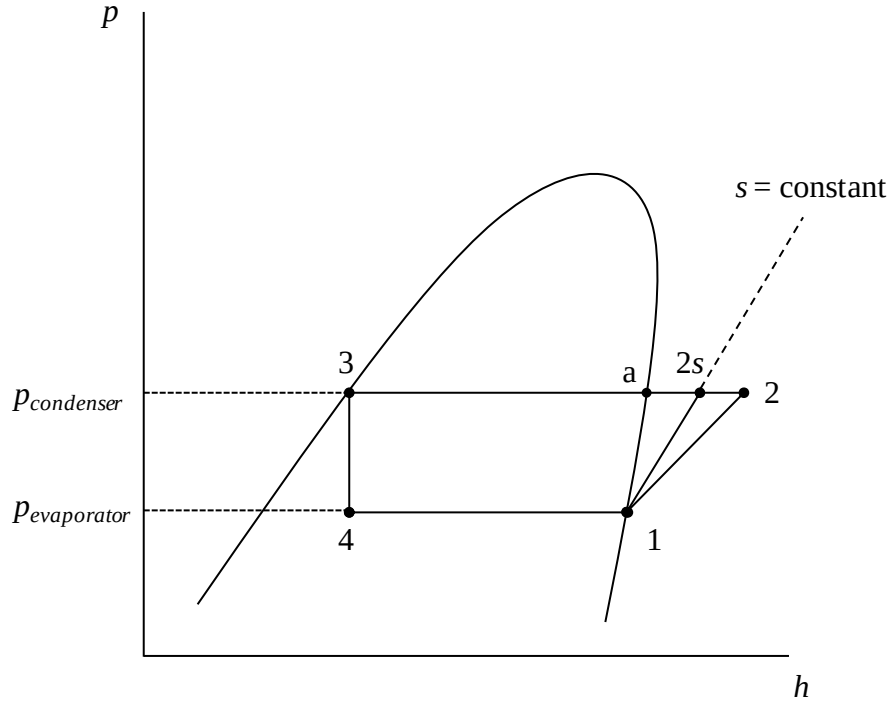


Figure 1.3. p - h Diagram of Ideal Generic Vapor Compression Refrigeration Cycle

Drawing separate control volumes around both the evaporator and the compressor and performing an energy balance on each while neglecting changes in potential and kinetic energy, it can be shown that the heat absorbed by the refrigerant in the evaporator and the work input to the compressor are, respectively, simply the differences in enthalpy across each device:

$$\frac{\dot{Q}_{in}}{\dot{m}} = h_1 - h_4 \quad (1-1)$$

$$\frac{\dot{W}_c}{\dot{m}} = h_2 - h_1 \quad (1-2)$$

The heat absorbed by the evaporator in Equation (1-1), $\dot{Q}_{in}$, is also known as the cooling capacity achieved by the cycle, and $\dot{m}$ is the mass flow rate of refrigerant.

The coefficient of performance (COP) of the refrigeration cycle is defined as the ratio of the cooling capacity to the work input:

$$COP = \frac{\dot{Q}_{in} / \dot{m}}{\dot{W}_c / \dot{m}} = \frac{h_1 - h_4}{h_2 - h_1} \quad (1-3)$$

The pressure ratio needed to be achieved by the compressor is the ratio of the condenser pressure to the evaporator pressure:

$$\Pi = \frac{p_2}{p_1} \quad (1-4)$$

1.3 Utilizing Water (R718) as a Refrigerant

With the ultimate goal of cooling an enclosed space, the evaporation temperature of the refrigerant used must be at or below the desired temperature-to-be of the enclosed region. At atmospheric pressure, water evaporates at 100°C. Therefore, in order to use water as a refrigerant, the pressures involved in the refrigeration cycle must be substantially lower than ambient.

In order to achieve appreciable cooling, the mass flow rate of R718 refrigerant must still be considerable. However since it operates under coarse vacuum pressure, the density is accordingly low, and therefore the volume flow rate required is substantially higher than would otherwise be necessary to meet the mass flow requirement. Additionally, even though pressure differences are very low under coarse vacuum pressure, the pressure ratio is very high; these two considerations lead naturally to the use of turbomachinery, in particular a high speed axial compressor, to achieve the required compression and volume flow.

The properties of water vapor are also such that a relatively high specific work input is required for a given pressure ratio, thereby requiring high compressor tip speeds [3], [4]. However these same properties also comfortably allow high tip speeds since the speed of sound

in water vapor is also relatively large[3], [4]; it is generally desirable to have the flow remain subsonic relative to the impeller to avoid choking and shock wave generation [3], [4].

1.3.1 Comparative Case Study: R718 vs. R134-a

To serve as an illustrative example of the performance benefits of using R718, a simple ideal refrigeration cycle analysis has been carried out comparing water vapor to the widely used refrigerant R134-a ($\text{CF}_3\text{CH}_2\text{F}$). The situation at hand is cooling an enclosed space to 75°F from an exterior temperature of 100°F, which is on the extreme end of cooling requirements encountered for air conditioners throughout the United States and much of the world during the height of summer. For the ideal cycle, the following assumptions have been made:

Table 1.1. Ideal Vapor Compression Refrigeration Cycle Assumptions

Device	Assumption
Evaporator	Isobaric, isothermal
Compressor	Isentropic
Condenser	Isobaric
Expansion Valve	Isenthalpic

In addition to Table 1.1, it is also assumed that the refrigerant entering the compressor is a saturated vapor, and the refrigerant exiting the condenser is a saturated liquid.

Tables 1.2 and 1.3 show the properties at each of the state points in the cycle for R134-a and R718, respectively. Note that state point *a* is the state of the saturated vapor in the condenser, and that the *s* associated with state point 2 signifies an isentropic compression process.

Table 1.2. Ideal Refrigeration Cycle Properties for R134-a in Case Study [2]

R134-a	<i>p</i>		<i>T</i>			<i>h</i>	<i>s</i>	<i>v</i>	<i>ρ</i>
State Point	(bar)	(kPa)	(°F)	(°C)	(K)	(kJ/kg)	(kJ/kg/K)	(m ³ /kg)	(kg/m ³)
1	6.44	643.60	75.0	23.9	297.0	260.39	0.9089	0.0318	31.433
2s	9.57	957.28	102.0	38.9	312.0	268.50	0.9089	0.0214	46.772
<i>a</i>	9.57	957.28	100.0	37.8	310.9	267.23	0.9048	0.0211	47.294
3	9.57	957.28	100.0	37.8	310.9	102.88	0.3762	0.0086	115.684
4	6.44	643.60	75.0	23.9	297.0	102.88	0.3786	0.0109	91.534

Table 1.3 Ideal Refrigeration Cycle Properties for R718 in Case Study [2]

R718	p		T			h	s	v	ρ
State Point	(bar)	(kPa)	(°F)	(°C)	(K)	(kJ/kg)	(kJ/kg/K)	(m ³ /kg)	(kg/m ³)
1	0.03	2.97	75.0	23.9	297.0	2545.19	8.5818	46.1820	0.022
2s	0.07	6.56	182.1	83.4	356.5	2656.43	8.5818	26.9572	0.037
<i>a</i>	0.07	6.56	100.0	37.8	310.9	2570.30	8.2993	21.8618	0.046
3	0.07	6.56	100.0	37.8	310.9	158.28	0.5428	0.0010	993.038
4	0.03	2.97	75.0	23.9	297.0	158.28	0.5472	1.0909	0.917

Information contained in the above tables is displayed graphically in the following figures on both T - s and p - h diagrams for each refrigerant. Portions of the vapor domes are also displayed in accordance with the scaling of the cycles themselves.

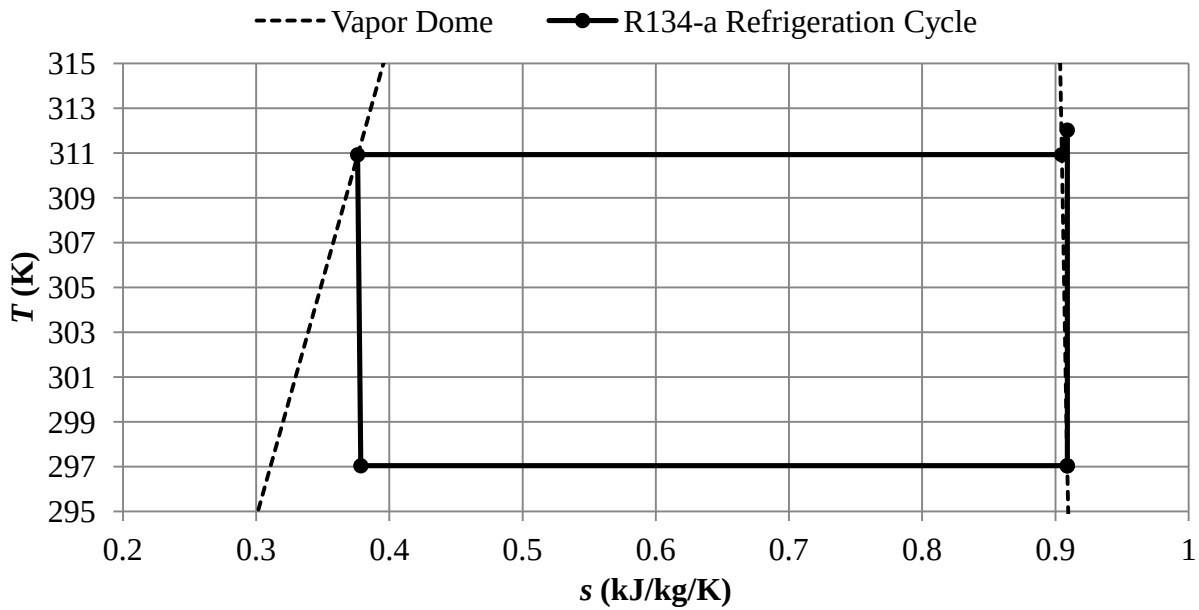


Figure 1.4. T - s Diagram for R134-a in Case Study

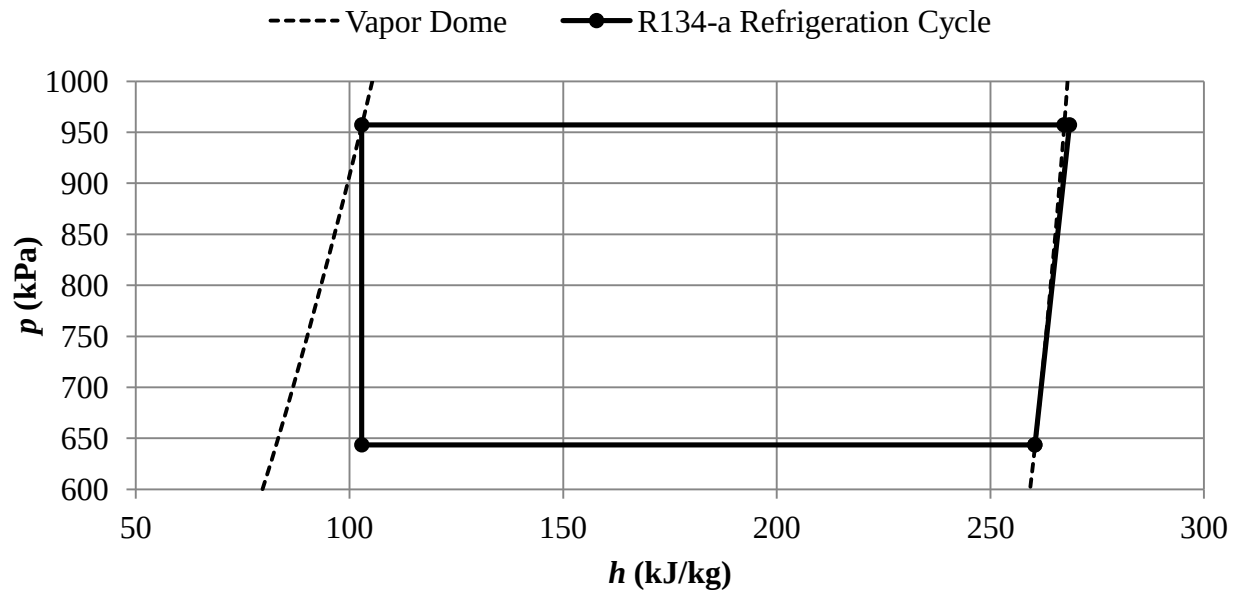


Figure 1.5. p - h Diagram for R134-a in Case Study

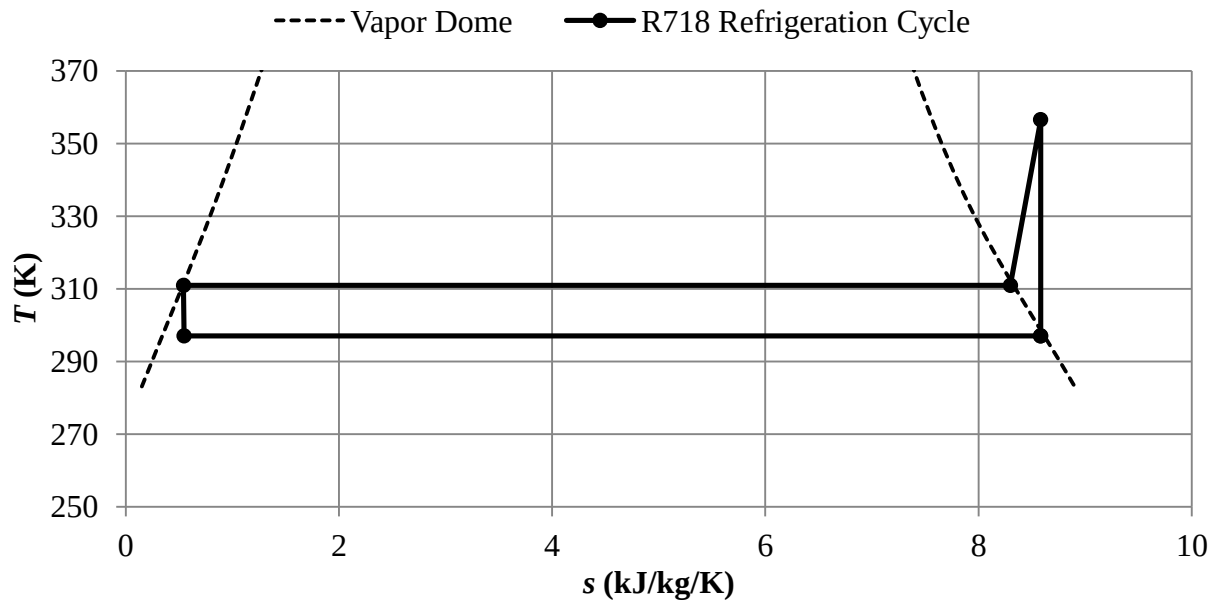


Figure 1.6. T - s Diagram for R718 in Case Study

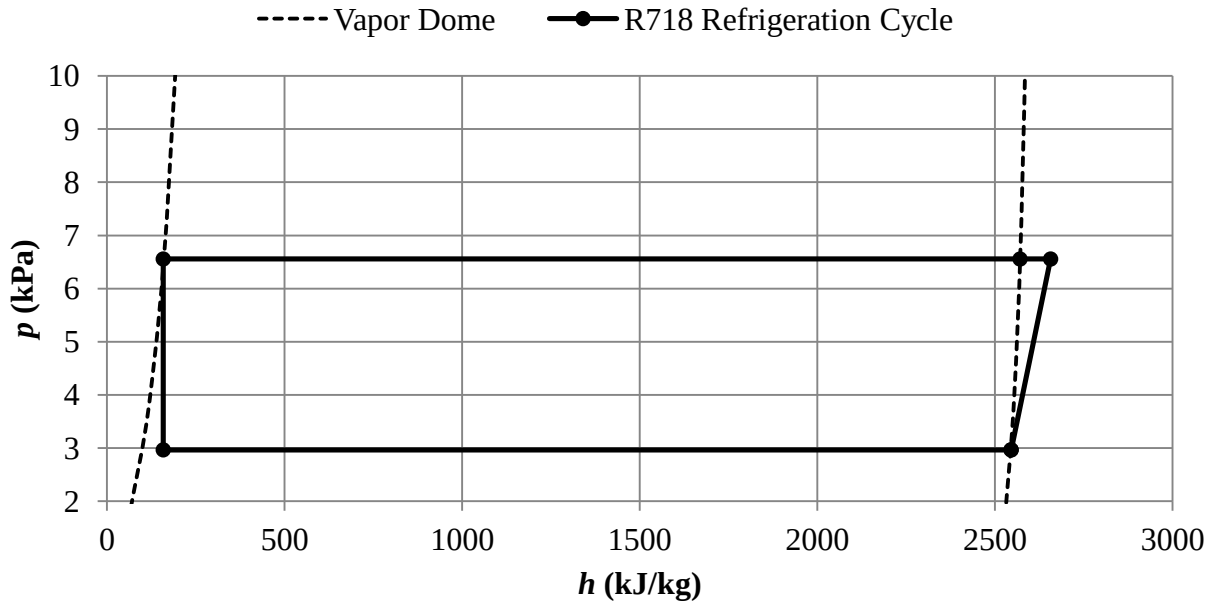


Figure 1.7. p - h Diagram for R718 in Case Study

Table 1.4 summarizes and compares much of the pertinent data presented in Tables 1.2 and 1.3.

Table 1.4. Performance Comparison of R134-a and R718 in Case Study				
		R134-a	R718	Ratio: R718/R134-a
W_c	kJ/kg	8.11	111.24	13.72
Q_{in}	kJ/kg	157.51	2386.91	15.15
Π	-	1.49	2.21	1.49
COP	-	19.43	21.46	1.10

As can be seen in Table 1.4, the cooling capacity of R718 is over 15 times that of R134-a in this case study. Although the specific work required by the compressor is over 13 times larger for water, the COP is still larger than that of R134-a. According to Natural Resources Canada [5], it takes an approximate average of 25btu/h per square foot (78.86W/m^2) to maintain a room at 24°C . It is therefore possible to determine the mass flow rate of refrigerant required to achieve this on a per-room-area basis since the cooling capacity is known as per Equation (1-1). Since the cycle is closed, this mass flow rate is the same for each device, including the compressor, for an individual refrigerant. The amount of power consumed by the compressor can then be

determined from Equation (1-2). With the mass flows and densities known at each station in the cycle, the volume flow rate, $\dot{V}$, is also known. Again this is on a per-room-area basis. This data is summarized in Table 1.5.

Table 1.5. Room-Area-Specific Flow and Compressor Properties in Ideal Case Study

		R134-a	R718	Ratio: R718/R134-a
$\dot{m}$	kg/s/m ²	500×10 ⁻⁶	30×10 ⁻⁶	0.066
$\dot{W}_c$	kW/m ²	0.0041	0.0037	0.905
$\rho_{c,avg}$	kg/m ³	39.102	0.029	0.0008
$\dot{V}_{c,avg}$	m ³ /s/m ²	13×10 ⁻⁶	0.0011	87.84

The entries labeled with the subscript *avg* are averaged properties between in inlet and outlet of the compressor (between state points 1 and 2). The telling column in Table 1.5 is the Ratio column: the R718 compressor uses almost 10% less power than the R134-a compressor while pushing nearly 90 times more volume flow to achieve the same cooling on a per-area basis. The lower compressor power requirement should of course be expected due to the larger *COP* of R718.

COP is always greater for R718 even when a non-isentropic compression process is undertaken. The isentropic efficiency of a compressor is defined as

$$\eta_{is,c} = \frac{h_{2s} - h_1}{h_2 - h_1} \quad (1-5)$$

which can be seen in Figures 1.2 and 1.3. Figure 1.8 shows plots of *COP* in the nearly-ideal cycle as a function of isentropic compressor efficiency.

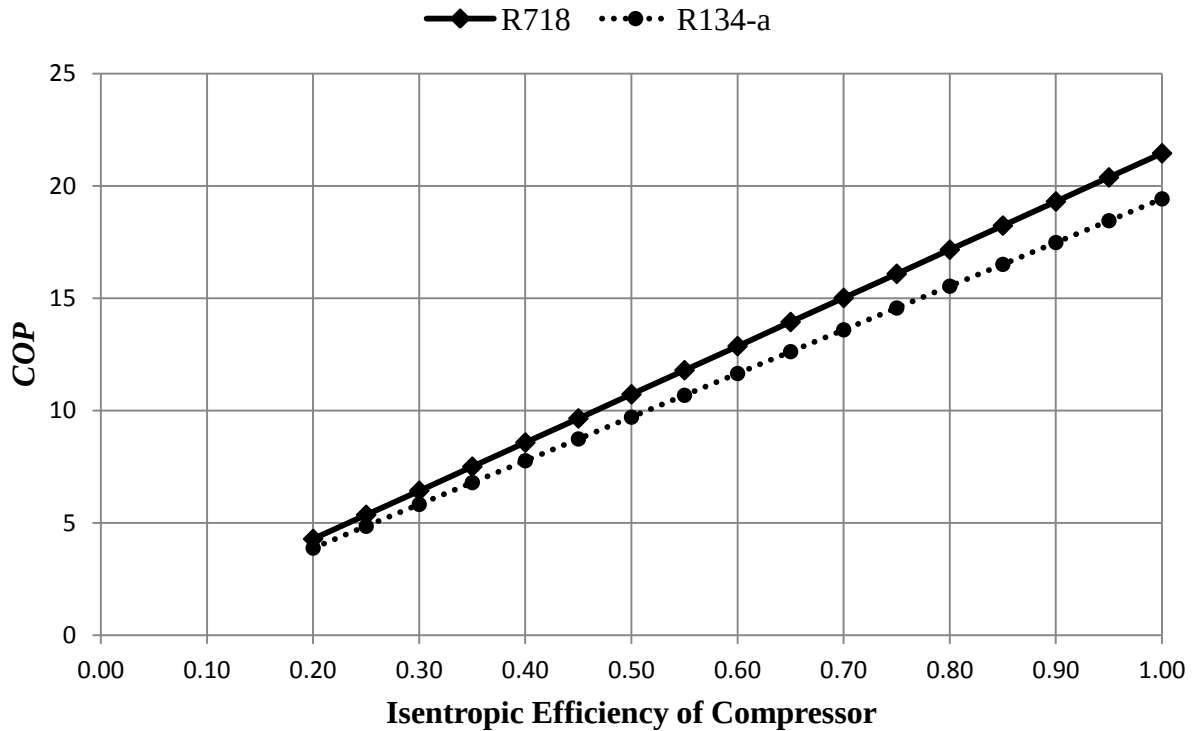


Figure 1.8. Comparison of COP for R718 and R134-a for different Compressor Efficiencies

For all isentropic compressor efficiencies shown in Figure 1.8, the COP for R718 is 1.1045 times that of the COP for R134-a. This implies the benefit of using R718 is large because the R718 cycle is intended for use with high efficiency axial compressors, thereby yielding even higher energy savings over an R134-a cycle that would normally use a lower efficiency centrifugal compressor [3], [4].

1.4 Woven Wheel Development

It is well documented that R718 generally out-performs most other common refrigerants in most major categories (cooling capacity, compressor power requirements, COP , etc) [3], [4], however similar to the case study in the previous section, the large specific volume of R718 requires comparatively large volume flow rates. Large volume flow rates through the compressor

inherently come with the need for large through-flow area in the impeller, which generally imply high rotational and tip speeds [3], [4].

1.4.1 Counter-Rotating Axial Compressors

Much research has been undertaken in order to determine the feasibility of developing compressors capable of producing the necessary pressure ratios and volume flows for using R718 as a refrigerant [3], [4], [6], [7], [8]. Wight et al. found that a single stage radial R718 compressor would need to be on the order of 6m in diameter with a tip speed of approximately 670m/s (approximately 2130rpm) for a 1000 ton cooling capacity (approximately 3517kW, which in terms of the case study presented in Section 1.3.1 would cool a 155,000ft² room from 100°F to 75°F using R134-a, or a larger 172,000ft² room from 100°F to 75°F using R718), and that a two-stage centrifugal compressor could operate with tip speeds just under 490m/s. These high tip speeds involve very large centrifugal forces and hence tremendous stresses in the rotor, and it was ultimately suggested that a six- or seven-stage axial machine with smaller diameter and tip speeds be considered instead (however this was deemed prohibitively expensive and complex by the authors).

Axial compressors are lauded for their high efficiency and power density [9]. Traditional multi-stage axial compressors consist of stationary guide vanes before and after each rotor in order to recapture static pressure and keep fluid velocities and rotor tip speeds subsonic; traditional systems also have all rotors spinning in the same direction at the same rotational speed as they are usually mounted on the same shaft. However, spinning the rotors in opposite directions can eliminate all interior stators. Figure 1.9 schematically illustrates how utilizing counter-rotation can serve to considerably increase the amount of work transmitted from the

rotors to the fluid in the same amount of space (increased power density) utilized by a traditional compressor since stators do not perform work.

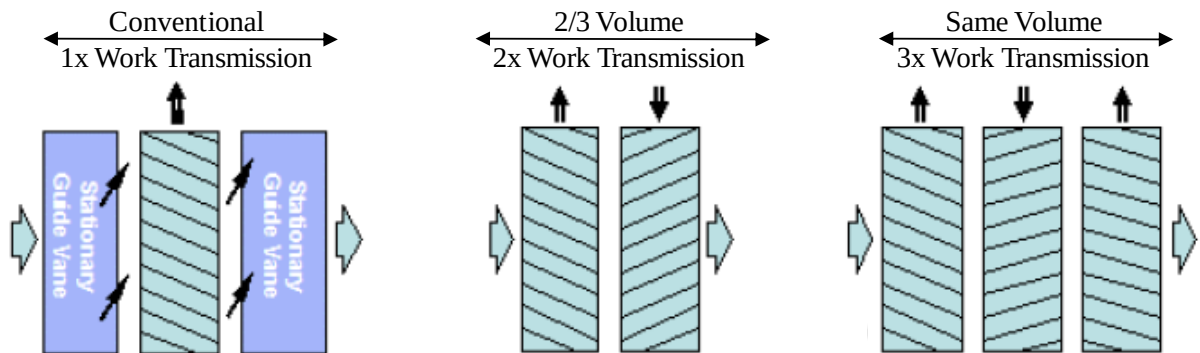


Figure 1.9. Comparison of Power Density for Axial Compressors

When comparing a counter-rotating axial compressor to a centrifugal compressor of the same capacity, the axial footprint is approximately the same, but the diameter requirement for a prescribed pressure gain reduces by a factor of 4 as shown in Figure 1.10. This implies reduced tip speeds and/or increased rotational speeds.

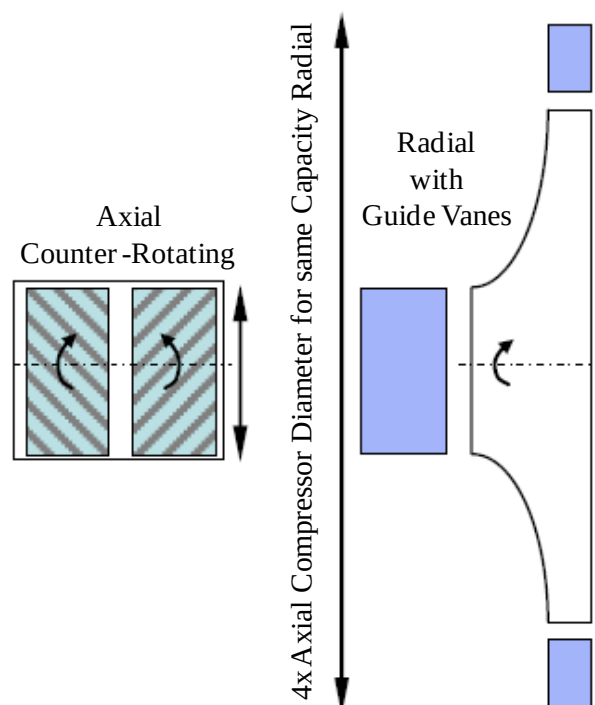


Figure 1.10. Comparison of Diameter for Equal Capacity Compressors

1.4.2 Original Woven Impeller

Since the volume flow and pressure gain requirements are both large for utilizing R718 in a vapor compression refrigeration cycle, the rotor tip speeds will inevitably be very large even for a counter-rotating axial compressor. Since R718 works under coarse vacuum, the fluid densities are very low (see Table 1.5) and so fluid forces on the impeller are not of major concern. However, low density inherently implies large specific volume and hence large volume flow rate, thus necessitating high rotational and tip speeds, which themselves lead to large centrifugal forces that create large stresses in the impeller. These stresses are large enough that special construction is required for the impeller [1], [3], [4].

Light-weight and high-strength materials are needed in order for the impeller to withstand the forces encountered during operation. This has led to the use of carbon fiber/epoxy matrix composites [10] for impeller fabrication, and also has the added advantage of low fabrication cost.

To utilize the full strength of the carbon fiber composite, the wheel is woven on a mandrel using a continuous fiber that also allows for the incorporation of an outer shroud in a novel, patented process owned by Müller and Michigan State University (MSU) [11]. All forces and stresses experienced by the impeller are therefore aligned (or nearly aligned) with the fiber direction (strongest) for both the blades (radial/centrifugal forces/tensile stress) and outer shroud (tangential forces/hoop stress/tensile stress) [12], [13].

The first generation mandrels are relatively simple (current generation mandrels will be discussed in Chapter 5) and are milled out of aluminum tubing using a CNC mill. They consist of any integer number of evenly-spaced slots that can be curved for aerodynamic enhancement. The impeller is woven by directing a fiber into a slot, through and across the interior of the mandrel,

and then exiting through a different slot at a specified number of openings away from the inlet slot. The fiber is then wound around the exterior of the mandrel for another specified amount of openings, which leads to the creation of the outer shroud, before repeating the process with a different set of slots. Repeating this process on a layer-by-layer basis in the direction of the rotational axis simultaneously creates the blades and the outer shroud. Figure 1.11 shows a candidate mandrel in the early days of impeller winding at MSU (circa 2009), while Figure 1.12 shows a mandrel with a partially wound wheel shortly after the onset of the winding process.

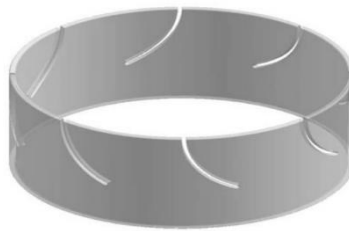


Figure 1.11. An 8-Slotted Mandrel

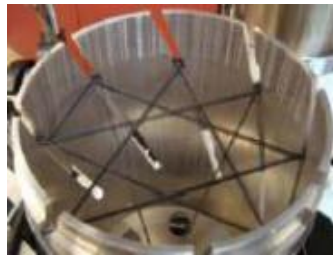
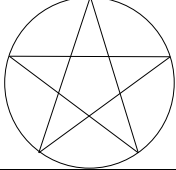
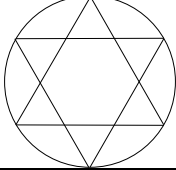
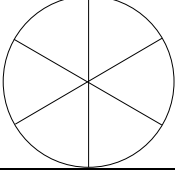
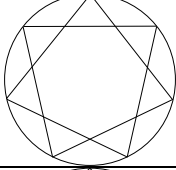
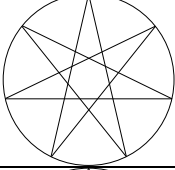
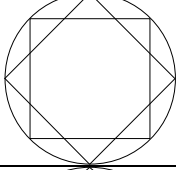
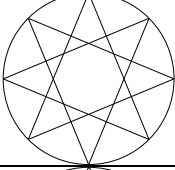
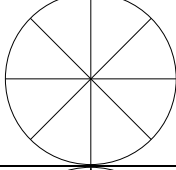
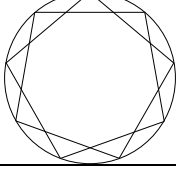
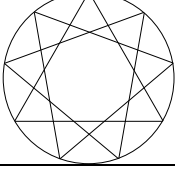
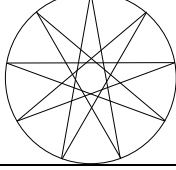


Figure 1.12. Mandrel with a Partially-Wound Impeller

Early investigations involved mandrels ranging between 5 and 9 slots. Different weaving patterns were also investigated [12]. Type “A” consists of skipping one slot across the interior of the mandrel, type “B” skips two, and type “C” skips three slots. These pattern combinations are summarized in Table 1.6.

Table 1.6. Winding Patterns for Continuous Fiber Impellers

Number of Mandrel Slots	Type A	Type B	Type C
5		---	---
6			---
7			---
8			
9			

Patterns 8B and 8C were selected for further analysis primarily due to ease-of- and repeatability-of-winding. Their geometries with differing hub radius to tip radius ratios (r_h/r_t) are shown in Figure 1.13. The analysis was undertaken by Li using the computational fluid dynamics (CFD) tool Fluent [12] to determine how the winding pattern and hub/tip ratio affect the compressor pressure ratio, isentropic efficiency (η_{is}) and volume flow rate. The results are summarized in Table 1.7.

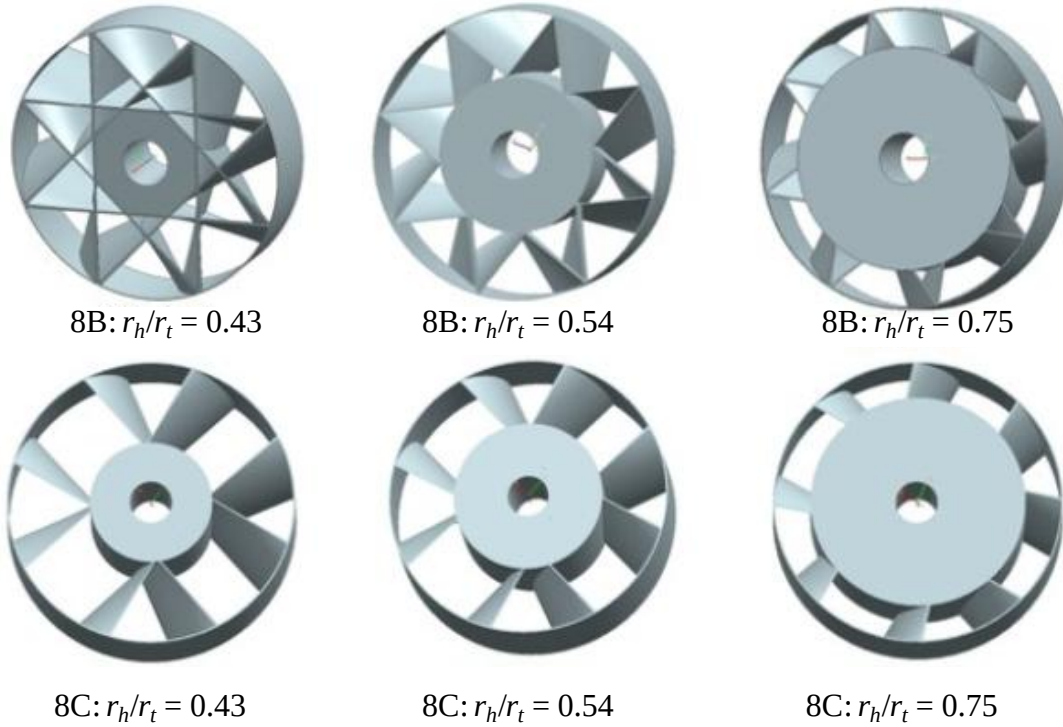


Figure 1.13. Geometries 8B and 8C with Differing Hub/Tip Ratios

Table 1.7. Performance Summary of Selected Impellers

Rotor	r_h/r_t	Π	η_{is}	Normalized Volume Flow
8B	0.75	1.162	68%	1.50
8B	0.54	1.121	65%	2.50
8B	0.43	1.107	64%	3.16
8C	0.75	1.163	73%	1.00
8C	0.54	1.124	65%	2.33
8C	0.43	1.115	66%	2.66

Pressure ratio is of secondary concern because the employment of multiple stages can make up for any deficiency in pressure gain (Figure 1.9). Large volume flow is paramount to generating sufficient cooling capacity using R718 as a refrigerant, and as Table 1.7 indicates, the impeller generating the largest volume flow is rotor pattern 8B with a hub/tip ratio of 0.43. This impeller type has been selected for fabrication and further testing and will be discussed in Chapter 4.

1.5 Geothermal Power Plant Application

In addition to making use of the woven impeller technology to enable utilization of R718 as the refrigerant in a vapor compression refrigeration cycle due to the high speed requirements, it is also well suited as an efficient means of removing non-condensable gases (NCG) from the primary condenser of geothermal power plants (GPP). This is because the operating environment is highly corrosive, and carbon fiber/epoxy composites have extremely high corrosion resistance [14]. The woven impeller technology is equally as employable as a replacement/retrofit for existing lower-efficiency NCG removal techniques or for new GPPs.

1.5.1 Geothermal Power Plants

Geothermal plants make use of the warm interior of the Earth by extracting either dry steam or hot geothermal liquid, known as brine, from an aquifer. Dry steam, steam produced from hot liquid/brine entering a flash chamber, or vapor of another fluid heated by the hot liquid is expanded through a turbine that spins an electric generator to produce power in a process very similar to the Rankine cycle. Flashed-steam GPPs are most common among GPPs since most viable geothermal sources yield high temperature liquid rather than dry steam [15]. These plants use either single- or double-flash chambers and comprise 61% of the globally installed plants. Around 59% of flash plants are single-flash [16], [17], [18]. Figure 1.14 shows a single-flash GPP schematic.

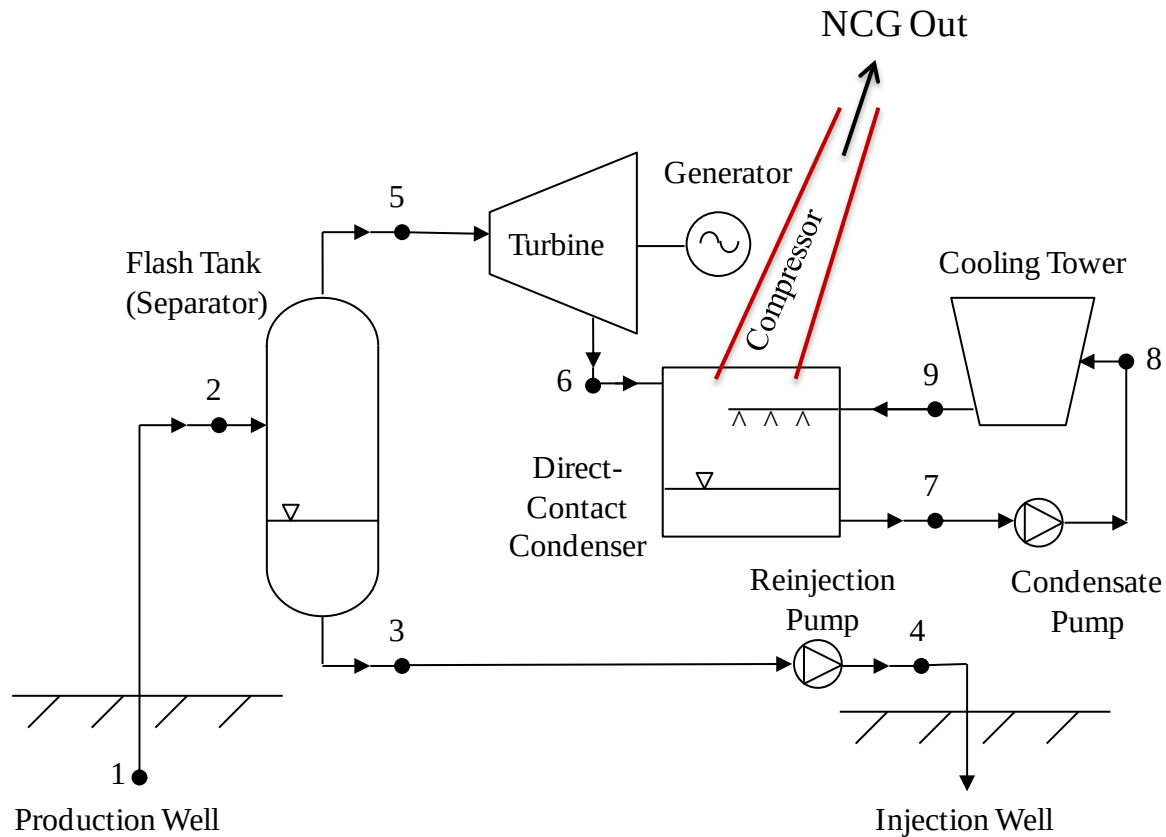


Figure 1.14. Single-Flash Geothermal Power Plant Schematic

When extracting the fluid from the geothermal well, many other types of substances are extracted along with it, including solids, ionic salts, and NCG. These additional undesirable materials result in many technical challenges for continuous smooth operation of the GPP: scales gradually formed by the salts (Ca_2Cl , KCl , NaCl , CaCO_3 , and Ca_2HCO_3) appear in the piping and flash chamber, and must be removed in order to prevent clogging and reduced effectiveness of heat exchange components; removal of scale is accomplished via dissolving using mild acid such as HCl . The scales themselves and the acid needed to remove them together contribute to a corrosive working environment, and hence the corrosion-resistant woven impeller is well suited to operation within this environment. Solids must also be removed to avoid additional clogging and component damage.

1.5.2 Non-Condensable Gases

The non-condensable gases that are extracted from a geothermal well along with the ionic salt-laden hot fluid consist partly of H_2S , CH_4 , NH_3 , N_2 , and C_2H_6 , but are a majority CO_2 . The levels of NCG present vary by GPP location, but are commonly around 1% by volume and in rare cases can reach over 10% by volume [19]. The presence of the NCG can have significant detrimental effects on overall GPP performance and efficiency. These gases pass through the turbine along with the flashed steam, but they do not condense into liquid inside the condenser. This means that they are not drawn out of the condenser by the condensate pump to the cooling tower. This leads to detrimental performance for a number of reasons [20]:

Firstly, the condenser pressure is far below ambient pressure to allow for maximum steam expansion and hence work and power extraction across the turbine. If the NCG were absent from the steam, the condenser operating pressure would be fixed by the cooling water temperature. Since they are in fact present, they serve to interfere with the heat transfer from the cooling water to the steam, be they direct- or indirect-contact condensers, with what is known as a “gas-blanketing” effect between the steam and cooling water. This then raises the temperature in the condenser, which in turn raises the operating pressure, thereby reducing work and power extraction potential from the turbine since there is a lower pressure difference across the turbine. This necessitates larger condensers with greater heat transfer area and hence higher capital costs to achieve the necessary cooling.

Secondly, if NCGs are allowed to accumulate in the condenser they contribute a partial pressure; this on its own raises the operating pressure inside the condenser creating a back pressure on the turbine above the designed outlet pressure, and thus reduces the potential work and power extraction.

Thirdly, acidic gases like CO₂ are water soluble and can cause corrosion in piping and other equipment that comes into contact with the steam and condensate if not removed.

Fourthly, NCG contain lower recoverable specific energy than steam. They dilute the steam and reduce turbine power extraction.

Lastly, cooling water consumption also increases as the amount of NCG increases due to the added cooling requirement. This then leads to increased cooling system costs as well. All of these issues contribute to the necessity of NCG removal for efficient and profitable operation of GPPs.

1.5.3 Current NCG Removal Techniques

There are a number of different vacuum devices used in today's existing GPPs to remove NCG from the primary condenser. These include

- Steam jet ejectors (SJE)
- Liquid ring vacuum pumps (LRVP)
- Radial/centrifugal blowers/compressors
- Hybrid systems (any combination of the above)
- Reboilers

Steam jet ejectors are the most common means of NCG removal, particularly in the case of single-flash GPPs. They use the venturi principle by accelerating high pressure “motive” (or “parasitic”) steam through a nozzle into a mixing chamber that then draws the NCG and any residual non-condensed steam out of the condenser (typically 75% to 99% of steam has condensed) [19], [20]. The mixture of motive and entrained gas is then decelerated in a diffuser, thereby increasing the mixture pressure to slightly above ambient so that it can be vented to the atmosphere. The dimensions of the SJE fix its capacity, so multiple ejectors can become

necessary in the case of larger NCG concentration and/or pressure gain requirements, and the use of inter-condensers can be employed to reclaim further condensate thereby reducing compression demands on the SJE system. Since SJE's have no moving parts, they are relatively inexpensive and easy to operate, but they are highly inefficient (generally between 10% and 15%), suited to narrow operating ranges (generally less than 3% volumetric NCG content) and prone to corrosion due to their steel or cast iron construction. They also require a large amount of parasitic steam that would otherwise be available for use by the turbine and hence reduce the overall power production capacity of the GPP [19], [20]. As the NCG fraction increases, the parasitic steam requirements also increase and operation can very quickly become uneconomical [19], [20]. Figure 1.15 shows a schematic of a SJE.

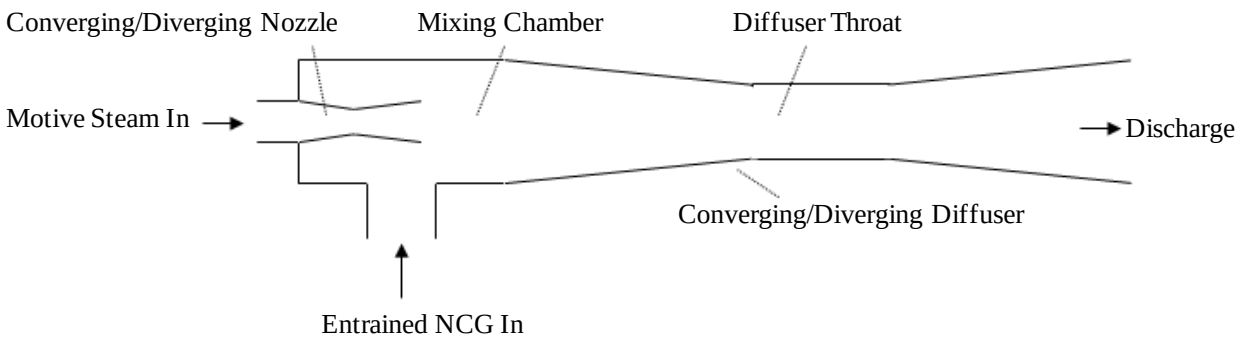


Figure 1.15. Steam Jet Ejector Schematic

Liquid ring vacuum pumps are electrically driven positive displacement machines that rotate an eccentric impeller surrounded by a ring of liquid to transfer energy to the pumped fluid. The liquid ring is formed by centrifugal force incurred by the impeller rotation and maintains its shape along the pump casing. This then seals off the space between the vanes of the impeller, thereby forming compression chambers. The eccentricity of rotation then causes the volume of the compression chambers to fluctuate cyclically. Gas is drawn through an inlet port as the liquid ring retreats from the impeller hub, increasing the volume of the compression chamber acting as a liquid piston. As the impeller rotates and the volume of the compression chamber decreases in

the reverse effect, the gas undergoes a compression process. When the compression chamber reaches sufficiently small volume that the gas is at high pressure (above ambient), it is then exposed to an exit port through which it naturally exhausts, and the process repeats. A typical LRVP is shown in Figure 1.16.

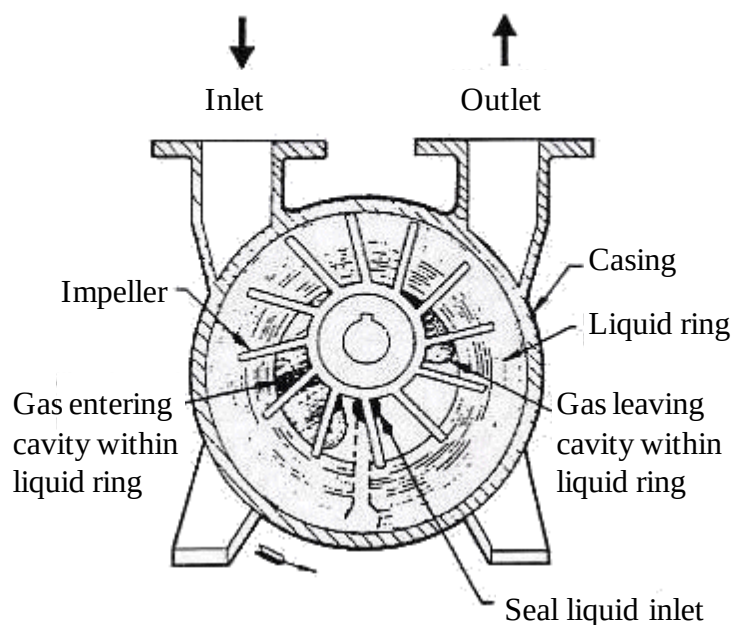


Figure 1.16. Counter-Clockwise Rotating Liquid Ring Vacuum Pump Schematic [21]

LRVP have better efficiencies than SJE and are on the order of around 50%, but they have much larger capital costs and can generally handle far lower volume flow rates and produce lower pressure ratios. As such, they are generally reserved for the final stage of NCG compression after it has passed through one or more SJE in what would be considered a hybrid system. Similar to SJE, the LRVP are constructed of primarily metallic components and so are prone to corrosion leading to high maintenance costs.

For large NCG content GPPs, centrifugal compressors are generally the preferred method of removal despite their complexity, large footprint, and large installation and maintenance costs. This is due to their ability to meet the volume flow and pressure ratio requirements combined with high operational efficiencies, which are on the order of 70% for a multi-stage system. This

is significantly higher than SJE, LRVP, or any combination of the two, but like the SJE and LRVP, the impellers and casings are manufactured from steel alloys and thus are given to corrosion. Figure 1.17 shows a 3-stage centrifugal compressor system for NCG removal.

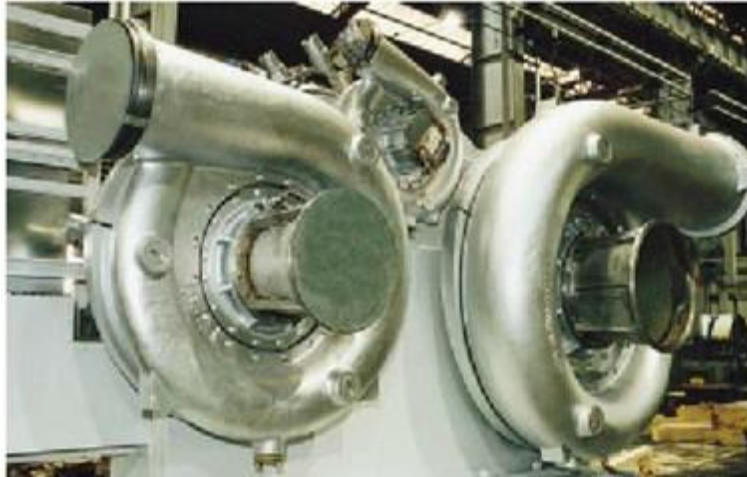


Figure 1.17. GE SRL 903 3-Stage Centrifugal Compressor [Source: GE Power Systems]

Reboiler systems use a fundamentally different technique from the preceding methodologies to improve GPP operation by separating the NCG from the bulk steam *before* feeding it through the turbine for power production. This is accomplished using a conventional vertical evaporator that both compresses and evaporates (via either direct- or indirect-contact) the geothermal steam. It discards high pressure steam containing significantly higher NCG concentration either directly to the atmosphere or through an emission control system prior to venting. This process however both degrades and reduces the amount of bulk steam that can enter the turbine, and although the bulk steam is not completely free of NCG, it has significantly reduced concentrations. This combination leads to great reductions in the vacuum equipment needs and hence reduced capital costs and power requirements for those systems, however the vent steam and degraded bulk steam entering the turbine cause reductions in power production across the turbine [19], [20].

1.6 Desalination Application

Numerous countries around the world along with some areas of the United States lack natural fresh water resources, while demand for fresh water is expected to only increase [22]. Approximately 97% of the water on earth is seawater, and about 2% is trapped in glaciers and icecaps. The available supply of fresh water on the planet is less than 0.5% of the planet's total water supply [23]. Since seawater is unsuitable for human consumption, agriculture, and a majority of industrial processes that require water, removing the salt from the essentially unlimited supply of seawater via desalination is becoming very important as a source of fresh water. As of 2002, over 10,000 desalination plants operate globally and produce nearly 38 million m³/day of fresh water, fulfilling the needs of around 75 million people [24].

The desalination of seawater is an energy intensive process that separates it into two streams: one of fresh water with a low salt concentration, and the other a concentrated brine solution to be rejected back into the sea. The two leading technologies used to accomplish this are Reverse Osmosis (RO) and MultiStage Flash distillation (MSF), accounting for 47% and 37% of global installations, respectively [24]. However, desalination is also possible using a Vapor Compression Distillation (VCD) technique to which the woven impeller technology is well suited, and with its high energy efficiency and low maintenance costs could serve to increase the market share of VCD in desalination.

1.6.1 Reverse Osmosis

Osmosis involves the movement of solvent molecules across a semi-permeable membrane to a region of higher solute concentration; the osmotic pressure is the applied external pressure necessary to achieve zero net movement of solvent across the membrane. Reverse Osmosis is then achieved by applying a pressure larger than the osmotic pressure on the seawater

that forces the solvent (water) across the membrane leaving behind a high concentration of solute (salt).

Pre-filtered seawater is pumped to anywhere from 50 to 80 bars using centrifugal pumps at a typical RO plant, thereby consuming a majority of the energy necessary for operation. The pressure variation is due to the type of membrane used. Common commercially used membranes are of the spiral wound and hollow fine fiber (HFF) configurations, where HFF is U-shaped. The membrane materials themselves are cellulose triacetate and polyamide [25], [26], [27], [28], [29].

1.6.2 Multistage Flash Distillation

MSF distillation works by flash evaporating seawater under low pressure rather than high temperature. Economical operation of MSF is achieved in practice by using regenerative heating whereby flashed seawater lends some of its heat to seawater undergoing the flashing process. This heat is then also used to gradually heat the incoming seawater feed itself. For true economic viability, MSF must be used with seawater heated with the output of a cogeneration power plant [30], [31], [32].

1.6.3 Vapor Compression Distillation

Vapor compression distillation works under the principle of decreased boiling point temperature due to reduced evaporator pressure. A compressor is used to evacuate and lower the pressure in an evaporator. Seawater exposed to the low pressure evaporates; the vapor is then drawn out through the compressor and is heated during the compression process. Heated vapor is condensed in a heat exchanger with the rejected heat serving to produce more vapor from additional seawater, which when compressed produces more heat, and so on. Figure 1.18 shows this schematically.

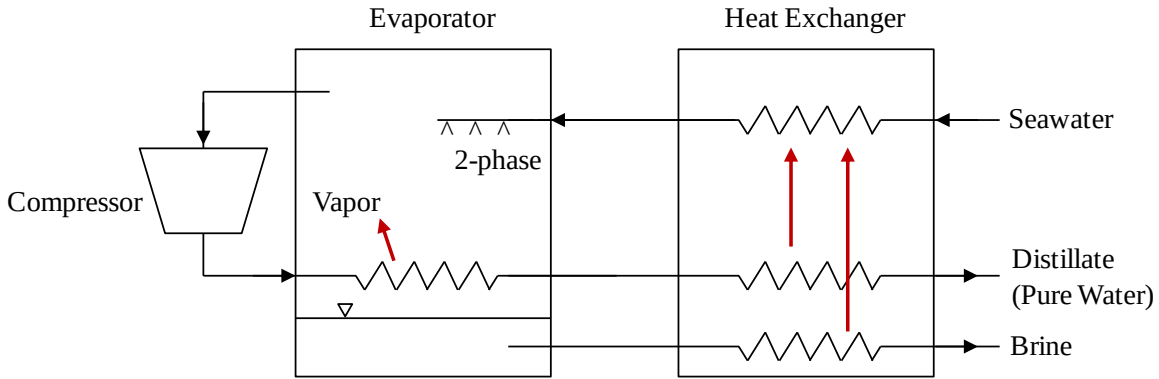


Figure 1.18. Vapor Compression Distillation for Desalination Schematic

The compressor is the only component requiring power, hence a high efficiency, high throughput axial compressor is an ideal candidate. This combined with the corrosive working environment lends itself perfectly to the employment of the corrosion resistant wound composite axial compressor proposed in this dissertation.

1.7 Wastewater Aeration Application

Municipal sewage and industrial wastewater treatment is commonly accomplished using the so-called “activated sludge” process. This process involves either air or oxygen being added to sewage along with bacteria and other organisms to reduce the organic content of the sewage by creating a type of biological floc. The mixture of sewage and biological material is known as “mixed liquor”. The aeration process maintains the desired dissolved oxygen (DO) levels in the mixed liquor (typically between 1 and 2 mg/L of water for stable biological activity in activated sludge systems) that the organisms breathe. The floc, i.e. the non-soluble particles brought out of suspension in the water, or “sludge blanket”, then settles out of the water, and the clarified water moves out of the aeration basin for further nutrient removal and disinfection before being returned to a receiving body of water. A majority of the sludge goes to sludge treatment before

being sent to a landfill, however a portion of it is recycled in order to reseed newly incoming wastewater [33]. An activated sludge treatment plant schematic is shown in Figure 1.19.

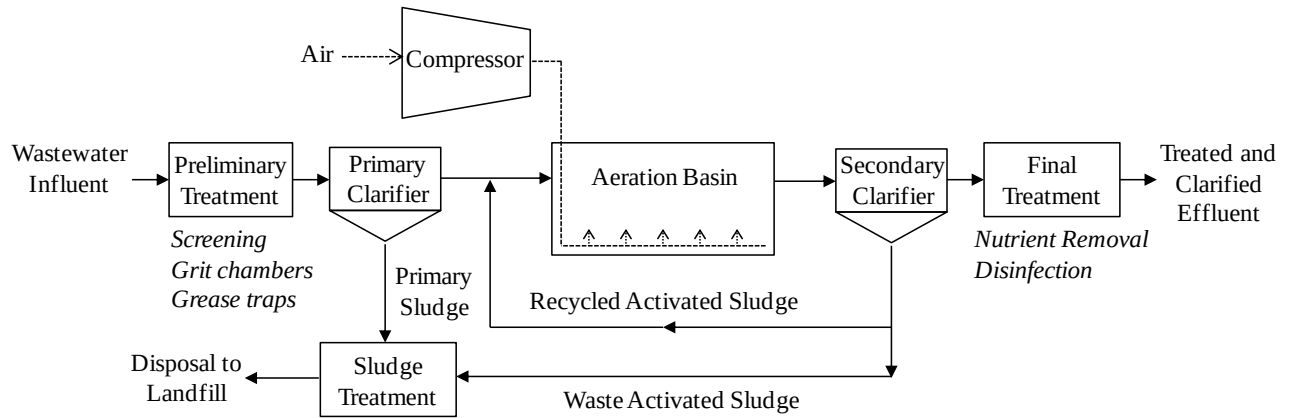


Figure 1.19. Activated Sludge Wastewater Treatment Schematic

Since the amount of wastewater entering a plant is generally variable, the amount of air or oxygen that needs to be supplied to the aeration basin to maintain the required DO levels is therefore also variable, and can account for anywhere from 25 to 60% of total treatment plant energy demands. The air or oxygen is typically delivered to the aeration basin anywhere between 0.35 and 1atm above ambient pressure [34].

Wastewater treatment plants currently employ either positive displacement blowers, or single or multistage centrifugal compressors for aeration, all of which have large capital and maintenance costs. However, the high efficiency, low capital cost axial compressor proposed in this dissertation has the potential to be an attractive and competitive alternative to currently used aeration blower technologies.

1.8 Mechanical Vapor Compression Application

The mechanical vapor compression (MVC) process is identical to that of the vapor compression distillation of seawater (Section 1.6), except it is generally performed on a much smaller scale to service industries producing consumer goods. It serves to concentrate solids

from a liquid solution by way of evaporation, compresses the vapor, and uses the additional heat produced from compressing the vapor to produce still more vapor. Figure 1.18 shows this schematically and Figure 1.20 shows this on a Mollier (enthalpy/entropy) diagram.

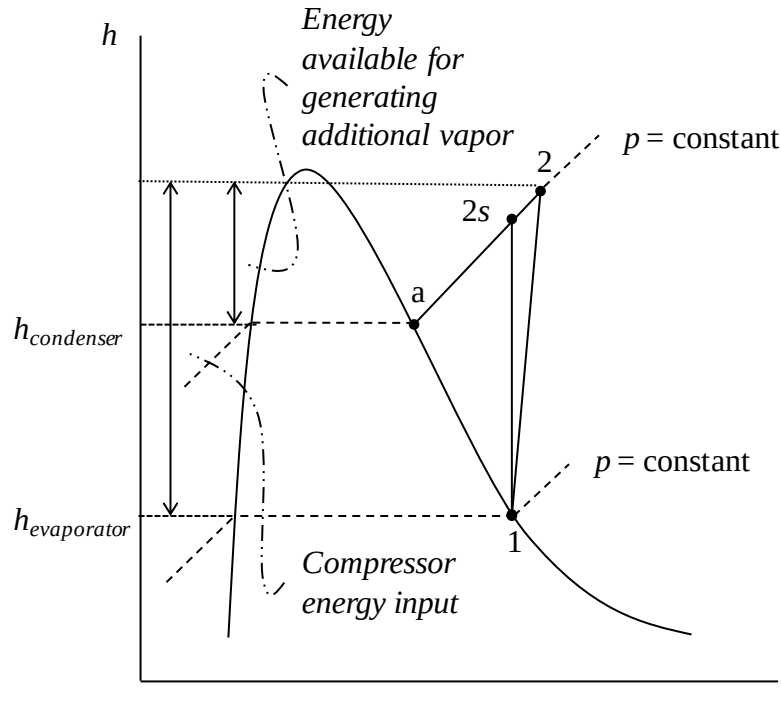


Figure 1.20. Mollier Diagram of MVC Process

MVC is used by the food processing industry to produce dried fruit, powdered milk, juice concentrates, soup packets, instant coffee, and many other foods that require rehydrating before consumption. Chemical processing, pharmaceutical products, and paper manufacturing also benefit from MVC [35]. The high efficiency woven impeller technology is a good alternative to conventional compressors used by processes requiring MVC.

1.9 Advantages of Woven Impeller Technology

The benefits of employing woven composites for impeller construction are multifaceted. Firstly, the cost of impeller manufacturing via winding is greatly reduced compared to traditional constructions for metal impellers that involve casting, machining, or some combination of the

two. Complex shapes required for the impeller are also more robust and less expensive to construct with winding compared to traditional composite lay-up methodologies.

Secondly, the winding process is able to be fully automated on a CNC-based winding platform that is compatible with both pre-impregnated (“pre-preg-” (wet-)) and dry-winding techniques, which allows for even further reductions in cost of construction.

Thirdly, the mandrel used as the base for winding can be either machined or printed from various materials using a 3D printer. Mandrel elements that are outside of the flow path can either remain as structural elements or can be removed. This removal process can work in a variety of ways: in the case of a metallic mandrel, a release agent must be applied before winding (wet-wound) or matrix infusion (dry-wound); for a 3D-printed plastic (ABS, PLA, etc) mandrel, the mandrel elements may simply be broken out of the cured impeller; mandrels may also be printed from dissolvable plastic (HIPS) so that mandrel removal is purely a chemical process rather than a labor-intensive one.

Fourthly, the blade winding process naturally lends itself to the incorporation of an outer shroud, and thus having all fibers aligned with the direction of the major forces (centrifugal, tangential) experienced by the impeller during operation. In addition to enhanced structural strength, the outer shroud serves to reduce tip leakage by eliminating tip clearance and reduces blade tip vortices. The added strength in the tangential direction also reduces vibration [13].

Fifthly, the carbon fiber composite is highly corrosion resistant [14] and is therefore very well suited to long term operation in environments that would quickly degrade and ultimately destroy its metallic counterparts.

Lastly, a permanent magnet motor can be integrated with an impeller. This feature is what allows for implementation of counter-rotation for enhanced performance and reduced

footprint by eliminating the need for stators between rotating stages. Each rotor rotates about its own short, non-rotating shaft (or a longer single non-rotating shaft) contained within the impeller hub, and requires its own variable frequency drive (VFD) to control the current through the motor stator windings that interact with the magnetic material incorporated into the hub (or shroud) of the impeller, thus allowing for any rate of rotation (within the capabilities of the VFD) in either direction about the shaft axis. Since there is no shaft rotation, each impeller can rotate independently, and thus counter-rotation is realized. This is possible without the need for rotating single or concentric shafts that inherently necessitate that all components mounted to them rotate at the same speed and in the same direction.

1.10 Research Objective and Dissertation Outline

The aim of this research is to design and analyze axial compressors that can be constructed from wound composite material using the patented technology created at Michigan State University. First- and second-generation impellers embody the scope of investigations performed. First-generation impellers originally conceived for their simplicity of construction are to be fully characterized geometrically, and performance is to be determined analytically, numerically, and experimentally with air under vacuum pressure. Enhanced sophistication of second-generation mandrels has led to greater control of wound impeller geometry and it is hypothesized that the second-generation impeller will exhibit improved performance characteristics. A second-generation impeller with similar mean geometry to first-generation models is to be compared in terms of performance both analytically and with numerical simulation at the same conditions.

Using stationary shafts and integrated permanent magnet motors makes the woven impeller modular. For a multistage counter-rotating machine, this widens the breadth of potential

operation significantly as rotating stages are not required to spin at the same rate and can operate independently. To that end, this research seeks to identify a regime to impart the maximum amount of energy to the working fluid in a multistage counter-rotating axial compressor based on inlet conditions and the number of stages employed. A robust design tool is to be developed in order to identify this regime.

The feasibility of using the woven impeller technology for these applications has already been established [12], [13], and this research will serve to gain deeper understanding into the details and properties of the flow characteristics during steady state operation. This will be accomplished using the commercial software ANSYS CFX in conjunction with other program modules within the ANSYS Workbench to simulate flow conditions and gauge performance.

This dissertation is divided into the following chapters:

Chapter 2: Modeling and Design of Axial Compressors

This chapter introduces the thermodynamics and fluid mechanics necessary to model turbomachinery in general, focusing on axial compressors. The various geometries of the types of woven impellers investigated are derived and defined along with the equations necessary to make performance predictions from a quasi-2-dimensional standpoint.

Chapter 3: Simulation of Axial Compressors

This chapter introduces the computational methodologies necessary to perform 3-dimensional numerical simulations of axial compressor systems. The Reynolds-averaged Navier-Stokes (RANS) equations are introduced in conjunction with turbulence modeling techniques.

Chapter 4: Star Pattern Impeller Performance

Star pattern impellers are wound on a first generation mandrel made from aluminum tubing with curved slots cut by a CNC-mill. This chapter details the analytical performance predictions made by expanding upon the techniques detailed in Chapter 2 along with CFD simulations that are compared with experimental results from a first generation star pattern impeller. A mathematical description of the star pattern geometry is developed and it is found to operate under the “forced vortex” flow regime. Flow field phenomena and their effect on performance are discussed.

Chapter 5: Free-Vortex Impeller Performance

Affordable 3D printing has allowed for the development of more sophisticated second generation mandrels that facilitate the winding of free-vortex blade impellers. This chapter takes the primary geometry and area-averaged blade turning of the first generation star pattern impeller investigated in Chapter 4 and applies it to the second generation free-vortex blade impeller operating at the same speed with the same inlet conditions. Flow field phenomena and their effect on performance are discussed, and a performance comparison is made between first and second generation impellers.

Chapter 6: Performance Enhancement of Multistage Counter-Rotating Axial Compressors

This chapter expands on the use of single stage free-vortex blade impellers to multistage counter-rotating machines. The independent variables and constraints are identified for a comprehensive search of the design space with the objective of imparting maximum specific work to the fluid. As water vapor under vacuum pressure is the working medium of most of the applications envisioned in Chapter 1 for the wound impeller, a study of up

to 7 counter-rotating stages has been performed for compressing water vapor under vacuum pressure at 22 different saturation temperatures. Inlet Mach numbers, flow angle, hub/tip ratio, and blade turning for each rotor are determined that yield maximum work transfer for each configuration. The corresponding non-dimensional coefficients are determined along with the tip speed ratios and work contributions of each stage. Numerical simulation of select cases has been performed in order to incorporate the effects of turbulence and viscosity into steady-state operation for more accurate performance predictions. Exhaustive design space searches have been performed analytically for seven other gas mixtures at a variety of inlet conditions in order to determine similarities and differences in best-point operation. This work serves to set the guide lines for fully independent rotation of subsonic counter-rotating multistage axial compressor operation.

Chapter 7: Conclusions

The main conclusions from Chapters 4-6 are recapitulated, and the contribution of this research is enumerated.

CHAPTER 2 MODELING AND DESIGN OF AXIAL COMPRESSORS

2.1 Thermodynamics and Gas Dynamics of Axial Compressors

2.1.1 Continuity and 1-Dimensional Compressible Flow

An axial compressor when viewed on its own is considered an “open system” from a thermodynamic standpoint. This means that there is no mass storage within the device, and hence any mass that flows into the compressor must ultimately flow out since mass can be neither created nor destroyed. This principle, when applied to a fixed, finite control volume as shown in Figure 2.1, leads to the Continuity Equation.

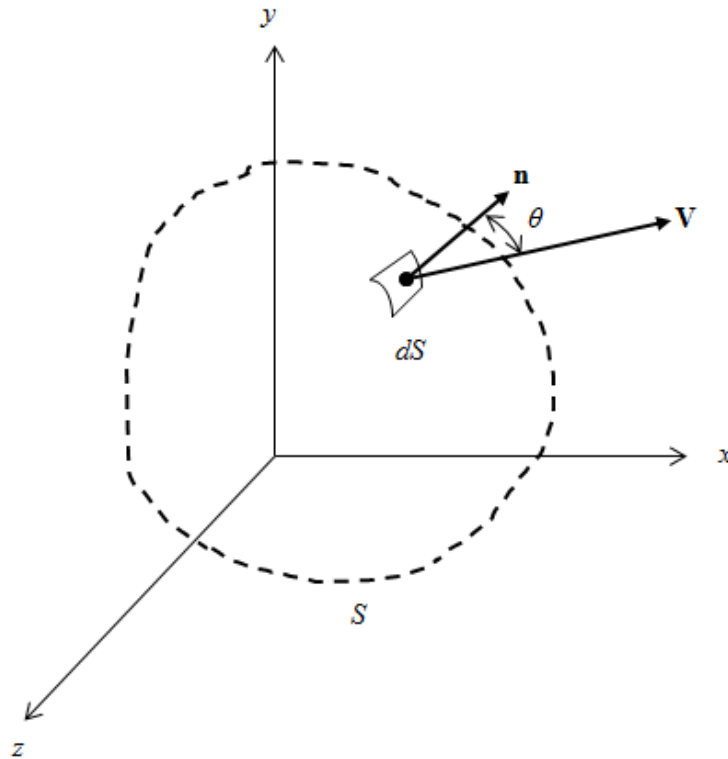


Figure 2.1. Fixed Finite Control Volume

The following development is based on that detailed by Anderson [36]. If the control volume has a volume $\mathcal{V}$ with a surface area S , then an elemental area on the surface is dS .

Defining $\mathbf{n}$ as an outward-pointing unit vector normal to the surface at dS allows for the further definition that $d\mathbf{S} = \mathbf{n}dS$, where bold characters indicate a vector quantity. If $\mathbf{V}$ and ρ are the local velocity and density, respectively, then the mass flow, $\dot{m}$, through any elemental surface that is arbitrarily oriented in a fluid flow is the product of the density, the velocity component normal to the surface, and the area of the surface:

$$\dot{m} = \rho(V \cos \theta)dS = \rho \mathbf{V} \cdot d\mathbf{S} \quad (2-1)$$

where θ is the angle between the velocity vector and the surface normal vector.

The overall mass flow into the control volume is the sum of the mass flows across each elemental surface area:

$$-\oint_S \rho \mathbf{V} \cdot d\mathbf{S} \quad (2-2)$$

where the negative sign is indicative of an inflow since $\mathbf{n}$ points out of the control volume.

The control volume itself is made up of smaller volumes dV that have a mass of ρdV . The total mass inside the control volume is then

$$\oint_V \rho dV \quad (2-3)$$

and hence the time rate of change of mass inside the control volume is

$$\frac{\partial}{\partial t} \oint_V \rho dV \quad (2-4)$$

Since mass can neither be created nor destroyed, the net mass flow into the control volume from Equation (2-2) must be the same as the rate of increase of mass inside the control volume as stated in Equation (2-4), resulting in (2-5), which is known as the Continuity Equation:

$$-\oint_S \rho \mathbf{V} \cdot d\mathbf{S} = \frac{\partial}{\partial t} \iiint_V \rho dV \quad (2-5)$$

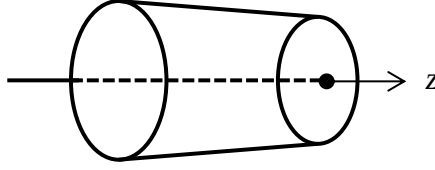


Figure 2.2. Quasi-One-Dimensional Control Volume

An axial compressor, as the name implies, has flow that is compressible and that is predominantly parallel to the axis of rotation. As a starting point for axial compressor modeling, a simplifying assumption can be made that the flow properties pressure, density, temperature and velocity only change only in the axial direction, z , and hence the flow can be considered one-dimensional (1D). Additionally, if the area changes gradually with z , $A = A(z)$, then the flow can be considered quasi-one-dimensional. Since the axial compressor is an open system and no fluid mass accumulates within the compressor during operation, i.e. the flow through the compressor is steady, all time derivatives vanish, and hence the right-hand-side of Equation (2-5) becomes zero. If the control volume is a considered to be tapered cylinder as shown in Figure 2.2, and a uniform flow is assumed perpendicular to the circular inlet (state point 1) and outlet (state point 2) faces, Equation (2-5) can be drastically simplified to

$$-\rho_1 V_1 A_1 + \rho_2 V_2 A_2 = 0 \quad (2-6)$$

where the negative sign again represents an inflow. Equation (2-6) can be rewritten as

$$\rho_1 V_1 A_1 = \rho_2 V_2 A_2 \quad (2-7)$$

where $A_2 < A_1$ since the fluid is compressed across the compressor. Equation (2-7) can also be expressed more familiarly as

$$\rho V A = \dot{m} = \text{constant} \quad (2-8)$$

2.1.2 Mixture Composition

The properties of the fluid mixture to be compressed must be known upstream of the impeller inlet. These properties include:

- The molecular weight of all fluid mixture components, MW_i
- The mass and mole fractions of each component, mf_i and y_i , respectively

With these quantities known, the molecular weight of the n -component mixture can be determined:

$$MW_{mix} = \sum_{i=1}^n y_i MW_i \quad (2-9)$$

This allows for the determination of the gas constant of the mixture:

$$R_{mix} = \frac{\bar{R}_u}{MW_{mix}} \quad (2-10)$$

where $\bar{R}_u = 8.314 \text{ kJ/kmol/K}$ is the Universal gas constant. For single-component fluid flows, the mass and mole fractions are 1, and Equation (2-10) holds by simply dropping the *mix* subscript.

For mixture components that can be assumed to be ideal gases, the specific heat capacity at constant pressure, c_p , of each component is modeled as a 4th-order polynomial in terms of temperature:

$$c_p = R(\alpha + \beta T + \gamma T^2 + \delta T^3 + \varepsilon T^4) \quad (2-11)$$

The coefficients α , β , γ , and ε are specific to individual gases and are used in accordance with [37].

Similar to Equation (2-9), the specific heat of the mixture can be determined based on the mole fraction of the mixture components:

$$c_{p,mix} = \sum_{i=1}^n y_i c_{p,i} \quad (2-12)$$

The result of Equations (2-10) and (2-12) can be combined to determine the specific heat capacity of the mixture at constant volume, c_v ,

$$c_{v,mix} = c_{p,mix} - R_{mix} \quad (2-13)$$

and the ratio of specific heats, γ , is defined as

$$\gamma_{mix} = \frac{c_{p,mix}}{c_{v,mix}} \quad (2-14)$$

Dividing both sides of (2-13) by c_p and invoking (2-14) leads to the useful result that

$$c_p = \frac{R\gamma}{\gamma - 1} \quad (2-15)$$

2.1.3 Thermodynamic Laws and Isentropic Relations

Much as continuity dictates that mass is conserved in a system, the First Law of Thermodynamics dictates that energy must be conserved. The following development is based on [36] Simply put, the heat, q , added to a system and the work, w , done on the system cause a change of energy, e , in the system:

$$\delta q + \delta w = de \quad (2-16)$$

The Second Law of Thermodynamics dictates in which direction energy a change of energy will take place by introducing the state variable entropy, s , defined as

$$ds = \frac{\delta q_{reversible}}{T} \quad (2-17)$$

where $\delta q_{reversible}$ is an incremental amount of heat added to the system in a reversible manner and T is the system temperature. The quantity $\delta q_{reversible}$ can be used to relate the initial and

final points of an irreversible process with an actual amount of heat added, δq , and adding to it an irreversible change in entropy:

$$ds = \frac{\delta q}{T} + ds_{irreversible} \quad (2-18)$$

Equation (2-18) states that a change in entropy is the heat added divided by the temperature, plus an increase in entropy due to a heat increase from dissipative phenomena such as viscosity. The second law states that any dissipative irreversibilities always serve to increase the entropy and is written

$$ds \geq \frac{\delta q}{T} \quad (2-19)$$

Referring to the first law in Equation (2-16), the work term for a reversible process can be written $\delta w = -pdv$, where v is specific volume, which describes compression or expansion work due to fluid pressure. This allows (2-16) to be rewritten as

$$\delta q - pdv = de \quad (2-20)$$

Assuming the heat transfer is reversible, Equation (2-17) can be substituted into (2-20) giving

$$Tds - pdv = de \quad (2-21)$$

which is another form of the first law.

The definition of enthalpy, h , is the internal energy plus flow work

$$h = e + pv \quad (2-22)$$

and differentiating (2-22) leads to

$$dh = de + pdv + vdp \quad (2-23)$$

Substituting (2-23) in terms of de into (2-21) yields

$$Tds = dh - vdp \quad (2-24)$$

For a calorifically perfect ideal gas, the change in enthalpy is related to a temperature change proportional to the specific heat:

$$dh = c_p dT \quad (2-25)$$

Substituting (2-25) into (2-24) and dividing by temperature gives

$$ds = c_p \frac{dT}{T} - \frac{v dp}{T} \quad (2-26)$$

The ideal gas equation of state is

$$pv = RT \quad (2-27)$$

and substitution of (2-27) into (2-26) gives

$$ds = c_p \frac{dT}{T} - R \frac{dp}{p} \quad (2-28)$$

Integrating from state 1 to 2 yields

$$s_2 - s_1 = \int_{T_1}^{T_2} c_p \frac{dT}{T} - \int_{p_1}^{p_2} R \frac{dp}{p} \quad (2-29)$$

resulting in

$$s_2 - s_1 = c_p \ln \frac{T_2}{T_1} - R \ln \frac{p_2}{p_1} \quad (2-30)$$

With Equation (2-30), the isentropic relations, i.e. relations for when there is no change in entropy ($s_2 = s_1$), can be derived. Setting the left side of (2-30) to zero and solving for the pressure ratio in terms of the temperature ratio gives

$$\frac{p_2}{p_1} = \left(\frac{T_2}{T_1} \right)^{\frac{c_p}{R}} \quad (2-31)$$

Substitution of Equation (2-15) to the temperature ratio exponent allows (2-31) to be rewritten as

$$\frac{p_2}{p_1} = \left(\frac{T_2}{T_1} \right)^{\frac{\gamma}{\gamma-1}} \quad (2-32)$$

Using the definition $de = c_v dT$ for a calorifically perfect ideal gas and substituting into (2-21), a similar development gives the pressure ratio in terms of the density ratio for an isentropic process:

$$\frac{p_2}{p_1} = \left(\frac{\rho_2}{\rho_1} \right)^{\gamma} \quad (2-33)$$

2.1.4 Critical Speed of Sound

Given that the static (i.e. moving) fluid properties are not necessarily known *a priori* throughout the flow field, the stagnation conditions are used to determine flow properties since in general the compressor will draw from a stationary fluid and induce the flow. Mach number, the ratio of fluid speed to the speed of sound in that fluid, plays a large role in the pertinent properties, but until the speed of sound is known, Mach number itself has no reference point from which to impart any significance.

The speed of a sound wave, a , in a fluid is related to an isentropic change in pressure with respect to density:

$$a = \sqrt{\left(\frac{\partial p}{\partial \rho} \right)_s} \quad (2-34)$$

The isentropic process for a calorifically perfect ideal gas shown in Equation (2-33) can be rearranged in the more commonly used form of (2-35).

$$pv^{\gamma} = \frac{p}{\rho^{\gamma}} = \text{constant} \quad (2-35)$$

Solving Equation (2-35) in terms of p and differentiating with respect to ρ as per (2-34) leads to an alternative form of a :

$$a = \sqrt{\frac{\gamma p}{\rho}} = \sqrt{\gamma RT} \quad (2-36)$$

where the equation of state for an ideal gas (2-27) has been used to arrive at the result on the right hand side of (2-36).

The Mach number is defined as the ratio of fluid speed in an inertial reference frame, c , to the speed of sound in that fluid:

$$M = \frac{c}{a} \quad (2-37)$$

The critical Mach number, M^* , is defined as where the flow changes from subsonic to supersonic at the sonic condition, namely at $M^* = 1$.

$$1 = \frac{c}{a} = \frac{c}{\sqrt{\gamma RT}} = \frac{a_c^*}{\sqrt{\gamma RT}} \quad (2-38)$$

Applying the first law of thermodynamics to a fluid moving through an arbitrary but fixed control volume of volume $\mathcal{V}$ with a closed surface area of S as shown in Figure 2.1 leads to the energy equation:

$$\begin{aligned} & \underbrace{\dot{Q}}_{(a)} + \underbrace{\dot{W}_{shaft}}_{(b)} + \underbrace{\dot{W}_{viscous}}_{(c)} - \underbrace{\oint_S p \mathbf{V} \cdot d\mathbf{S}}_{(d)} + \underbrace{\oint_{\mathcal{V}} \rho (\mathbf{f} \cdot \mathbf{V}) d\mathcal{V}}_{(e)} \\ &= \underbrace{\oint_{\mathcal{V}} \frac{\partial}{\partial t} \left[\rho \left(e + \frac{c^2}{2} \right) \right] d\mathcal{V}}_{(f)} + \underbrace{\oint_S \rho \left(e + \frac{c^2}{2} \right) \mathbf{V} \cdot d\mathbf{S}}_{(g)} \end{aligned} \quad (2-39)$$

The terms in Equation (2-39) are as follows in order from left to right:

(a) $\dot{Q}$ represents the combination of the rate of heat added across the control surface due to thermal conduction, diffusion, and radiation.

(b) $\dot{W}_{shaft}$ is the rate of work done on the fluid in the control volume by a rotating shaft crossing the control surface.

(c) $\dot{W}_{viscous}$ is the rate of work done on the control surface by viscous stresses.

(d) The first surface integral on the left hand side is the rate of work done on the fluid due to pressure forces on the control surface, where $\mathbf{V}$ is the three-component velocity vector within the coordinate frame, and $d\mathbf{S}$ is a differential surface element perpendicular to a unit vector normal to that surface element.

(e) The first volume integral on the left hand side is the rate of work done on the fluid due to body forces, where $\mathbf{f}$ is the force per unit mass.

(f) The first volume integral on the right hand side is the time rate of change of the flow field variables inside the control volume, where e is the specific internal energy.

(g) The first surface integral on the right hand side is the net rate of energy flow across the control surface [36].

For a steady, one-dimensional flow with no shaft or viscous work, no heat addition or generation, and with negligible changes in potential energy, the terms (a)-(c), (e), and (f) can be eliminated, resulting in a much-simplified form of the energy equation:

$$h_1 + \frac{c_1^2}{2} = h_2 + \frac{c_2^2}{2} \quad (2-40)$$

where h is the specific enthalpy of the flow as defined in (2-22), and subscripts 1 and 2 denote the inlet and outlet control surfaces, respectively, of the control volume perpendicular to the flow direction.

Equation (2-25) allows Equation (2-40) to be written as

$$c_p T_1 + \frac{c_1^2}{2} = c_p T_2 + \frac{c_2^2}{2} \quad (2-41)$$

The concept of total properties (i.e. temperature, pressure, density, enthalpy), also known as the stagnation conditions, are defined as the properties the fluid attains when brought to rest isentropically. Therefore, if the fluid at state point 1 has static temperature T and velocity c , then $T_1 = T$ and $c_1 = c$. If the fluid at location 2 is brought to rest isentropically, then $c_2 = 0$ and $T_2 = T_t$ by definition, yielding

$$c_p T_t = c_p T + \frac{1}{2} c^2 \quad (2-42)$$

and

$$h_t = h + \frac{1}{2} c^2 \quad (2-43)$$

Solving (2-42) for total temperature gives

$$T_t = T + \frac{c^2}{2c_p} \quad (2-44)$$

Returning to Equation (2-38) and substituting in Equation (2-44) yields the following:

$$a_c^{*2} = \gamma R T = \gamma R \left(T_t - \frac{c^2}{2c_p} \right) \quad (2-45)$$

Further substitution of Equation (2-15) as well as $c^2 = a_c^{*2}$ into (2-45) gives

$$a_c^{*2} = \gamma R T_t - \frac{\gamma R a_c^{*2} (\gamma - 1)}{2\gamma R} \quad (2-46)$$

Canceling terms and rearranging yields

$$a_c^{*2} + \frac{a_c^{*2} (\gamma - 1)}{2} = \gamma R T_t \quad (2-47)$$

$$a_c^{*2} \left(1 + \frac{\gamma - 1}{2} \right) = \gamma R T_t \quad (2-48)$$

$$a_c^{*2} \left(\frac{\gamma + 1}{2} \right) = \gamma R T_t \quad (2-49)$$

so that

$$a_c^* = \sqrt{\frac{2}{\gamma + 1} \gamma R T_t} \quad (2-50)$$

The critical Mach number, M_c^* , is then

$$M_c^* = \frac{c}{a_c^*} \quad (2-51)$$

2.1.5 Flow Property Ratios

It is convenient to define the ratio of a static property to its corresponding total property, and this can be done in terms of Mach number. Returning to Equation (2-44) and dividing by static temperature gives

$$\frac{T_t}{T} = 1 + \frac{c^2}{2c_p T} \quad (2-52)$$

Substituting (2-15) for c_p and subsequently (2-36) for $\gamma R T$ yields

$$\frac{T_t}{T} = 1 + \frac{c^2 (\gamma - 1)}{2a^2} \quad (2-53)$$

Using the definition of Mach number in (2-37) results in

$$\frac{T_t}{T} = 1 + \frac{(\gamma - 1)}{2} M^2 \quad \text{or} \quad \frac{T}{T_t} = \frac{1}{1 + \frac{(\gamma - 1)}{2} M^2} \quad (2-54)$$

However it is even more convenient to write the static-to-total temperature ratio in terms of critical Mach number. Beginning with Equation (2-41) and similarly substituting in Equations (2-15) and (2-36) for c_p and γRT , respectively, yields

$$\frac{a_1^2}{\gamma-1} + \frac{c_1^2}{2} = \frac{a_2^2}{\gamma-1} + \frac{c_2^2}{2} \quad (2-55)$$

Bringing state 2 isentropically to Mach 1 by definition means both a_2 and c_2 are at the critical speed of sound. Dropping the subscripts for states 1 and 2 since they are no longer necessary gives

$$\frac{a^2}{\gamma-1} + \frac{c^2}{2} = \frac{a^{*2}}{\gamma-1} + \frac{a^{*2}}{2} \quad (2-56)$$

Combining like terms on the right-hand-side of (2-56) and dividing both sides by c^2 allows for Mach number to be written in terms of critical Mach number:

$$M^2 = \frac{2}{\frac{\gamma+1}{M^{*2}} - (\gamma-1)} \quad (2-57)$$

Equation (2-57) can be substituted into Equation (2-54), and upon multiplying the numerator and denominator by M^{*2} and combining like terms, the temperature ratio in terms of critical Mach number is written as

$$\frac{T}{T_t} = 1 - \frac{\gamma-1}{\gamma+1} M^{*2} \quad (2-58)$$

Equation (2-58) can then be substituted into Equations (2-32) and (2-33) to similarly relate pressure and density ratio in terms of critical Mach number:

$$\frac{p}{p_t} = \left(1 - \frac{\gamma-1}{\gamma+1} M^{*2} \right)^{\frac{\gamma}{\gamma-1}} \quad (2-59)$$

and

$$\frac{\rho}{\rho_t} = \left(1 - \frac{\gamma-1}{\gamma+1} M^{*2}\right)^{\frac{1}{\gamma-1}} \quad (2-60)$$

2.2 Euler's Equation of Turbomachinery

2.2.1 Moment of Momentum

The power consumed by a compressor produced by a turbine is determined by Newton's second law as it applies to the moments of forces: a balance of the moment of momentum (a conservation of angular momentum) is needed. Momentum is the product of mass and velocity, hence moment of momentum, $\mathbf{H}$, about a point is the vector from the point to the mass, i.e.

$$\mathbf{H} = \mathbf{r} \times m\mathbf{V} \quad (2-61)$$

With $\mathbf{V} = \dot{\mathbf{r}}$, (2-61) becomes

$$\mathbf{H} = \mathbf{r} \times m\dot{\mathbf{r}} \quad (2-62)$$

Differentiating (2-62) with respect to time gives

$$\dot{\mathbf{H}} = \dot{\mathbf{r}} \times m\dot{\mathbf{r}} + \mathbf{r} \times m\ddot{\mathbf{r}} \quad (2-63)$$

Since a vector crossed with itself is zero, (2-63) simplifies to

$$\dot{\mathbf{H}} = \mathbf{r} \times m\ddot{\mathbf{r}} \quad (2-64)$$

Newton's second law states that the force on an object is the product of its mass and acceleration:

$$\mathbf{F} = m\ddot{\mathbf{r}} \quad (2-65)$$

Crossing both sides of (2-65) by $\mathbf{r}$ gives

$$\mathbf{r} \times \mathbf{F} = \mathbf{r} \times m\ddot{\mathbf{r}} \quad (2-66)$$

Since $\mathbf{r} \times \mathbf{F}$ is by definition a moment, $\mathbf{M}$, and invoking the definition in (2-64), Equation (2-66) can be written

$$\mathbf{M} = \dot{\mathbf{H}} \quad (2-67)$$

or in other words, the vector sum of the moments exerted by external forces on a system of mass about some arbitrary point or axis is the time rate of change of its angular momentum [38].

Writing (2-67) for a moving fluid as

$$\mathbf{M} = \dot{\mathbf{H}} = \frac{D}{Dt}(\mathbf{r} \times m\dot{\mathbf{r}}) \quad (2-68)$$

where $\frac{D}{Dt} = \frac{\partial}{\partial t} + \mathbf{V} \cdot \nabla$ is defined as the Substantial Derivative for a moving fluid: $\frac{\partial}{\partial t}$ is the local derivative, accounting for the time rate of change at a fixed point, and $\mathbf{V} \cdot \nabla$ is the convective derivative, accounting for the movement of fluid from one point in the flow field to another.

The Reynolds Transport Theorem states that the substantial derivative of a field quantity (mass, velocity, angular momentum, etc) that is a function of both space and time, $\mathbf{F}(\mathbf{x}, t)$, in a control volume can be written in terms of the volume and surface integrals corresponding to the control volume [39]:

$$\frac{D\mathbf{F}(\mathbf{x}, t)}{Dt} = \iiint_V \frac{\partial}{\partial t} f(\mathbf{x}, t) dV + \iint_S f(\mathbf{x}, t) \mathbf{V} \cdot \mathbf{n} dS \quad (2-69)$$

As an example, the relationship between $\mathbf{F}$ and f is easily demonstrated as the relationship between mass (m or $\mathbf{F}$ in (2-69)) and density (ρ or f in (2-69)):

$$m = \iiint_V \rho dV \quad (2-70)$$

Applying the Reynolds transport theorem in (2-69) to the conservation of angular momentum in (2-68) and making use of (2-70) gives

$$\frac{D}{Dt}(\mathbf{r} \times m\dot{\mathbf{r}}) = \iiint_V \frac{\partial}{\partial t} (\mathbf{r} \times \rho \dot{\mathbf{r}}) dV + \iint_S (\mathbf{r} \times \rho \dot{\mathbf{r}}) \mathbf{V} \cdot \mathbf{n} dS \quad (2-71)$$

For steady flow, the first term on the right hand side of (2-71) vanishes, hence

$$\frac{D}{Dt}(\mathbf{r} \times m\dot{\mathbf{r}}) = \iint_S (\mathbf{r} \times \rho\dot{\mathbf{r}}) \mathbf{V} d\mathbf{S} \quad (2-72)$$

where, as was done for the continuity equation, the definition $d\mathbf{S} = \mathbf{n}dS$ has been invoked.

Looking at the cross product inside the surface integral of (2-72) and working within a mutually orthogonal three-coordinate system, $\hat{e}_r, \hat{e}_\theta, \hat{e}_z$, where $\hat{e}_r$ is radial to, $\hat{e}_\theta$ is tangential to, and $\hat{e}_z$ is axially parallel to the axis of a turbomachine rotating within the control volume,

$$\mathbf{r} \times \rho\dot{\mathbf{r}} = \begin{vmatrix} \hat{e}_r & \hat{e}_\theta & \hat{e}_z \\ r & \theta & z \\ \rho c_r & \rho c_u & \rho c_m \end{vmatrix} = \hat{e}_r \rho(\theta c_m - z c_u) - \hat{e}_\theta \rho(r c_z - z c_r) + \hat{e}_z \rho(r c_u - \theta c_r) \quad (2-73)$$

Substituting the result of (2-73) into (2-72) gives

$$\mathbf{M} = \frac{D}{Dt}(\mathbf{r} \times m\dot{\mathbf{r}}) = \iint_S (\hat{e}_r(\theta c_m - z c_u) - \hat{e}_\theta(r c_z - z c_r) + \hat{e}_z(r c_u - \theta c_r)) \rho \mathbf{V} d\mathbf{S} \quad (2-74)$$

The moment about the turbomachine axis, M_{ax} , is the component of the moment of momentum about that axis. For 1D flow, the radial velocity component c_r must be zero, and so

$$M_{ax} = \hat{e}_z \cdot \mathbf{M} = \iint_S r c_u \rho \mathbf{V} d\mathbf{S} \quad (2-75)$$

Referring again to the tapered cylindrical control volume in Figure 2.2 and assuming uniform flow across the circular inlet (state 1) and outlet (state 2) faces, the surface integrals in (2-75) reduce to

$$M_{ax} = \dot{m}(r_2 c_{u2} - r_1 c_{u1}) \quad (2-76)$$

The power, $\tilde{P}$, consumed (or produced in the case of a turbine) is the product of the axial moment with the rotation rate, ω .

$$\tilde{P} = M_{ax} \omega = \dot{m} \omega (r_2 c_{u2} - r_1 c_{u1}) = \dot{m} (u_2 c_{u2} - u_1 c_{u1}) \quad (2-77)$$

where $u = r\omega$ is the turbomachine tangential blade speed at radius r .

Dividing the power by the mass flow rate gives the specific work, $\tilde{e}$, imparted to (from) the fluid by (to) the compressor (turbine), which is known as Euler's equation of turbomachinery:

$$\tilde{e} = \frac{\tilde{P}}{\dot{m}} = (u_2 c_{u2} - u_1 c_{u1}) \quad (2-78)$$

2.2.2 Thermodynamic Work Transfer

Returning to the energy equation in (2-39), it has been shown that for the control volume in Figure 2.2, Equation (2-40) applies for a general fluid, and Equation (2-41) applies for an ideal gas. Considering now that an axial compressor is enclosed within the control volume and connected to a shaft that crosses the control surfaces, the shaft work term in (2-39) must not be neglected. Note that power, $\tilde{P}$, and $\dot{W}_{shaft}$ are equivalent. Rewriting:

$$\dot{W}_{shaft} - \iint_S p \mathbf{V} \cdot d\mathbf{S} = \iint_S \rho \left(e + \frac{c^2}{2} \right) \mathbf{V} \cdot d\mathbf{S} \quad (2-79)$$

or

$$\dot{W}_{shaft} - (-p_1 c_1 A_1 + p_2 c_2 A_2) = -\rho_1 \left(e_1 + \frac{c_1^2}{2} \right) c_1 A_1 + \rho_2 \left(e_2 + \frac{c_2^2}{2} \right) c_2 A_2 \quad (2-80)$$

Rearranging gives

$$\dot{W}_{shaft} + p_1 c_1 A_1 + \rho_1 \left(e_1 + \frac{c_1^2}{2} \right) c_1 A_1 = \rho_2 \left(e_2 + \frac{c_2^2}{2} \right) c_2 A_2 + p_2 c_2 A_2 \quad (2-81)$$

Dividing the left side of (2-81) by the left side of (2-7) and the right side of (2-81) by the right side of (2-7) substituting c for V yields

$$\frac{\dot{W}_{shaft}}{\rho_1 c_1 A_1} + \frac{p_1}{\rho_1} + e_1 + \frac{c_1^2}{2} = \frac{p_2}{\rho_2} + e_2 + \frac{c_2^2}{2} \quad (2-82)$$

Noting that the specific volume is the reciprocal of density, invoking the definition for enthalpy

from Equation (2-22), using the definition of total enthalpy, and letting $\frac{\dot{W}_{shaft}}{\rho_1 c_1 A_1} = \frac{\dot{W}_{shaft}}{\dot{m}} = \tilde{e}$,

Equation (2-82) becomes

$$\tilde{e} = h_{t2} - h_{t1} \quad (2-83)$$

For an ideal gas, (2-83) becomes

$$\tilde{e} = c_p (T_{t2} - T_{t1}) \quad (2-84)$$

Factoring out T_{t1} :

$$\tilde{e} = c_p T_{t1} \left(\frac{T_{t2}}{T_{t1}} - 1 \right) \quad (2-85)$$

Equation (2-32) relates the pressure to the temperature at any two locations in an isentropic flow in terms of their ratios. Substituting the stagnation pressure ratio for the stagnation temperature ratio gives

$$\tilde{e} = c_p T_{t1} \left[\left(\frac{p_{t2}}{p_{t1}} \right)^{\frac{\gamma-1}{\gamma}} - 1 \right] \quad (2-86)$$

Equation (2-86) is for an ideal (isentropic, reversible) turbomachine. In an actual compressor, irreversibilities in the compression process lead to entropy production, and as will be shown, this leads directly to total pressure loss [39].

The quantity ratios in Equation (2-30) are static values. The static pressure ratio can be related to the total pressure ratio as

$$\frac{p_2}{p_1} = \frac{\left(\frac{p_2}{p_{t2}}\right)}{\left(\frac{p_1}{p_{t1}}\right)} \left(\frac{p_{t2}}{p_{t1}}\right) \quad (2-87)$$

Using the isentropic relation $pv^\gamma = \text{constant}$ allows for the pressure ratios to be expressed in terms of temperature ratios as per (2-32):

$$\frac{p_2}{p_1} = \frac{\left(\frac{T_2}{T_{t2}}\right)^{\frac{\gamma}{\gamma-1}}}{\left(\frac{T_1}{T_{t1}}\right)^{\frac{\gamma}{\gamma-1}}} \left(\frac{p_{t2}}{p_{t1}}\right) \quad (2-88)$$

Assuming the total temperature stays constant during the entropy change, i.e. $T_{t2} = T_{t1}$, Equation (2-88) simplifies to

$$\frac{p_2}{p_1} = \left(\frac{T_2}{T_1}\right)^{\frac{\gamma}{\gamma-1}} \left(\frac{p_{t2}}{p_{t1}}\right) \quad (2-89)$$

Substituting (2-89) into Equation (2-30) gives

$$s_2 - s_1 = \Delta s = c_p \ln\left(\frac{T_2}{T_1}\right) - R \frac{\gamma}{\gamma-1} \ln\left(\frac{T_2}{T_1}\right) - R \ln\left(\frac{p_{t2}}{p_{t1}}\right) \quad (2-90)$$

Using the relation from (2-15) cancels the first two terms on the right hand side of Equation (2-90) giving

$$\Delta s = -R \ln\left(\frac{p_{t2}}{p_{t1}}\right) = -R \ln\left(\frac{p_{t1} - \Delta p_t}{p_{t1}}\right) \quad (2-91)$$

As indicated in Equation (2-91), zero change in total pressure, Δp_t , results in zero entropy change, and a non-zero change in entropy necessitates a non-zero Δp_t .

Since the actual compressor will generate entropy and incur pressure losses due to wall friction and boundary layer formation on the blades and casing, as well as flow separation losses behind the trailing edge and in areas of stall as examples, it is convenient to use a single isentropic efficiency, η_{is} , to represent all of the losses at once. This implies that while the quantities in Equations (2-83) and (2-84) are true regardless of incurred loss, Equation (2-86) must be modified to reflect these losses:

$$\tilde{e} = \frac{c_p T_{t1}}{\eta_{is}} \left(\left(\frac{p_{t2}}{p_{t1}} \right)^{\frac{\gamma-1}{\gamma}} - 1 \right) \quad (2-92)$$

However, while using isentropic efficiency is generally valid, it can be misleading when analyzing compressors with large pressure ratios and/or multiple stages. If a single stage is broken into a large number of smaller stages of equal efficiency, the overall compressor will have a lower efficiency than that of the small stages [54]. This small stage efficiency is the polytropic efficiency, η_{pt} , and is discussed in detail in Appendix A2.

Equations (2-78), (2-83), (2-92), and (A2-24) then equate changes in rotor and tangential flow velocities with the change in total enthalpy and temperature, while factoring in isentropic or polytropic efficiency to determine the corresponding change in total pressure. Recapitulating,

$$\begin{aligned} \tilde{e} = u_2 c_{u2} - u_1 c_{u1} = h_{t2} - h_{t1} = c_p (T_{t2} - T_{t1}) &= \frac{c_p T_{t1}}{\eta_{is}} \left(\left(\frac{p_{t2}}{p_{t1}} \right)^{\frac{\gamma-1}{\gamma}} - 1 \right) \\ &= c_p T_{t1} \left(\left(\frac{p_{t2}}{p_{t1}} \right)^{\frac{\gamma-1}{\gamma \eta_{pt}}} - 1 \right) \end{aligned} \quad (2-93)$$

where the polytropic efficiency (shown in the lower term of (2-93)) is larger than the corresponding isentropic efficiency.

2.3 Circular Arc Blade Description

2.3.1 Rotor Blades

Turbomachines in general rotate continuously and impart work to or from the fluid by the dynamic action of rotating blades. While Equation (2-93) relates enthalpy and hence temperature and pressure changes to changes in tangential rotor and fluid velocity, it does not describe how to implement those velocity changes. The blades of the axial compressor used to impart work to the fluid that are studied in this dissertation are circular arc segments. The curving of the blades serves to change the tangential velocity of the fluid as it traverses axially from the inlet of compressor to the outlet.

Different 3D blade arrangements that have been investigated will be detailed in Chapters 4 and 5. However both begin with the definition of a circular arc at the outer shroud radius.

Figure 2.3 shows a circular arc blade segment at the outer shroud of the impeller where the blade angle, β , is shown at the inlet relative to the tangential direction, and the axial length of the rotor, z_{rot} , is depicted.

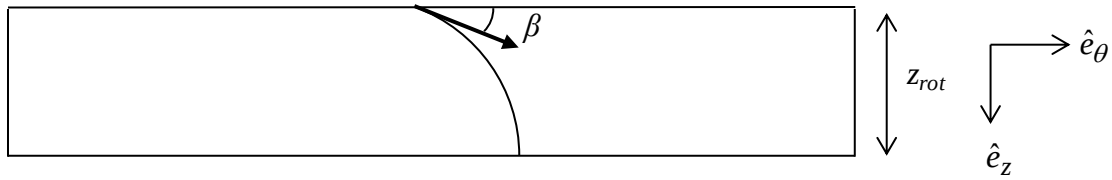


Figure 2.3. Circular Arc Blade Segment

Since the viewing plane in Figure 2.3 is at the outer shroud, the radial direction, $\hat{e}_r$, points into the page in keeping with a right-handed coordinate system. This holds with convention of the radial direction emanating from the central axis of rotation as the view provided in Figure 2.3 should be taken as looking at the inner side of the outer shroud as seen from the center of the impeller. Also note that impeller rotates in the $-\hat{e}_\theta$ direction.

A more detailed version of the circular arc is presented in Figure 2.4. If β_{in} and β_{out} , the inlet and outlet blade angles, respectively, are both known along with z_{rot} , then the blade angles across the entire axial span of the rotor can be prescribed because the circular arc defines them. Note that the subscripts *in* and *out* are interchangeable with subscripts 1 and 2 from preceding sections, respectively.

Referring to Figure 2.4, the camber angle, ϕ , and the chord length, ch , are

$$\phi = \beta_{out} - \beta_{in} \quad (2-94)$$

$$ch = 2r_b \sin\left(\frac{\phi}{2}\right) \quad (2-95)$$

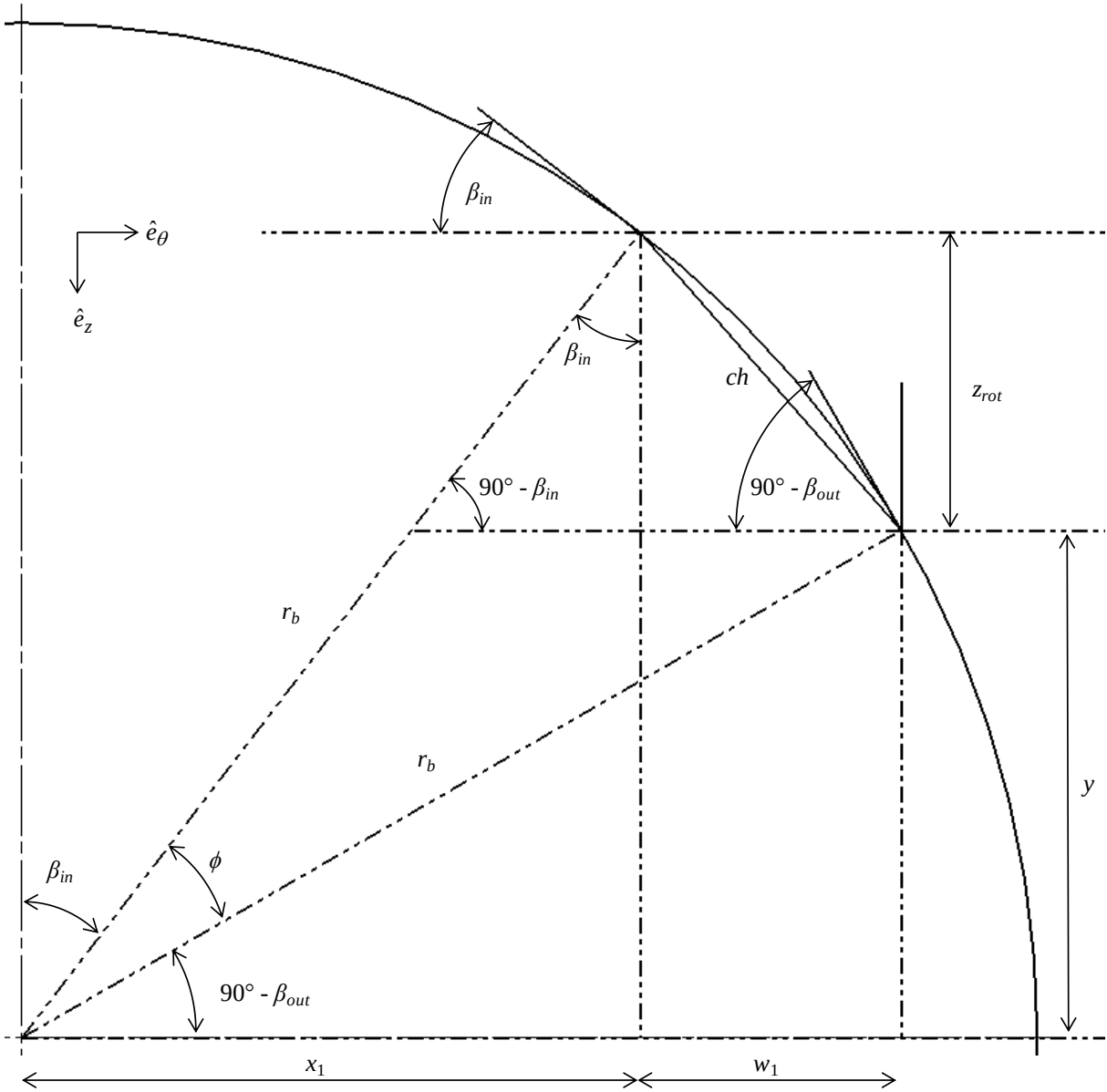


Figure 2.4. Detail of Circular Arc Blade Segment

However, the blade radius, r_b , is as of yet still an unknown quantity. Using standard geometrical principals, the following relations can be determined:

$$\cos(90^\circ - \beta_{out}) = \frac{x_1 + w_1}{r_b} \quad (2-96)$$

$$\sin(\beta_{in}) = \frac{x_1}{r_b} \quad (2-97)$$

$$\tan(\beta_{in}) = \frac{x_1}{z_{rot} + y} \quad (2-98)$$

$$\tan(90^\circ - \beta_{out}) = \frac{y}{x_1 + w_1} \quad (2-99)$$

With the inlet and outlet blade angles known, Equations (2-96) – (2-99) can be assembled into a system of Equations and solved for the unknown quantities r_b , w_1 , x_1 , and y . This assembled system is

$$\underbrace{\begin{bmatrix} \cos(90^\circ - \beta_{out}) & -1 & -1 & 0 \\ \sin(\beta_{in}) & 0 & -1 & 0 \\ 0 & 0 & -1 & \tan(\beta_{in}) \\ 0 & \tan(90^\circ - \beta_{out}) & \tan(90^\circ - \beta_{out}) & -1 \end{bmatrix}}_{\tilde{A}} \underbrace{\begin{bmatrix} r_b \\ w_1 \\ x_1 \\ y \end{bmatrix}}_{\tilde{B}} = \underbrace{\begin{bmatrix} 0 \\ 0 \\ -z_{rot} \tan(\beta_{in}) \\ 0 \end{bmatrix}}_{\tilde{C}} \quad (2-100)$$

The system in (2-100) can be solved for the unknown quantities, $\tilde{B}$, using standard linear algebra techniques:

$$\tilde{A}\tilde{B} = \tilde{C} \Rightarrow \tilde{B} = \tilde{A}^{-1}\tilde{C} \quad (2-101)$$

where the resulting expression for the blade radius is

$$r_b = \frac{z_{rot}}{\cos(\beta_{in}) - \cos(\beta_{out})} \quad (2-102)$$

and hence the chord length in Equation (2-95) can be determined.

The radius of the circular arc blade also allows the interior blade angles, β_i , to be defined as a function of axial location, z_i , as

$$\beta_i = 90^\circ - \sin^{-1}\left(\frac{z_{rot} - z_i + y}{r_b}\right) \quad (2-103)$$

where y is as shown in Figure 2.4 and is also determined from solving (2-100):

$$y = \frac{z_{rot} \cos(\beta_{out})}{\cos(\beta_{in}) - \cos(\beta_{out})} \quad (2-104)$$

2.3.2 Inlet Guide Vane Blades

The inlet guide vanes (IGV) are a set of non-rotating axially configured blades; the IGV does not consume any work since it does not rotate and hence has no tangential blade speed, therefore $\tilde{e}$ in (2-93) is zero. They do however serve to impart pre- or counter-swirl to the flow before entering the downstream rotor to enhance the rotor's performance.

Flow upstream of the IGV is assumed to be swirl-free, and so flow enters the IGV perpendicular to the tangential direction. This simplifies the blade radius calculation because the inlet angle is 90° . Figure 2.5 shows the circular arc segment for the IGV blades.

Note that the directional differences in the coordinate system used in Figures 2.4 and 2.5 are simply for convenience in using the portion of the circle in the first quadrant in a Cartesian plane, and that care must be taken to ensure that the blade angles work appropriately with one another from IGV to rotor, and from one rotor to the next.

Also note that the IGV angles (α) and the rotor blade angles (β) differ in name because they are referenced from inertial and rotating reference frames, respectively. This is discussed in detail in Section 2.4.

The camber angle and chord length are as in Equations (2-94) and (2-95), respectively, making the appropriate substitution of α for β . However the blade radius is simply

$$r_{b,IGV} = \frac{z_{IGV}}{\sin(\phi)} \quad (2-105)$$

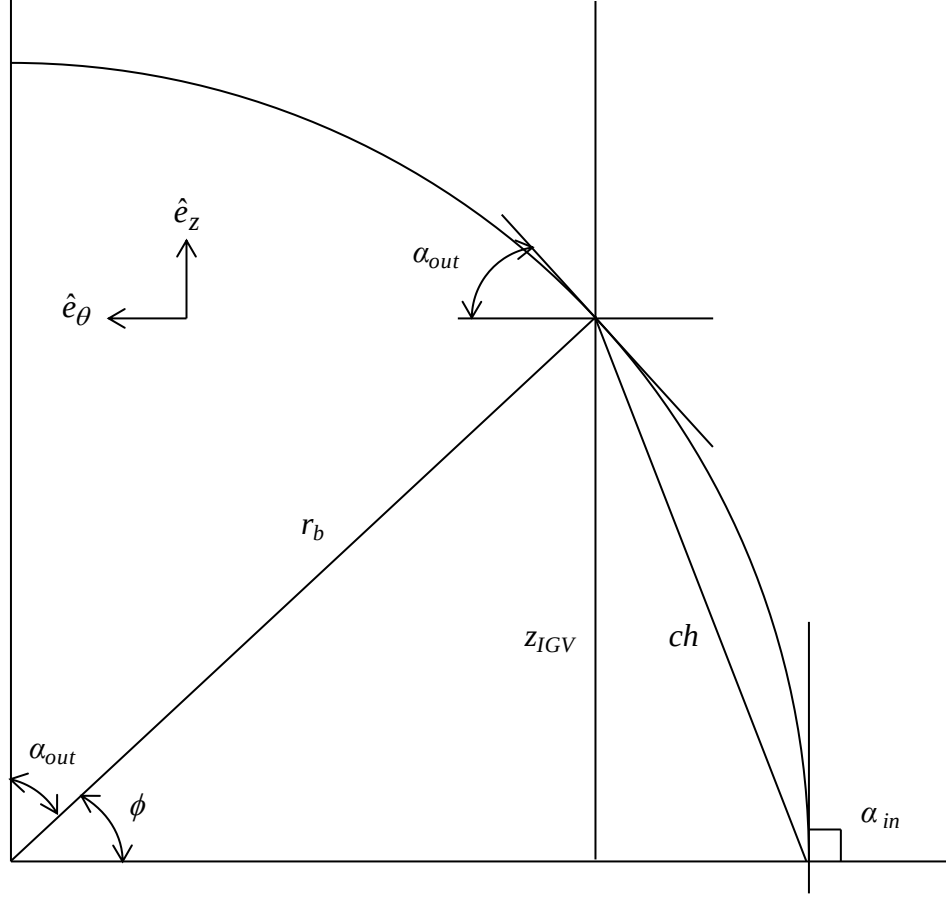


Figure 2.5. Circular Arc Blade Segment for IGV

The IGV blade angle (flow angle), α_i , at any axial location, z_i , is

$$\alpha_i = \alpha_{in} - \sin^{-1}\left(\frac{z_i}{r_b}\right) \quad (2-106)$$

Bearing in mind the directional change between Figures 2.4 and 2.5, the IGV shown in Figure 2.5 serves to impart counter-swirl to the flow entering the rotor shown in Figure 2.4.

2.4 Velocity Diagrams and Reference Frames

2.4.1 Inertial Reference Frame

The IGV is viewed purely from an inertial reference frame (“absolute system”) since it does not rotate. Flow velocities observed from the inertial reference frame, be they within the IGV or not, are called $\mathbf{c}$ (bold indicating a vector quantity such that $\mathbf{c}$ has magnitude c), where

$c_m \hat{e}_z$ is the velocity component in the meridional (for axial compressors, approximately axial) direction, $c_u \hat{e}_\theta$ is the velocity component in the tangential direction, and $c_r \hat{e}_r$ is the velocity component in the radial direction, and

$$\mathbf{c} = c_m \hat{e}_z + c_u \hat{e}_\theta + c_r \hat{e}_r \quad (2-107)$$

The velocity diagrams viewed at the inlet and outlet of the IGV are shown in Figure 2.6. The absolute flow angles, α , are measured relative to the tangential direction, and the radial component, generally assumed to be negligible¹ for simplicity, is not depicted.

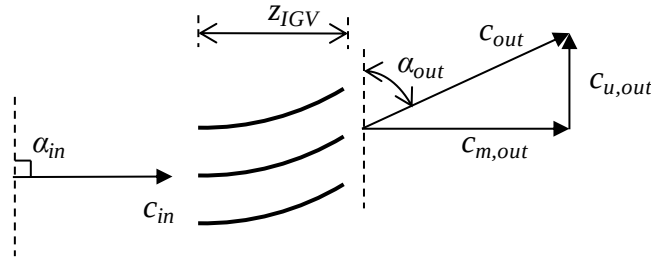


Figure 2.6. IGV Velocity Diagrams

2.4.2 Rotating Reference Frame

In contrast to the IGV where the blade curvature is matched by the velocity in the absolute system, the blade curvature in the rotor needs to be matched by the velocity as viewed from the rotating frame of reference (“relative system”) of the rotor itself. Since the rotation of the rotor is only in the tangential direction, the relative velocity vector, $\mathbf{w}$ (with magnitude w), differs from $\mathbf{c}$ by definition only in terms of the tangential velocity component, w_u , where

$$w_u = u - c_u \quad (2-108)$$

and u is the tangential velocity of the rotor at a given radius.

The entire relative velocity vector is then

¹ Radial equilibrium, i.e. zero radial velocity, is imposed for certain blade designs. This is discussed in Chapter 5.

$$\mathbf{w} = w_m \hat{e}_z + w_u \hat{e}_\theta + w_r \hat{e}_r \quad (2-109)$$

where w_m and w_r are equal to c_m and c_r , respectively.

The velocity diagrams at the rotor inlet and rotor outlet are shown in Figure 2.7 with the corresponding circular arc blades matched corresponding to the relative velocities. Note that the absolute velocity at the IGV outlet depicted in Figure 2.6 is the same as the absolute velocity at the rotor inlet in Figure 2.7.

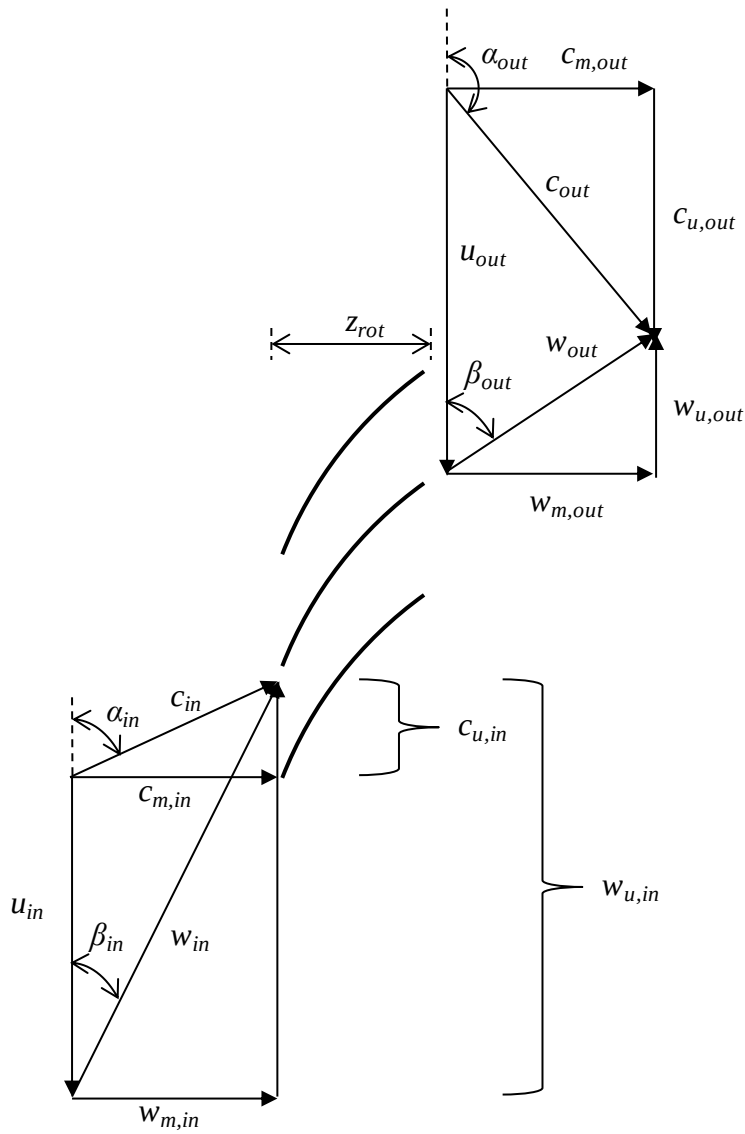


Figure 2.7. Rotor Velocity Diagrams

The IGV shown in Figure 2.6 imparts counter-swirl to the flow before entering the rotor depicted in Figure 2.7 since the tangential velocity component, c_u , is in the opposite direction of the rotation of the rotor. This results in a smaller inlet angle, β_{in} , than would be required if the IGV were to impart pre-swirl (in the same direction of the rotor rotation), or were simply absent (a swirl-free rotor inlet). The swirl-free inlet angle would be between the counter- and pre-swirl angles given the same c_m and u values.

As was briefly discussed in Section 1.4.1, counter rotation of impellers allows for increased power density in an axial compressor. As the term implies, the tangential velocity vectors for rotors and counter-rotors point in opposite directions. This also entails that the relative frame of reference, and hence β , switch accordingly. Figure 2.8 shows the velocity diagrams at the inlet and outlet of the counter-rotor. Note that the absolute velocity at the outlet of the rotor in Figure 2.7 is the same as the absolute velocity at the inlet of the counter-rotor in Figure 2.8. Any subsequent rotating stages will rotate in the opposite direction of the previous stage, and the absolute velocity vector at the inlet of a subsequent stage will be the same as the outlet absolute velocity vector of the previous stage.

In traditional multistage axial compressors, rotors are separated by non-rotating stators in order to reduce the swirl in the flow, decreasing the absolute tangential velocity, and in doing so increase the static pressure so that the subsequent rotor can add further swirl to the flow and increase the total pressure as per Equation (2-93). This is done because all rotors are on the mounted on the same shaft and so rotate at the same speed. Without the presence of the intermediate stators, subsequent rotor blade angles that need to match the relative velocity can then only add more swirl in the same direction creating larger and larger absolute velocities. Eventually the blade angles would become so steep that they would close off the flow passages

entirely or else direct the flow back upstream towards previous rotors. In either case this is undesirable, and so both α and β are limited to range between 15° and 165° .

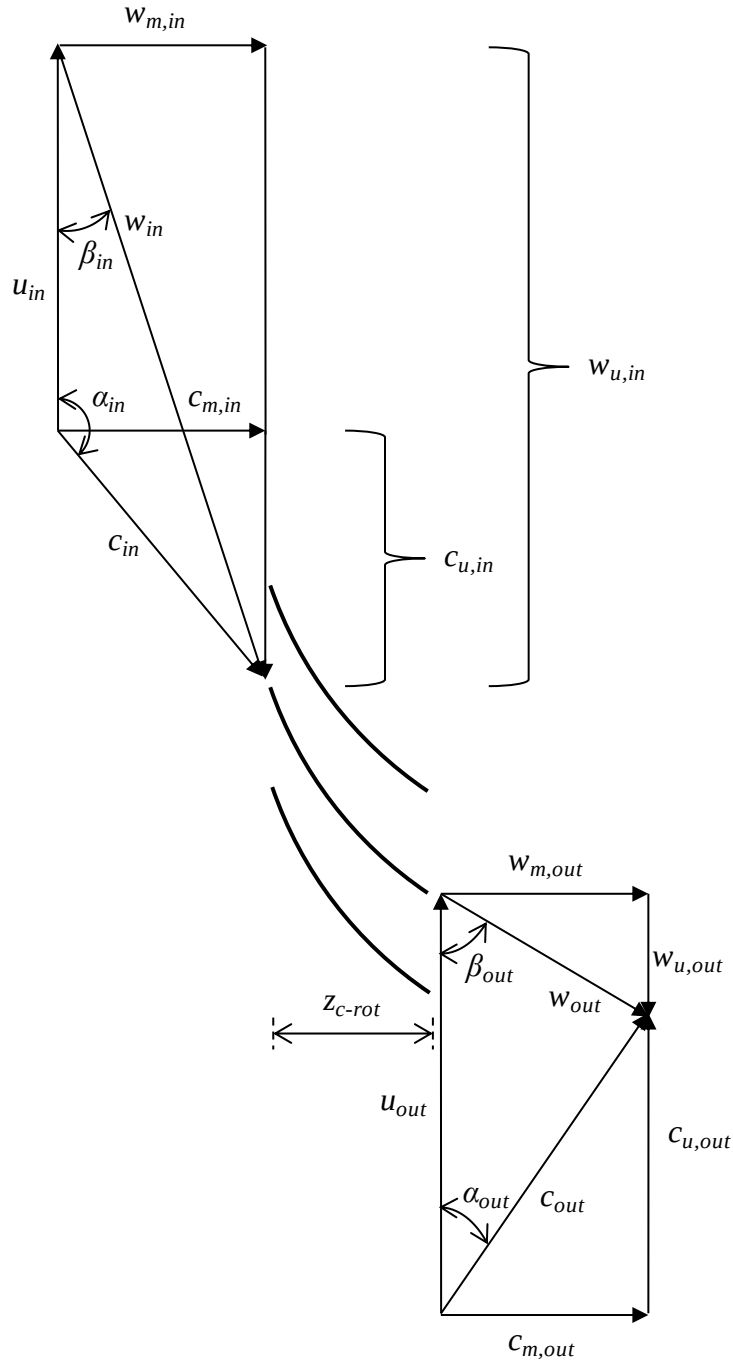


Figure 2.8. Counter-Rotor Velocity Diagrams

Additionally, while the relative flow velocity could be moderate, the absolute flow velocity is very likely to be supersonic. Supersonic (as well as transonic) flow can lead to the

formation of shocks and hence shock loss; Mach numbers greater than unity also contribute to other factors leading to decreases in operational efficiency [40]. This is the case whether viewing the flow from either the absolute or relative system. However, employing counter-rotation allows both relative and absolute velocities to be kept subsonic without the use of stators and without having exceedingly steep flow and blade angles. Figure 2.9 schematically illustrates how rotors with the same speed and rotation direction (all u vectors are the same) without intermediate stators quickly diminish the axial velocity with exceedingly steep blade angles. One can imagine that if the axial velocity were preserved, the absolute velocity vector magnitude would be very large indeed.

Figure 2.10 schematically illustrates how the employment of intermediate stators serves to keep both blade angles and absolute velocities moderate.

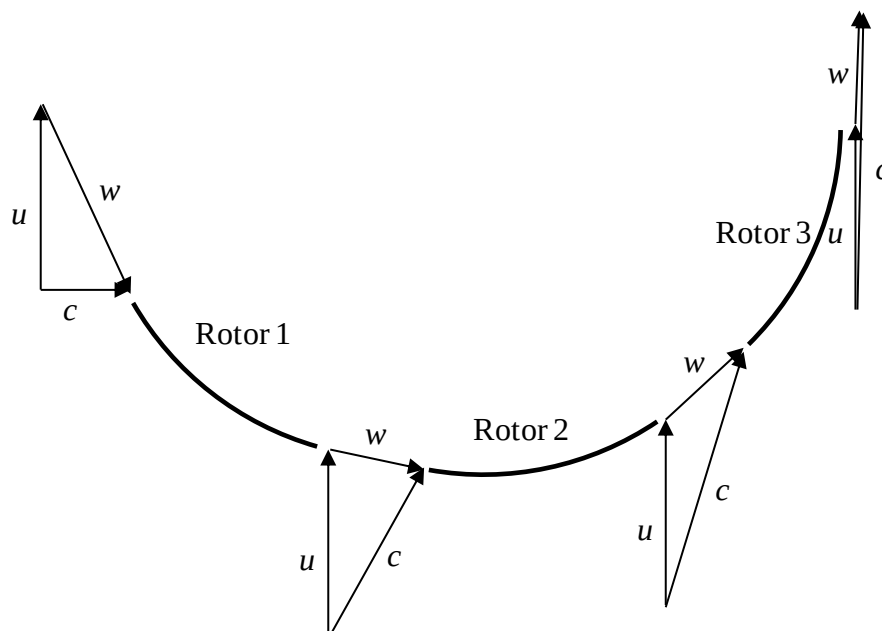


Figure 2.9. Schematic of Unidirectional Rotation and Singular Speed without Intermediate Stators

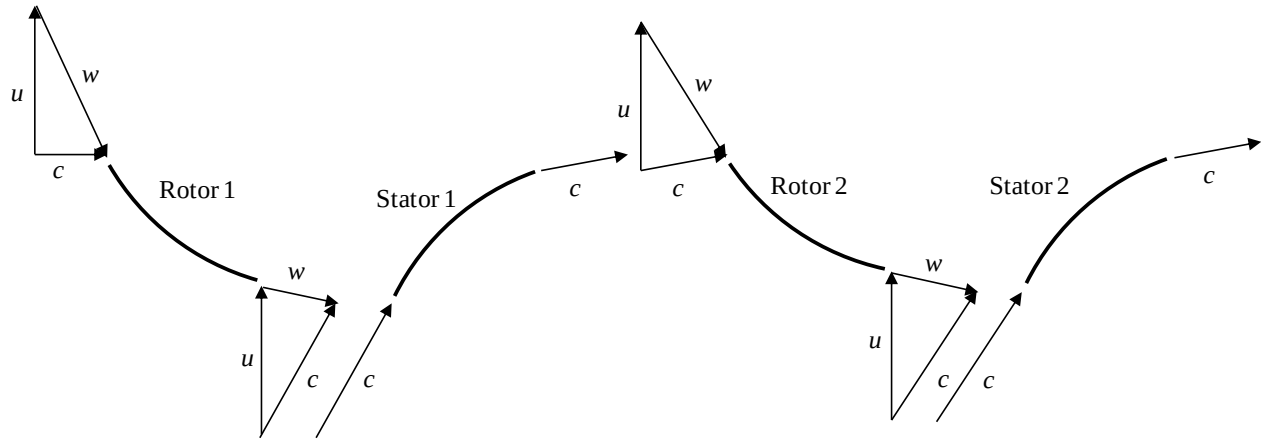


Figure 2.10. Schematic of Unidirectional Rotation and Speed with Intermediate Stators

Figure 2.11 schematically illustrates how counter rotation can avoid both steep blade angles and diminished axial velocity without the need for stators by having large relative velocity at each rotor inlet and merely changing the direction of the absolute velocity (all c vectors the same between rotors, different w vectors. The u vectors are unchanged between inlet and outlet for a given rotor.)

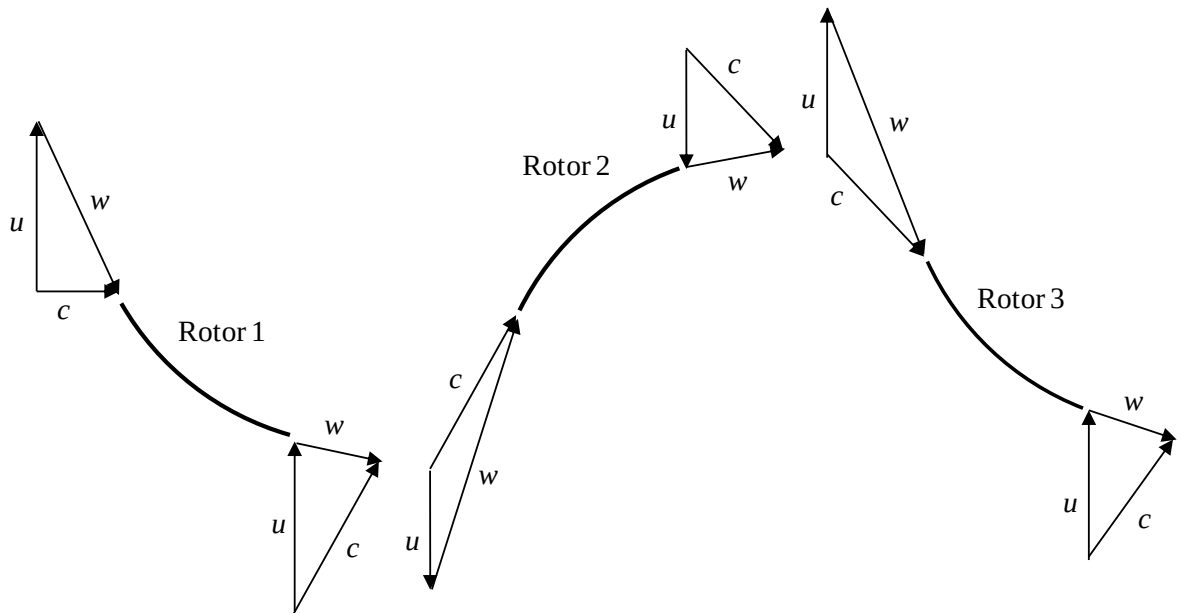


Figure 2.11. Schematic of Counter Rotation without Intermediate Stators

However, care must be taken to ensure the relative velocity at the inlet of each rotor and counter rotor is not excessively large; this is ensured by considering the relative Mach number.

The absolute Mach number is defined in Equation (2-37), and the absolute critical Mach number is defined in Equation (2-51); the total temperature used in (2-51) is based on the absolute velocity and is defined in (2-44).

The total (stagnation) temperature in the relative system, T_{tw} , is similar to that of (2-44) but uses the difference between the squares of the relative and absolute velocities such that

$$T_{tw} = T_{tc} + \frac{w^2 - c^2}{2c_p} \quad (2-110)$$

and T_{tc} has been replaced by (and is equivalent to) T_t in (2-44) for clarity. The critical speed of sound in the relative system, a_w^* , is then similar to (2-50) except uses the relative total temperature:

$$a_w^* = \sqrt{\frac{2}{\gamma + 1} \gamma R T_{tw}} \quad (2-111)$$

The relative Mach number, M_w^* , is then

$$M_w^* = \frac{w}{a_w^*} \quad (2-112)$$

Since it is desirable to keep both absolute and relative Mach numbers below 1 [40], it is convenient to select them as compressor design parameters. The working fluid and the total temperature in the absolute reference frame at the compressor inlet are generally known quantities; these allow for the absolute critical speed of sound to be determined from (2-50), and specifying an absolute critical Mach number in (2-51) determines the absolute velocity. However, simply specifying a relative Mach number to determine the relative velocity is not as straight forward because the relative total temperature upon which to base the relative speed of

sound itself depends on the relative velocity it is attempting to determine (see Equations (2-110)-(2-112)).

If the rotor blade speed is specified to determine the relative velocity, then imposing a relative Mach number over constrains the system. Therefore imposing both the absolute and relative Mach numbers sets the blade and hence rotational speed of the rotor. To do so, the definition of relative critical speed of sound in (2-111) must be substituted into (2-112) and squaring both sides. Solving for w^2 gives

$$w^2 = \frac{M_w^{*2} 2\gamma R T_{tw}}{\gamma + 1} \quad (2-113)$$

Substituting w^2 from (2-113) into (2-110) gives the relative total temperature in terms of relative critical Mach number, the absolute velocity, and the absolute total temperature. After some manipulation, the result is shown in (2-114).

$$T_{tw} = \frac{T_{tc} - \frac{c^2}{2c_p}}{1 - \frac{M_w^{*2} \gamma R}{c_p (\gamma + 1)}} \quad (2-114)$$

This then determines the relative critical speed of sound in (2-111), and (2-112) determines the relative velocity.

2.5 0- and 1-Dimensional Design Procedure

Thus far Chapter 2 has served to provide the necessary mathematical background and equations necessary to describe the flow thermodynamics and kinematics, as well as some blade geometry. Considering these at the inlet and outlet of an IGV and subsequent counter-rotating stages constitutes a 0 dimensional, or 0D, design. Selecting a span-wise radius between (or at) the hub and shroud and incrementally moving from inlet to outlet in the stream-wise direction

constitutes a 1D design. This section details the design variables selected, determined, and the constraints to which they must adhere. Table 2.1 shows an overview of Sections 2.5.1-2.5.3.

Table 2.1. 0D and 1D Design Considerations

Flow Properties	Geometry and Kinematics	Thermodynamics and Aerodynamics	Design Constraints
Gas Mixture Composition	Blade (Rotational) Speed	Equation of State	Circular Arc Blades
Compressible Flow	Blade Angles	Continuity	Constant Tip Radius
Total Properties	Flow Angles	Conservation of Energy	Constant Blade Thickness
Mas114s Flow Rate	Absolute and Relative Velocities	Conservation of Momentum	Subsonic in Relative and Absolute Systems
	Absolute and Relative Mach Numbers	Local Properties	Deceleration and Diffusion Limits

2.5.1 Inlet Conditions and Fluid Properties

First and foremost the composition of the working gas medium needs to be known. It is assumed that the compressor draws from a stationary fluid, hence the total temperature and pressure need also be known. Table 2.2 summarizes the inlet conditions.

Table 2.2. Compressor Inlet Conditions

Quantity	Variable	Unit	Equation
<i>Known Value</i>			
Component Mole Fraction	y_i	-	
Component Molecular Weight	MW_i	kg/kmol	
Total Temperature	T_{tc}	K	
Total Pressure	p_t	kPa	
Mass Flow Rate	$\dot{m}$	kg/s	
<i>Determined Value</i>			
Mixture Molecular Weight	MW	kg/kmol	(2-9)
Mixture Gas Constant	R	kJ/kg/K	(2-10)
Component Specific Heat	$c_{p,i}$	kJ/kg/K	(2-11)
Mixture Specific Heat	c_p	kJ/kg/K	(2-12)
Specific Heat Ratio	γ	-	(2-15)
Total Density	ρ_t	kg/m ³	(2-27)

Note that Equation (2-15) must be written in terms of γ . Also note that the ideal gas equation of state in (2-27) holds equally for total properties as for local properties, and that the total density, ρ_t , is the reciprocal of the total specific volume v_t .

2.5.2 Geometric and Kinematic Considerations

The fluid is assumed to enter the IGV swirl-free, i.e. with zero tangential velocity in the absolute reference frame. The selected inlet flow angle in Table 2.3 amounts to the outlet flow angle from the IGV, which is the inlet flow angle to the rotor as depicted in Figures 2.6 and 2.7. In the absence of an IGV, the inlet flow angle to the rotor is simply 90° .

The total temperature in the absolute reference frame determines the critical speed of sound. Defining M_c^* then sets the absolute flow velocity as well as all static properties as per Equations (2-58)-(2-60). Since α at the rotor inlet is defined, the meridional and tangential components of the absolute flow velocity are therefore set at the inlet according to the velocity diagrams shown in Figures 2.6 and 2.7, namely

$$\frac{c_m}{c} = \sin(\alpha) \quad (2-115)$$

$$\frac{c_u}{c} = \cos(\alpha) \quad (2-116)$$

Defining M_w^* determines the relative total temperature and hence the relative critical speed of sound as per Equations (2-114) and (2-111), respectively. Equation (2-112) then determines the relative velocity. Since the meridional velocity is not reference frame-dependent, i.e. $w_m = c_m$, the relative tangential velocity and relative flow angle (no incidence is assumed for simplicity and so the blade angle is the same as the relative flow angle) are also known from the velocity diagrams in Figures 2.6 and 2.7:

$$\frac{w_m}{w} = \sin(\beta) \quad (2-117)$$

$$\frac{w_u}{w} = \cos(\beta) \quad (2-118)$$

With the absolute and relative flow velocity diagrams fully defined, the tangential blade speed is known:

$$u = c_u + w_u \quad (2-119)$$

With the mass flow rate, meridional velocity, and static density known, continuity (Equation (2-8)) sets the cross-sectional area perpendicular to the rotor inlet. The cross-sectional area itself is the ring shaped segment between the hub and shroud minus the thickness of the blades:

$$A = \pi(r_t^2 - r_h^2) - N_b t_b (r_t - r_h) \quad (2-120)$$

N_b and t_b are the number of blades and blade thickness while r_h and r_t are the hub and shroud radii, respectively. Due to the construction of the woven impeller, the blade thickness is constant in both the span-wise and stream-wise directions. Both a blade number and thickness are selected. Equation (1-120) is quadratic in terms of both hub and tip radius. Rewriting (1-120) in terms of hub/tip ratio, r_h/r_t , (in effect substituting Equation (1-122) for r_h) and selecting its value to be between 0 and 1, the quadratic formula determines tip radius.

$$r_t = \frac{(1 - r_h/r_t)t_b N_b \pm \sqrt{(- (1 - r_h/r_t)t_b N_b)^2 - 4\pi(1 - (r_h/r_t)^2)(-A)}}{2\pi(1 - (r_h/r_t)^2)} \quad (2-121)$$

The positive r_t value is selected as the physically viable solution. The hub/tip ratio then determines the hub radius.

$$r_h = (r_h/r_t) r_t \quad (2-122)$$

The area-mean radius, r_m , selected for calculation of the blade speed, work transfer, etc is

$$r_m = \sqrt{\frac{r_t^2 + r_h^2}{2}} \quad (2-123)$$

and the rotational speed of the impeller, ω , is

$$\omega = \frac{u}{r_m} \quad (2-124)$$

Thus the rotor inlet (IGV outlet) is determined. With the selection of aspect ratio and number of blades, the IGV inlet can be determined by working backwards to the inlet by satisfying continuity, keeping the tip radius constant, and imposing the 90° inlet flow angle.

The rotor outlet is also determined by satisfying continuity and keeping tip radius constant. This is done using either of two methods (the method used will be discussed as it arises in subsequent chapters). The first is simply to keep c_m constant at all stream-wise and span-wise locations throughout the flow field. The second is temporarily holding c_m constant as a reference and setting the outlet blade angle determines the relative tangential velocity at the outlet. With constant tip radius, the blade speed is also unchanged. This allows the outlet c_u value to be determined by Equation (2-119). This determines the amount of work done on the fluid at the tip and all other radii¹, and hence the outlet total temperature and pressure as per Equation (2-93).

With the outlet temperature known, the outlet a_c^* is known. Instead of keeping c_m constant, it is rather the axial Mach number, M_{cm}^* , that is held constant from inlet to outlet.

$$M_{cm}^* = \frac{c_m}{a_c^*} \quad (2-125)$$

¹ This is ensured by the Free Vortex Condition, which imparts equal work to all radii across the span. This is discussed in Chapter 5.

This determines the outlet c_m value, which is slightly faster than at the inlet due to the increase in speed of sound, and since the outlet c_u and w_u values are known, the completed relative and absolute velocity diagrams and their corresponding Mach numbers are all known. The new cross-sectional area is therefore also determined, and so the new hub and mean radii are known. The new hub radius is determined by solving the quadratic formula of (2-120) in terms of hub radius since the tip radius is known. Again the positive radius value is selected as the physically viable result.

$$r_h = \frac{-t_b N_b \pm \sqrt{(t_b N_b)^2 + 4\pi(\pi_t^2 - r_t t_b N_b - A)}}{-2\pi} \quad (2-126)$$

For a 1D design the stream-wise direction is traversed incrementally from inlet to outlet, thus requiring an axial length. The aspect ratio, AR , of the rotor is defined as ratio of the difference of tip and hub radii to the axial length from inlet to outlet. Selecting AR , the axial rotor length, z_{rot} , is then

$$z_{rot} = \frac{r_t - r_h}{AR} \quad (2-127)$$

and the blade radius can be determined in Equation (2-102) with the definition of an outlet blade angle.

The outlet blade angle is indirectly selected by defining a blade turning angle, $\Delta\beta$, which is added to the value of the calculated inlet angle as

$$\beta_{out} = \beta_{in} + \Delta\beta \quad (2-128)$$

where $\Delta\beta \geq 0$ to ensure that the relative tangential velocity decreases across the blade, thereby compressing the fluid ($\Delta\beta < 0$ would serve to increase the relative tangential velocity and expand the fluid). The same process outlined for the 0D design is then followed with the exception that the blade angles at each stream-wise location are determined from (2-103).

Table 2.3 shows a summary of the known, or in many cases selected design variables. Table 2.4 shows the values that these selections determine. Note that in the absence of an IGV, all entries in the Location column that denote an IGV location are instead rotor locations.

Table 2.3. Compressor Geometry and Flow Kinematics – Design Selections			
Quantity	Location	Variable	Unit
Flow Angle	First Rotor Inlet	α	°
Absolute Mach Number	IGV Inlet	M_c^*	-
Hub/Tip Ratio	IGV Inlet	r_h/r_t	-
Aspect Ratio	IGV and Each Rotor	AR	-
Blade Thickness	IGV and Each Rotor	t_b	m
Number of Blades	IGV and Each Rotor	N_b	-
Relative Mach Number	All Rotor Inlets	M_w^*	-
Outlet Blade Angle	All Rotor Outlets	β	°

For subsequent counter rotors and rotors, all *Known/Selected* values in Table 2.3 are again chosen with the exception of α and M_c^* , which are taken as the outlet values from the previous rotor as depicted in Figure 2.8.

Table 2.4. Compressor Geometry and Flow Kinematics – Determined Values			
Quantity	Variable	Unit	Equation
<i>Rotor Inlet/Outlet</i>			
Absolute Velocity	c	m/s	(2-51)
Meridional Velocity	c_m	m/s	(2-115)/(2-125)
Absolute Tangential Velocity	c_u	m/s	(2-116)/(2-119)
Relative Velocity	w	m/s	(2-112)
Relative Tangential Velocity	w_u	m/s	(2-118)
Blade Tangential Velocity	u	m/s	(2-119)
Blade Angle	β	°	(2-117)/(2-128)
Blade Radius	r_b	m	(2-102)
Total Density	ρ_t	kg/m ³	(2-27)
Static Density	ρ	kg/m ³	(2-60)
Cross-Sectional Area	A	m ²	(2-8)
Hub Radius	r_h	m	(2-122)/(2-126)
Tip Radius	r_t	m	(2-121)
Axial Length	z	m	(2-127)
Total Pressure	p_t	kPa	(2-93)

Table 2.4 (cont'd)

Static Pressure	p	kPa	(2-59)
Total Absolute Temperature	T_{tc}	K	(2-93)
Total Relative Temperature	T_{tw}	K	(2-114)/(2-110)
Static Temperature	T	K	(2-58)
Specific Shaft Work Input	$\tilde{e}$	kJ/kg	(2-93)
Power Input	$\tilde{P}$	kW	(2-78)
Absolute Speed of Sound	a_c^*	m/s	(2-50)
Absolute Mach Number	M_c^*	-	(2-51)
Absolute Meridional Mach Number	M_{cm}^*	-	(2-125)
Relative Speed of Sound	a_w^*	m/s	(2-111)
Relative Mach Number	M_w^*	-	(2-112)

2.5.3 Variable Constraints

The selection of the design variables described in Table 2.3 must be made in such a way that the compressor does not underperform. Underperformance can happen in one of two ways: either the compressor is under-designed such that it is capable of producing a higher pressure ratio with additional power input, or it is over-designed in the sense that it is asked to do too much and the flow will separate from the blades. In the second circumstance, the input power serves to heat the fluid but not increase its pressure; additional power input would only serve to heat it further without any additional pressure increase.

As mentioned in section 2.4.2, keeping both the absolute and relative critical Mach numbers below 1 avoids the potential for choking and shock formation and the performance losses associated with the entropy increase across the shock [40].

Unlike turbines which serve to expand and accelerate the flow in a favorable pressure gradient regime, compressors compress and decelerate the flow in an adverse pressure gradient regime. The adverse pressure gradient flow is much more prone to boundary layer separation than flow in a favorable pressure gradient [41]. In order to avoid separation, the deceleration

must be limited. De Haller [42] suggested that the ratio of the outlet to inlet relative velocities, dH , be no less than 0.72:

$$dH = \frac{w_2}{w_1} \geq 0.72 \quad (2-129)$$

It is for this reason that M_{cm}^* was selected to remain constant rather than simply c_m as outlined in Section 2.5.2: if $c_m = w_m$ is allowed to increase, then the relative tangential velocity has more room to decrease since it is the entire vectorial magnitude from Equation (2-109) being considered in (2-129). As per Equation (2-119), a decrease in w_u means an increase in c_u for a given u , and an increase in c_u results in an increase in work transfer as per Equation (2-93).

In addition to the relative flow deceleration, flow diffusion within the rotor blade row must also be limited. The diffusion factor, DF , is defined as [43]

$$DF = \left(1 - \frac{w_2}{w_1}\right) + \frac{w_{u2} - w_{u1}}{2\sigma w_1} \quad (2-130)$$

The term in parentheses in (2-130) is essentially the deceleration ratio described by the de Haller number, which is essentially a local diffusion factor [44]. The second term in (2-130) accounts for blade turning (numerator), and blade solidity, σ , which is a measure of how much of the flow area is occupied by blades:

$$\sigma = \frac{ch}{t} \quad (2-131)$$

where ch is the chord length as defined in (2-95), and the blade stagger, t , is

$$t = \frac{2\pi}{N_b} \quad (2-132)$$

As the solidity increases, the flow better follows the blades [44], and an increase serves to decrease diffusion as per (2-130). Likewise, a decrease in blade turning, which reduces the

loading on the blades, also serves to reduce diffusion. Values of DF should be kept below 0.45 to avoid large losses in total pressure [43].

Table 2.5 summarizes the constraints discussed in this section.

Table 2.5. Summary of Constraints	
Quantity	Variable Constraint
Absolute Mach Number	$M_c^* \leq 0.7$
Relative Mach Number	$M_w^* \leq 0.8$
de Haller Deceleration	$dH \geq 0.72$
Diffusion Factor	$DF \leq 0.45$
Hub/Tip Ratio	$r_h/r_t < 1$
Aspect Ratio	$0.5 < AR < 2.0$
Absolute Flow Angle	$15^\circ < \alpha < 165^\circ$
Relative Flow/Blade Angle	$15^\circ < \beta < 165^\circ$

All of the constraints mentioned in Table 2.5 that can vary radially must be satisfied at all span-wise locations (hub, tip, mean radii).

Of course the question becomes how to select the variables in Table 2.3 such that all constraints are satisfied without leaving any room for improvement in terms of work transmission and total pressure ratio gained by the fluid. This is discussed in Chapter 6.

Chapters 4 and 5 discuss the “star-pattern” blade design and the “free-vortex” blade design, respectively. Each of these is a 2D design that describes the blade shapes and corresponding flow properties at *all* stream-wise and span-wise locations, rather than at a single radius and either all stream-wise locations (1D) or simply inlet and outlet (0D).

All 0D, 1D, and 2D designs are baseline designs, and while the flow constraints in Table 2.5 serve as good guidelines to achieve desirable compressor performance with limited losses, 3D flow simulation is required in order to account for and capture realistic 3D flow features, namely turbulence, and more accurately predict the performance of the 0D, 1D, and 2D design-based compressor. Chapter 3 discusses 3D flow simulation, and Chapters 4 and 5 present the 3D

simulated results compared to the baseline performance predictions for their respective blade types.

CHAPTER 3 SIMULATION OF AXIAL COMPRESSORS

Historically speaking, the study of fluid dynamics as they are understood today was started in seventeenth century France and England and was based purely on experiment. Theoretical fluid dynamic principles were then developed over the course of the eighteenth and nineteenth centuries, and up until around the 1960's, fluid dynamics on the whole were studied entirely from an experimental and theoretical standpoint. The development of numerical algorithms to solve physical problems and the arrival of the high speed digital computer on which to implement them has led to the latest approach to studying fluid dynamics, namely Computational Fluid Dynamics (CFD) [45].

This chapter serves to introduce the governing flow equations, the techniques used to model their behavior, and the solution methodology implemented by the solver used for compressor simulation in this dissertation: ANSYS CFX.

3.1 Computational Fluid Dynamics – Governing Equations

CFD as it relates to numerical compressor simulation takes into account all of the governing equations and principles from Chapter 2 that apply to compressor modeling and fluid dynamics in general, but uses them in their 3D and transient entirety to get a more realistic picture of the flow behavior. The governing equations and principles repeated here are

• Conservation of mass	Continuity
• Conservation of linear and angular momentum	Newton's second law
• Conservation of energy	First law of thermodynamics

The governing equations are derived in *conservation form*, i.e. using a control volume that is fixed in space, and are collectively known as the transport equations. Since the 1D

versions have been described in Chapter 2, details of the full (albeit similar) 3D derivations can be found in [45].

The continuity equation for a compressible fluid in an inertial reference frame is

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{V}) = 0 \quad (3-1)$$

where $\mathbf{V}$ is the three dimensional velocity vector with components u , v , and w .

The momentum equations as they pertain to fluid dynamics are known as the Navier-Stokes equations, so called to honor the Frenchman Navier and the Englishman Stokes who independently obtained the equations in the nineteenth century [45]. The equations account for body forces acting on the fluid mass within the control volume (gravitational, electrical, magnetic) and surface forces (pressure imparted from neighboring fluid elements as well as normal and shear stresses imparted due to friction). They are a set of scalar equations written in vector form as

$$\frac{\partial(\rho \mathbf{V})}{\partial t} + \nabla \cdot (\rho \mathbf{V} \otimes \mathbf{V}) = -\nabla p + \nabla \tau + S_M \quad (3-2)$$

where p is pressure, τ is stress, and S_M is a momentum source. Writing (3-2) as a set of scalar equations, the x -, y -, and z -components are

$$\frac{\partial(\rho u)}{\partial t} + \nabla \cdot (\rho u \mathbf{V}) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + \rho f_x \quad (3-3)$$

$$\frac{\partial(\rho v)}{\partial t} + \nabla \cdot (\rho v \mathbf{V}) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + \rho f_y \quad (3-4)$$

$$\frac{\partial(\rho w)}{\partial t} + \nabla \cdot (\rho w \mathbf{V}) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} + \rho f_z \quad (3-5)$$

where τ_{ij} is a stress in the j direction perpendicular to the i -axis. Normal stresses are denoted when $i = j$ as depicted by example in Figure 3.1 and are related to the change in volume with

respect to time. Shear stresses are when $i \neq j$ and are similarly depicted by example in Figure 3.2 and are related to the rate of shear deformation with respect to time. Body forces per unit mass acting in the i direction are f_i .

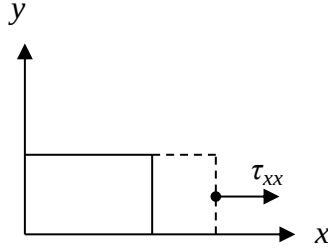


Figure 3.1. Normal Stress Acting on Fluid Element

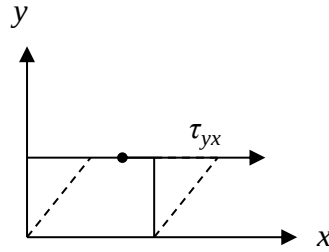


Figure 3.2. Shear Stress Acting on Fluid Element

In the case of a Newtonian fluid, the fluid stresses are related proportionally to the time rate of strain (velocity gradients). The relationships between the stresses and the velocity gradients were also obtained by Stokes [45]:

$$\tau_{xx} = \lambda(\nabla \cdot \mathbf{V}) + 2\mu \frac{\partial u}{\partial x} \quad (3-6)$$

$$\tau_{yy} = \lambda(\nabla \cdot \mathbf{V}) + 2\mu \frac{\partial v}{\partial y} \quad (3-7)$$

$$\tau_{zz} = \lambda(\nabla \cdot \mathbf{V}) + 2\mu \frac{\partial w}{\partial z} \quad (3-8)$$

$$\tau_{xy} = \tau_{yx} = \mu \left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \right) \quad (3-9)$$

$$\tau_{xz} = \tau_{zx} = \mu \left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z} \right) \quad (3-10)$$

$$\tau_{yz} = \tau_{zy} = \mu \left(\frac{\partial w}{\partial y} + \frac{\partial v}{\partial z} \right) \quad (3-11)$$

where μ is molecular (dynamic) viscosity, and although still not definitively confirmed at present day, the widely used Stokes hypothesis states that

$$\lambda = -\frac{2}{3}\mu \quad (3-12)$$

Equations (3-6) – (3-12) can be expressed concisely in tensor form:

$$\boldsymbol{\tau} = \mu \left(\nabla \mathbf{V} + (\nabla \mathbf{V})^T - \frac{2}{3} \delta \nabla \cdot \mathbf{V} \right) \quad (3-13)$$

where δ is the identity matrix (Kronecker Delta function where $\delta = 1$ for $i = j$ and $\delta = 0$ for $i \neq j$).

The energy equation relates the rate of energy change inside a fluid element to the net heat flux into the element and the rate of work done on the element by body and surface forces.

Expressed in terms of total enthalpy, h_t , the total energy equation is

$$\frac{\partial(\rho h_t)}{\partial t} - \frac{\partial p}{\partial t} + \nabla \cdot (\rho \mathbf{V} h_t) = \nabla \cdot (\lambda \nabla T) + \nabla \cdot (\mathbf{V} \cdot \boldsymbol{\tau}) + \mathbf{V} \cdot \mathbf{S}_M + \rho \dot{q} \quad (3-14)$$

where

$$h_t = h + \frac{1}{2}(\mathbf{V} \cdot \mathbf{V}) \quad (3-15)$$

$\nabla \cdot (\mathbf{V} \cdot \boldsymbol{\tau})$ is the work done due to viscous stress, $\mathbf{V} \cdot \mathbf{S}_M$ is work done by external momentum sources, and $\rho \dot{q}$ is the volumetric heat addition [45], [46]. Note that the pressure change with respect to time is subtracted on the left-hand-side of (3-14) precisely because the energy equation is expressed in terms of enthalpy rather than internal energy (see Equation (2-22)).

3.2 RANS Equations

Laminar flow is characterized as flow with smooth parallel streamlines in which viscous forces play a large role and there is no significant mixing between neighboring fluid particles; turbulent flows are characterized as flows with irregular fluid motions that fluctuate in both space and time where inertial forces mostly out-weigh viscous forces and with significant mixing occurring between neighboring fluid particles [47]. The Navier-Stokes equations (3-2) fully describe both laminar and turbulent flows (including transition from one to the other) simultaneously. Due to the complexity captured by the Navier-Stokes equations, they can only be solved analytically for a limited number of idealized simple flows.

Since turbulent flows are stochastic, the mathematics become more tractable when dealing with flow quantities in an average sense. The Reynolds Decomposition serves to separate the 3D turbulent velocity field into its mean and fluctuating components by subtracting the mean portion from the full flow field:

$$v_i = V_i - \bar{V}_i \Rightarrow V_i = \bar{V}_i + v_i \quad (3-16)$$

where v_i is the fluctuating velocity in the i -direction, and the average velocity in the i -direction is defined as

$$\bar{V}_i = \frac{1}{\Delta t} \int_t^{t+\Delta t} V_i dt \quad (3-17)$$

with Δt being much larger than the largest turbulent time scale. The CFX solver additionally weights the averages according to the local density.

Substituting (3-16) into the transport equations and taking the mean over a long period of time leads to the mean continuity equation and Reynolds Averaged Navier-Stokes (RANS)

equations. The continuity equation (3-1) is effectively unchanged in form by this substitution, but is stated here in terms of mean velocity using index notation:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho \bar{V}_i) = 0 \quad (3-18)$$

The mean momentum (RANS) equations become

$$\frac{\partial (\rho \bar{V}_i)}{\partial t} + \frac{\partial}{\partial x_j} (\rho \bar{V}_i \bar{V}_j) = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} (\tau_{ij} - \rho \overline{v_i v_j}) + S_M \quad (3-19)$$

There is a distinct difference between (3-19) and (3-2), namely the additional $\rho \overline{v_i v_j}$ term, which are known as the “turbulent” or “Reynolds” stresses. These arise mathematically from distributing the decomposed velocity in the nonlinear second term on the left-hand-side of (3-2) and subtracting the averaged product of the fluctuating components to the other side. They act as stresses similar to the way that the viscous stresses arise from momentum transfer at the molecular level, so the Reynolds stresses arise from momentum transfer by the fluctuating velocity field [41].

The Reynolds averaged total energy equation is

$$\frac{\partial (\rho \bar{h}_t)}{\partial t} - \frac{\partial \bar{p}}{\partial t} + \frac{\partial}{\partial x_j} (\rho \bar{V}_j \bar{h}_t) = \frac{\partial}{\partial x_j} \left(\lambda \frac{\partial T}{\partial x_j} - \rho \overline{v_j h} \right) + \frac{\partial}{\partial x_j} (\bar{V}_i (\tau_{ij} - \rho \overline{v_i v_j})) + \rho \dot{q} \quad (3-20)$$

Much as the RANS equations contain the additional Reynolds stress term, the Reynolds averaged total energy equation contains this same term as well as an additional flux term $\rho \overline{v_j h}$, both of which arise in this case from distributing the decomposed velocity in the nonlinear third term on the left-hand-side of Equation (3-14). The kinetic energy portion of the mean total enthalpy now includes a contribution from the turbulent kinetic energy, k

$$\bar{h}_t = \bar{h} + \frac{1}{2} (\bar{V}_i \bar{V}_i) + k \quad (3-21)$$

where

$$k = \frac{1}{2} \overline{v_i^2} \quad (3-22)$$

The $\frac{\partial}{\partial x_j} (\overline{V_i} (\tau_{ij} - \rho \overline{v_i v_j}))$ term in (3-20) is the viscous work term that can be enabled or disabled within the ANSYS CFX preprocessing software, CFX-Pre. All simulations performed for this dissertation include the viscous work term.

3.3 Turbulence Modeling

Turbulent flow is comprised of random fluctuations in the velocity field with respect to both space and time where inertial forces dominate over viscous forces. The dimensionless Reynolds number, Re , relates the characteristic length scale, time scale, and viscosity of a flow to characterize it as laminar, turbulent, or transitional.

$$Re = \frac{VL\rho}{\mu} \quad (3-23)$$

Turbulent flows are therefore flows with large Reynolds number. The characteristic length used in axial compressor modeling is the axial chord length (Equation (2-95)), and the characteristic velocity is the relative velocity (Equation (2-109)).

These characteristic dimensions used to determine Reynolds number are the largest scales, but are by no means the only scales of importance. The large scales are in the energy-containing range that provide the energy to the turbulent fluctuations. The concept of the energy cascade dictates that large turbulent motions, or “eddies”, tend to break up and transfer their energy to smaller eddies, which in turn break up and transfer their energy to still smaller eddies. This “energy cascade” continues to a scale sufficiently small such that the molecular viscosity is able to dissipate the kinetic energy and the eddies become stable [41].

Numerical simulation of turbulent fluid flow necessitates a discretization of the flow field into a mesh of finite volumes. However creating a sufficiently fine mesh with length scales small enough to capture the effects of the energy cascade in the dissipation range is computationally prohibitive even with modern super computers [46]. It is for this reason that much effort has been undertaken in the CFD community to model turbulent flow behavior without having to resolve an overly fine mesh.

Solving the Reynolds averaged equations (Equations (3-18) – (3-20)) rather than the instantaneous equations (Equations (3-1), (3-2), (3-14)) throughout a finite volume mesh serves to alleviate significant computational effort, but the averaging process introduces additional unknowns to the equations in what is known as the “closure problem”.

The closure problem is (of course) problematic because there are four governing equations for the mean flow (mean continuity and RANS), but where the instantaneous equations have four unknowns (the three velocity components and pressure), the averaged equations have the three mean velocity components, mean pressure, and additional unknowns in the form of the averaged products of fluctuating components, namely the Reynolds stresses.

A system of equations with more unknowns than there are equations is said to be unclosed. In order to close the system, the Reynolds stresses need to be modeled by additional equations of known quantities.

3.3.1 Turbulent Viscosity Hypothesis

The turbulent (eddy) viscosity hypothesis was postulated by Boussinesq in 1877 and states that the momentum transfer of turbulent eddies can be modeled by a turbulent viscosity in the same manner that momentum transfer by molecular motion in a fluid can be attributed to

molecular viscosity. It is mathematically analogous to the stress/strain rate relationship for a Newtonian fluid as per Equations (3-6) – (3-11).

Thus eddy viscosity models serve to relate the Reynolds stresses to the mean velocity gradients via a proportionality constant equal to the turbulent viscosity, μ_t :

$$-\overline{\rho v_i v_j} = \mu_t \left(\frac{\partial \bar{V}_i}{\partial x_j} + \frac{\partial \bar{V}_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \left(\rho k + \mu_t \frac{\partial \bar{V}_k}{\partial x_k} \right) \quad (3-24)$$

The turbulent viscosity itself then must be modeled. The following sections detail what are known as statistical turbulence models due to their use of averaged flow quantities that approximate the viscous contribution of the turbulent eddies.

3.3.2 Zero and One Equation Turbulent Viscosity Models

Zero equation models are the simplest of the turbulent viscosity models. They use an algebraic equation to compute a uniform value for μ_t rather than solving any additional transport equations, hence the name zero equation. The zero equation model used in ANSYS CFX is one proposed by Prandtl and Kolmogorov that is the product of a turbulent length scale, l_t , and a turbulent velocity scale U_t :

$$\mu_t = \rho f_\mu U_t l_t \quad (3-25)$$

The velocity scale is taken as the largest velocity in the flow field, f_μ is a proportionality constant, and the length scale is

$$l_t = \frac{\Psi^{1/3}}{7} \quad (3-26)$$

where Ψ is the volume of the fluid domain. The model however only accurately models free shear flow and is unsuitable for compressor modeling [41], [46].

One equation models solve a single additional transport equation in terms of either the turbulent kinetic energy, k (3-22), or turbulent energy dissipation rate, ε . The turbulent viscosity is modeled as

$$\mu_t = c\rho k^{1/2}l_m \quad (3-27)$$

where l_m is the mixing length and c is a model constant. These models are not widely used because the mixing length, itself a function of time and space, is not modeled but rather must be specified based on the flow regime prior to solving the equations throughout the flow field and so are insensitive to changes in length scale and dissipation rate [41]. One equation models were not used in this dissertation to simulate axial compressor flow.

3.3.3 Two Equation Turbulent Viscosity Model – k - ε

Two equation eddy viscosity models solve two transport equations of turbulent quantities in addition to the mean continuity and RANS equations. Historically nearly all two equation models select the turbulent kinetic energy, k (3-22), as one of the variables with the other being either the product kL (L being a characteristic length scale), or ω^2 (square of turbulent frequency), or τ (turbulent time scale) as examples. The most widely used two equation model is the k - ε model, which solves the additional transport equations for the eponymous turbulent quantities k and ε , the turbulence dissipation rate. Length and time scales can be formed directly from k and ε ($L = k^{3/2}/\varepsilon$ and $\tau = k/\varepsilon$, respectively) and so the k - ε model has the distinct advantage over one equation models in that a flow-dependent mixing length need not be specified [41].

The following development is based on [46]. With the addition of the variables k and ε to the system of equations, the mean continuity equation is unchanged and as is shown in Equation (3-18). Implementing the turbulent viscosity hypothesis and substituting for the Reynolds stresses as in Equation (3-24), the momentum equations are then

$$\frac{\partial(\rho\bar{V}_i)}{\partial t} + \frac{\partial}{\partial x_j}(\rho\bar{V}_i\bar{V}_j) = -\frac{\partial p'}{\partial x_i} + \frac{\partial}{\partial x_j} \left(\mu_{eff} \left(\frac{\partial \bar{V}_i}{\partial x_j} + \frac{\partial \bar{V}_j}{\partial x_i} \right) \right) + S_M \quad (3-28)$$

where p' is a modified pressure defined as

$$p' = p + \frac{2}{3}\rho k + \frac{2}{3}\mu_{eff} \frac{\partial \bar{V}_k}{\partial x_k} \quad (3-29)$$

and the effective viscosity is

$$\mu_{eff} = \mu + \mu_t \quad (3-30)$$

The k - ε model relates the turbulent viscosity to k and ε themselves as

$$\mu_t = C_\mu \rho \frac{k^2}{\varepsilon} \quad (3-31)$$

where C_μ is an empirically derived model constant.

The values for turbulent kinetic energy and turbulence dissipation rate come directly from their respective additional transport equations:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho\bar{V}_j k)}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right) + P_k - \rho\varepsilon \quad (3-32)$$

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \frac{\partial(\rho\bar{V}_j \varepsilon)}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right) + \frac{\varepsilon}{k} (C_{\varepsilon 1} P_k - C_{\varepsilon 2} \rho\varepsilon) \quad (3-33)$$

where $C_{\varepsilon 1}$, $C_{\varepsilon 2}$, σ_k , and σ_ε are also empirically derived model constants. Table 3.1 gives the values for all k - ε model constants. The term P_k is the turbulence production due to viscous forces and is modeled as

$$P_k = \mu_t \left(\frac{\partial \bar{V}_i}{\partial x_j} + \frac{\partial \bar{V}_j}{\partial x_i} \right) \frac{\partial \bar{V}_i}{\partial x_j} - \frac{2}{3} \frac{\partial \bar{V}_k}{\partial x_k} \left(3\mu_t \frac{\partial \bar{V}_k}{\partial x_k} + \rho k \right) \quad (3-34)$$

With six equations ((3-18), (3-28), (3-32), (3-33)) and six unknowns (p , $\bar{V}_i$, k , ε), the system is closed and the flow field can be fully solved throughout the mesh of finite volumes (discussed in Sections 3.4 and 3.5). The k - ε model can be applied to any turbulent flow but it does not perform optimally in near-wall and stagnation regions due to the need to resolve k and ε in these regions whose gradients are very strong. It therefore requires nonlinear damping functions and production limiters to give marginally accurate results in these areas. It does however give highly accurate results in free shear flow and the core flow of an axial compressor [41], [46].

Table 3.1. k - ε Model Constants

C_μ	$C_{\varepsilon 1}$	$C_{\varepsilon 2}$	σ_k	σ_ε
0.09	1.44	1.92	1.0	1.3

3.3.4 Two Equation Turbulent Viscosity Model – k - ω

Similar to the k - ε model, the k - ω model developed by Wilcox [48] solves two additional transport equations for the turbulence quantities k and ω , the turbulent frequency, along with the mean continuity and RANS equations. The turbulence frequency is defined as

$$\omega \equiv \frac{\varepsilon}{k} \quad (3-35)$$

The transport equations for k and ω , respectively, are

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho \bar{V}_j k)}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right) + P_k - \beta' \rho k \omega \quad (3-36)$$

and

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho \bar{V}_j \omega)}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\left(\mu + \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right) + \alpha \frac{\omega}{k} P_k - \beta \rho \omega^2 \quad (3-37)$$

The production term P_k is as described in Equation (3-34), and the turbulent viscosity used in the k - ω model is

$$\mu_t = \rho \frac{k}{\omega} \quad (3-38)$$

The Reynolds stresses are modeled in the k - ω model identically to the k - ε model as per Equation (3-24). The k - ω model constants are given in Table 3.2.

Table 3.2. k-ω Model Constants				
β'	α	β	σ_k	σ_ω
0.09	5/9	0.075	2.0	2.0

Just as the k - ε model can be applied to any turbulent flow, so can the k - ω model. The k - ω model does not perform well at non-turbulent free-stream boundaries where there is an intermittent region of turbulent and non-turbulent flow for a given spatial location. It therefore requires a non-zero and hence non-physical boundary condition for ω in the intermittent region (both k and ε are zero in non-turbulent fluid, see Equation (3-35)), and the resulting flow calculations are sensitive to how this boundary condition is set [41], [49].

However the k - ω model performs very well in boundary layer flows. It is very accurate in near-wall regions in dealing with viscous effects as well as dealing with the effects of stream-wise pressure gradients. For this reason it is the second most commonly used two equation turbulent viscosity model after k - ε [41], [49] and is suitable for use in axial compressor modeling.

3.3.5 Two Equation Turbulent Viscosity Model – Shear Stress Transport

Since the k - ε model performs very well in the free-stream but not in near-wall regions, and the k - ω model performs very well in near-wall regions, it is almost natural that an effort be made to combine the two. Menter [50] undertook just such an effort to blend the two models

such that their best behavior is exhibited in all near-wall and free-stream locations. Menter's model has the ability to accurately account for the transport of the principle turbulent shear stresses in adverse pressure gradient boundary layers and so named it the Shear Stress Transport (SST) model.

The SST model uses a blending function, F_1 , that transitions from a value of $F_1 = 1$ in the near-wall region (thereby fully implementing $k-\omega$) to $F_1 = 0$ outside the boundary layer in the free-stream (thereby fully implementing $k-\varepsilon$). This takes the form of

$$\phi_3 = F_1\phi_1 + (1 - F_1)\phi_2 \quad (3-39)$$

where ϕ_1 represents the $k-\omega$ model and its constants, and ϕ_2 represents the $k-\varepsilon$ model and its corresponding constants.

Both ϕ_1 and ϕ_2 have a transport equation in terms of k , so adding them together in (3-39) (i.e. adding Equations (3-32) and (3-36)) poses no problem. However the transport equations for ε and ω are mismatched dimensionally (energy/time and 1/time, respectively), so adding them in their forms of (3-33) and (3-37) is not possible. Therefore, the $k-\varepsilon$ model must first be transformed to a $k-\omega$ formulation.

Starting from the definition of turbulent frequency in Equation (3-35), the dissipation is put in terms of k and frequency:

$$\omega = \frac{\varepsilon}{k} \Rightarrow \varepsilon = k\omega \quad (3-40)$$

Differentiating dissipation with respect to time gives

$$\frac{d\varepsilon}{dt} = \omega \frac{dk}{dt} + k \frac{d\omega}{dt} \quad (3-41)$$

Writing (3-41) in terms of the frequency time derivative gives

$$\frac{d\omega}{dt} = \frac{1}{k} \frac{d\varepsilon}{dt} - \frac{\omega}{k} \frac{dk}{dt} \quad (3-42)$$

Substituting the ε transport equation (3-33) into $\frac{d\varepsilon}{dt}$ of (3-42) and the k transport equation (3-32)

into $\frac{dk}{dt}$ of (3-42) results in the transformed k - ε equations into a k - ω formulation. These

transformed equations are

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho \bar{V}_j k)}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\left(\mu + \frac{\mu_t}{\sigma_{k2}} \right) \frac{\partial k}{\partial x_j} \right) + P_k - \beta' \rho k \omega \quad (3-43)$$

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho \bar{V}_j \omega)}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\left(\mu + \frac{\mu_t}{\sigma_{\omega 2}} \right) \frac{\partial \omega}{\partial x_j} \right) + \frac{2\rho}{\omega \sigma_{\omega 2}} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} + \alpha_2 \frac{\omega}{k} P_k - \beta_2 \rho \omega^2 \quad (3-44)$$

It is Equations (3-43) and (3-44) that can be utilized in Equation (3-39). This gives the set of equations ϕ_3 :

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho \bar{V}_j k)}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\left(\mu + \frac{\mu_t}{\sigma_{k3}} \right) \frac{\partial k}{\partial x_j} \right) + P_k - \beta' \rho k \omega \quad (3-45)$$

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho \bar{V}_j \omega)}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\left(\mu + \frac{\mu_t}{\sigma_{\omega 3}} \right) \frac{\partial \omega}{\partial x_j} \right) + (1 - F_1) \frac{2\rho}{\omega \sigma_{\omega 2}} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} + \alpha_3 \frac{\omega}{k} P_k - \beta_3 \rho \omega^2 \quad (3-46)$$

The blending function designed by Menter is

$$F_1 = \tanh(\arg_1^4) \quad (3-47)$$

where

$$\arg_1 = \min \left(\max \left(\frac{\sqrt{k}}{\beta' \omega y}, \frac{500\nu}{y^2 \omega} \right), \frac{4\rho k}{CD_{k\omega} \sigma_{\omega 2} y^2} \right) \quad (3-48)$$

and

$$\nu = \frac{\mu}{\rho} \quad (3-49)$$

is kinematic viscosity, y is the distance to the nearest wall, and

$$CD_{k\omega} = \max\left(\frac{2\rho}{\omega\sigma_{\omega 2}} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}, 1 \times 10^{-10}\right) \quad (3-50)$$

is a cross diffusion term.

Since the k - ε and k - ω models on their own do not accurately account for the transport of turbulent shear stress and hence over-predict the turbulent viscosity, the SST model limits turbulent viscosity to achieve proper transport behavior:

$$\nu_t = \frac{a_1 k}{\max(a_1 \omega, SF_2)} \quad (3-51)$$

where

$$F_2 = \tanh\left(\arg_2^2\right) \quad (3-52)$$

and

$$\arg_2 = \max\left(\frac{2\sqrt{k}}{\beta' \omega y}, \frac{500\nu}{y^2 \omega}\right) \quad (3-53)$$

and the strain rate, S , is

$$S = \frac{1}{2} \left(\frac{\partial \bar{V}_i}{\partial x_j} + \frac{\partial \bar{V}_j}{\partial x_i} \right) \quad (3-54)$$

The SST model constants are summarized in Table 3.3.

Table 3.3. SST Model Constants

ϕ_1	ϕ_2	ϕ_3
α_1	α_2	$\alpha_3 = \alpha_1 + \alpha_2$
5/9	0.44	0.9956
β_1	β_2	$\beta_3 = \beta_1 + \beta_2$
0.075	0.0828	0.1578
σ_{k1}	σ_{k2}	$\sigma_{k3} = \sigma_{k1} + \sigma_{k2}$
1.1765	1	2.1765
$\sigma_{\omega 1}$	$\sigma_{\omega 2}$	$\sigma_{\omega 3} = \sigma_{\omega 1} + \sigma_{\omega 2}$
2	1.1682	3.1682
β'		
0.09		
a_1		
0.31		

The SST model gives very accurate predictions for the onset and amount of flow separation under adverse pressure gradients [41], [46], [50]. This method is included in ANSYS CFX and is the method of choice for all compressor simulations performed in this dissertation.

3.4 Meshing and Discretization

The transport equations described in the preceding sections are discretized and ultimately converted to algebraic equations that are then solved throughout the flow field. ANSYS CFX discretizes the spatial domain into a mesh using an element-based finite volume method.

The solution variables and fluid properties are stored at the mesh vertices (nodes) and a control volume is constructed around each node using the “median dual”. The median dual is defined by the lines that join the centers of the element edges to the centers of each of those elements sharing the common node. This is illustrated schematically in 2D in Figure 3.3 where the control volume is represented by the shaded area.

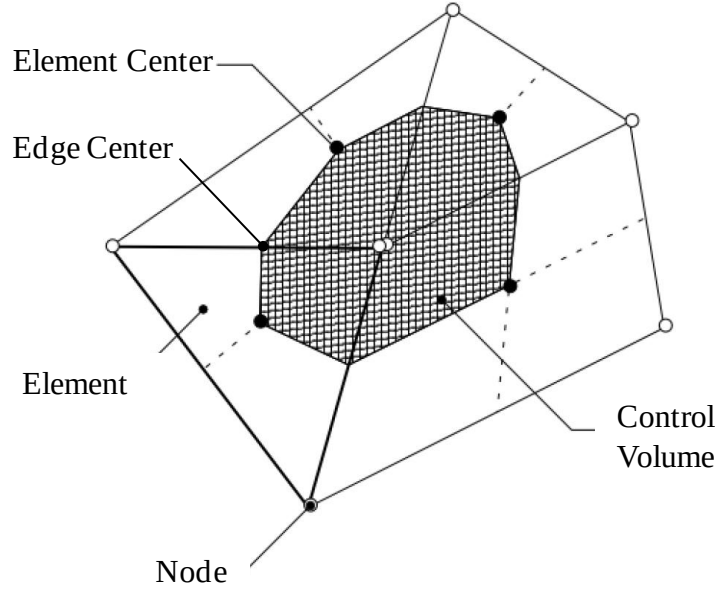


Figure 3.3. 2D Finite Volume Mesh Representation [46]

The transport equations are then integrated over each control volume in the mesh. Since the volumes do not change with time, the time derivatives are moved to the outside of the integrals, and volume integrals of divergence and gradient quantities are converted to surface integrals Gauss' Divergence Theorem. The Divergence theorem states [51]

$$\int_{\mathcal{V}} \nabla \cdot \mathbf{F} d\mathcal{V} = \int_s (\mathbf{F} \cdot \mathbf{n}) ds \quad (3-55)$$

where $\mathbf{F}$ is a vector field and $\mathbf{n}$ is the outward pointing unit normal vector from the surface, s . As examples, the mean continuity (3-18) and RANS with turbulent viscosity hypothesis (3-28) equations become

$$\frac{d}{dt} \int_{\mathcal{V}} \rho d\mathcal{V} + \int_s \rho \bar{V}_j dn_j \quad (3-56)$$

and

$$\frac{d}{dt} \int_{\mathcal{V}} \rho \bar{V}_i d\mathcal{V} + \int_s \rho \bar{V}_j \bar{V}_i dn_j = - \int_s p' dn_j + \int_s \mu_{eff} \left(\frac{\partial \bar{V}_i}{\partial x_j} + \frac{\partial \bar{V}_j}{\partial x_i} \right) dn_j + \int_{\mathcal{V}} S_{\bar{V}_i} d\mathcal{V} \quad (3-57)$$

where the volume integrals represent accumulations, the surface integrals represent the summations of the fluxes across the element surfaces, and dn_j are the differential Cartesian components of the outward pointing surface normal vector.

Individual elements are split into sectors by rays emanating from the element center to the center of each edge; the midpoints of the rays are used as integration points, ip_i . Figure 3.4 shows a 2D representation of this.

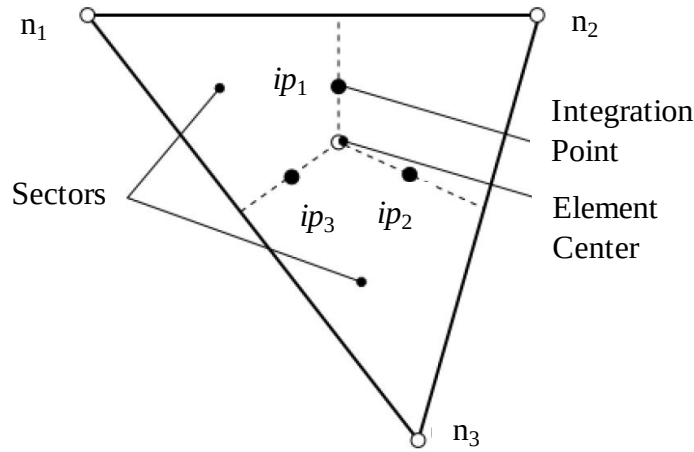


Figure 3.4. 2D Mesh Element [46]

The volume and surface integrals must then be discretized. Volume integrals are discretized within each element sector and accumulated to the control volume to which the sector belongs. Surface integrals are discretized at the integration points and distributed to the adjacent control volumes [46]. The discretization converts (3-56) and (3-57) to

$$\mathcal{V} \left(\frac{\rho - \rho^\circ}{\Delta t} \right) + \sum_{ip} \dot{m}_{ip} = 0 \quad (3-58)$$

and

$$\mathcal{V} \left(\frac{\rho \bar{V}_i - \rho^\circ \bar{V}_i^\circ}{\Delta t} \right) + \sum_{ip} \dot{m}_{ip} (\bar{V}_i)_{ip} = \sum_{ip} \left(\mu_{eff} \left(\frac{\partial \bar{V}_i}{\partial x_j} + \frac{\partial \bar{V}_j}{\partial x_i} \right) \Delta n_j \right)_{ip} + \bar{S}_{\bar{V}_i} \mathcal{V} \quad (3-59)$$

where $\dot{m}_{ip} = (\rho \bar{V}_j \Delta n_j)_{ip}$, $\forall$ is the control volume, Δt is the time step, Δn_j is the discrete outward surface vector, and summations are over all integration points of the control volume. The superscript $^\circ$ refers to the previous time level. These equations use a first-order backward Euler scheme. This scheme approximates values the current time step based on the Taylor series expansion of values at the previous time step and neglects second-order derivatives and higher (hence the first-order accuracy).

$$f^\circ = f + \frac{df}{dt}(-\Delta t) + \underbrace{\frac{d^2 f}{dt^2} \frac{(-\Delta t)^2}{2!} + \dots}_{\text{neglect}} \Rightarrow \frac{df}{dt} = \frac{f - f^\circ}{\Delta t} \quad (3-60)$$

Upon specifying an initial and boundary conditions, transient solutions can be solved over a specified duration. Steady-state solutions are not functions of time by definition. The CFD approach to solving steady-state problems is to introduce time as an extra variable and calculate the solution until the main variables no longer change with respect to time. ANSYS CFX uses second-order-accurate implicit schemes with pressure velocity coupling, further details are provided in [46].

The solution variables and other properties are stored at the mesh nodes. Solution gradients are approximated at the integration points, and ANSYS-CFX uses tri-linear finite-element shape functions to perform these approximations. For a variable φ , the variation within an element according to the shape functions is

$$\varphi = \sum_{i=1}^{nodes} N_i \varphi_i \quad (3-61)$$

where N_i is the shape function at node i , and φ_i is the value of φ at node i such that

$$\sum_{i=1}^{nodes} N_i = 1 \quad (3-62)$$

and at node j ,

$$N_i = \begin{cases} 1 & i = j \\ 0 & i \neq j \end{cases} \quad (3-63)$$

The element types used in this dissertation are hexahedral and tetrahedral. Figures 3.5 and 3.6 show representations of these elements, respectively, and Table 3.4 provides the corresponding shape functions.

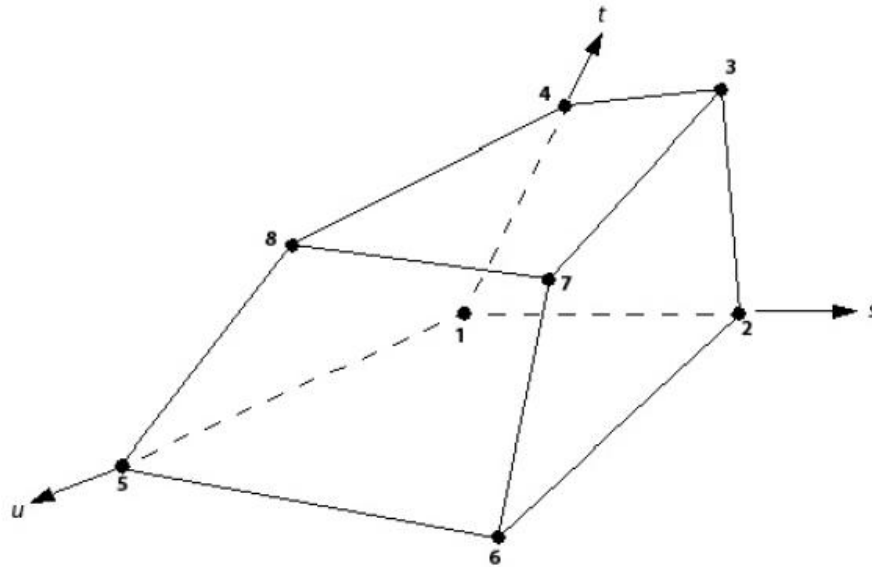


Figure 3.5. Hexahedral Volume Element [46]

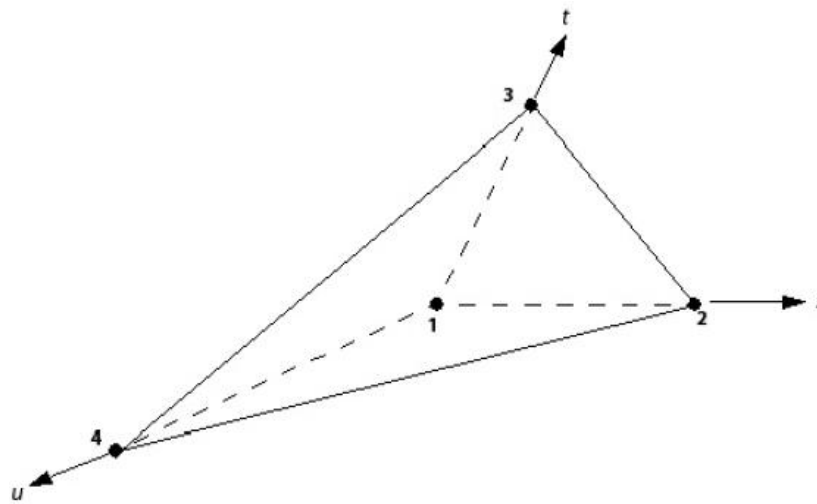


Figure 3.6. Tetrahedral Volume Element [46]

Table 3.4. Tri-Linear Element Shape Functions

	Hexahedral	Tetrahedral
$N_1(s,t,u)$	$(1-s)(1-t)(1-u)$	$1-s-t-u$
$N_2(s,t,u)$	$s(1-t)(1-u)$	s
$N_3(s,t,u)$	$st(1-u)$	t
$N_4(s,t,u)$	$(1-s)t(1-u)$	u
$N_5(s,t,u)$	$(1-s)(1-t)u$	
$N_6(s,t,u)$	$s(1-t)u$	
$N_7(s,t,u)$	stu	
$N_8(s,t,u)$	$(1-s)tu$	

The shape functions are then used as follows. For diffusion of φ in the x-direction,

$$\frac{\partial \varphi}{\partial x_{ip}} = \sum_n \frac{\partial N_n}{\partial x_{ip}} \varphi_n \quad (3-64)$$

where the summation is over all the shape functions of an element. The derivatives of the shape functions are expressed in terms of their local derivatives through the Jacobian transformation matrix:

$$\begin{pmatrix} \frac{\partial N}{\partial x} \\ \frac{\partial N}{\partial y} \\ \frac{\partial N}{\partial z} \end{pmatrix} = \begin{bmatrix} \frac{\partial x}{\partial s} & \frac{\partial y}{\partial s} & \frac{\partial z}{\partial s} \\ \frac{\partial x}{\partial t} & \frac{\partial y}{\partial t} & \frac{\partial z}{\partial t} \\ \frac{\partial x}{\partial u} & \frac{\partial y}{\partial u} & \frac{\partial z}{\partial u} \end{bmatrix}^{-1} \begin{pmatrix} \frac{\partial N}{\partial s} \\ \frac{\partial N}{\partial t} \\ \frac{\partial N}{\partial u} \end{pmatrix} \quad (3-65)$$

Applying the finite volume method (FVM) to all elements in the fluid domain leads to a discrete set of conservation equations of the form

$$\sum_{nb_i} a_i^{nb} \varphi_i^{nb} = b_i \quad (3-66)$$

where i is the node number and nb means “neighbor” but also includes the central coefficient multiplying the solution at the i^{th} location. For a scalar equation, a , φ , and b are also scalar (e.g. enthalpy or turbulent kinetic energy), but for the 3D mass-momentum equations, these are 4×4 (a) and 4×1 (φ , b) where the four components are the three velocities and the pressure. The set

of equations starts with an approximate solution φ^n that is improved by a correction φ' leading to a better solution φ^{n+1} :

$$\varphi^{n+1} = \varphi^n + \varphi' \quad (3-67)$$

where φ' is a solution of

$$A\varphi' = r^n \quad (3-68)$$

with the residual r^n obtained from

$$r^n = b - A\varphi^n \quad (3-69)$$

Performing multiple iterations of this process will reduce the residuals and achieve the desired accuracy of solution [46].

3.5 Boundary Conditions

All simulations performed applied total pressure and total temperature boundary conditions in the stationary reference frame at the domain inlet with the flow direction normal to the inlet plane. The flow at the inlet is constrained to be subsonic, and the medium turbulence intensity, I , option at the inlet is selected using the stream-wise flow components as

$$I = \frac{\bar{u}}{U} = 0.05 \quad (3-70)$$

The outlet boundary is set either using an unconstrained mass flow rate, or selecting an average static pressure for the outlet plane.

CHAPTER 4 STAR PATTERN IMPELLER PERFORMANCE

The first generation of woven impellers was woven into a “star” pattern consisting of near-radial circular-arc blades. This was done based on the ease of manufacturing the mandrel (machined from metal or plastic/PVC) and the subsequent ease of winding. Based on the findings by Li [12], and eight-bladed 0.43 hub/tip ratio (r_h/r_t) impeller was selected for prototyping and testing (pattern 8B, see Tables 1.6, 1.7, and Figure 1.13). However, a geometrical description of the impeller is necessary to make meaningful performance predictions.

4.1 Circular Arc Blade Description at Shroud in Global Coordinates

The carbon fiber composite impeller was wound on an eight-slotted mandrel. The angular space between each blade slot is

$$\Phi_b = \frac{360^\circ}{8 \text{ blade slots}} = \frac{45^\circ}{\text{blade slot}} \quad (4-1)$$

However the winding process is achieved by extending fibers across every third mandrel slot as shown in Figures 4.1 and 4.2.

The mandrel slots are segments of circular arcs, and hence the blades created by winding fibers through these slots are also circular-arc. This is accomplished by layering the fibers in the axial direction (into page, referring to Figure 4.1) during winding. The outer shroud is simultaneously created during the winding process; the fibers are also wound around the outside of the mandrel and sent back through the interior.

Section 2.3 describes in detail the relevant considerations for circular arc blade geometry, however only one radius (tip radius) is considered (Figures 2.3 and 2.4). Section 4.1 serves to determine the full 3D impeller geometry at all interior points.

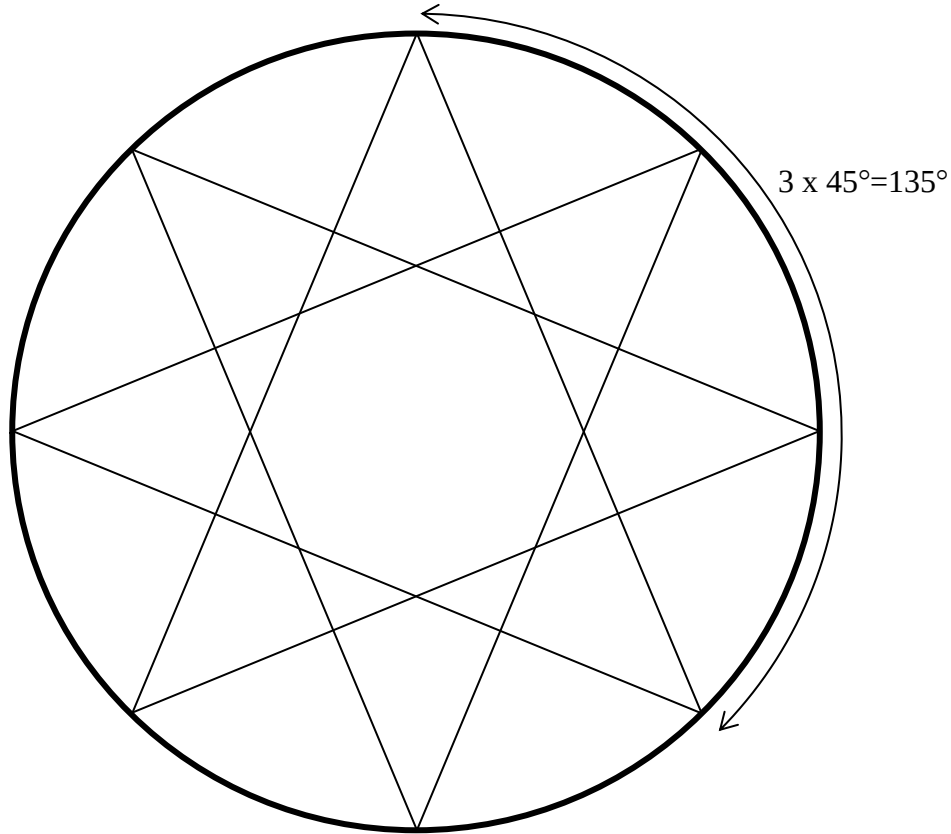


Figure 4.1. Fiber Pattern of Star Impeller

As seen in Figures 2.3 and 2.4, the circular-arc blade has a “width” in the X -direction of w_1 . This same distance w_1 is shown in Figure 4.2 looking into the face of the impeller, where a global coordinate system (X,Y,Z) is defined with reference to mandrel slots A and B , with Z pointing into the page.

The geometry of the star impeller is discretized with respect to the axial direction into n segments with $n + 1$ nodes. The length per element is then z / n , and the axial location of the i^{th} discrete point is $z_i = z_{i-1} + z / n$. The axial length of the rotor is z_{rot} , and the node $i = 1$ is located at the rotor inlet.

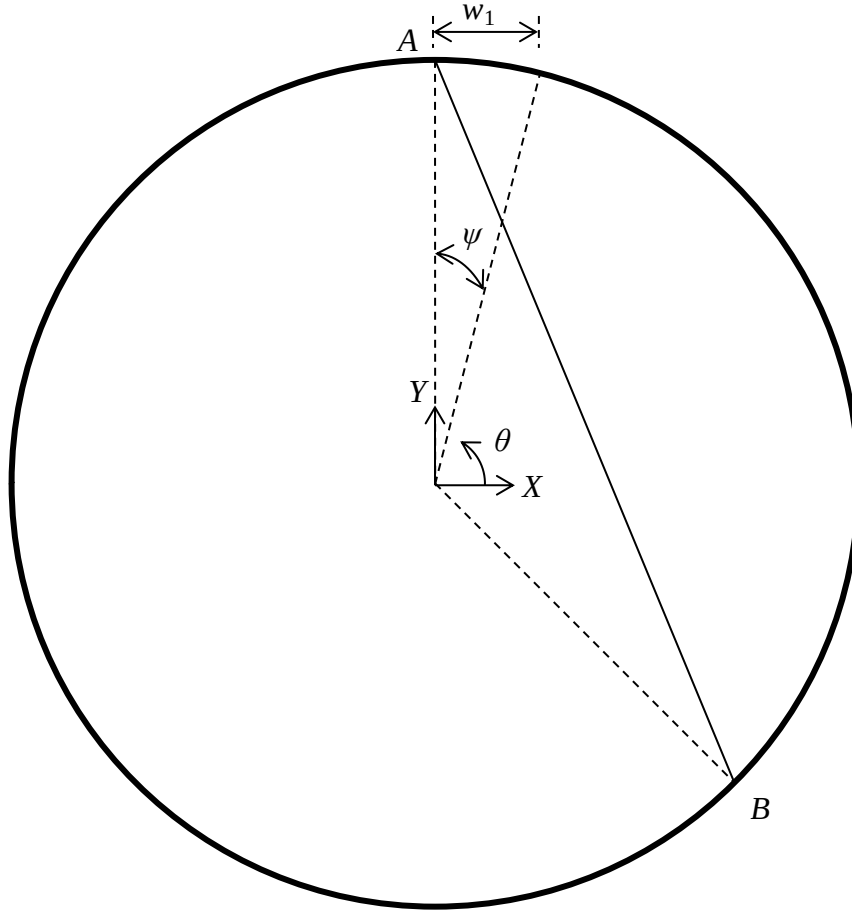


Figure 4.2. Frontal View of Star Impeller Showing One Blade

Note that end point A is where the fiber enters the mandrel during winding, and end point B is where it exits. Figure 4.3 shows a generic case of the axial view for one blade with additional necessary parameters.

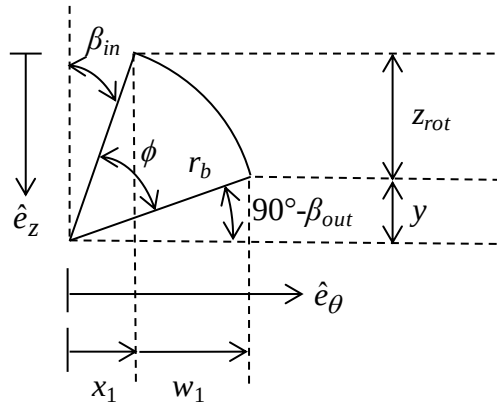


Figure 4.3. Axial View of One Blade at Outer Shroud

The geometry in Figure 4.3 (detailed in Figure 2.4) is described by the system of equations given in (2-100), and the solutions to the pertinent unknown values are given in Equations (2-102) – (2-104). For the star pattern impeller considered in Chapter 4, the outlet blade angle has been selected as 90° .

To map the interior points on the blade in the selected (X, Y, Z) coordinates (where Z is into the page in Figure 4.2), the end points at the outer shroud (tip radius, r_t) must be considered. Using the subscript i after w , x , ψ and ϕ allows the width and angle to be determined at each axial location along the outer shroud. In this sense, for the reader's reference, ϕ_1 , ψ_1 , and w_{11} are all zero, and ϕ_n , ψ_n , and w_{1n} are all as pictured in Figure 4.2.

As can be seen in Figure 4.3,

$$x_{1i} = r_b \cos(\phi + 90^\circ - \beta_{out} - \phi_i) \quad (4-1)$$

where

$$\phi_i = \phi_{i-1} + \beta_i - \beta_{i-1}, \quad i \geq 2 \quad (4-2)$$

and

$$\beta_i = 90^\circ - \sin^{-1}\left(\frac{z - z_i + y}{r_b}\right) \quad (4-3)$$

It then follows that

$$w_{1i} = x_{1i} - x_{11} \quad (4-4)$$

The angle ψ_i at each axial location is then

$$\psi_i = \sin^{-1}\left(\frac{w_{1i}}{r_t}\right) \quad (4-5)$$

With the result from Equation (4-5), the (X, Y) coordinates of the blades along the outer shroud at each axial location are

$$X_{i,A} = r_t \sin(\psi_i) \quad (4-6)$$

and

$$Y_{i,A} = r_t \cos(\psi_i) \quad (4-7)$$

The coordinates specified by Equations (4-6) and (4-7) are true only for the portion of the blade at the outer shroud pictured at the top Figure 2.4, called location A. The other end location, B, must also be determined. Since all of the slots in the mandrel are the same, the fibers will take the same path in the winding process as at point A, but with an angular offset of -135° . Therefore, with the appropriate trigonometric identities, the coordinates of point B at the outer shroud at each axial location are

$$X_{i,B} = r_t \cos(45^\circ + \psi_i) \quad (4-8)$$

and

$$Y_{i,B} = -r_t \sin(45^\circ + \psi_i) \quad (4-9)$$

4.2 Circular Arc Blade Description at Interior Points

With the blade angles (Equation (4-3)) and the coordinates of the blades defined at each end point at the outer shroud (Equations (4-6) – (4-9)), it is possible to determine the coordinates and hence blade angles of every interior point along the blade for all axial locations. Since the fiber paths taken to create the blades are straight lines (although a circular fiber path is taken to create outer shroud), it is possible to define the equation for each line at all axial locations referencing the coordinate system shown in Figure 4.2.

First the slope, m , must be determined:

$$m = \frac{\Delta Y}{\Delta X} \Rightarrow m_i = \frac{Y_{i,B} - Y_{i,A}}{X_{i,B} - X_{i,A}} \quad (4-10)$$

The Y-intercept, b , is then determined from the generic equation of a line from the slope and two known coordinates:

$$b_i = Y_{i,A} - m_i X_{i,A} \quad (4-11)$$

With the equation of each line known, interior points along each line can be determined. Discretizing each line into q intervals with $q + 1$ nodes gives any location, j , on the length of the blade at any axial location, i :

$$X_{i,j} = \left(\frac{X_{i,B} - X_{i,A}}{q} \right) + X_{i,j-1}, \quad 2 \leq j \leq q \quad (4-12)$$

$$Y_{i,j} = m_i X_{i,j} + b_i, \quad 2 \leq j \leq q \quad (4-13)$$

Equations (4-12) and (4-13) can be used to conveniently convert to polar coordinates (r , θ) using the standard where θ is the angle of the ray r extending from the origin above the positive X -axis in the counter-clockwise direction in the X - Y plane (Figure 4.2):

$$X_{i,j} = r_{i,j} \cos(\theta_{i,j}) \quad (4-14)$$

$$Y_{i,j} = r_{i,j} \sin(\theta_{i,j}) \quad (4-15)$$

and so

$$r_{i,j} = \sqrt{X_{i,j}^2 + Y_{i,j}^2} \quad (4-16)$$

$$\theta_{i,j} = \sin^{-1} \left(\frac{Y_{i,j}}{r_{i,j}} \right) \quad (4-17)$$

With all blade coordinates defined on a polar basis, it is then possible to determine the blade angles at all locations. Figures 4.4 and 4.5 show the blade profiles at the outer shroud at locations A and B , respectively.

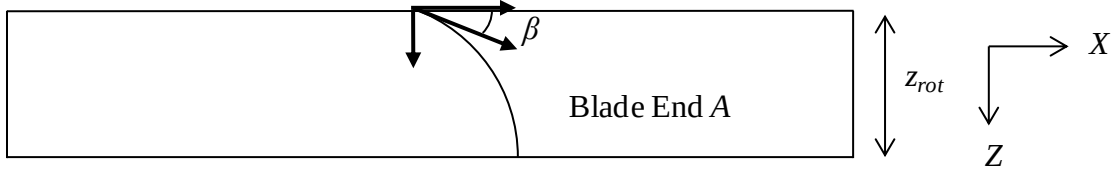


Figure 4.4. Blade Profile at the Outer Shroud for Location A

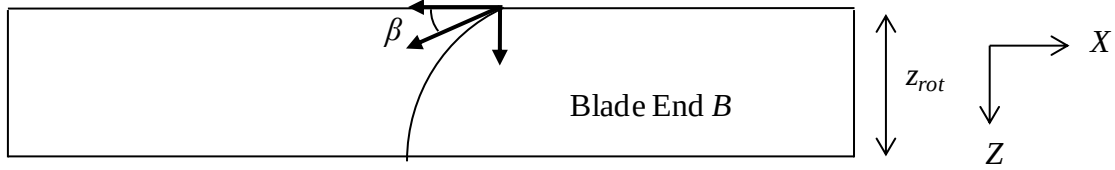


Figure 4.5. Blade Profile at the Outer Shroud for Location B

Equations (4-14) – (4-17) are necessary for determining the blade angles at all i and j locations because it is only possible to determine β from the inverse tangent of the ratio of changes in X to Z at the tip. It would only be possible to use the same relationship at the interior points if the blades were purely radial (i.e. passed through the center of the impeller from slot to slot during winding). Since they are not (see Figure 4.2), the blade angles must be determined using the change in arc length at any particular r value rather than change in X . The blade angles are then

$$\beta = 90^\circ - \tan^{-1} \left(\frac{|r \Delta \theta|}{\Delta z} \right) \quad (4-18)$$

more specifically,

$$\beta_{i,j} = 90^\circ - \tan^{-1} \left(\frac{|r_{i,j}(\theta_{i+1,j} - \theta_{i,j})|}{z_{i+1} - z_i} \right) \quad (4-19)$$

However it is only truly necessary to use Equation (4-19) at the inlet in order to get a span-wise approximation for all inlet angles (for $i = 1$), i.e.

$$\beta_{1,j} = 90^\circ - \tan^{-1} \left(\frac{|r_{1,j}(\theta_{2,j} - \theta_{1,j})|}{z_2 - z_1} \right) \quad (4-20)$$

With the outlet angle known (90° at all spans), and the inlet angle closely approximated by Equation (4-20), the system of equations in (2-100) can be solved q times at all remaining interior j locations enabling an extended version of Equation (4-3) to be utilized to accurately determine all interior blade angles.

$$\beta_{i,j} = 90^\circ - \sin^{-1} \left(\frac{z - z_i + y}{r_{b,j}} \right), i \geq 2 \quad (4-21)$$

Since all of the mandrel slots are the same and evenly spaced, all blade angles as determined by Equations (4-20) and (4-21) are the same for all blades, and all (X,Y) coordinates of the remaining blades can be determined by using the polar description in (4-16) and (4-17) when increasing the value of $\theta_{i,j}$ by increments of $\Phi_b = 45^\circ$ and then converting back to (X,Y) coordinates.

It must be noted that to this point the fibers crossing from mandrel slot A to slot B have been referred to as a single “blade”. In actuality, there are two blades formed in this region because each slot is the same: at approximately half way between points A and B (both located at the outer shroud), the blade “flips” (at what can be considered the “hub”) thus in effect creating two separate blades in terms of flow interaction, but with only one set of fibers passing through the center of the mandrel. Figure 4.6 shows $n + 1 = 16$ fibers (lines) connected from points A to B in the $X-Y$ plane, with each fiber being in a different axial location.

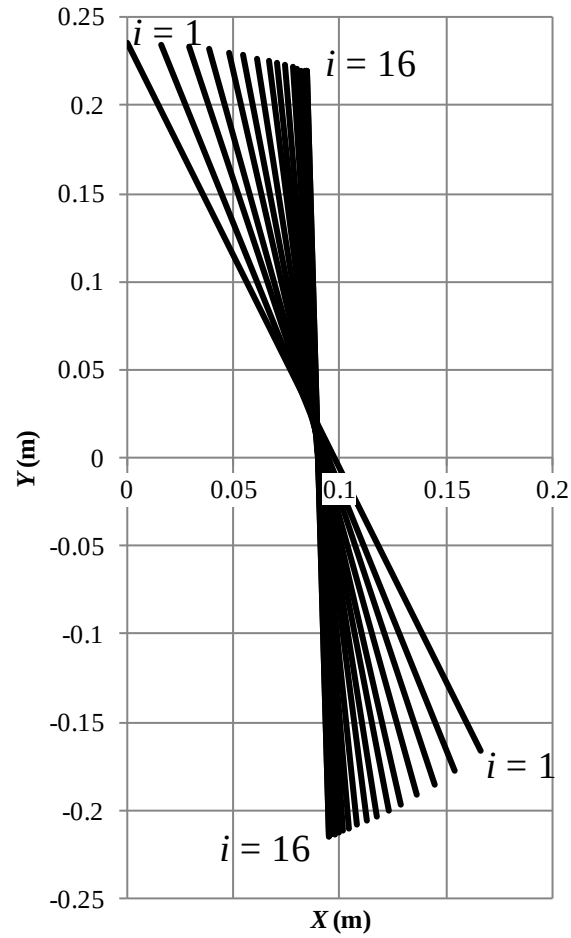


Figure 4.6. Individual Fiber Paths Crossing a Single Set of Mandrel Slots

For an observer looking into the inlet face of the impeller down the axis of rotation as shown in Figure 4.6, the suction side of the blade is seen at all times. When this winding pattern is completed for all mandrel slots, the 8-slotted mandrel yields a star-patterned 16-bladed impeller. Figure 4.7 depicts this without the inclusion of the outer shroud.

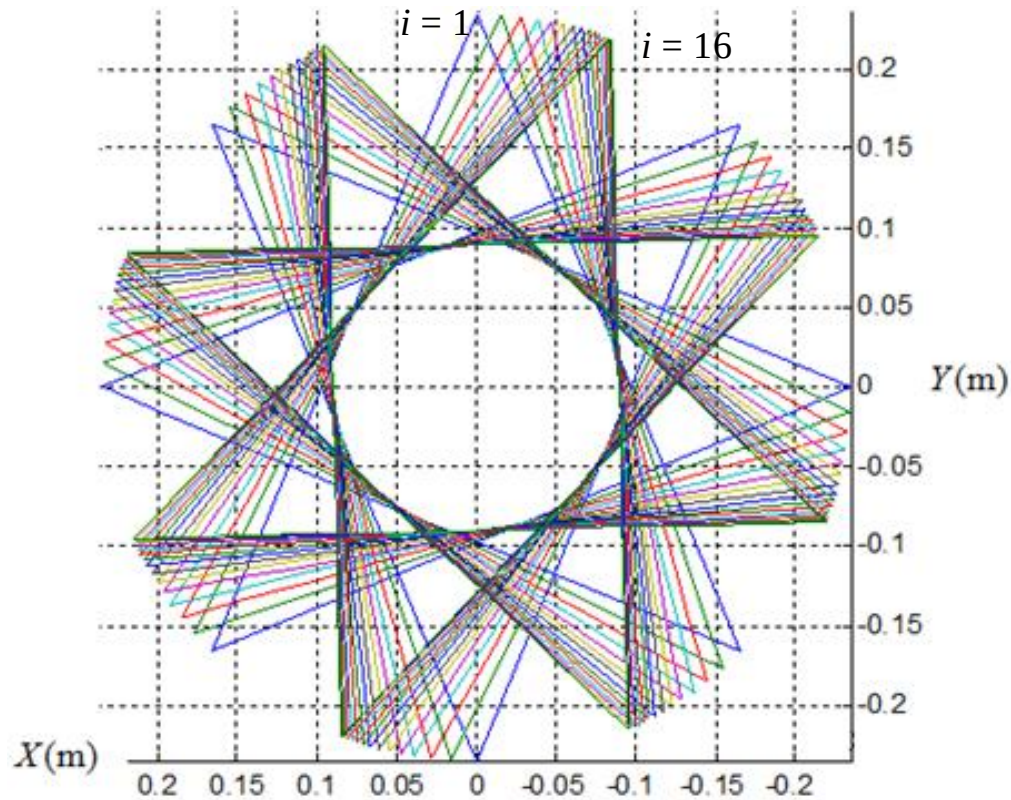


Figure 4.7. Individual Fiber Paths Crossing All Mandrel Slots

Since each individual fiber is represented by a line as per Equations (4-10) – (4-13), all of the fiber intersections and the “flip” point (blade inflection point) at any axial location can be determined by setting the equations for the lines in question equal to each other and solving for the (X,Y) coordinates. These are shown in Figure 4.8 for the primary fibers passing between mandrel slots A and B only. Note that only the blade inflection points and end points A and B involve only the primary fibers passing through slots A and B ; the other four points are truly intersection points with fibers crossing through from other slots.

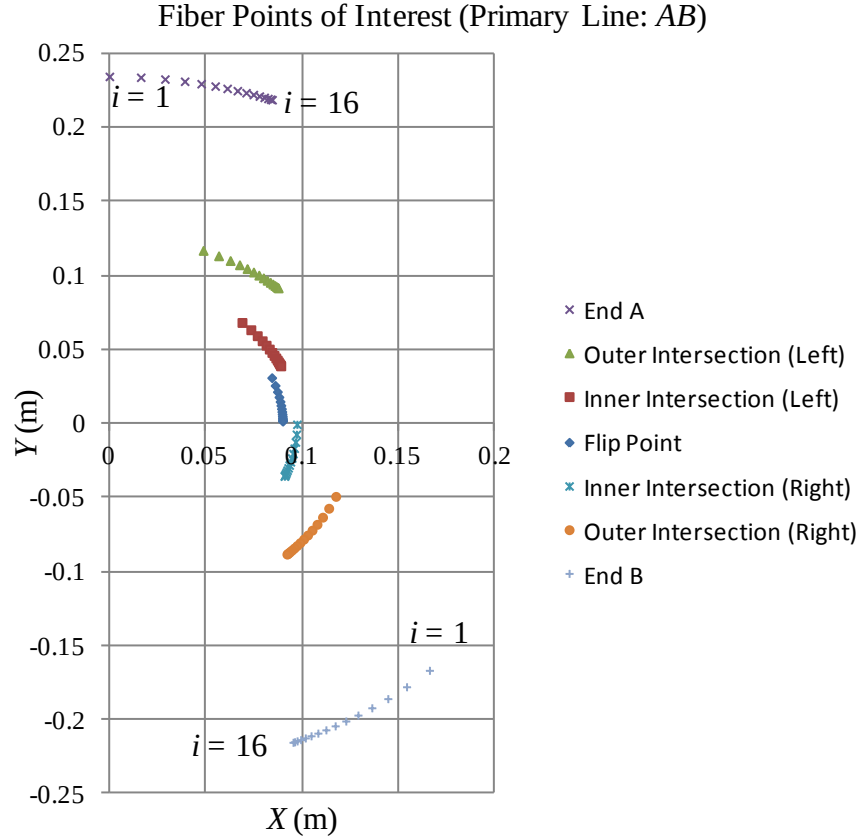


Figure 4.8. Fiber Intersections, Inflections, and End Points at Different Axial Locations

An inherent result of this winding pattern is that the blade radius is smallest at the outer shroud and increases in the direction of the “hub”. It increases to the point where there is no longer any curvature (infinite blade radius, i.e. a flat blade) at the blade inflection point. Curvature then recurs but in the opposite direction, thereby creating a separate blade despite consisting of the same set of fibers.

4.3 Cross Sectional Area of Star Pattern Impeller

Although the blade shapes are constructed using traditional circular arcs as their basis, the blade pattern is non-traditional, and the cross-sectional area the flow encounters must account for the space occupied by the blades. The cross-sectional area is divided into concentric rings and the area occupied by the blades is subtracted from the ring area. Considering a single blade (i.e.

half of a set of fibers passing between two slots) from end *A* to the inflection point of fiber set *AB*, there are two intersections with neighboring blades. This entails that some ring segments will have not only different blade lengths to subtract, but also different numbers of blades. Figure 4.9 shows the full scale wheel with three sets of fibers and their names. Figure 4.10 shows a close-up view between end *A* and the blade inflection point of fiber set *AB*.

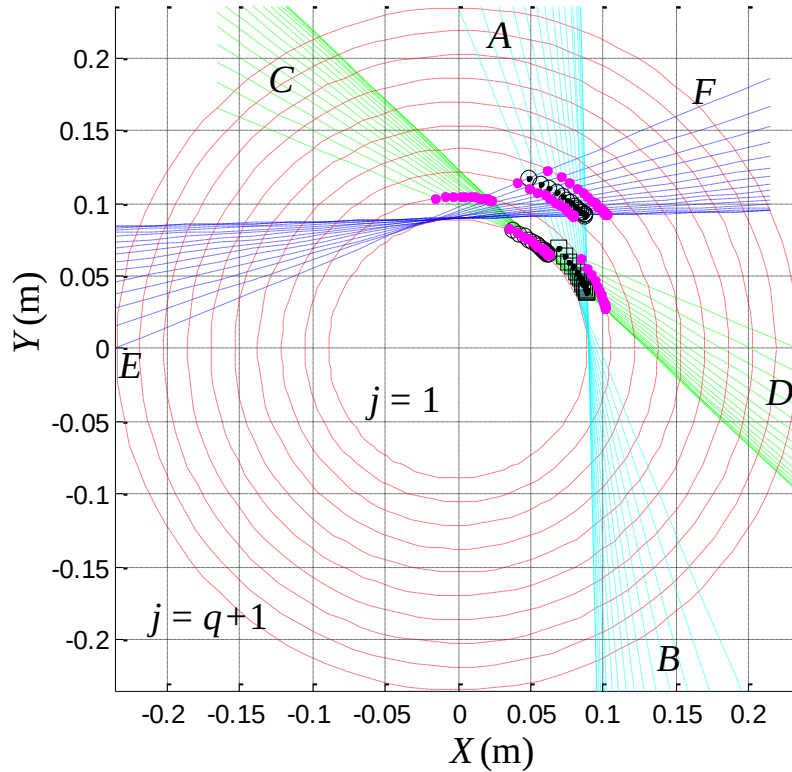


Figure 4.9. Full-Wheel View of Fibers Intersecting a Single Blade

In Figures 4.9 and 4.10, the black indicates points of fiber intersection (and the inflection point as indicated in Figure 4.10); pink indicates sing segments containing more than one blade. Furthermore, this discretization is radial, whereas the initial discretization in the previous section performed to determine the equations of the lines the fibers follow was done in the fiber direction.

The radial discretization uses $q = 9$ ring segments (10 concentric circles).

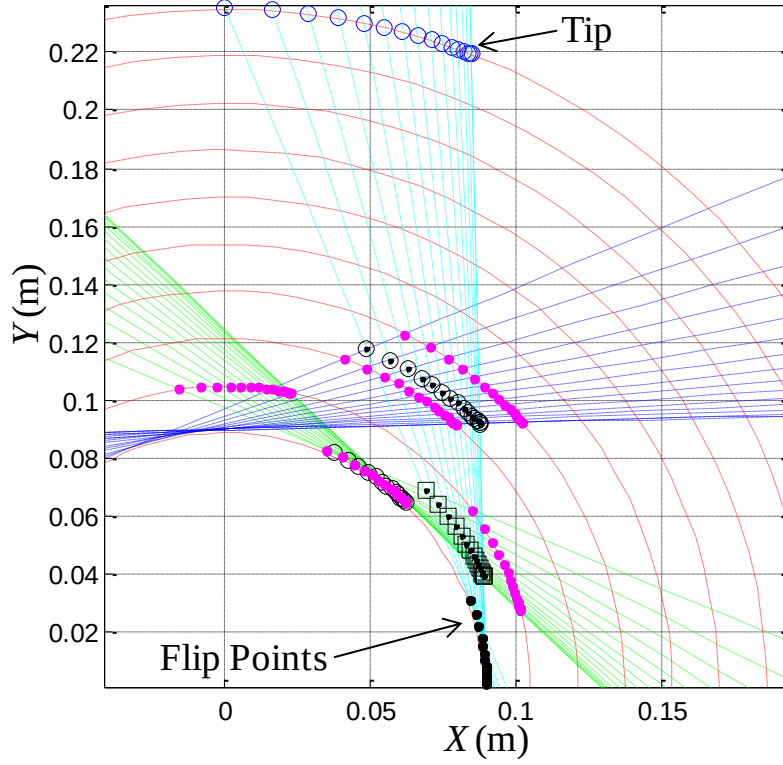


Figure 4.10. Single Blade View with Intersecting Fibers

$$\begin{aligned}
 A_j = & \pi \left(r_{ring,j+1}^2 - r_{ring,j}^2 \right) \\
 & - t_b N_b \sqrt{\left(X_{ringAB,j+1} - X_{ringAB,j} \right)^2 + \left(Y_{ringAB,j+1} - Y_{ringAB,j} \right)^2} \\
 & - \begin{cases} t_b N_b \sqrt{\left(X_{ringCD,j+1} - X_{ringCD,j} \right)^2 + \left(Y_{ringCD,j+1} - Y_{ringCD,j} \right)^2} & \text{for } j = 1 \\ t_b N_b \sqrt{\left(X_{ringEF,j+1} - X_{ringEF,j} \right)^2 + \left(Y_{ringEF,j+1} - Y_{ringEF,j} \right)^2} & \text{for } j = 3 \end{cases}
 \end{aligned} \tag{4-22}$$

Note that even if only fibers from line AB are considered, any ring segment will have two blades to subtract from the area since the one other half of the blade exists in the same ring 135° away. Equation (4-22) accounts for this by using only the (X,Y) coordinates for half of the blade and setting the number of blades, N_b , to be twice the number of mandrel slots (8 slots gives $N_b = 16$), and the rings with the intersections will in fact be accounting for four blades.

4.4 Star Pattern Impeller Flow Calculations

With the geometry fully described, it is possible to make 2D (stream-wise and span-wise) predictions of the flow behavior based on the process outlined in Chapter 2 albeit with some modifications. The process in Chapter 2 specifies both the absolute and relative critical Mach numbers and thus determines the rotational speed. Rotational speed is given for the star pattern impeller.

The Chapter 2 methodology also holds the meridional velocity constant at all stream-wise and span-wise locations; it then determines the corresponding increase in hub radius corresponding to the cross sectional area decrease due to compression and density increase (tip radius is held constant). For the star pattern impeller, the geometry is fully prescribed, and so the meridional velocity must be determined iteratively such that continuity is satisfied.

The star pattern calculations assume a swirl-free inlet to the impeller (i.e. no IGV, hence the tangential absolute velocity component is zero). With the inlet blade angles from Equation (4-20) known and a given rotational speed (Equation (2-124)), the span-wise velocity profile is set (Equations (2-108), (2-117), (2-118)). Figure 4.11 shows a representative velocity diagram.

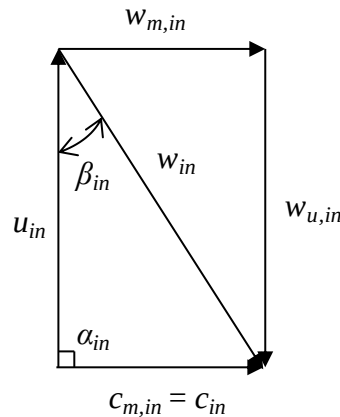


Figure 4.11. Representative Star Pattern Impeller Inlet Velocity Diagram

With the working medium, total pressure, and temperature at the inlet also known the inlet is then fully specified (total density, critical absolute and relative speeds of sound, Mach numbers, static properties). The total mass flow rate at the inlet is determined in each ring segment (Equation (2-8) with (4-22)). Table 4.1 provides inlet data and relevant global geometric properties for the star pattern impeller rotating at 7500rpm studied in this chapter. A polytropic efficiency¹ of 80% has been assumed for these calculations.

Table 4.1. Star Pattern Impeller Inlet Conditions and Geometry		
Fluid		Air
R	kJ/kg/K	0.287
c_p	kJ/kg/K	1.0042
γ	-	1.4001
p_t	kPa	22.753
T_{tc}	K	301.15
ρ_t	kg/m ³	0.2633
r_t	m	0.235
r_h	m	0.0889
$\beta_{in,tip}$	°	25
β_{out}	°	90
u_{tip}	m/s	184.6
$\dot{m}$	kg/s	3.5441
Axial Segments	-	15
Radial Segments (Rings)	-	9

The remainder of the flow field is determined at all spans at each axial location moving sequentially to the outlet. Changes in total properties are determined by Equation (2-93). With the blade speed and blade angle known at all locations, the velocity ratios are determined (Equations (2-117), (2-118)), but the actual values must be determined iteratively. The c_m value within a given ring segment is allowed to vary (thereby adjusting c_u and c , hence a_c^* , M_c^* , ρ and ultimately $\dot{m}$), but the total mass flow rate (the sum of the mass flows through each ring segment) at all axial locations must match that of the inlet in order to preserve continuity.

¹ Polytropic efficiency is a more accurate means of characterizing multistage turbomachines than isentropic. Since this dissertation will ultimately study counter-rotating multistage axial compressors, polytropic efficiency is used here for consistency. This is discussed thoroughly in Chapter 6 and in Appendix A2.

Figures 4.12 and 4.13 plot flow property contours vs. global (X,Y) coordinates (a) and vs. percent stream-wise and percent span-wise location (b), respectively. Note that the contours plotted in the global coordinate system display contours over the same image of the impeller blade depicted in Figure 4.10 but omit intersecting fibers from different blades.

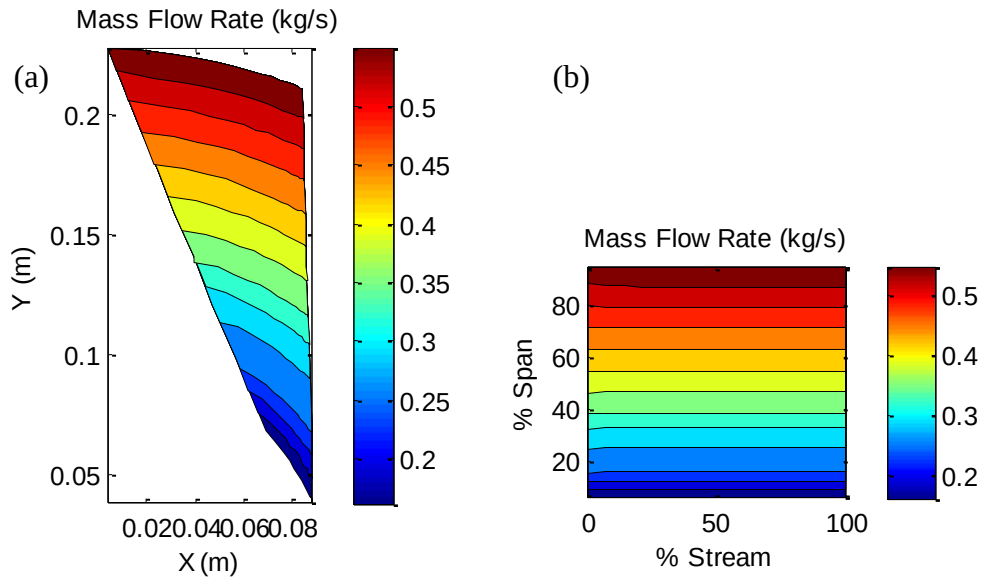


Figure 4.12. Mass Flow Rate Contours (a) vs. Global Coordinates (b) vs. Percent Stream and Span

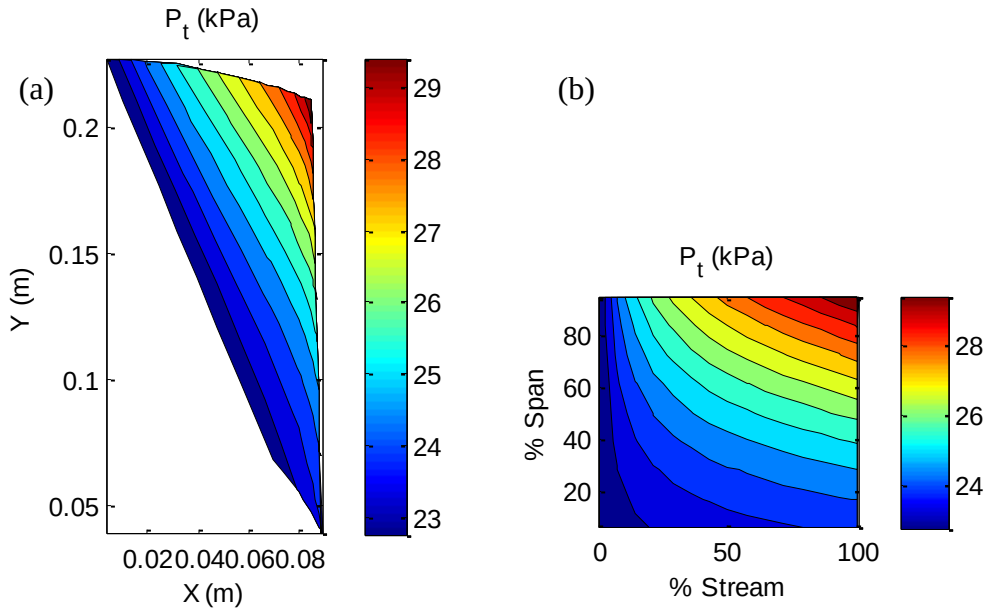


Figure 4.13. Total Pressure Contours (a) vs. Global Coordinates (b) vs. Percent Stream and Span

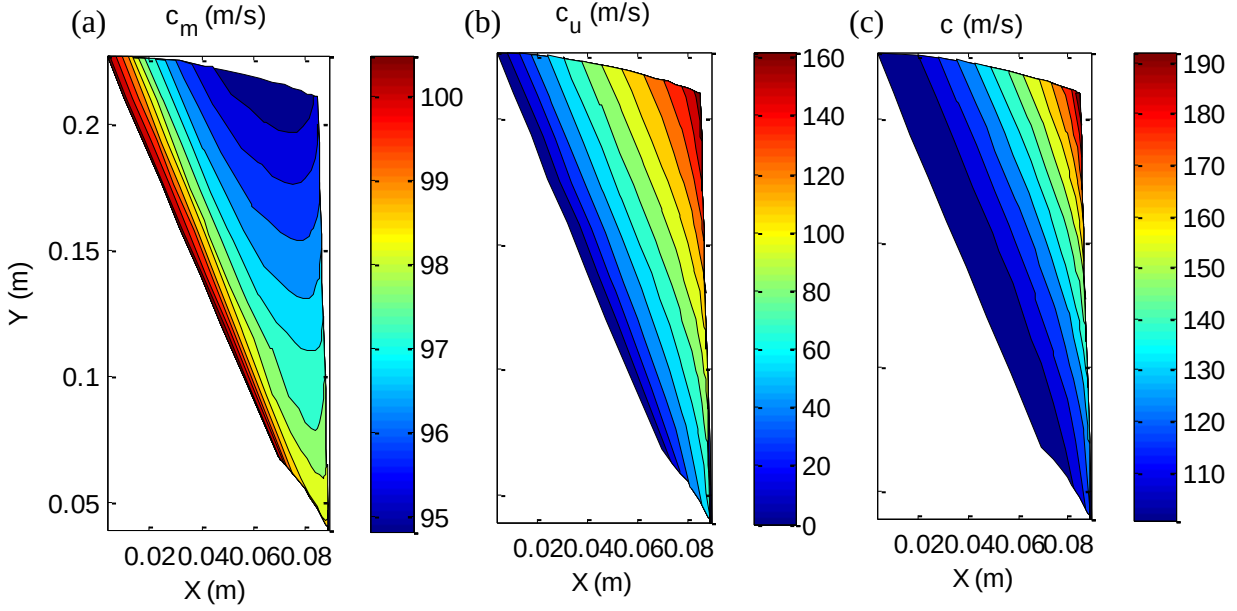


Figure 4.14. Absolute Velocity Contours vs. Global Coordinates

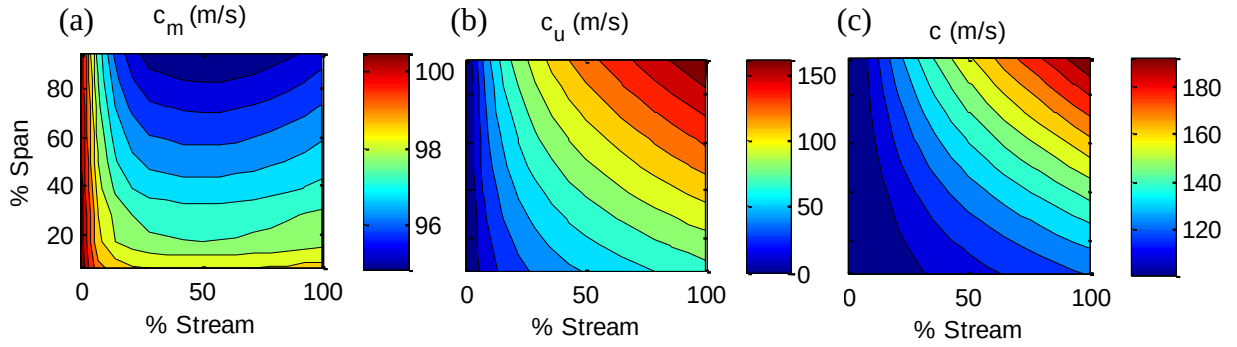


Figure 4.15. Absolute Velocity Contours vs. Percent Stream and Span

As can be seen in Figures 4.14-(a) and 4.15-(a), the meridional velocity is not constant. There is a range of variation throughout the flow field of up to approximately 4%, with the velocities near the hub exceeding those near the shroud at all axial locations. Interestingly, at all spans, c_m decreases from the inlet to around 47% stream, at which point c_m increases to the outlet. This behavior is caused by the swirl imparted to the flow by the impeller and is required to satisfy continuity. The c_u component increases from zero at the inlet to values ranging from just under 75% to 175% the magnitude of c_m at the outlet. Mach numbers used to determine static values are referenced to the full velocity vector, c . While there is modest change in c_u from

0 to 47% stream, there is compression and an associated increase in density; continuity is preserved by reducing c_m . However beyond 47% stream, c_u and hence c become much larger, increasing the Mach number and hence reducing the static density as per Equation (2-60). With sufficient reduction in ρ , continuity must be preserved by an increase in c_m . Indeed, the contours of ρ mirror the contours of c_m with the opposite trend, as shown in Figure 4.16.

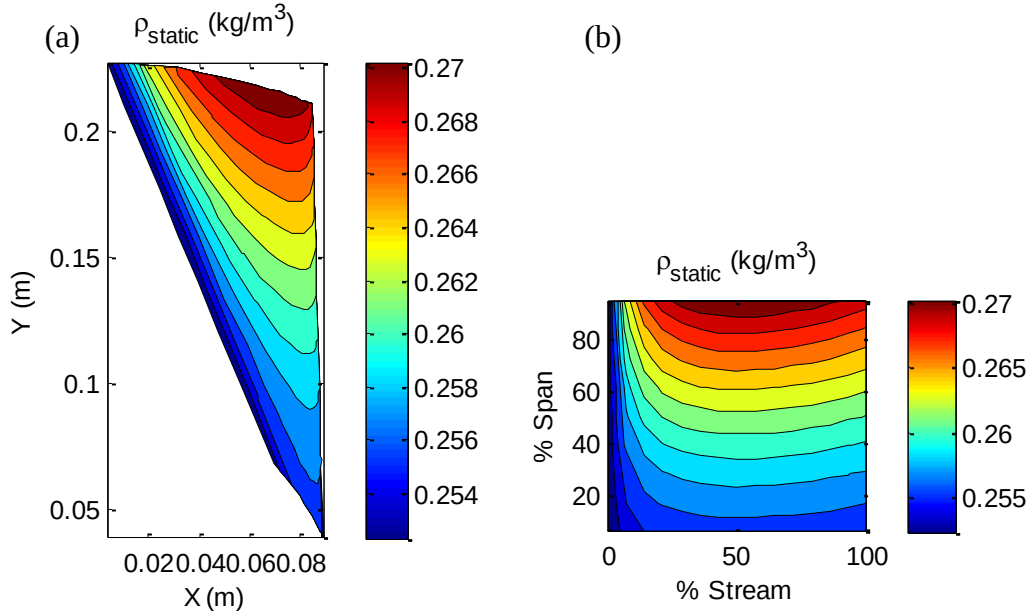


Figure 4.16. Static Density Contours (a) vs. Global Coordinates (b) vs. Percent Stream and Span

The swirl distribution within the rotor is described by a parabola at each stream-wise location, indicating a solid-body type of swirl is present. This is very closely approximated by the “Forced Vortex” condition, i.e.

$$c_u r = r^2 K \quad \text{or} \quad \frac{c_u}{r} = K \quad (4-23)$$

where K is a constant. Figure 4.17 plots values of $c_u r$ at different stream-wise locations for different spans.

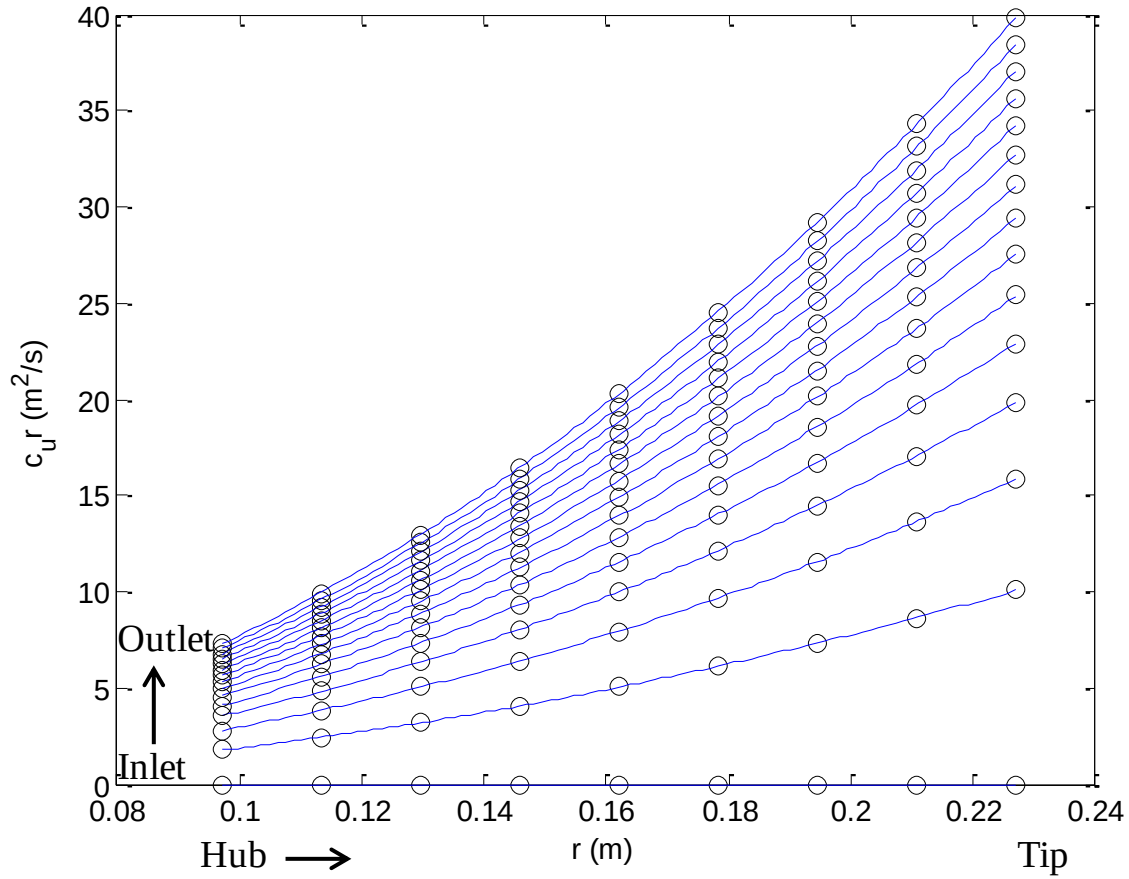


Figure 4.17. Star Pattern Swirl Distribution

Table 4.2 provides the values of K at the hub and tip from inlet to outlet.

% Stream	K_{hub}	K_{tip}	$K_{\text{Area-Weighted Avg}}$	% Diff $K_{\text{tip-hub}}$
Outlet (100%)	186.7	195.4	192.4	0.43%
92%	295.8	307.9	303.6	0.38%
85%	373.1	385.5	381.0	0.32%
77%	433.1	444.8	440.6	0.26%
69%	482.5	492.8	489.0	0.21%
62%	525.0	534.2	530.8	0.17%
54%	562.9	570.8	567.9	0.13%
46%	597.4	604.0	601.6	0.11%
38%	629.4	634.9	632.9	0.08%
31%	659.7	664.1	662.5	0.06%
23%	688.8	692.0	690.8	0.05%
15%	716.9	719.2	718.4	0.03%
8%	744.5	745.8	745.3	0.02%
Inlet (0%)	771.8	772.2	772.0	0.01%

As can be seen in Table 4.2, the differences in the K values between hub and tip are at worst less than half a percentage point, and at best a mere one-hundredth of a percent. This means that the forced vortex condition can be used to reasonably describe the swirl within the star pattern rotor. Doing so can ease the flow calculation process for star pattern geometries of different patterns and sizes since the c_u component can be determined without first iterating c_m .

Figures 4.18 and 4.19 show the calculated contours of w_u and w . Since the meridional velocity does not change with reference frame ($c_m = w_m$), these contours are not included a second time.

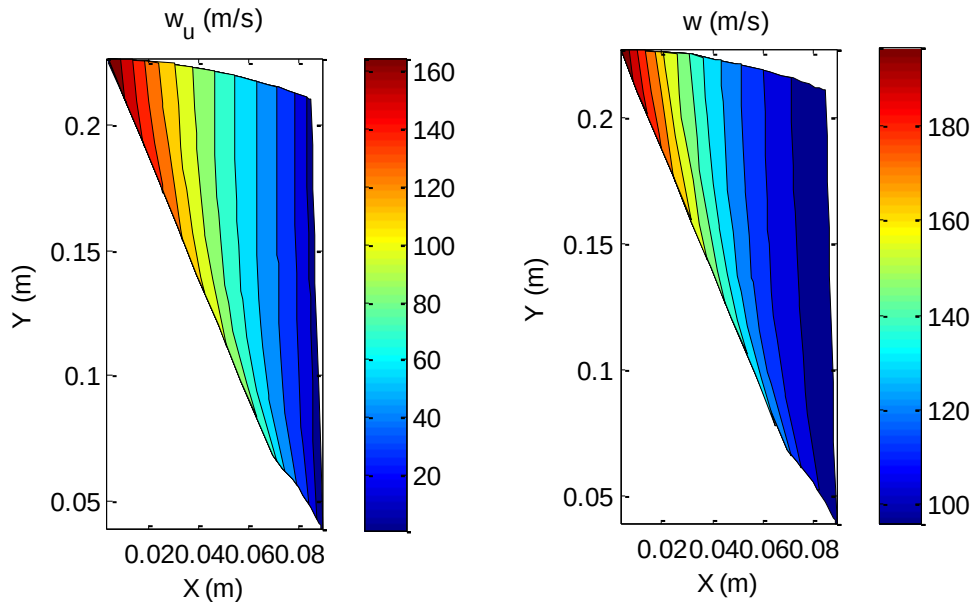


Figure 4.18. Relative Velocity Contours vs. Global Coordinates

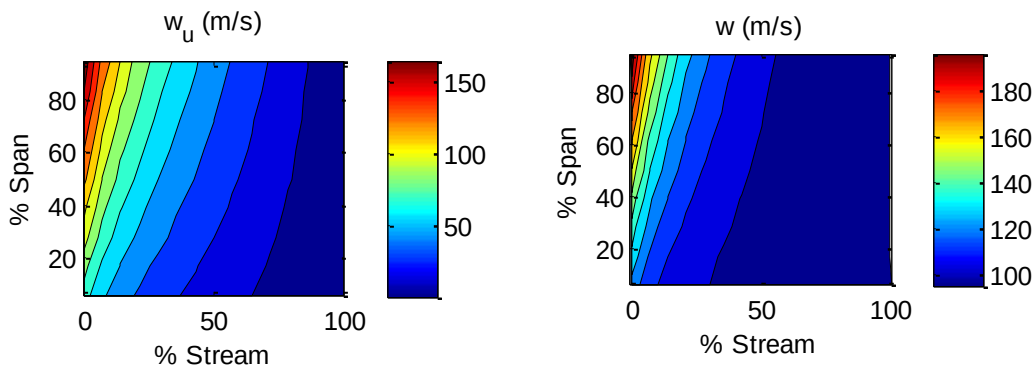


Figure 4.19. Relative Velocity Contours vs. Percent Stream and Span

Figures 4.20 and 4.21 show contours of absolute and relative critical Mach numbers. They behave as expected in that the absolute Mach number increases, and the relative Mach number decreases with increasing percent stream indicating that the flow is following the blades. The largest values occur at larger radii, and they of course reflect the respective contours of c and w . They also both fall below their maximum allowable values as per the constraints outlined in Table 2.5 (i.e. $M_c^* \leq 0.7$ and $M_w^* \leq 0.8$ everywhere).

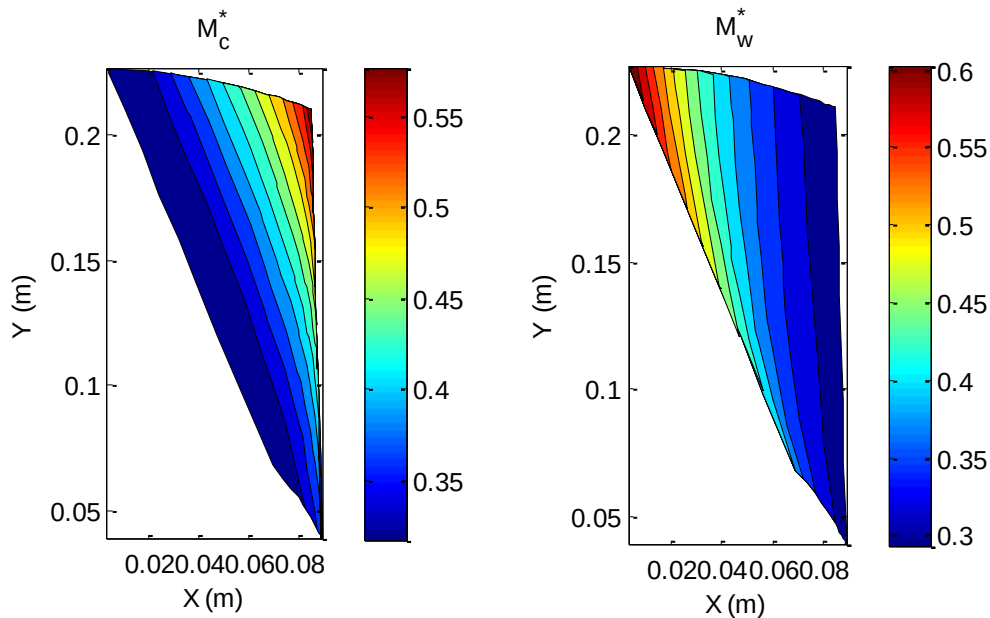


Figure 4.20. Absolute and Relative Critical Mach Number Contours vs. Global Coordinates

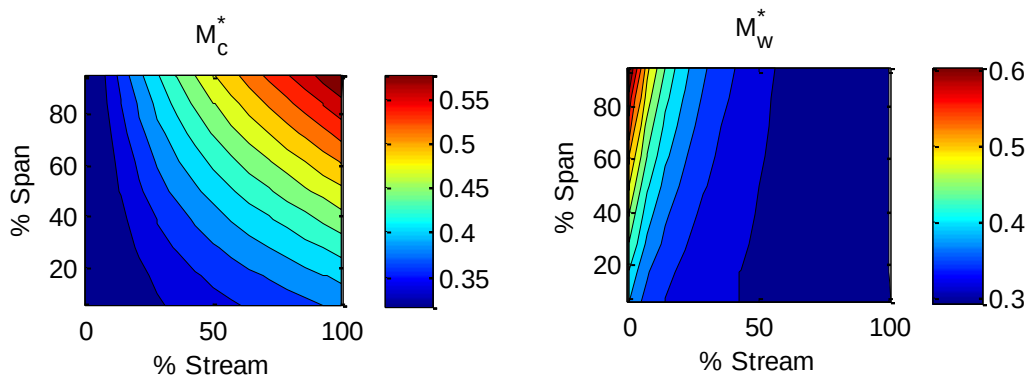


Figure 4.21. Absolute and Relative Critical Mach Number Contours vs. Percent Stream and Span

Table 4.3 summarizes the performance characteristics determined by the 2D calculations in this section. Geometric quantities are also included. Data is presented ranging from tip to hub, and the values presented are in reference to the entire stream-wise direction.

Table 4.3. 2D Performance and Geometry Summary

Span	Π_t	$\tilde{e}$	dH	DF	$\Delta\beta$	r_b	ch	t	σ
%	-	kJ/kg	-	-	°	m	m	m	-
Tip (94.5)	1.3170	31.265	0.4668	0.7787	58.8	0.158	0.155	0.089	1.743
83.4	1.2700	26.951	0.4952	0.7308	57.0	0.161	0.154	0.083	1.858
72.3	1.2273	22.957	0.5263	0.6797	55.1	0.165	0.152	0.076	1.992
61.3	1.1890	19.295	0.5601	0.6258	52.9	0.169	0.150	0.070	2.147
50.1	1.1547	15.944	0.5972	0.5680	50.4	0.175	0.149	0.064	2.334
39.0	1.1241	12.911	0.6377	0.5065	47.5	0.182	0.147	0.057	2.559
28.0	1.0974	10.207	0.6821	0.4409	44.3	0.192	0.145	0.051	2.839
16.9	1.0740	7.816	0.7281	0.3052	40.6	0.206	0.143	0.015	9.783
Hub (5.8)	1.0541	5.743	0.7802	0.3037	36.4	0.225	0.141	0.040	3.532
Area-Weighted Average	1.1938	19.573	0.5762	0.5997	51.5	0.175	0.150	0.067	2.789

As can be seen highlighted in Table 4.3, only three near-hub segments of the nine do not exceed their maximum diffusion factor constraints ($DF \leq 0.45$), and only two near-hub segments do not have excessive relative deceleration and exceed the de Haller criteria ($dH \geq 0.72$). Since these criteria are imposed to place reasonable expectations on performance, a fully 3D simulation accounting for turbulence and viscous effects must be performed.

4.4.1 A Note on Blade Stagger, Solidity, and Diffusion Factor

Due to the unique construction of the woven star-pattern impeller, the blade stagger (and hence blade solidity and diffusion factor) do not uniformly increase in general as would be the case for a conventional impeller. The fully wound impeller is axisymmetric, but within a given

volume between periodic surfaces (see Section 4.5), there are two distinct flow regions. Figure 4.22 shows these two regions and their differing values of blade stagger at varying span, depicted as red and black dotted lines.

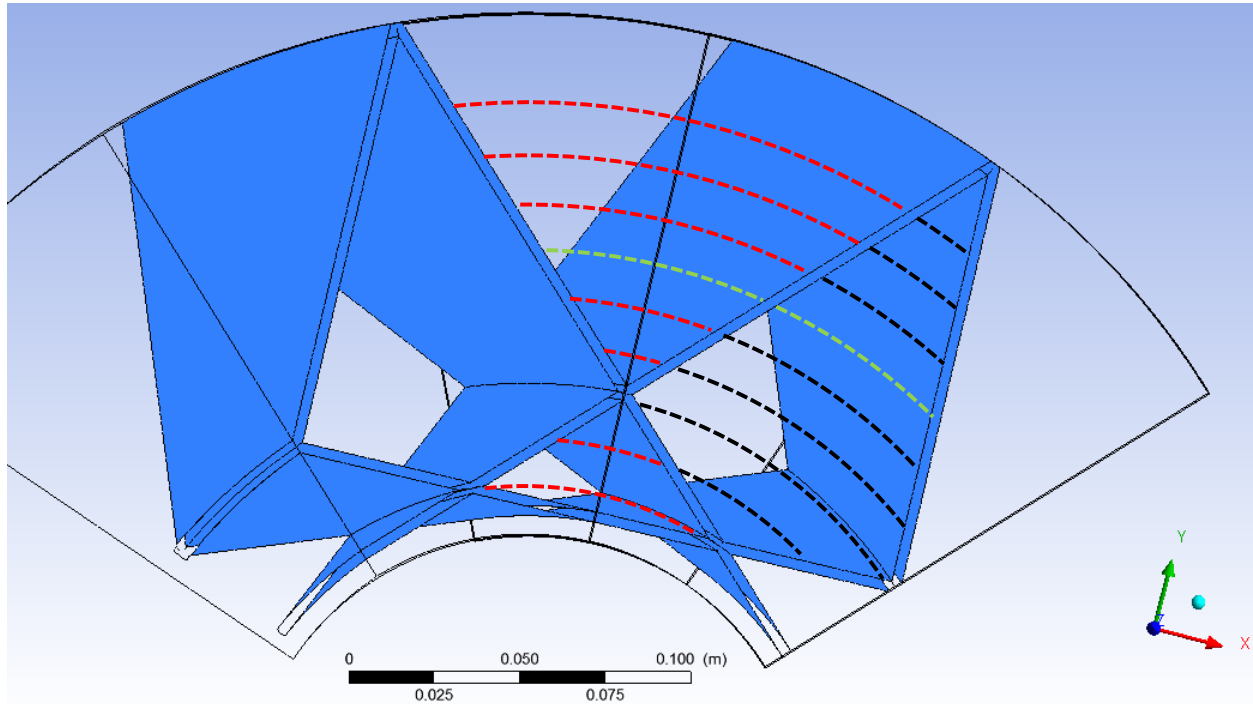


Figure 4.22. Differing Blade Stagger at Single Span-wise Locations

Moving from low span to high, black lines exhibit small stagger with rapid increase, and then more gradually decreases to zero stagger at the tip. Red lines exhibit the opposite behavior with the largest blade stagger at any span occurring at the tip. Since each span has constant chord length, blade solidity exhibits the inverse behavior (see Equation (2-131)), and the values of t , σ , and DF given in Table 4.3 are therefore averaged quantities at their respective spans. The green lines in Figure 4.22 are located at 50.1% span where the blade stagger is equal in each region. Table 4.4 provides the values of t , σ , and DF for each region.

Table 4.4. Region-Dependent Blade Stagger, Solidity, and Diffusion Factor

Span	t_{red}	t_{black}	t_{avg}	σ_{red}	σ_{black}	σ_{avg}	DF_{red}	DF_{black}	DF_{avg}
%	m	m	m	-	-	-	-	-	-
Tip (94.5)	0.172	0.007	0.089	0.9	23.0	1.7	0.5517	1.0057	0.7787
83.4	0.145	0.020	0.083	1.1	7.5	1.9	0.5605	0.9011	0.7308
72.3	0.118	0.034	0.076	1.3	4.4	2.0	0.5662	0.7932	0.6797
61.3	0.091	0.049	0.070	1.6	3.1	2.1	0.5690	0.6826	0.6258
50.1	0.064	0.064	0.064	2.3	2.3	2.3	0.5677	0.5682	0.5680
39.0	0.035	0.080	0.057	4.2	1.8	2.6	0.5623	0.4506	0.5065
28.0	0.005	0.097	0.051	31.6	1.5	2.8	0.5529	0.3289	0.4409
16.9	0.030	0.059	0.045	4.8	2.4	3.2	0.4069	0.3404	0.3737
Hub (5.8)	0.078	0.002	0.040	1.8	88.8	3.5	0.2231	0.3844	0.3037
Area-Weighted Average	0.095	0.043	0.069	1.6	3.5	2.2	0.5338	0.6762	0.6050

Since the diffusion factors for spans less than 39% are below the upper limit of 0.45 in the region denoted with black lines, it is expected that the flow will exhibit somewhat healthier behavior than in all other regions. However, the de Haller criterion is still not met in any region above 16.9% span, and so healthiest flow behavior (i.e. no flow separation) is expected to be exhibited in this near-hub region.

4.4.2 Boundary Layer Growth Prediction

It is possible to use the flat-plate approximation on a curved surface as derived in [47] to estimate the boundary layer thickness, δ , as a function of span, i , and axial location, j as

$$\delta_{i,j} = 0.38 x_{i,j} \text{Re}_{x_{i,j}}^{-1/5} \quad (4-24)$$

where the Reynolds number, Re , is calculated with reference to the relative velocity as

$$\text{Re}_{x_{i,j}} = \frac{w_{i,j} x_{i,j} \rho_{i,j}}{\mu} \quad (4-25)$$

and the blade curvature in the axial direction denoted here as $x_{i,j}$ is

$$x_{i,j} = r_{b,j} \phi_{i,j} \quad (4-26)$$

which gives the actual arc length of the curved blade at a span- and stream-wise location relative to the blade leading edge. Any particular ϕ (see Figure 4.3) is

$$\phi_{i,j} = \phi_{i-1,j} + \beta_{i,j} - \beta_{i-1,j} \quad (4-27)$$

and

$$\beta_{i,j} = 90^\circ - \sin^{-1} \left(\frac{z_{rot} - z_i + y}{r_{b,j}} \right) \quad (4-28)$$

The boundary layer thickness increases in the axial direction on all blade surfaces. Of particular interest is the black-lined region shown in Figure 4.22 near the tip (circled in Figure 4.23) where boundary layer growth from each blade surface can be large enough so as to intersect with one another. Figure 4.23 depicts the linear distance between, $\mathcal{A}$, different blades emanating from the same mandrel slot at different spans (blue lines between green and pink circles), and these values are presented numerically in Table 4.5. The value of $\mathcal{A}$ does not change with stream-wise position.

Since the boundary layer thickness increases with downstream distance, the boundary layer thickness at the outlet is considered here to determine the largest thickness.

Table 4.5. Boundary Layer Thickness at Rotor Outlet

Span %	$\delta_{out,j}$ mm	$2\delta_{out,j}$ mm	$\mathcal{A}$ mm
Tip (94.5)	5.2	10.5	6.6
83.4	5.2	10.4	20.2
72.3	5.1	10.3	34.1
61.3	5.1	10.2	43.8

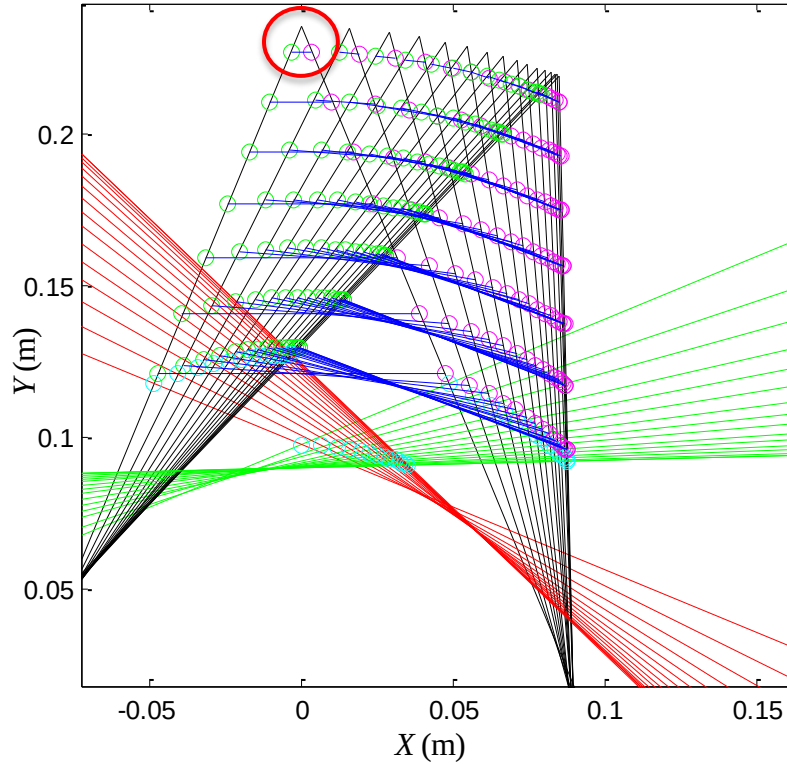


Figure 4.23. Linear Distance A Between Blades Emanating from One Mandrel Slot

Table 4.5 indicates that $2\delta_{out,j} > A$ only in the very-near tip region. Here it is expected that the boundary layers from each blade surface will grow thick enough to intersect with one another, indicating a complete absence of core flow in this region. At span 94.5%, Table 4.6 provides the boundary layer thickness as it increases in the axial direction in terms of %-M' (normalized stream-wise location).

Table 4.6. Near-Tip Boundary Layer Thickness vs. Normalized Stream-wise Location

%-M'	$2\delta_{out,j}$ (mm)	A (mm)
0	0	6.6
20.6	2.7	6.6
36.5	4.4	6.6
50.3	5.9	6.6
63.2	7.1	6.6
75.5	8.3	6.6
87.8	9.4	6.6
100	10.5	6.6

The two thickening boundary layers are predicted to intersect at around 57%M' at 94.5% span. For stream-wise locations greater than 57%M', the span of intersection is then expected to

decrease as stream-wise location increases. This effect is however outpaced by Δ , which increases at a greater rate than twice the boundary layer thickness, indicating that the intersecting boundary layers should be seen only in the very near-tip region. In this region of intersecting boundary layers, the flow is not expected to produce useful pressure ratio or see significant through-flow as there is a complete absence of core flow.

The boundary layer intersection in the near-tip region in combination with the excessive de Haller numbers and Diffusion Factors at spans over 17% lead to the expectation that the core flow will exhibit healthy behavior in the near-hub region where there is both less blade turning and on average higher blade solidity due to the fiber crossings of multiple blades. Furthermore, it is expected that a larger range of spans will exhibit healthy core flow nearer the inlet (less boundary layer growth as well as less blade and flow turning), and this range of spans containing healthy core flow should decrease with increasing stream; in particular this decrease should recede from the tip, with large spans exhibiting signs of boundary layer separation

Including the boundary layer thickness at the outlet on each blade surface in addition to the blade thickness gives a more accurate approximation of the expected mass flow exiting the rotor. This is achieved by replacing the t_b terms in Equation (4-22) with $t_b+2\delta$ at the corresponding span. The adjusted mass flow rate of the core flow accounting for boundary layer thickness is $\dot{m}_{adj}=2.733\text{kg/s}$, a 22.8% reduction with respect to the initially calculated 3.544kg/s .

4.5 Star Pattern Impeller 3-Dimensional Simulation

ANSYS CFX is used to simulate the flow conditions in 3D and they are compared to the 2D calculations of the previous section. Since computing resources are finite, common practice in turbomachinery simulation is to simulate only a wedge-shaped slice of the flow field

surrounding the impeller rather than the entire wheel, thus enabling higher flow field resolution with potentially less overall computational expense. In order to implement this, a representative repeating section must be selected, and it should be selected such that the faces aligned with the axis of rotation are parallel to the blade faces, equally spaced between blades, and centrally encompass a third blade. These repeating sections are bounded by periodic surfaces where it is assumed that all flow properties on one surface exactly match those of the other.

When weaving the star pattern impeller, each slot in the mandrel serves as the origin for one blade (blue) and the terminus of another (black); this is depicted in Figure 4.24.

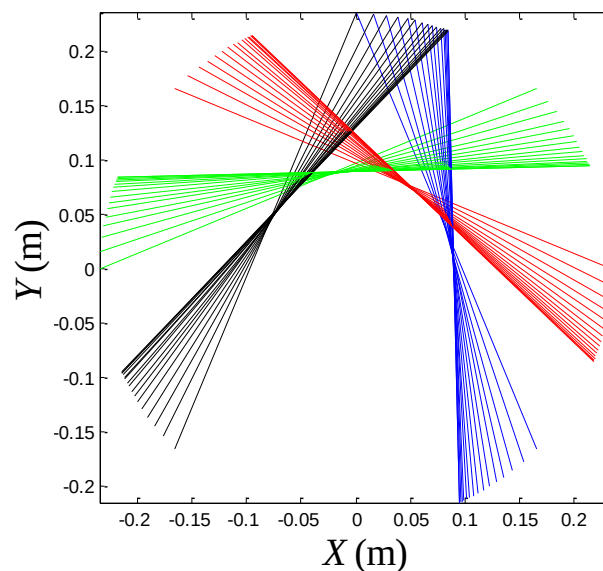


Figure 4.24. Two Blades Emanating from One Mandrel Slot

The geometry shown in Figure 4.24 is replicated in Siemens NX and the repeating section is created. Figures 4.25 – 4.27 show the 3D-modeled single blade, four blades, and the periodic sections, respectively.

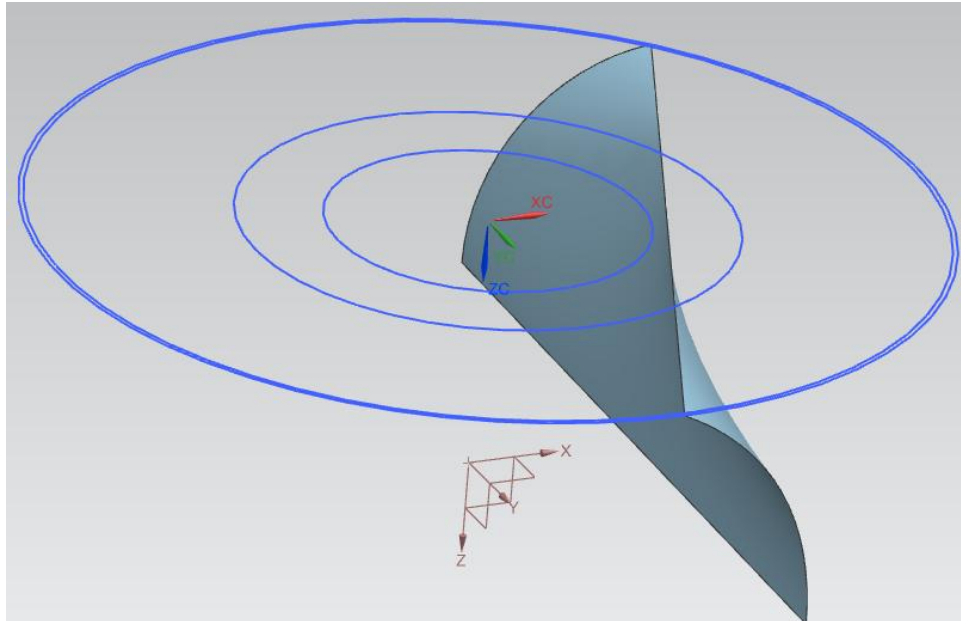


Figure 4.25. Single 3D Blade

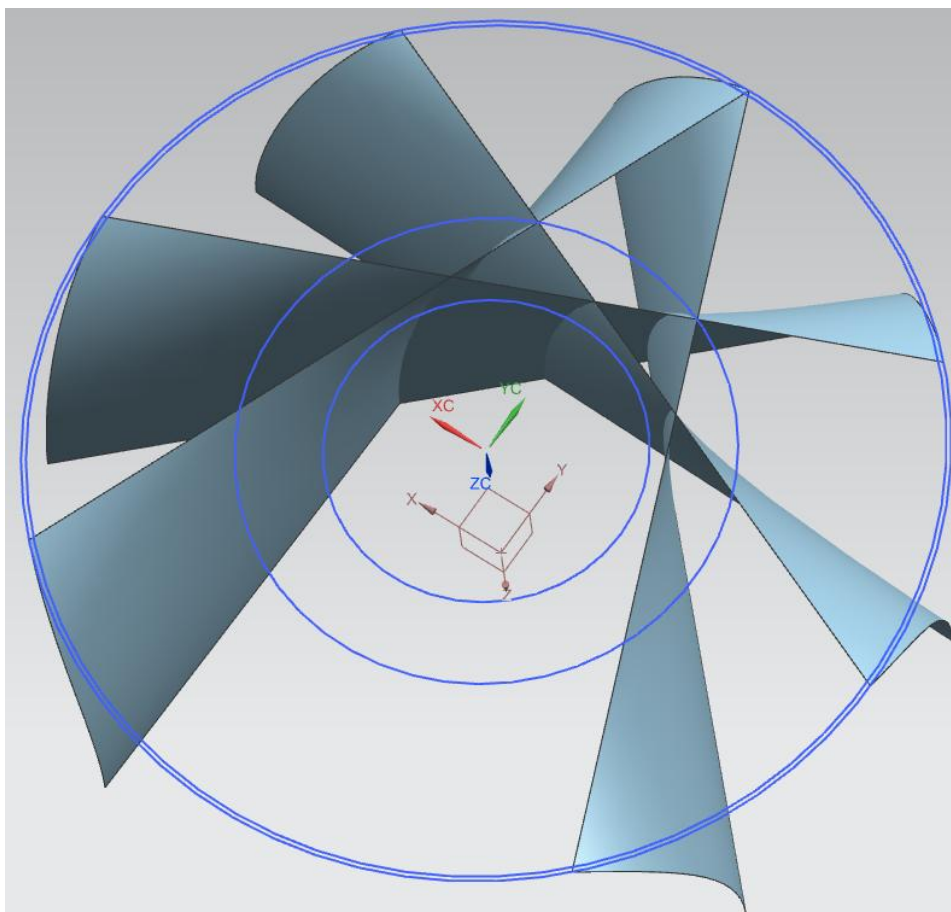


Figure 4.26. Four 3D Blades

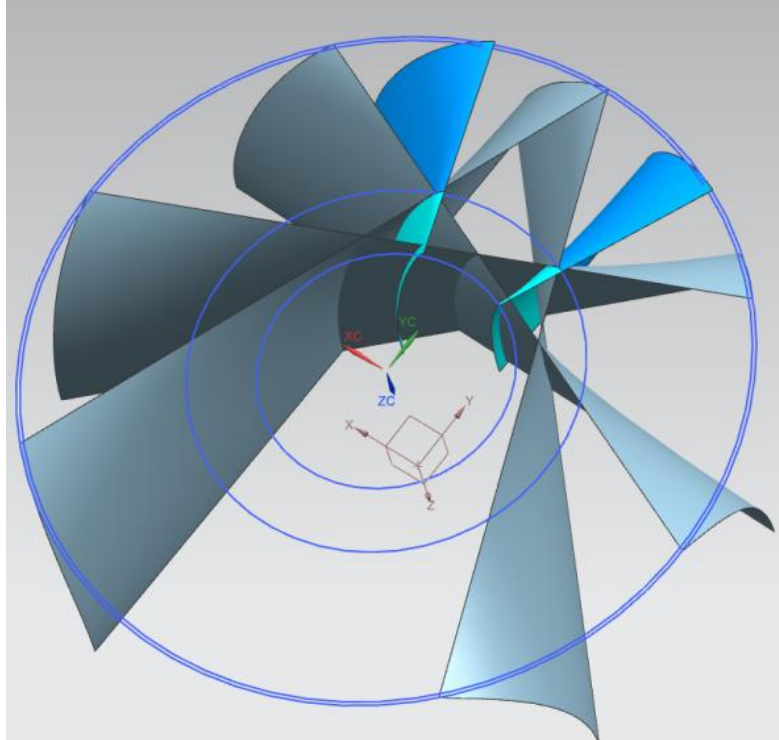


Figure 4.27. Four 3D Blades with Periodic Sections

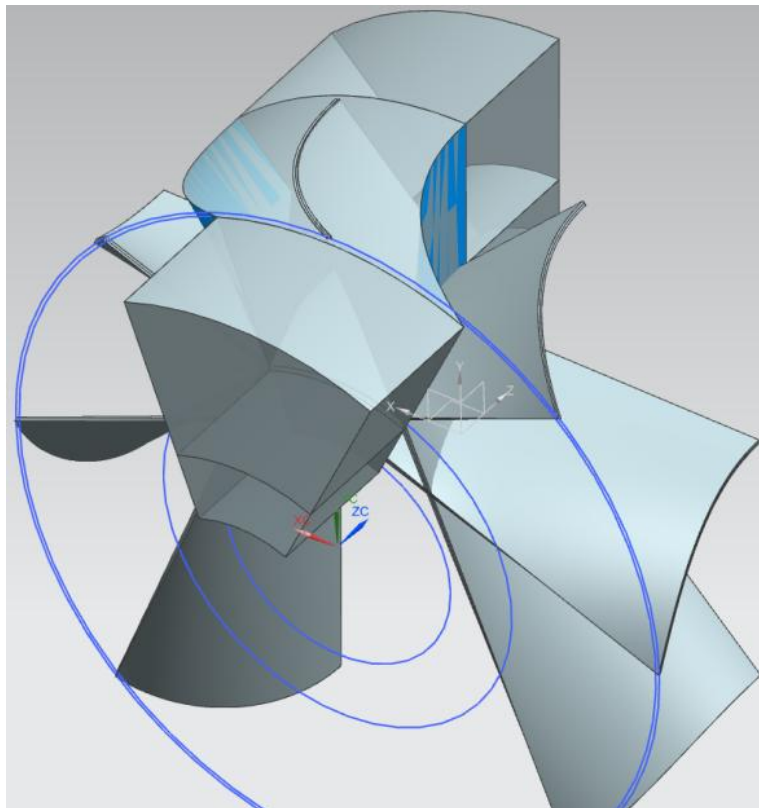


Figure 4.28. Enclosed Volume with Inlet and Outlet Regions

Thickness is added to the blades and the periodic sections in combination with hub and shroud create a closed volume. An inlet and an outlet region are also added as shown in Figure 4.28. The blades are subtracted from this volume leaving behind the fluid region, shown in Figure 4.29.

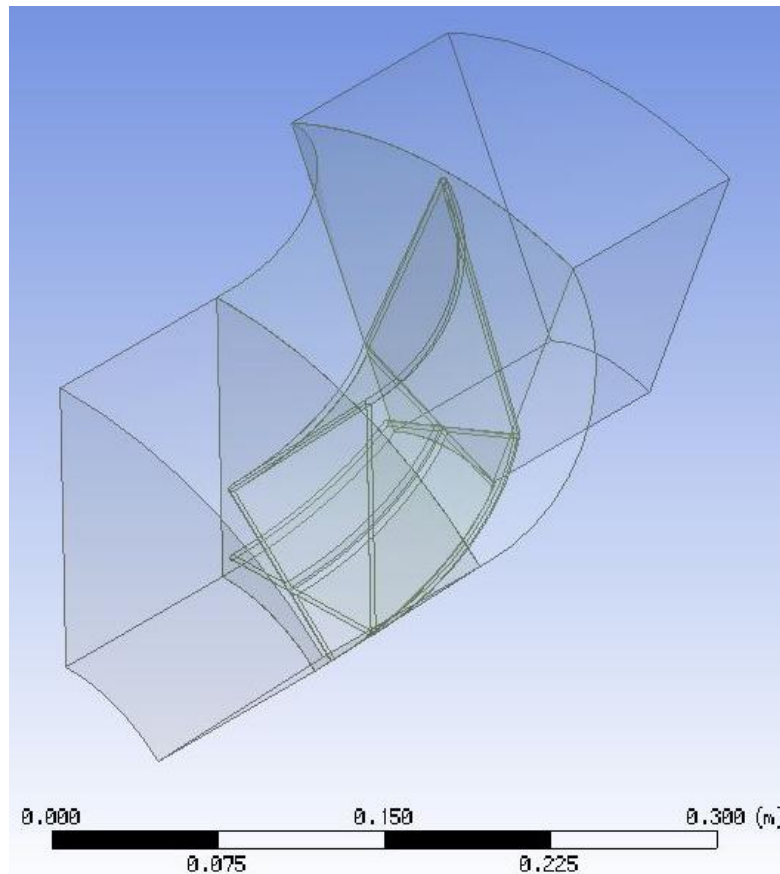


Figure 4.29. Fluid Region for Simulation

The fluid region must be meshed so that the differential equations describing the flow field (Chapter 3) can be converted to algebraic equations and solved at each discrete point (node). The inlet and outlet regions are meshed using the Sweep Method to fill the region with hexahedral elements. The complex rotor geometry is such that a finite hexahedral mesh is not possible to sweep without overlapping elements (resulting in negative element volumes); for this reason the rotor region was meshed with tetrahedrons using the Patch Conforming Method [52].

4.5.1 Mesh Independence

Since the discretization from differential transport equations to algebraic equations inherently introduces error to the flow calculations, it must be ensured that the mesh itself is not one of those sources of error. Mesh independence is achieved when a variable of interest is within a certain percent-change on meshes of different density. The coarsest mesh for which this is true may then be selected for use to expend the least amount of computational effort. Figure 4.30 shows very coarse, medium, and very fine meshes used to test for mesh independence

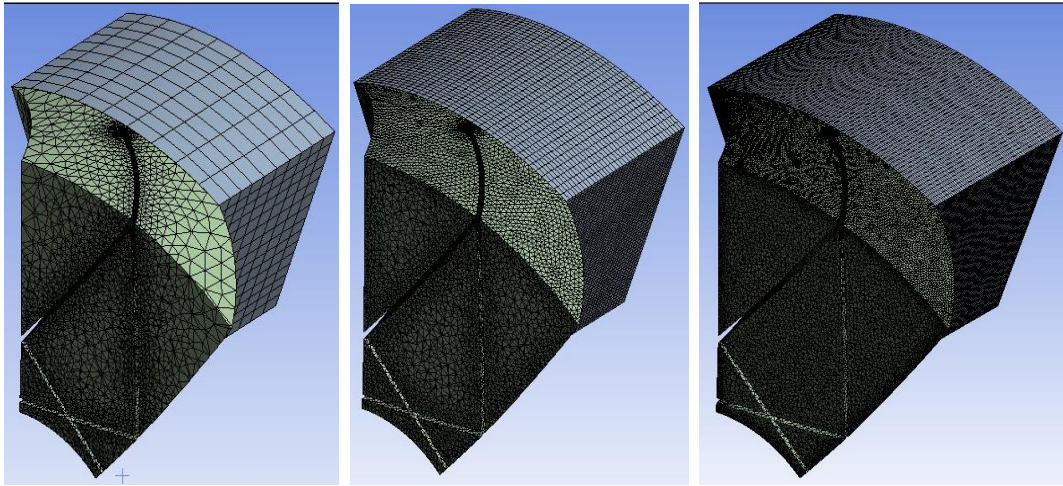


Figure 4.30. Very Coarse (Left), Medium (Center), and Very Fine (Right) Meshes

The efficiency (both polytropic and isentropic) was the variable of choice for monitoring since it is a reflection of both the thermodynamics and kinematics of the flow. Mesh independence was considered achieved when the efficiency values varied by 1% or less when successively doubling (or $\times 1.5$) the mesh density, i.e. maximum allowable element face size was successively halved (or $\times 2/3$). Figure 4.31 plots this progression. Further details of the mesh independence study including numerical residual histories are provided in Appendix A1.

The medium mesh comprised of just under 2 million elements (over 500,000 nodes) shown in Figure 4.32 in its entirety was ultimately used.

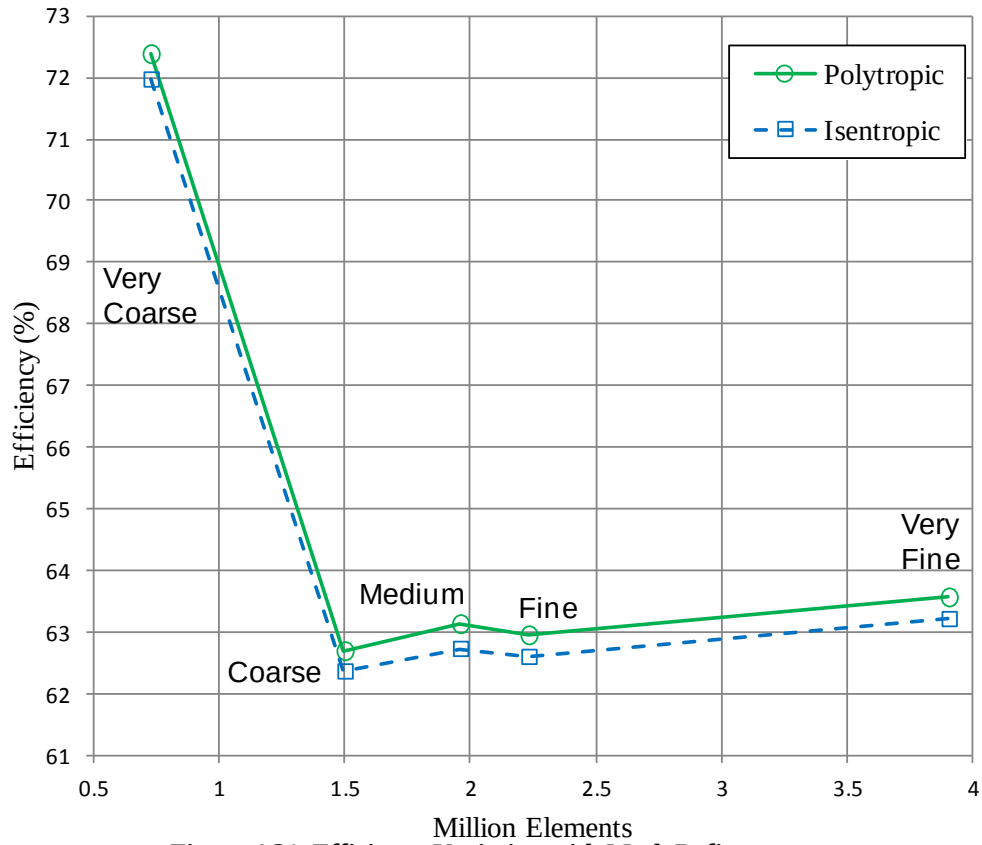


Figure 4.31. Efficiency Variation with Mesh Refinement

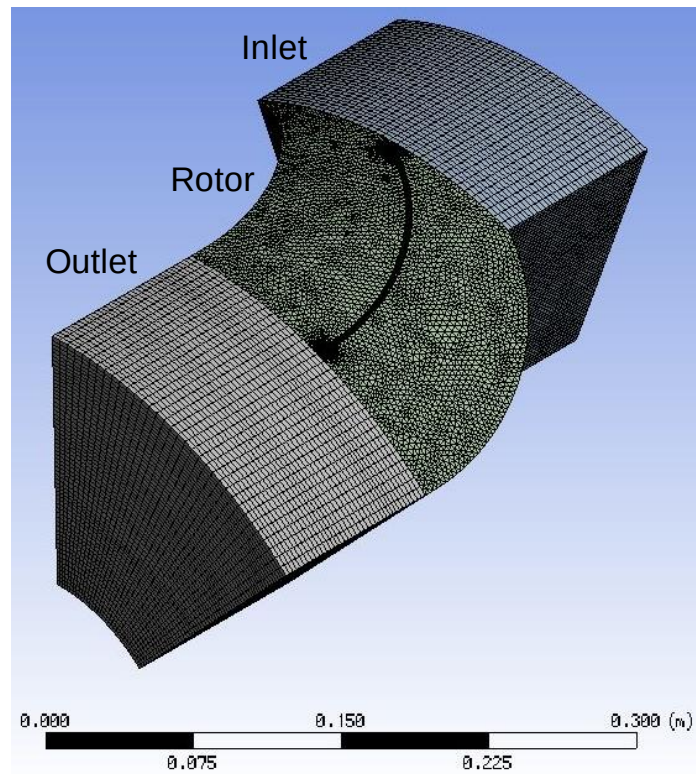


Figure 4.32. Medium-Density Mesh

4.5.2 Simulation Results

Table 4.7 summarizes the main simulation results and compares them with the analytical calculations in terms of their area-weighted averages.

Table 4.7. Main Simulation Results				
		Calculation	Simulation	Difference
$\tilde{e}$	kJ/kg	19.57	11.15	43.0%
$\dot{m}$	kg/s	3.54	2.58	27.3%
$\dot{m}_{adj}$	kg/s	2.73	2.58	5.6%
η_{pt}	%	80.0%	63.1%	16.9%
Π_t	-	1.193	1.074	10.0%
$\tilde{P}$	kW	53.5	28.7	46.4%

The largest discrepancy between the simulation results and analytical calculations is in the power consumption: the simulation has a lower-than-predicted mass flow rate and specific work transfer to the fluid, hence the power consumption has been over predicted by over 46% (refer to the left sides of Equation (2-78)). However, the adjusted mass flow rate accounting for boundary layer thickness is only just over 5% larger than the simulated mass flow rate. This is in contrast to the initially calculated 27% over-prediction of the mass flow rate when not accounting for the boundary layer. The total pressure ratio achieved was only over predicted in by 10%. Figure 4.33 shows contours of total pressure in the meridional view plane.

Note that in the following simulation contours, the rotor region is between the solid black vertical lines. Additionally, the lower -(b) contours are the calculation contours presented in Section 4.4 shown here again for convenience.

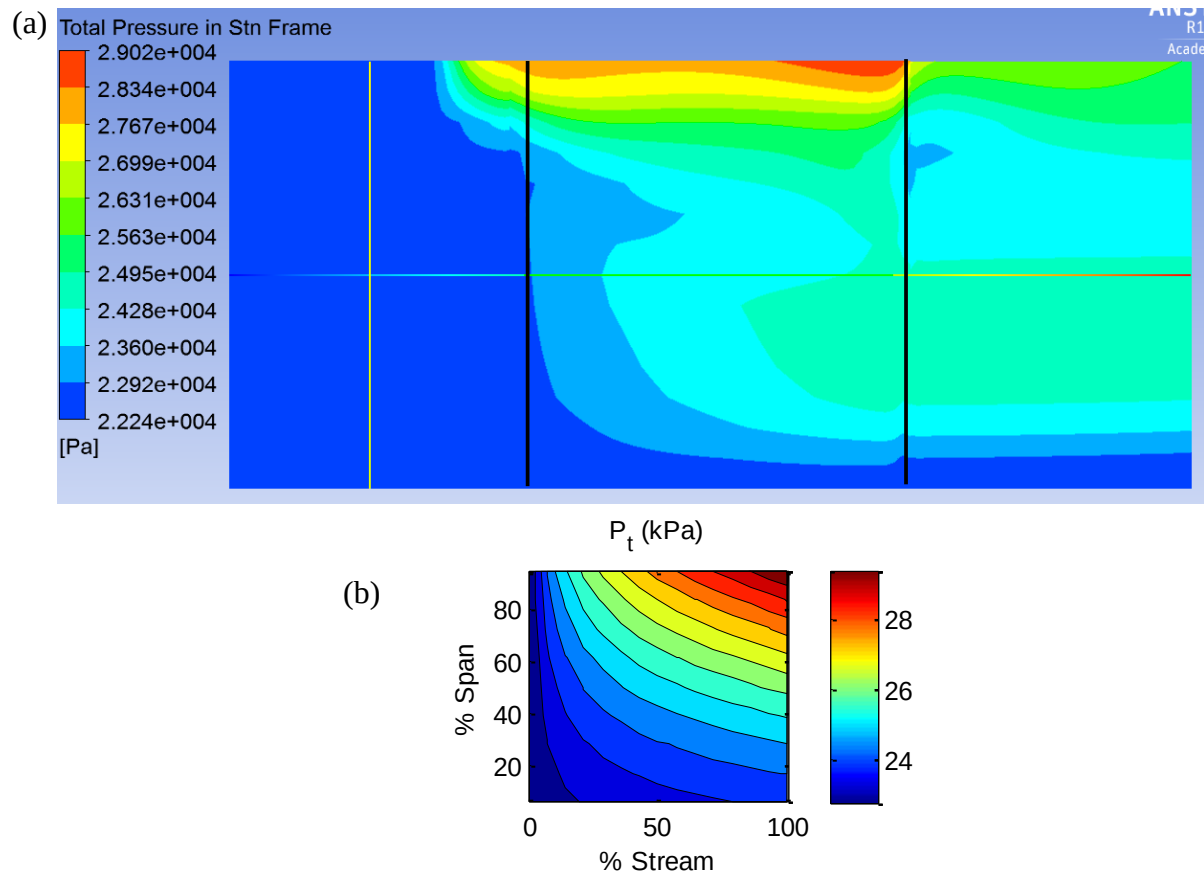


Figure 4.33. Meridional View of Total Pressure Contours (a) Simulation (b) Calculation

Total pressure increases from inlet to outlet just as it does in the calculation contours. However, in the near-tip region within the rotor, the pressure is large and approximately constant from inlet to 80% stream, at which point there is a slight pressure increase approaching the outlet. Yet once the flow exits the rotor, much of this total pressure gain is abruptly lost, particularly for high span. This is because much of the total pressure gained is due to stagnated fluid buildup in the near tip region due to boundary layer separation. There is very little through-flow in this region as shown in Figure 4.34-(a), and there is a radial velocity component near the exit in the tip direction as shown in Figure 4.35 (there was zero radial velocity assumed in the calculated velocity profile). While pressure may build up within the rotor, it is unable to move down stream in useful fashion as the fluid behind the rotor simply expands to fill the space that

the rotor is not occupying. There is also large entropy generation in this region (see Figure 4.36), which leads to total pressure loss as discussed in Section 2.2.2.

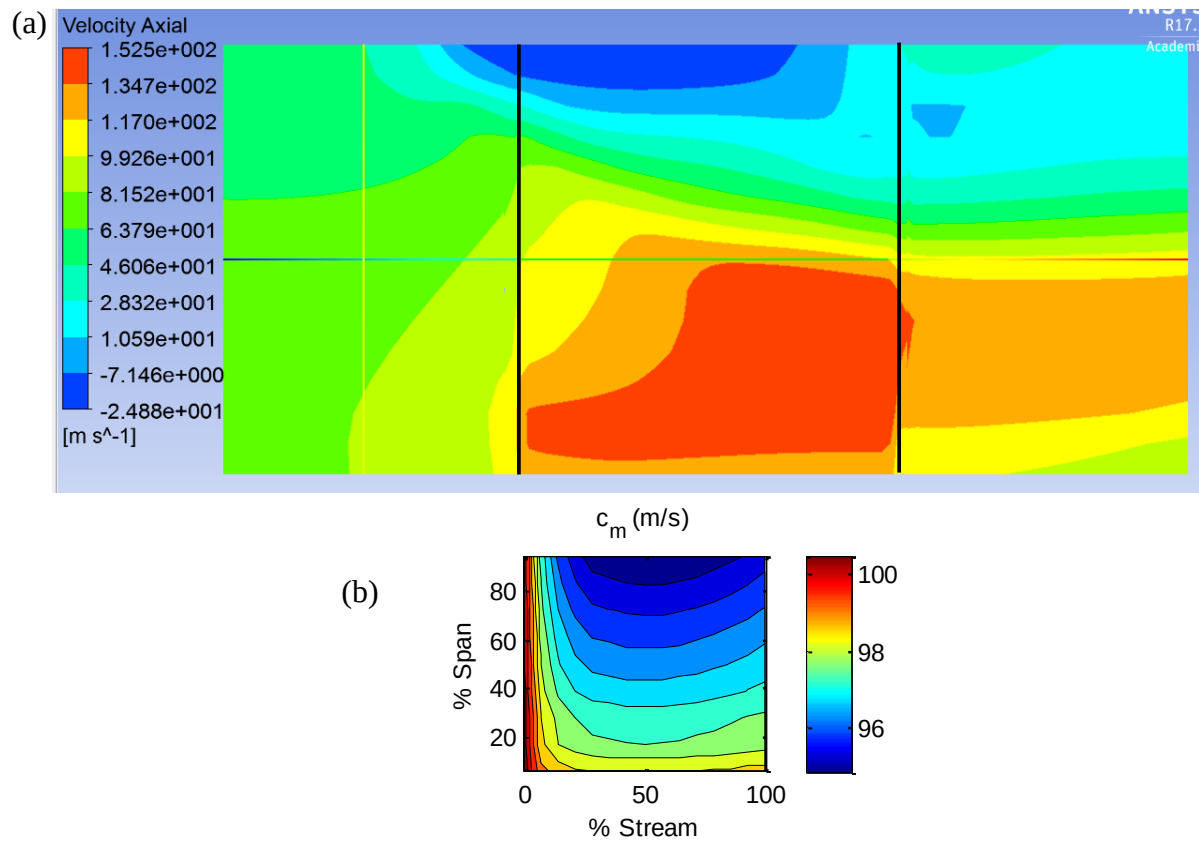


Figure 4.34. Meridional of Axial Velocity Contours (a) Simulation (b) Calculation

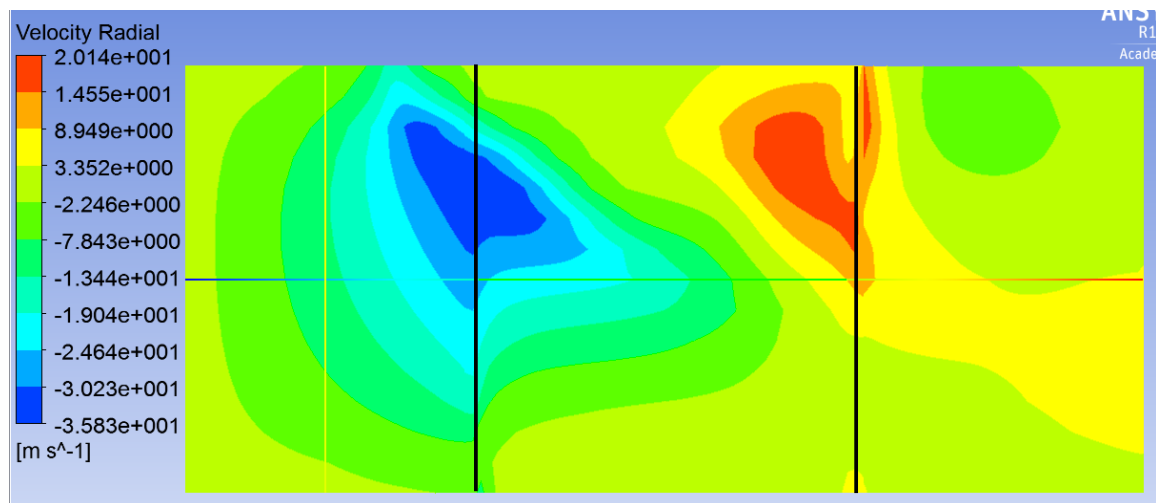


Figure 4.35. Meridional View of Radial Velocity Contours

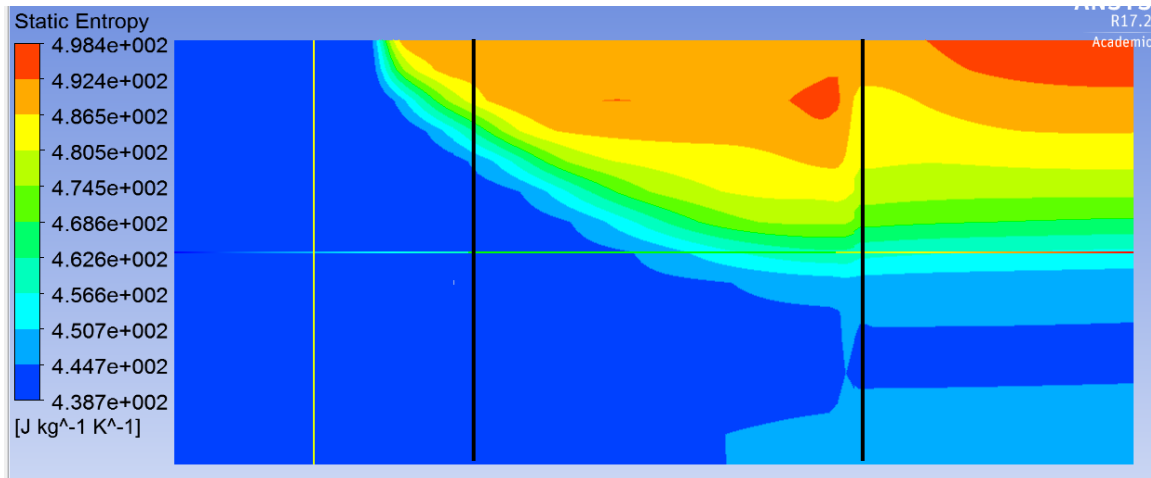


Figure 4.36. Meridional View of Entropy Contours

The presence of reversed flow at large spans (negative axial velocity shown in Figure 4.34-(a)) indicates that there is a large boundary layer in the flow that has grown thick enough to separate as it is wont to do in an adverse pressure gradient flow with wall surface curvature [55].

As was predicted based on the data in Tables 4.3 and 4.4, the flow is well behaved in that it properly follows the blades at low span, but the contours from Figures 4.34 and 4.36 indicate that there is healthy flow up to around 50% span for all stream-wise locations. Tables 4.3 and 4.4 suggest that spans above approximately 17% would exhibit unhealthy behavior.

This is likely due to largely in part to the differences in the meridional velocity profiles between the calculated and simulated flows. The contour in Figure 4.34-(b) indicates that there should be a deceleration in calculated axial velocity upon entering the rotor, but then it should gradually accelerate toward the rotor outlet after reaching approximately 47% stream with velocities ranging from around 95m/s to 100m/s. Figure 4.34-(a) indicates that below 50% span, the axial velocity only increases from inlet to outlet with speeds ranging from approximately 100m/s to 150m/s, where above 50% span it only decreases, and in many cases actually becomes negative, indicating a reversal in flow, particularly near the tip. This much-accelerated flow below 50% span is what accounts for the simulated mass flow rate being only 27.3% less than

the simulated mass flow (Table 4.7) rather than by much more than that, as would generally be expected when experiencing such large regions of back flow.

The absolute tangential velocity behaves much as predicted (Figure 4.37), although the simulated maximum tangential velocity is over 200m/s rather than 150m/s as predicted. As some total pressure is lost in the tip region after exiting the rotor, so too is some of the tangential velocity component. However this serves to further illustrate that there is boundary layer separation in that essentially all of the flow is in the tangential direction in this region (Figure 4.37-(a)).

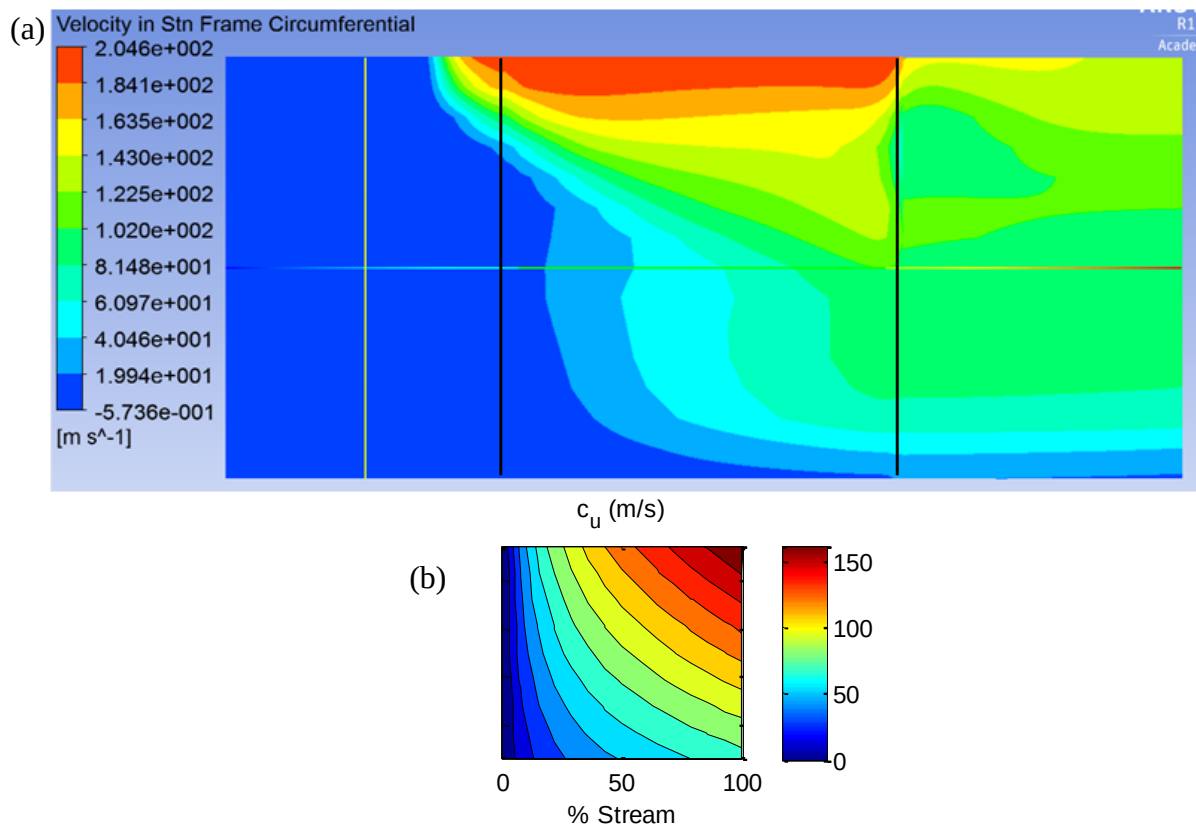


Figure 4.37. Meridional View of Absolute Tangential Velocity Contours (a) Simulation (b) Calculation

The abrupt change in relative tangential velocity in Figure 4.38-(a) from nearly 200m/s to around 0m/s (thickening of tangential flow near a wall) implies that the velocity is essentially normal to the blade surface at this point (all tangential), and therefore the boundary layer wants

break away from the blade surface. This then leads to separated flow and back flow as neighboring fluid from upstream moves to fill the vacated region [55].

Furthermore, the calculations assumed $c_u = 0\text{m/s}$ at all spans at the rotor inlet. It can be seen that near the tip region, some swirl component is propagated back up stream prior to entering the rotor. This appears to impact the relative tangential velocity as shown in Figure 4.38-(a), and the overall result is nearly opposite to the predicted behavior for w_u .

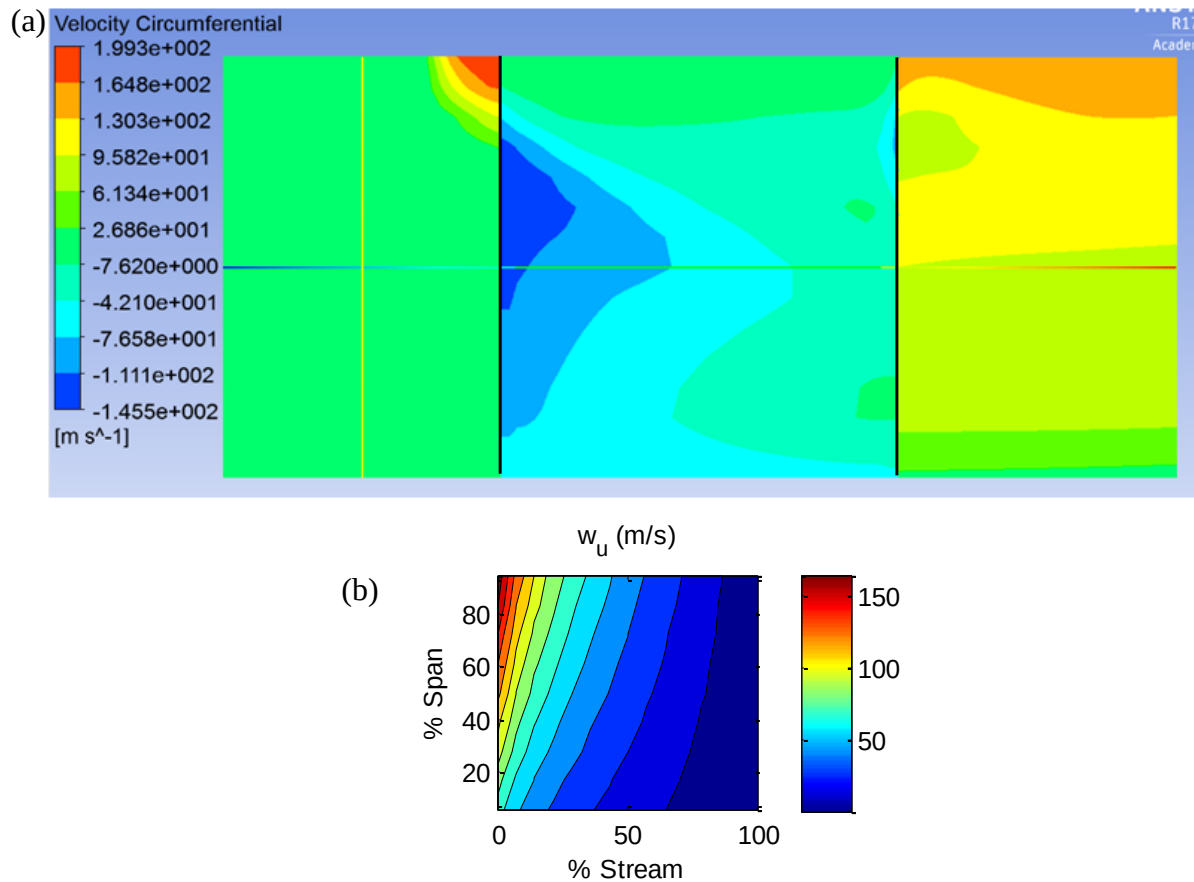


Figure 4.38. Meridional View of Relative Tangential Velocity Contours (a) Simulation (b) Calculation

When considering the boundary layer growth prediction in Section 4.4.2, the predicted trend of growth in the axial direction and the intersection of layers in the tip region that then extend towards the hub as %-stream increases seem to be confirmed, but the extent to which this happens is vastly under predicted. It was calculated that this would be the case for spans larger than 94% at stream-wise locations larger than 57% (see Tables 4.5 and 4.6), however it the

simulation results from Figure 4.34-(a) indicate that there is reverse flow at all stream locations for large spans, and this region expands to smaller spans as stream-wise location increases. Blade-to-blade views of meridional velocity for spans 0.2 to 0.9 are shown in the following figures to provide more detail.

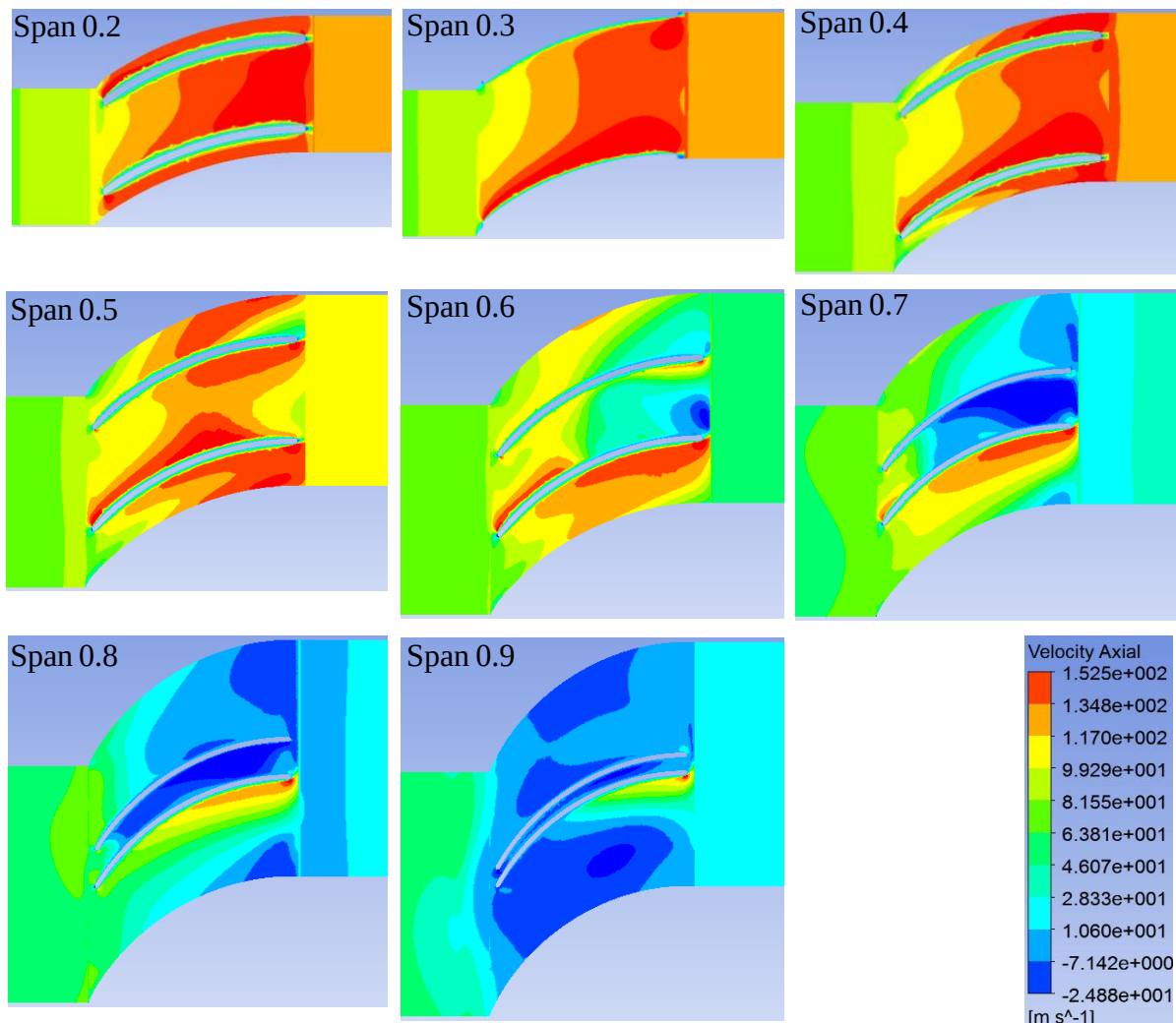


Figure 4.39. Blade-to-Blade View of Axial Velocity Contours

Regions of negative axial velocity can be seen forming near the outlet at span 0.6. For increasing span, negative axial velocities are present further upstream. At span 0.9 (and larger), nearly all of the axial velocity in the rotor is zero or negative. Figure 4.40 provides full-span views of the axial contours at the rotor inlet, mid-span, and outlet.

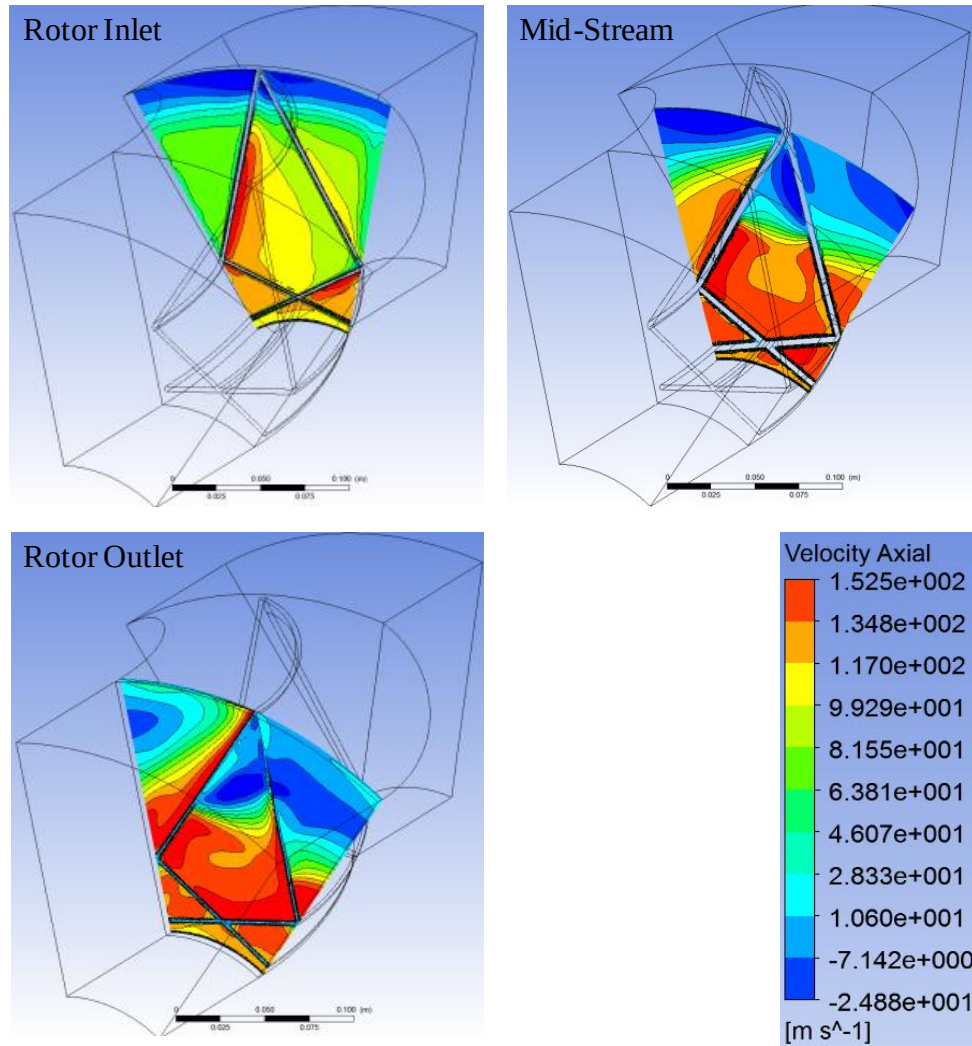


Figure 4.40. Full-Span View of Axial Velocity Contours

Table 4.8 provides de Haller numbers and Diffusion Factor values based on the simulated velocities.

Table 4.8. Simulated de Haller and Diffusion Factor Values

Span	dH	DF
Tip (94.5%)	0.9295	0.1389
83.4%	0.6385	0.4655
72.3%	0.3844	0.7608
61.2%	0.4884	0.6992
50.1%	0.7617	0.3640
39.0%	1.0392	0.0642
28.0%	0.9694	0.1323
16.9%	1.0605	0.0188
Hub (5.8%)	0.8979	0.1332
Area Weighted Average	0.8308	0.2701

The highlighted values above 50% span in Table 4.8 are values beyond the imposed limits and differ from the calculation values by an average of 30% and 46% for the de Haller Numbers and Diffusion Factors, respectively. The simulation values in the near-tip region are acceptable, but only superficially. This is because the full relative velocity includes the axial component, which is of course negative and does not vary greatly. The views presented in Figures 4.34-(a), 4.39, and 4.40 make it clear that despite the values in Table 4.8, the near-tip region exhibits some of the least desirable flow behavior.

Maximum absolute and relative Mach numbers were however well below their imposed limits in both calculation and simulation. This implies that losses in total pressure and operating efficiency compared to predicted values do not arise due to shock formation and the entropy increase inherently present behind them. Figures 4.41 and 4.42 show absolute and relative Mach number contours, respectively.

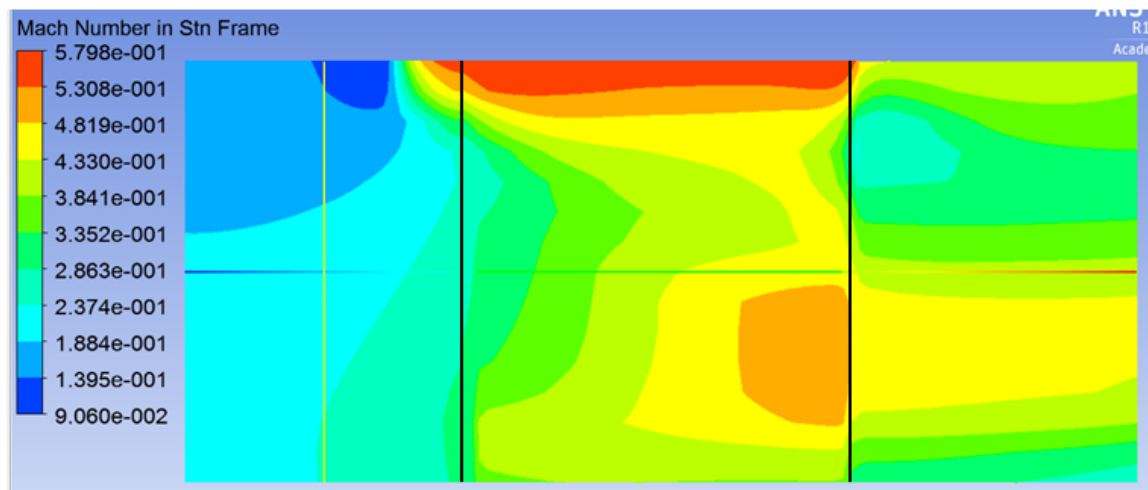


Figure 4.41. Meridional View of Absolute Mach Number Contours

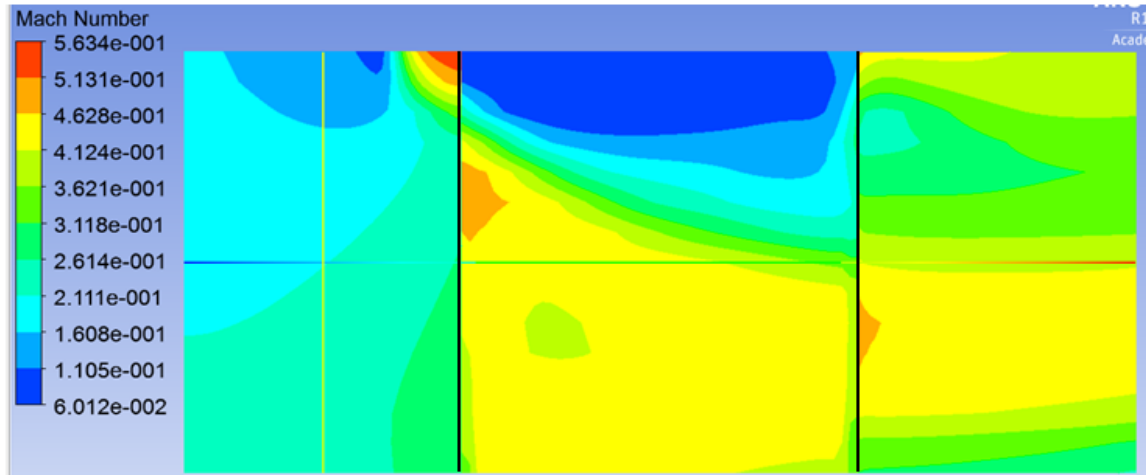


Figure 4.42. Meridional View of Relative Mach Number Contours

The zero and negative axial velocities of the meridional component in combination with a non-zero radial component in most steam- and span-wise locations along with the opposite behavior of the relative tangential velocity are what lead to differences in mass flow rate, total pressure ratio, and operating efficiency between the simulated results and the calculated predictions. These simulated values were however expected to be lower than calculated due to the excessive de Haller numbers and Diffusion Factors at larger spans, which predict the onset flow separation and back flow seen. Indeed, even the simulated de Haller numbers and Diffusion Factors indicate that flow separation should be present, which is easily seen in the figures presented in this section.

Experiments were performed to gauge the real behavior of the star pattern woven impeller. These are discussed in Section 4.6.

4.6 Experimental Apparatus and Results

An experimental test loop was constructed in the Composite Vehicle Research Center (CVRC) at MSU to measure performance characteristics exhibited by a prototype woven star pattern impeller. The prototype was wound on a mandrel using the technique described in

Sections 4.1 and 4.2 with the resulting dimensions as given in Table 4.1. The prototype rotor is shown in Figure 4.42 (right) alongside the 3D model created in NX (left).

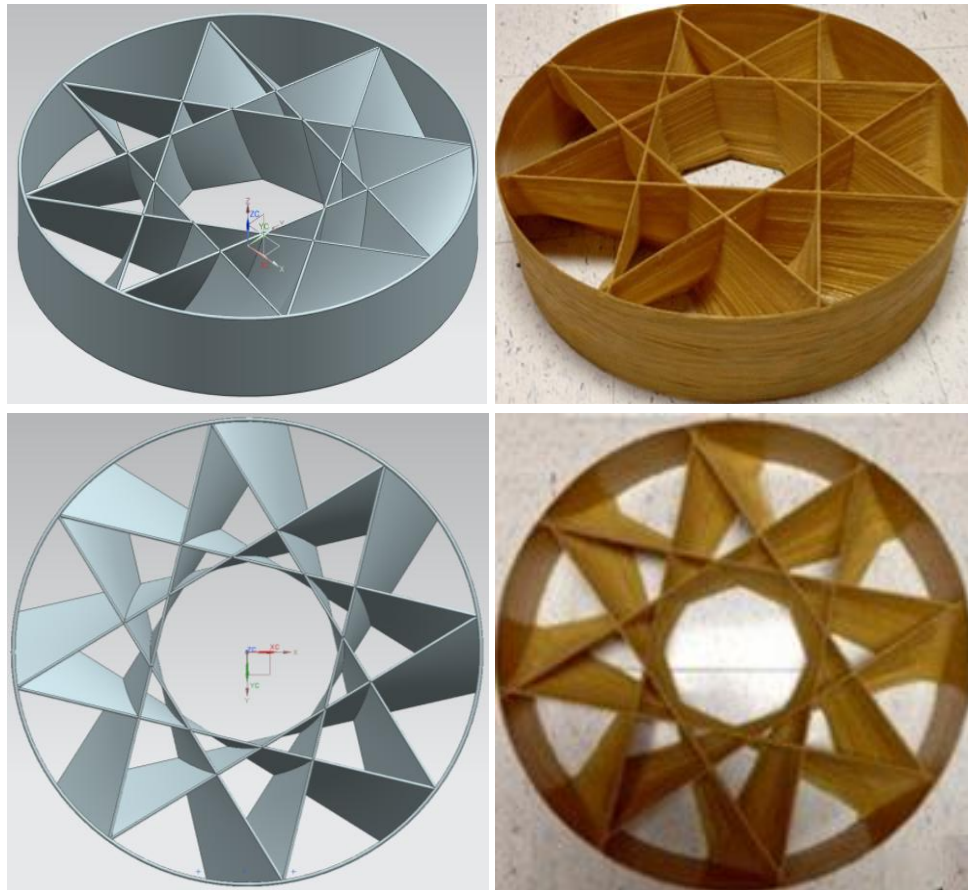


Figure 4.42. Star Pattern Impeller Prototype

The prototype construction was accomplished by winding Kevlar-49 fiber on an aluminum mandrel (Figure 4.43) that was slotted with eight circular arcs using a CNC mill.



Figure 4.43. A Mandrel with 8 Circular Arc Slots [13]

Prior to winding, the mandrel is coated with a mold release agent. The mandrel is then wet-wound. As the name implies, wet-winding consists of passing the fibers through an epoxy bath that consists of a resin (SC15) and hardening agent before being wound to create the 8B pattern (see Table 1.6). The wound composite impeller is then cured at room temperature for approximately 24 hours. When the epoxy is fully hardened, the mandrel is removed and the impeller is ready to be fitted with a shaft and installed in the test loop.

The test loop, shown open in Figure 4.44, is designed to operate under vacuum pressure. Its steel construction allows it to easily withstand the upto 1 bar pressure difference that a very good vacuum would create between the loop interior and the ambient environment. The bends in the loop act as a pressure loss mechanism [47] so that any pressure gained across the compressor will be lost by the time the flow completes a circuit of the loop, thereby maintaining the rotor inlet pressure. There are flow-straightening vanes in the corners to straighten out (de-swirl) the flow entering the rotor, and a butterfly valve located downstream of the rotor used to control the massflow rate. The shaft and impeller are centered using three threaded downstream struts.

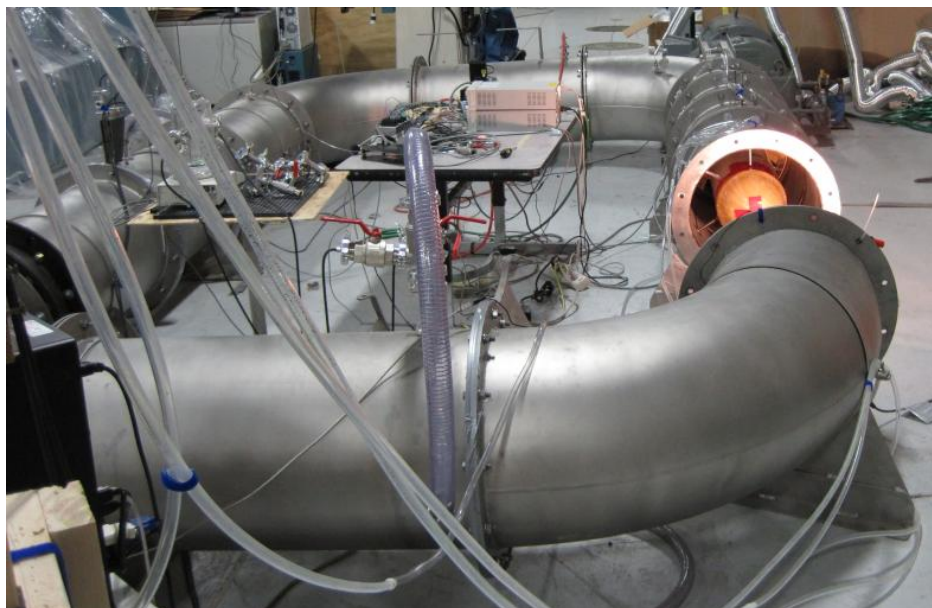


Figure 4.44. Open Test Loop Showing Installed Impeller

Total-to-total pressure difference is measured across the compressor using Pitot-static tubes placed at mid-span upstream and downstream of the compressor. Total-to-static pressure difference is measured on the opposite side of the loop at zero span (loop center) to determine the average flow velocity and volume flow rate. The Pitot-static tubes are read with the aid of inclined manometers.

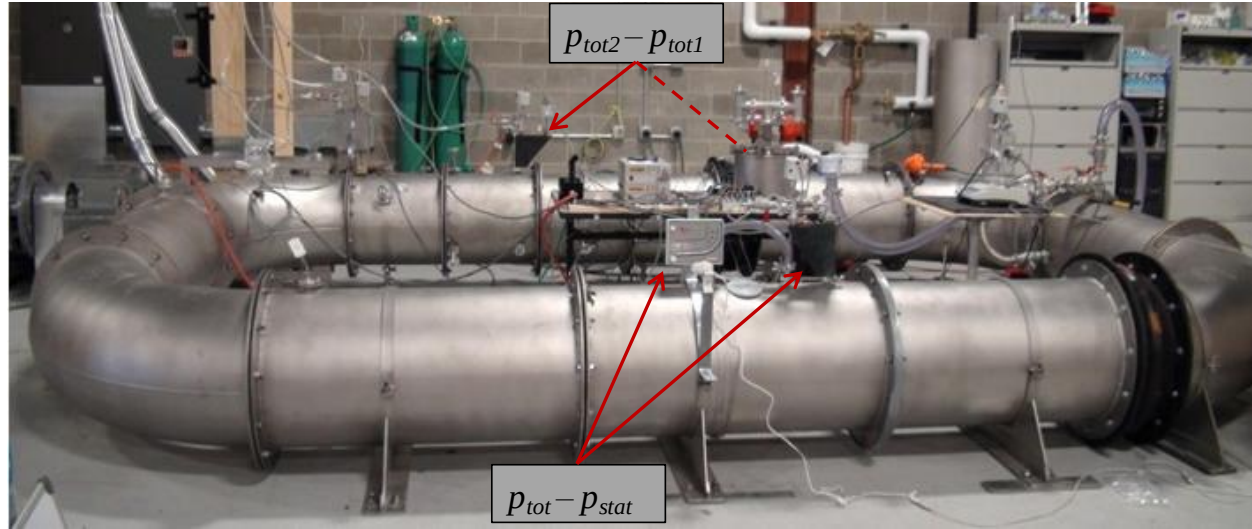


Figure 4.45. Closed Test Loop Showing Pressure Measurement Probes

The compressor was rotated at 7500rpm at the same pressure and temperature conditions as listed in Table 4.1. A “flow hub” of $r_h=88.9\text{mm}$ was used to block the flow through the center of the impeller and thereby direct it through the bladed region of the impeller. Table 4.9 provides the measured performance as well as a comparison to the calculated and simulated values at these conditions presented earlier in this chapter.

Table 4.9. Experimental Results and Predicted Performance Comparison

		Calculation	Simulation	Experiment	%Diff _{sim-calc}	%Diff _{exp-calc}	%Diff _{exp-sim}
$\tilde{e}$	kJ/kg	19.573	11.154	11.319	43.0%	42.2%	1.5%
$\dot{m}$	kg/s	2.73	2.58	2.86	5.5%	4.8%	10.9%
η_{pt}	%	80.0%	63.1%	26.1%	17.0%	53.9%	36.9%
Π_t	-	1.193	1.075	1.034	9.9%	13.3%	3.8%
$c_{m,avq}$	m/s	98.7	83.3	57.1	15.6%	42.1%	31.5%

The average axial velocity was determined in the following manner from the total-to-static pressure difference measurement. Prior to spinning the impeller, a vacuum pump was used to reduce the pressure inside the loop from ambient to 22.8kPa (absolute total pressure). The electric motor used to drive the compressor is then turned on and the speed is increased to 7500rpm. The total-to-static pressure difference reading is then used to convert to static-to-total pressure ratio as shown in Equation (4-29) using $p_t=22.8\text{kPa}$.

$$\frac{p}{p_t} = \frac{p_t - \Delta p_{t-stat}}{p_t} \quad (4-29)$$

The static-to-total pressure ratio is then substituted into Equation (2-59) and the absolute critical Mach number, M_c^* , is solved for algebraically. Knowing the mixture composition and total temperature (measured as ambient in the laboratory), Equation (2-50) gives the critical absolute speed of sound. Equation (2-51) is then used to determine the axial velocity as the Pitot tube measuring total-to-static pressure difference is parallel with the central axis of the loop and so does not capture the tangential velocity component.

Based on the de Haller numbers and Diffusion Factors of the simulated results in Table 4.8, it is reasonable to assume that the compressor is doing useful work and creating pressure rise for spans below 50.1%. Indeed, area-averaging the calculated work transfer in this region gives the experimental value for $\tilde{\epsilon}$ provided in Table 4.9, which is only 1.5% different from the simulation. The polytropic efficiency corresponding to this $\tilde{\epsilon}$ and measured total pressure ratio can then be solved for algebraically from Equation (2-93). Unfortunately, this polytropic efficiency is over 50% lower than the initially assumed 80% operating efficiency, and 37% lower than simulation.

As can be seen in Table 4.9, the experimental performance is not as good as was simulated, which in turn was not as good as initially predicted. This is to be expected as simulation results account for more loss mechanisms than the initial calculations do (viscous shear, more accurate prediction of boundary layer growth and separation), but they still do not account for many flow effects that the prototype actually experiences. There is imperfect sealing between the outer shroud of the impeller and the test loop walls, and the actual blade surfaces of the composite impeller have very large surface roughness compared to the perfectly smooth blades used in simulation. The wheel itself was wound by hand such that there are likely unevenly distributed fibers when considering layering around mandrel slot corners, and the wet-wound epoxy is likely not distributed perfectly.

The measurements themselves will be another source of discrepancy between simulation and experiment, as Pitot tube placement in both the stream-wise and span-wise directions will impact the pressure readings. Additionally, the calculations and simulations were performed using perfectly dry air, but the air in the loop has non-zero levels of relative humidity.

Since de Haller numbers and Diffusion Factors are beyond the recommended limits at spans over 50%, it is recommended that blade turning at the tip be reduced, thereby reducing the blade and flow turning everywhere else. Although the initial calculations will produce less pressure ratio, it is likely that the impeller will be able to perform better in simulation and experiment by being able to utilize more of the span to perform useful work on the fluid, thereby increasing the operating efficiency and overall work transfer.

4.7 Chapter Summary and Conclusions

A full 3D characterization of the star pattern geometry has been performed. This has been accomplished without the use of projected circular arcs onto the curved surface of the outer

shroud/mandrel (simpler from a CAD standpoint) thereby ensuring circular fidelity of the arcs comprising the blades. This applies to the 8B pattern described here in detail, and in general for this and all other star patterns of any diameter with any amount of tip blade turning. Methods to then predict flow behavior and performance characteristics are also developed and described. It has been verified that the star pattern impeller can in general be reasonably described by the forced-vortex condition.

The geometry used to predict the performance was simulated using ANSYS CFX and experiments were performed under the same conditions resulting in the in following conclusions:

- Although the calculations predicted excessive de Haller numbers and Diffusion Factors for spans greater than 17% and 28%, respectively, the simulations show that this is in fact only true for spans greater than 50%.
- This implies that there is excessive blade turning at spans greater than 50%.
- At spans greater than 50%, there are large regions of back flow due to boundary layer separation. The boundary layer separation is a result of excessive flow turning/blade surface curvature in an adverse pressure gradient.
- The separated boundary layer serves as flow blockage that has the effect of a nozzle that accelerates the remainder of the flow through the regions at less than 50% span. This is what accounts for the only 27% reduction in mass flow rather than the 50% that might reasonably be assumed for these conditions.
- Non-zero radial velocities are present within the rotor, particularly at spans above 50%.
- Absolute and Relative Mach numbers within acceptable range indicating there are no shock formations that might cause additional entropy generation.

- Taking the area average of the calculated specific work transfer for spans less than 50% provides very good agreement with the simulated results (within 1.5%).
- The 1.5% accuracy with the simulated results leads to the supposition that the experiments are providing similar work transfer to the fluid. However, for the measured pressure ratio, this corresponds to a polytropic efficiency of 26.1% for the prototype (as opposed to 63.1% in simulation, down from an initially assumed 80%).

However, it is likely that there are significant errors within the measurements themselves that lead to the drastic differences in polytropic efficiency. Possible sources of error that may account for the discrepancy between simulation and experiment include the following:

- The surface roughness and fiber/epoxy consistency of the manufactured impeller is very different from the perfectly curved and smooth blade surfaces used in simulation.
- Pressure measurement device placement (stream-wise, span-wise location) may not have perfectly captured pressure rise. A span-wise traverse of the Pitot tubes would give a more accurate pressure profile.
- Support struts were not included in simulation, likely influencing total pressure difference measurement across the compressor. It is likely that the struts increase the measurement sensitivity to probe placement.
- Simulations performed using dry air whereas experiments used air with non-zero relative humidity.

It should be possible to increase efficiency in both simulation and experiment by reducing the amount of blade turning, thereby increasing the span of useful work transfer to beyond 50% resulting in an overall larger measured pressure ratio. However, the forced-vortex distribution requires that decreasing the tip blade turning decreases blade turning at all other spans, hence reducing the potential work transfer there. This creates the question of how much tip turning to sacrifice before the rest of the wheel stops performing useful work, thereby creating an intensive optimization problem based in simulation and experiment.

In regard to the evolution of the woven impeller, it was decided that this first generation star pattern, originally conceived of for its ease of construction and stability [13], can be modified to avoid this specific optimization problem. Chapter 5 details the next generation of woven impeller.

CHAPTER 5 FREE-VORTEX BLADE IMPELLER PERFORMANCE

The star pattern impeller discussed in Chapter 4 was conceived based on the simplicity of its construction. A CNC mill was used to cut circular arc slots into aluminum tubing and winding proceeded from there. However, developments made in recent years in the field of 3D printing technology have resulted in the cost of acquiring a 3D printer being reduced substantially. This has led to the printing of significantly more varied and sophisticated mandrels that were simply not possible with the CNC machining of aluminum tubing.

Figure 5.1 shows a mandrel model suitable for 3D printing. The blade slots have circular arc curvature at the outer radius as the aluminum tubing did, and winding still facilitates the automatic incorporation of an outer shroud. Now however, the blade slots are more like blade channels, and there is a different circular arc at the inner radius as well as a blade profile everywhere in between. The inner columns, or “fingers”, allow for the incorporation of a true inner shroud, all while still using a single continuous fiber tow. This also makes it possible for the blades to consist of curved fibers in addition to straight fibers enabling direct control of the blade profile, although this aspect is not considered in this work.



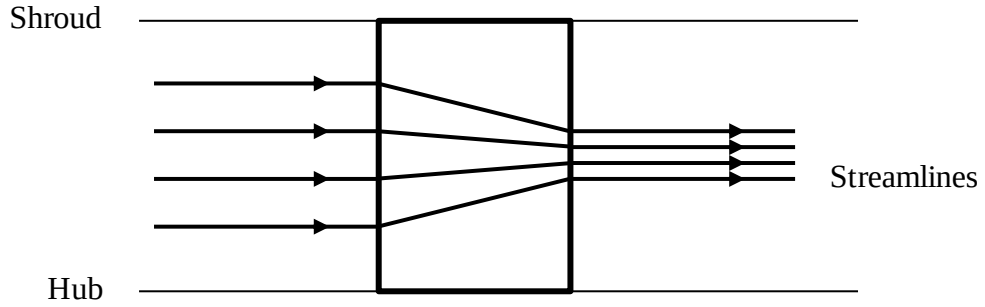
Figure 5.1. A Second Generation Mandrel

The significance of having a different circular arc at the hub that can be imposed (rather than simply naturally larger curve radius as was the case for the star pattern impeller) is that it is possible to achieve an entirely different swirl-distribution. The forced-vortex swirl distribution in the star pattern impeller has excessive swirl at the tip, but reducing the swirl at the tip necessarily reduces swirl everywhere else and thereby creates an optimization problem involving trade-offs as was discussed at the end of Chapter 4 (too much swirl: low through-flow and reduced work transfer, pressure ratio, and efficiency. Less swirl: more through-flow but less potential work transfer). However the second generation mandrels allow for the winding of impellers that have Free-Vortex Blades (FVB) and exhibit a free-vortex type swirl distribution that has what are in essence the opposite characteristics of the forced-vortex condition: at large span there is less blade and flow turning and at low span there is more. This also allows for the blades to be evenly loaded rather than having higher loading at the tip and lower loading at the hub as is the case with the forced-vortex blades. Sections 5.1 and 5.2 provide the requisite mathematics.

5.1 Theory of Radial Equilibrium

The free-vortex condition arises out of the theory of radial equilibrium. As was seen in Chapter 4, non-zero radial velocities exist within the blade row. These arise from centrifugal forces on the fluid being balanced by the radial pressure distribution transporting the fluid particles as necessary to maintain equilibrium. The following development is adapted from [54].

Radial equilibrium requires the assumption that radial flows occur only within the blade row and that flow outside the blade row is in radial equilibrium as shown in Figure 5.2. It is further assumed that the flow is axisymmetric and that the effect of a discrete number of blades is not transferred to the flow.



Rotation Axis -----

Figure 5.2. Radial Equilibrium Flow Across a Blade Row

A differential fluid element of mass $dm = \rho dA = \rho r dr d\theta$ (unit depth) as shown in Figure 5.3 is considered to be in radial equilibrium when the pressure forces balance the centrifugal forces acting on the element, thereby ensuring zero radial velocity.

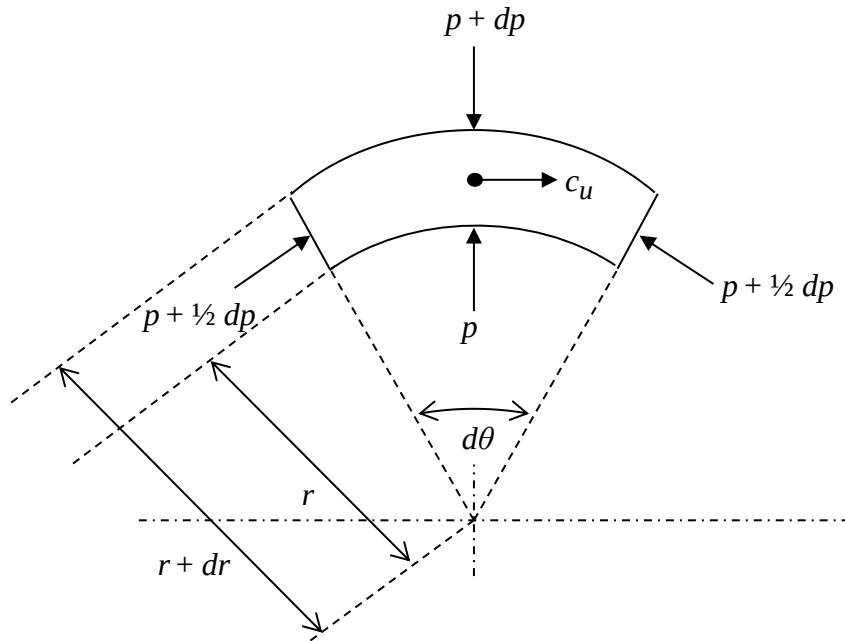


Figure 5.3. Fluid Element in Radial Equilibrium

Taking the radially inwards direction as positive, the radial force balance is written

$$(p + dp)(r + dr)d\theta - \rho dr d\theta - (p + \frac{1}{2} dp)dr d\theta = dm \frac{c_u^2}{r} \quad (5-1)$$

The first two terms in Equation (5-1) are simply the pressure acting over the corresponding area (arc length multiplied by unit depth). The term on the right-hand-side is the centrifugal force, which is of course depends on the fluid element mass, tangential speed, and its distance from its rotational center. The third term on the left-hand-side bears further mention. Figure 5.4 shows a decomposition of the forces acting on the sides of the unit-depth fluid element.

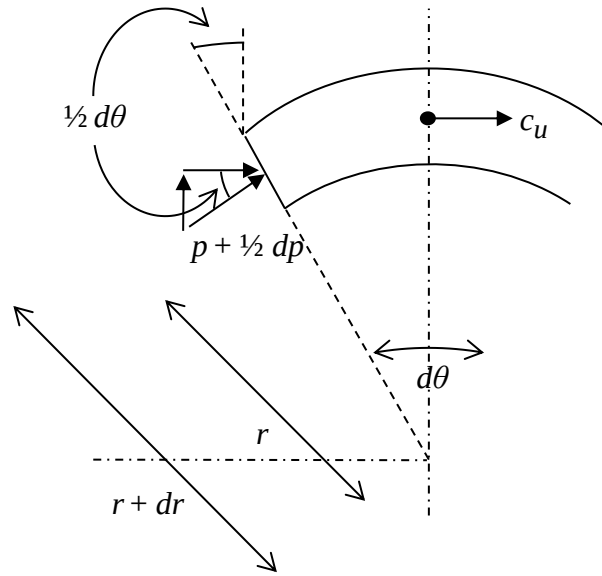


Figure 5.4. Fluid Element Side-Wall Pressure Force Decomposition

The vertical component of the fluid element side-wall pressure force is the component of interest since it acts radially. Denoting this component p_{left} allows for the expression

$$p_{left} = \left(p + \frac{1}{2} dp\right) \sin\left(\frac{d\theta}{2}\right) \quad (5-2)$$

For small θ , $\sin(\theta) \approx \theta$, therefore

$$p_{left} = \left(p + \frac{1}{2} dp\right) \frac{d\theta}{2} \quad (5-3)$$

The right side of the fluid element mirrors the left. Summing these two components gives

$$p_{left} + p_{right} = 2\left(p + \frac{1}{2} dp\right) \frac{d\theta}{2} = \left(p + \frac{1}{2} dp\right) d\theta \quad (5-4)$$

These forces in Equation (5-4) act over the distance dr and unit depth. Since they point radially outward, they receive a negative sign in the force summation in Equation (5-1). Distributing the terms in Equation (5-1), neglecting products of differentials and simplifying gives the expression

$$\frac{1}{\rho} \frac{dp}{dr} = \frac{c_u^2}{r} \quad (5-5)$$

Integrating Equation (5-5) in the span-wise direction gives

$$p_{tip} - p_{hub} = \int_{r_{hub}}^{r_{tip}} \rho \frac{c_u^2}{r} dr \quad (5-6)$$

The stagnation enthalpy in Equation (2-43) can be written in terms of its velocity components. With zero radial velocity, this yields

$$h_t = h + \frac{1}{2} (c_m^2 + c_u^2) \quad (5-7)$$

and differentiating with respect to r gives

$$\frac{dh_t}{dr} = \frac{h}{dr} + c_m \frac{dc_m}{dr} + c_u \frac{dc_u}{dr} \quad (5-8)$$

Writing Equation (2-24) in terms of density rather than specific volume gives

$$Tds = dh - \frac{1}{\rho} dp \quad (5-9)$$

and differentiating this with respect to r yields

$$T \frac{ds}{dr} = \frac{dh}{dr} - \frac{1}{\rho} \frac{dp}{dr} \quad (5-10)$$

Subtracting Equation (5-10) from (5-8) and substituting the relation in (5-5) with some manipulation gives

$$\frac{dh_t}{dr} - T \frac{ds}{dr} = c_m \frac{dc_m}{dr} + \frac{c_u}{r} \frac{d(c_u r)}{dr} \quad (5-11)$$

For blade loading in the span-wise direction to be even, changes in stagnation enthalpy and entropy must be zero in the span-wise direction, i.e. $\frac{dh_t}{dr} = 0$, $\frac{ds}{dr} = 0$, and so Equation (5-11) reduces to

$$c_m \frac{dc_m}{dr} + \frac{c_u}{r} \frac{d(c_u r)}{dr} = 0 \quad (5-12)$$

Equation (5-12) expresses the condition of radial equilibrium for rotors imparting equal work to the fluid at all radii.

5.2 Free-Vortex Flow

The free-vortex condition is one in which the axial component of vorticity, ω , is zero. As will be shown, the result is that the product of the tangential velocity and radius is constant (as opposed to the quotient being constant in the case of forced-vortex flow). The following continues the development in [54].

Looking at an inviscid fluid element rotating about a fixed axis as shown in Figure 5.5, the circulation, Γ , is defined as the line integral of tangential velocity around a curve that encloses area A such that

$$\Gamma = \oint \mathbf{c} \cdot d\mathbf{s} \quad (5-13)$$

Stokes' theorem [56] states that the circulation of the fluid particle is the area integral of the vorticity:

$$\Gamma = \oint \mathbf{c} \cdot d\mathbf{s} = \int_A \nabla \times \mathbf{c} \cdot d\mathbf{A} \quad (5-14)$$

where the vorticity is $\boldsymbol{\omega} = \nabla \times \mathbf{c}$.

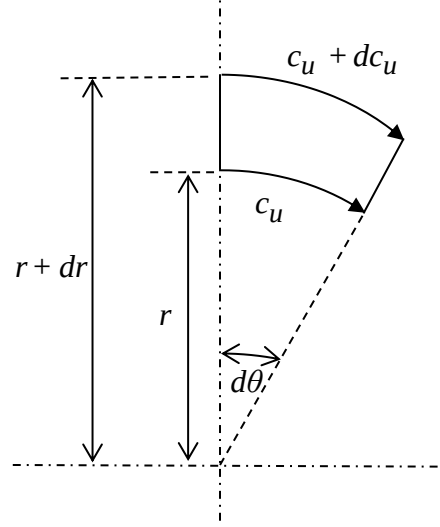


Figure 5.5. Circulation about a Fluid Element

The vorticity at a point is then defined as the limiting value of circulation divided by a vanishingly small area so that

$$\omega = \frac{d\Gamma}{dA} \quad (5-15)$$

For the fluid element in Figure 5.5 with no radial velocity component, the circulation is

$$d\Gamma = d\oint \mathbf{c} \cdot d\mathbf{s} = (c_u + dc_u)(r + dr)d\theta - c_u r d\theta \quad (5-16)$$

and ignoring products of differentials gives

$$d\Gamma = \left(\frac{dc_u}{dr} + \frac{c_u}{r} \right) r dr d\theta \quad (5-17)$$

Substituting Equation (5-17) into (5-15) with $dA = r dr d\theta$, simplifying, and rearranging gives

$$\omega = \frac{1}{r} \frac{d(c_u r)}{dr} \quad (5-18)$$

Free-vortex flow requires that axial vorticity be zero, so Equation (5-18) can be solved:

$$\int 0 = \int \frac{d(c_u r)}{dr} \quad (5-19)$$

and so

$$c_u r = K \quad (5-20)$$

where K is a constant at any given stream-wise location. Equation (5-20) is the defining characteristic of free-vortex flow.

Substituting the free-vortex condition in Equation (5-20) into the radial equilibrium setting described in Equation (5-12) dictates the axial velocity profile for free-vortex flow:

$$c_m \frac{dc_m}{dr} + \frac{c_u}{r} \frac{d(K)}{dr} = 0 \quad (5-21)$$

Canceling terms and integrating gives

$$\int \frac{dc_m}{dr} = \int 0 \quad (5-22)$$

which leads to the result

$$c_m = \text{constant} \quad (5-23)$$

Thus the absolute velocity is known for the entire span at any stream-wise location: the radial velocity component is assumed to be zero thereby allowing the axial velocity component to be constant (Equation (5-23)), and the product of radius and tangential velocity is also constant (Equation (5-20)).

Since any given rotor has one rotational speed, i.e. ω is constant, Equation (5-20) can be extended to include ω

$$c_u r \omega = K \quad (5-24)$$

and since blade speed is the product of r and ω , it follows that

$$c_u u = K \quad (5-25)$$

at all spans for a given stream-wise location. Applying Equation (5-25) to Equation (2-93) results in the same work transfer, and hence increase in total pressure ratio, regardless of span between any two stream-wise locations.

5.3 2- and 3-Dimensional Circular Arc Blade Description

The free-vortex blades are first constructed in 1D (axial direction) from circular arcs at the area mean line as described in Chapter 2. The free-vortex condition then facilitates the determination of the blade curvature in the span-wise direction extending away from the mean-line towards both the hub and the outer shroud: K is found from Equation (5-20) and so the tangential velocity is determined at all known radii. With the rotational speed of the rotor, Equation (2-119) then gives the relative tangential fluid speed thereby completing the velocity diagram at all spans. This results in different circular arcs with different curvatures, radii, and chord lengths at each span, but they each have the same axial length in keeping with the winding pattern required by the mandrel (the same amount of fiber must be used at both hub and tip, and so the same axial length is used at all spans to avoid fiber bunching).

5.3.1 Geometry Selection

The FVB geometry is selected such that it is as similar as possible to that of the star pattern impeller discussed in Chapter 4 to facilitate a fair performance comparison. In order for the FVB to satisfy continuity, the hub radius was allowed to change with stream-wise position as described in Equation (2-126) (increasing with decreased cross-sectional area to account for increases in density and pressure due to the compression process) while the tip radius was kept constant. Table 5.1 shows the major geometrical parameters for the FVB rotor alongside those of the star pattern impeller.

Table 5.1. Geometrical Parameter Comparison of FVB and Star Pattern Impellers

		Star Pattern	FVB	Difference
z_{rot}	m	0.133	0.133	0.0%
r_t	m	0.235	0.235	0.0%
$r_{h,in}$	m	0.089	0.089	0.0%
$r_{h,out}$	m	0.089	0.099	10.8%
t_b	m	0.002	0.002	0.0%
N_b	-	16	16	0.0%
$\beta_{in,mean}$	°	37.1	37.1	0.0%
$\beta_{out,mean}$	°	90.0	90.0	0.0%
$\Delta\beta_{tip}$	°	65.0	24.5	-62.3%
$\Delta\beta_{mean}$	°	52.9	52.9	0.0%
$\Delta\beta_{hub}$	°	37.1	93.2	151.3%
σ_{mean}	-	2.2	2.1	-3.2%

The inlet and outlet mean blade angles in Table 5.1 are the area-averaged values for the star pattern impeller, and so these values were used at the area-mean radius in the FVB impeller. The forced-vortex and free-vortex swirl distributions account for the respective differences in blade turning at the tip and hub; at the tip, the FVB turning is more than 60% less than that of the star pattern impeller, while the hub turning is over 150% greater. The hub turning will always be larger when comparing an FVB impeller to a star pattern impeller with very similar geometry, but in this case it is exceedingly large due to the relatively small hub/tip ratio. For larger hub/tip ratios, there is less change in r , and hence there is less compensatory change in c_u to maintain the constant K . This small hub/tip ratio has led to an outlet angle at the hub of just under $\beta_{hub,out} = 150^\circ$.

Since none of the outlet blade angles in the star pattern impeller are greater than 90° , the hub outlet angle on the FVB has likewise been limited to 90° . This does not ensure true free-vortex flow below mid span, but it does serve to provide a more fair performance comparison. Figure 5.6 shows blade-to-blade views at the tip and mid-span. Figure 5.7 shows blade-to-blade views at the hub of both the 90° -limited case and the 150° outlet case for reference (with the rotor inlet on the left). Note that in Figure 5.7, the inlet angles are identical.

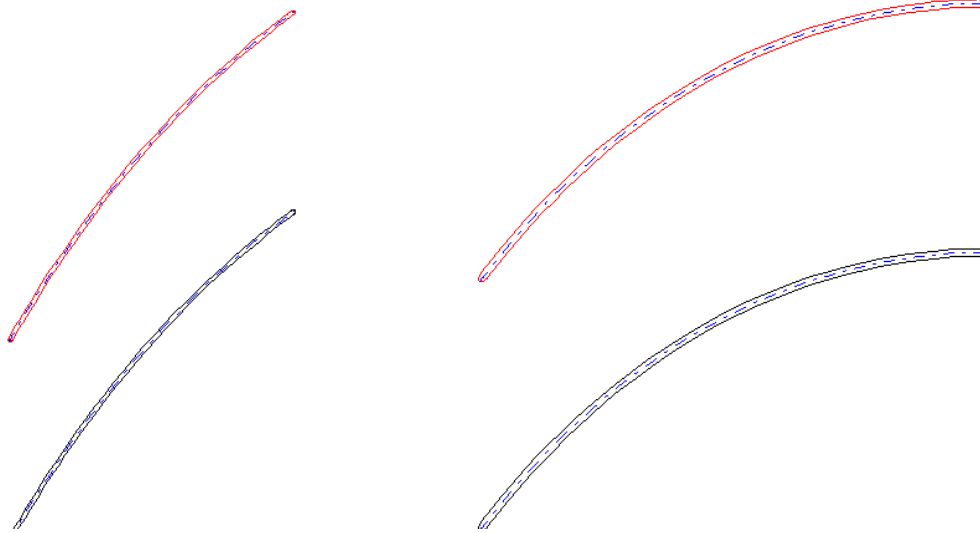


Figure 5.6. Blade-to-Blade View of FVB Circular Arcs at Left: Tip, Right: Mid-Span



Figure 5.7. Blade-to-Blade View at Hub of Left: 90°-Limited Case, Right: Full Free-Vortex Turning

Figures 5.8 and 5.9 show a full single blade and the entire rotor, respectively, for both cases.

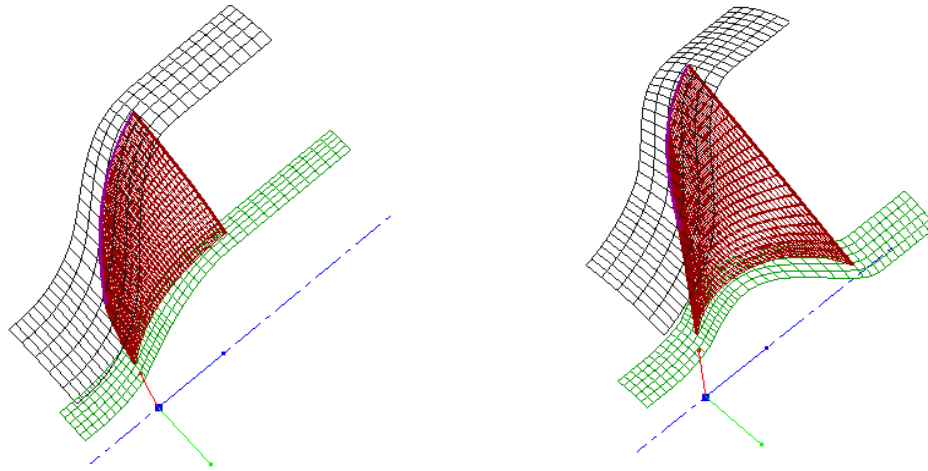


Figure 5.8. A single Free-Vortex Blade for Left: 90°-Limited Case, Right: Full Free-Vortex Turning

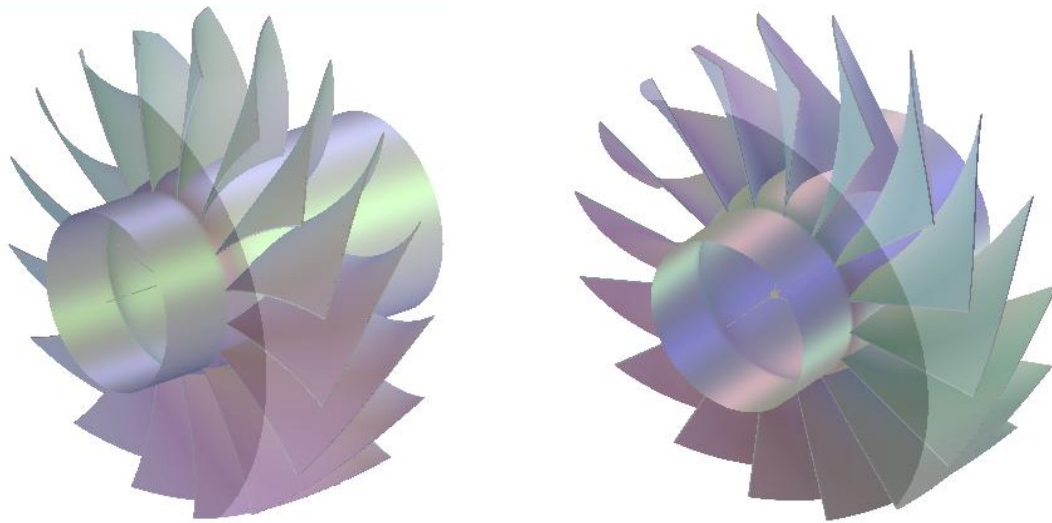


Figure 5.9. Full FVB Rotor View for Left: 90°-Limited Case, Right: Full Free-Vortex Turning

5.4 Free-Vortex Blade Impeller 3-Dimensional Simulation

The 90°-limited case was simulated in ANSYS CFX under the same total temperature, total pressure, and rotational speed conditions as the star pattern impeller. The mesh however was created in the TurboGrid module using the ATM Optimized method and consists of 282,458 purely hexahedral elements with 304,500 nodes. Figure 5.10 shows the representative mesh surfaces for the hub (blue), shroud (purple), high periodic face (yellow), inlet (green), and blade (gray) that when extended inward (outward from the blade) comprise the fluid region.

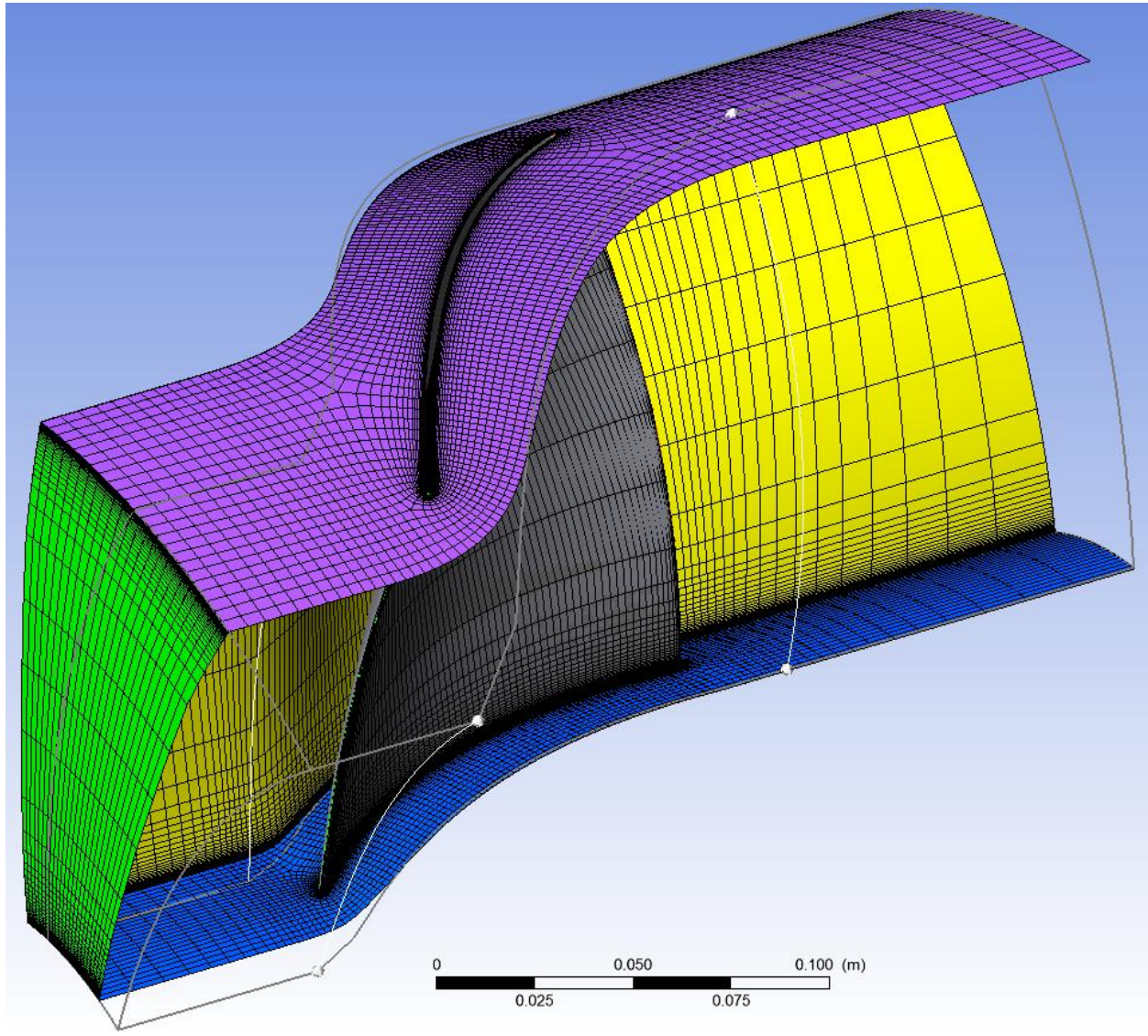


Figure 5.10. Mesh Surfaces for FVB Simulation

The node number is reduced compared to the star pattern impeller because a significant portion of the elements are in the near-wall region to capture boundary layer effects. The star pattern impeller has over 6.2% of the modeled fluid passage occupied by blades since it has two main blades (at high span, emanating from the encompassed mandrel slot) and two secondary blades (at low span, crossing between neighboring mandrel slots) passing between periodic surfaces as shown in Figure 4.29. The FVB configuration only has a single blade passing between periodic surfaces and so the blades only account for 3.1% of the meshed volume. The

single blade presence between periodic surfaces also allows the surfaces to be twice as close together as with the star pattern impeller, thereby reducing the hub and shroud surface areas to be meshed. This allows roughly just over half the number of nodes to be safely used without performing a rigorous mesh independence study as was done for the star pattern (see Section 4.5.1 and Appendix A1) since the mesh densities are similar (the FVB a has higher average node density of 255 nodes/cm³ vs. 234 nodes/cm³ for the star pattern mesh used in Chapter 4).

The fluid properties and inlet conditions are the same as those provided in Table 4.1. Similar to Section 4.4.2, boundary layer thickness was estimated on each blade surface and subtracted from the cross-sectional area of the core flow. Table 5.2 provides the predicted and simulated performance of the FVB impeller.

Table 5.2. Performance of FVB Impeller				
		Calculation	Simulation	Difference
$\tilde{e}$	kJ/kg	20.02	12.79	36.1%
$\dot{m}$	kg/s	3.822	3.234	15.4%
$\dot{m}_{adj}$	kg/s	3.218	3.234	-0.5%
η_{pt}	%	80.0%	74.8%	5.2%
Π_t	-	1.197	1.111	7.2%
$\tilde{P}$	kW	64.43	41.36	35.8%
$c_{m,avg}$	m/s	105.6	97.0	8.1%

The largest discrepancy between calculated performance and simulation is the specific work transfer to the fluid. This is due to the under-turned blades at the hub region not performing with true free-vortex behavior, hence there is less potential work transfer below mid span. However there are less than 10% differences for polytropic efficiency, total pressure ratio, average and axial velocity. The boundary layer-adjusted mass flow rate slightly under-predicts the mass flow achieved in simulation, but it remains within 1%. The slim under-prediction is what accounts for the difference in power consumption from calculation to simulation being slightly less than that of the specific work transfer.

Similar to Chapter 4, the upper contours in the following figures are results from simulation, while the lower contours illustrate the predicted values from calculation. The rotor region in the meridional plane of the simulation results is contained between the solid black vertical lines.

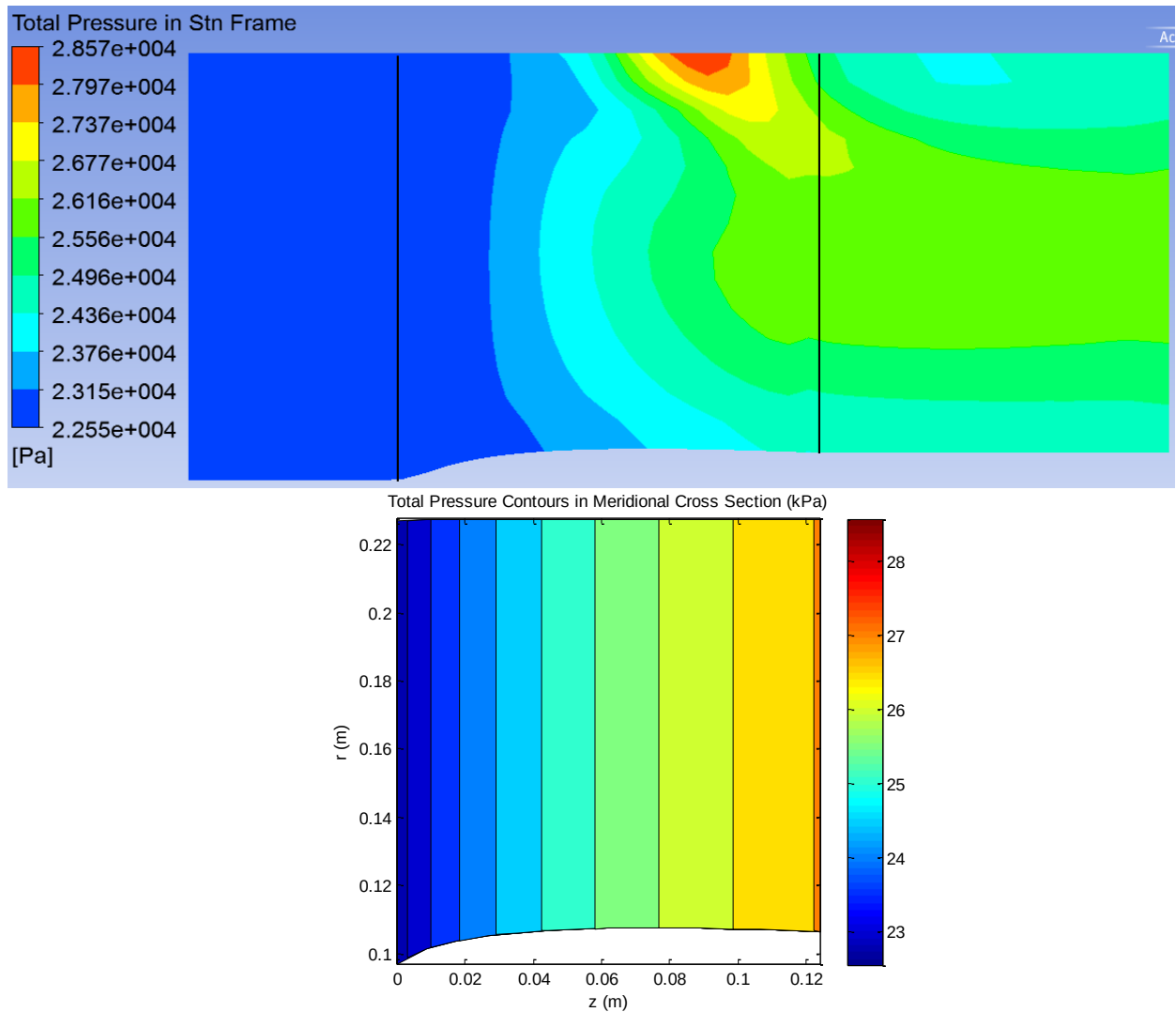


Figure 5.11. Meridional View of Total Pressure Contours. Upper: Simulation, Lower: Calculation

The total pressure ratio gained is 7.2% lower in simulation than calculation. As can be seen in the calculation contour of Figure 5.11 (lower), the total pressure ratio increase is only in the stream-wise direction; it is the same value for all spans as indicated by Equations (5-20) and (5-25) in conjunction with Equation (2-93). In simulation this also holds mostly true from the

rotor inlet to approximately 55%-stream, at which point there is some span-wise variation as indicated by non-vertical contour lines. A high pressure pocket near the tip is evident between 55%- and 85%-stream for spans larger than 85%. The loss of total pressure beyond 85% stream at these spans indicates that this is not useful pressure gain, and this loss region can account for the discrepancy between calculated and simulated total pressure ratios.

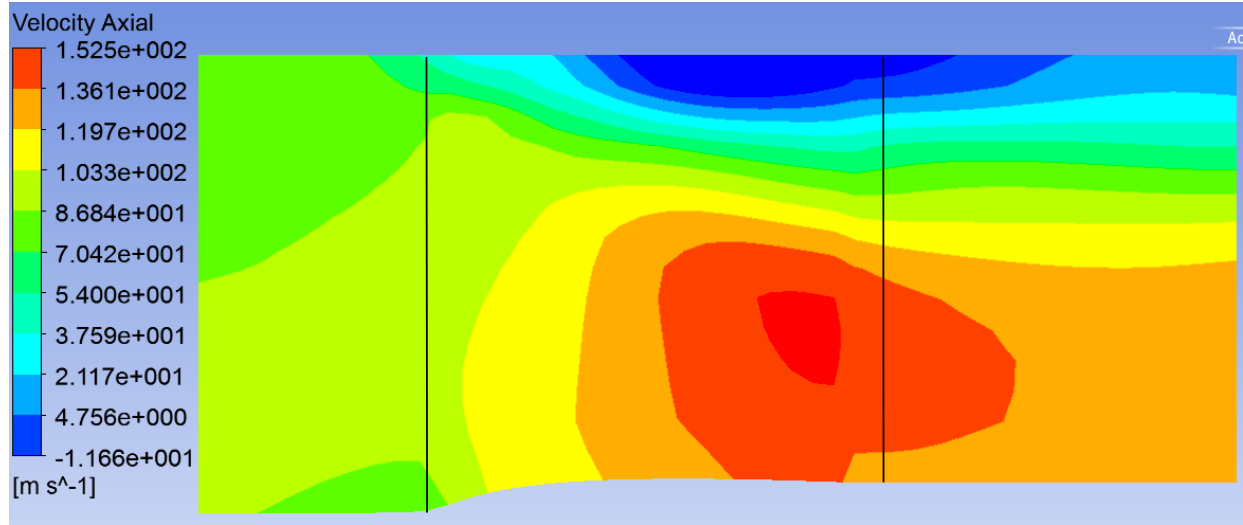


Figure 5.12. Meridional View of Simulated Axial Velocity Contours

Equation (5-23) gives the span-wise axial velocity distribution as constant. It was also assumed that the stream-wise axial velocity distribution would remain constant for simplicity of calculation. However, Figure 5.12 shows a large variation in axial velocity, particularly near the tip where there are negative axial velocities seen, indicating the presence of back-flow.

Back flow region causes the remainder of the core flow to see reduced cross-sectional flow area, creating a nozzle effect. This increase in c_m offsets the negative c_m values, hence the difference of only 8.1% on an area-average basis between simulation and calculation. Boundary layer growth (seen in Figure 5.17 at high spans) and excessive blade curvature account for the presence of the separated region with its accompanying back flow.

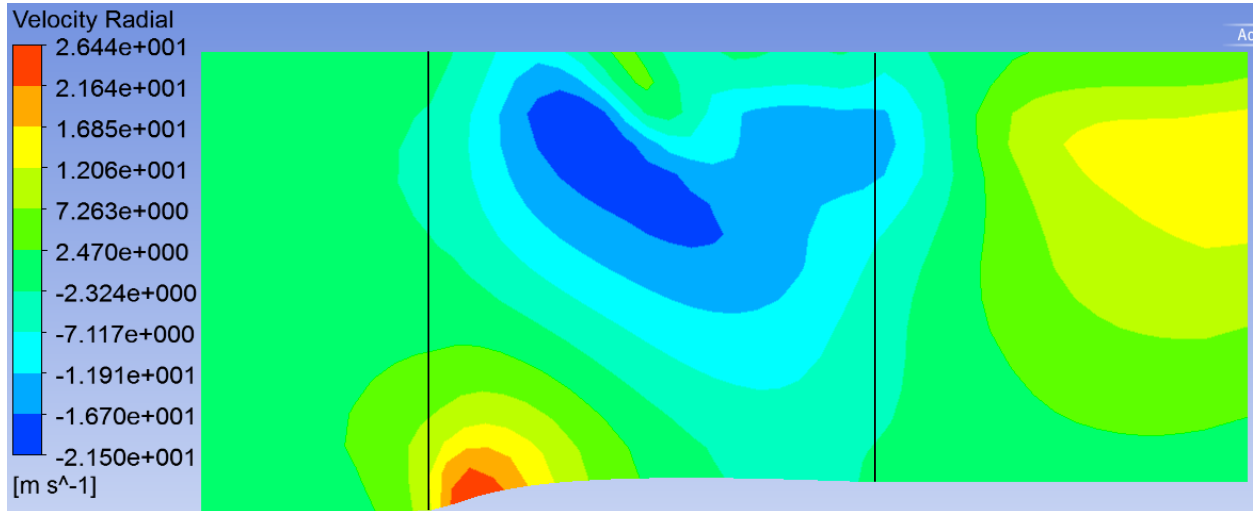


Figure 5.13. Meridional View of Simulated Radial Velocity Contours

The non-uniform axial velocity profile gives rise to non-zero radial velocity. Core flow moving down stream at spans below 80% and separated flow moving back upstream at spans above 90% create the radial velocity gradients associated with non-zero circulation. The anti-parallel flow creates a fluid shear region that serves to circulate the flow as it transitions from flowing downstream to back upstream as it does not simply reflect upstream, but must curve back upstream. Flow near the hub at the inlet is brought downstream and radially outwards toward the shroud, some of which circulates back upstream and radially inwards towards mid-span. This seems to be exacerbated by the trajectory of the hub profile at the inlet; the largest change in hub radius is where the largest outward radial velocities are seen in Figure 5.13, although this is to be expected as the streamlines are directed radially outward around the radially increasing hub wall.

Figure 5.14 shows contours of entropy, which as has been established in Chapter 2 are associated with losses in total pressure. The majority of the increases in entropy are seen above 80% span beginning near 30% stream and continuing downstream past the outlet of the rotor. The large entropy region downstream of the rotor trailing edge corresponds with the region of lower total pressure seen in Figure 5.11 at the same location.

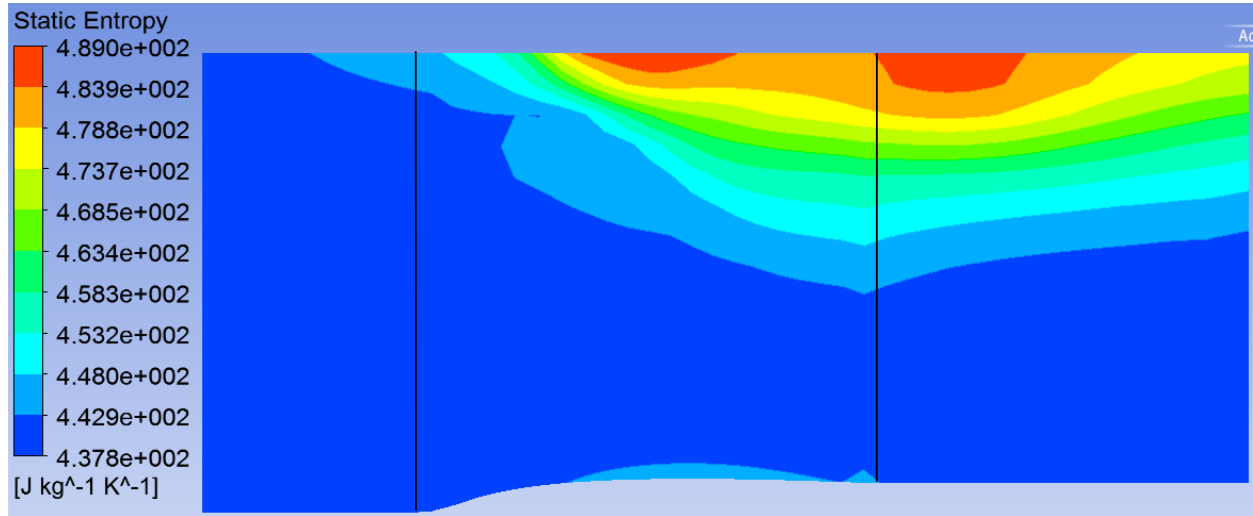


Figure 5.14. Meridional View of Simulated Entropy Contours

Despite the total pressure uniformly increasing with stream-wise location and the absolute tangential velocity increasing with both increasing stream and decreasing span, the trends of the two contours in simulation very closely mimic one another (see the upper contours of Figures 5.11 and 5.15). As seen in the lower contour of Figure 5.15, the flow should enter the rotor with zero c_u across the entire span since there is no IGV, and it should increase towards the outlet, particularly at the hub region according to the free-vortex distribution (this is indicated by the contour lines leaning towards the downstream direction). However, the simulation c_u contour lines are near vertical for spans below 80% nearer the inlet, and begin leaning upstream moving towards the outlet. The largest region of c_u is near the tip in the same location where the largest total pressure is observed rather than near the hub at the outlet. This indicates that the backflow region is not allowing for the expected tangential velocity behavior. The fluid is simply rotating with the blades in this region. Indeed, this is the region seen in Figure 5.16 of a concentration of very low relative tangential velocity.

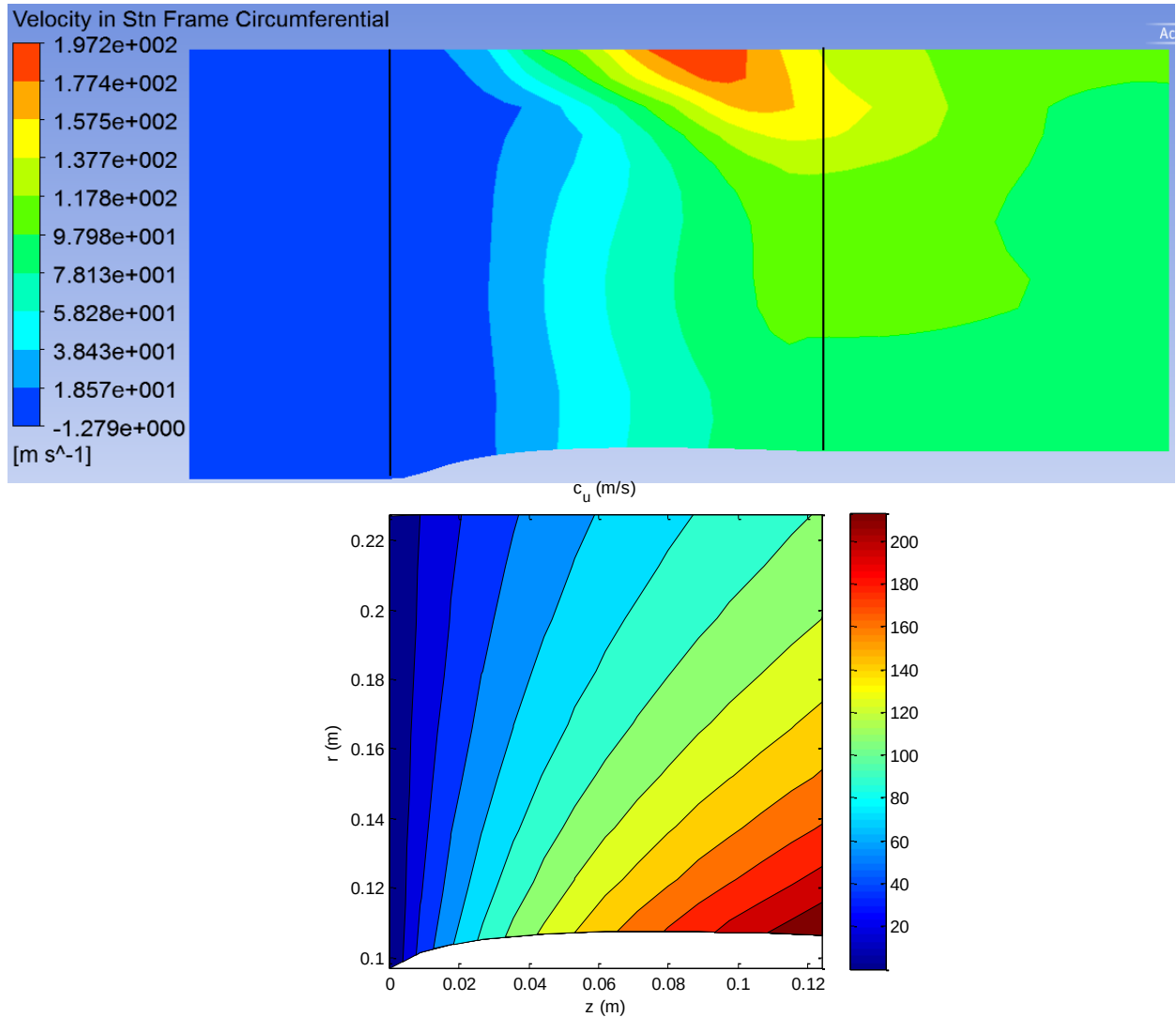


Figure 5.15. Meridional View of Absolute Tangential Velocity Contours. Upper: Simulation, Lower: Calculation

This is similar to the discussion in Chapter 4, where the region of high total pressure due to flow “build up” at high span (it does not move downstream to the outlet leading to a region of concentrated fluid mass) is a result of boundary layer growth and subsequent separation due to excessive blade curvature. The separation is accompanied by a back flow region as the fluid moves to fill the void left by the separation. When the fluid velocity (nearly all tangential at high span beyond 40%-stream) very closely matches the nearby wall, there is very little to hold it to the wall, and hence it separates [55].

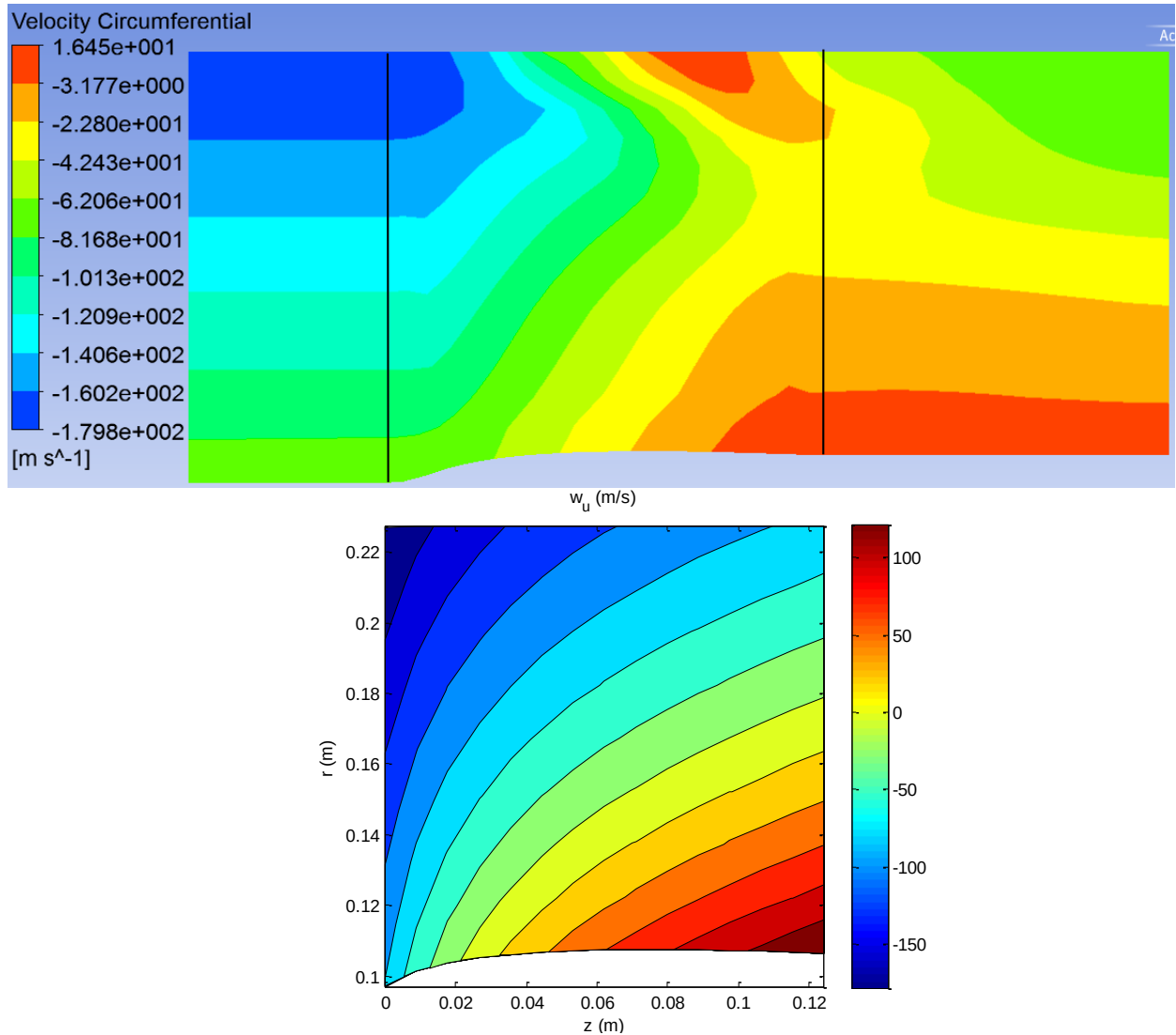


Figure 5.16. Meridional View of Relative Tangential Velocity Contours. Upper: Simulation, Lower: Calculation

There is however better agreement in relative tangential velocity contours from calculation to simulation than is the case for the absolute tangential velocity. Spans below 80% exhibit the same trend at all stream-wise locations thereby indicating that the flow follows the blades well, and this is a major contributor to the polytropic efficiency of the rotor losing only 5.2% from calculation to simulation.

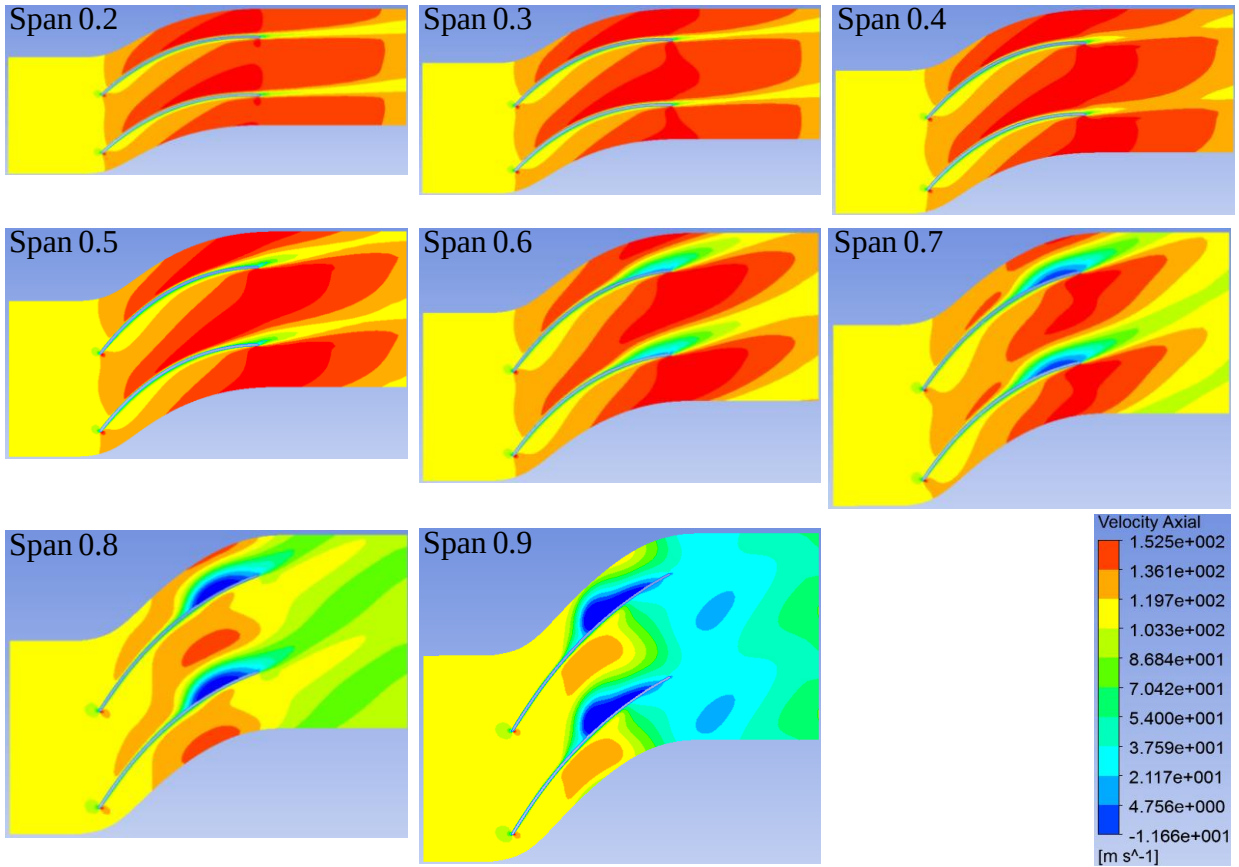


Figure 5.17. Blade-to-Blade View of Axial Velocity Contours

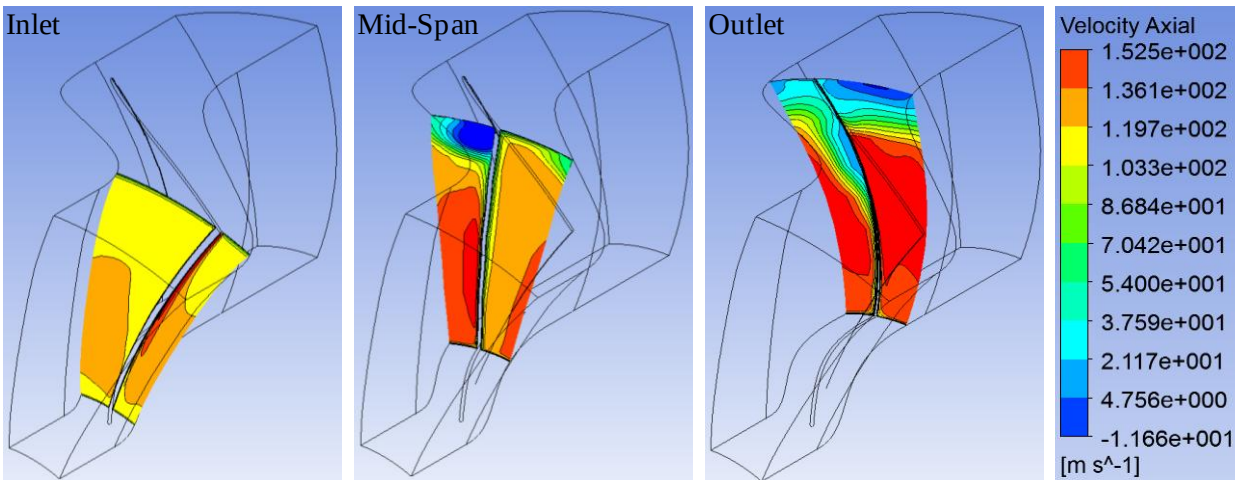


Figure 5.18. Full Span View of Axial Velocity Contours

Figure 5.17 shows the blade-to-blade view of axial velocity contours at representative spans. The boundary layer thickness can be seen increasing with both downstream location and increasing span. The separated region begins on the suction surface of the blade at approximately

50%-stream at spans above 50%. The separated region grows towards the outlet as the total pressure increases, and for spans beyond 87%, the separated region reaches across to the suction surface of the neighboring blade just upstream of the outlet plane to beyond the blade trailing edge. This is particularly evident in the full-span velocity contour at the outlet shown in Figure 5.18.

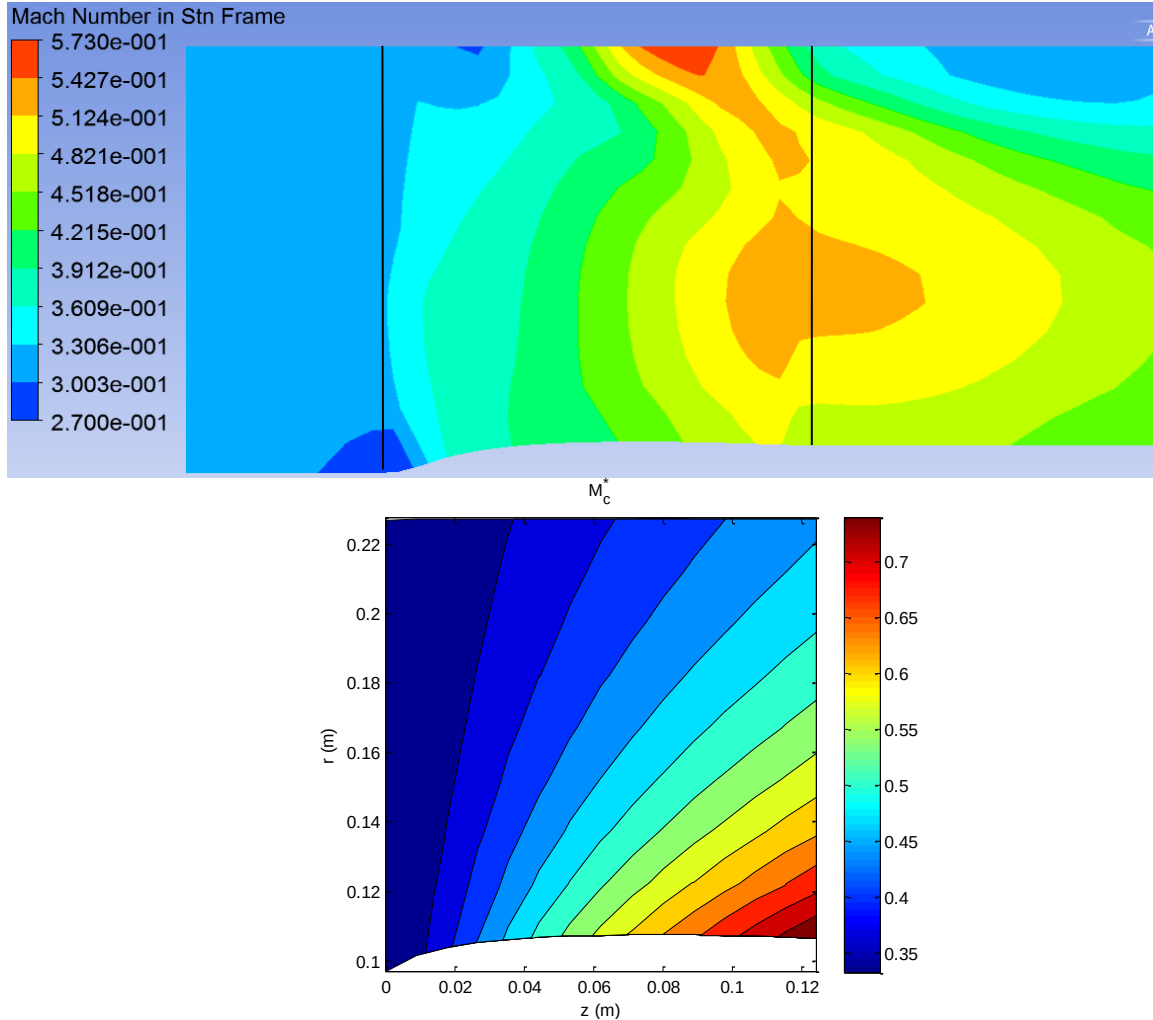


Figure 5.19. Meridional View of Absolute Mach Number Contours. Upper: Simulation, Lower: Calculation

Although full-turning calculations predict an absolute Mach number at the hub outlet of 0.85, the absolute and relative Mach numbers shown in Figures 5.19 and 5.20, respectively, remain below the recommended respective limits of 0.7 and 0.8 indicating that there are no shock losses contributing to the overall losses in total pressure and polytropic efficiency.

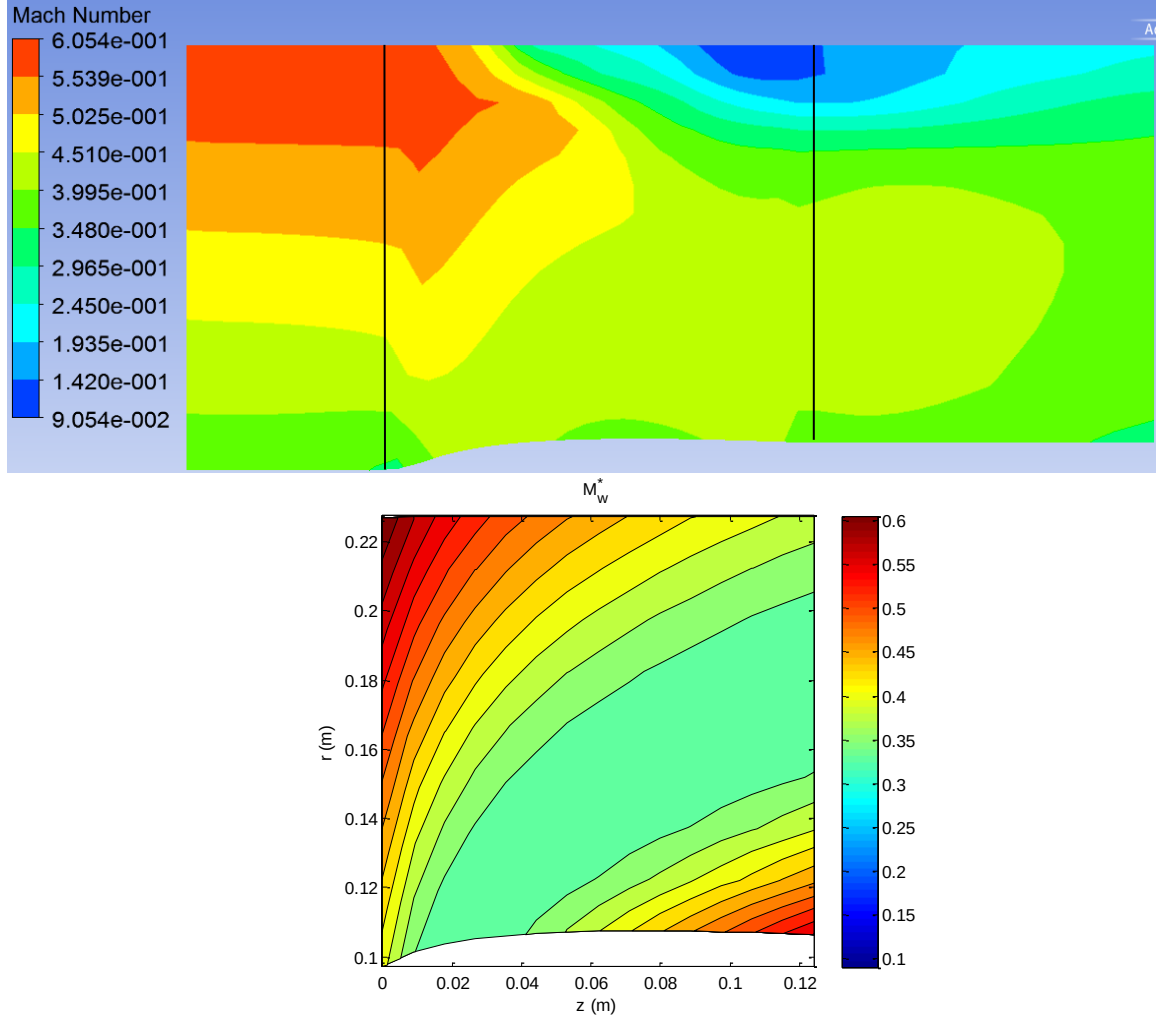


Figure 5.20. Meridional View of Relative Mach Number Contours. Upper: Simulation, Lower: Calculation

Other losses can be attributed to exit loss, which arise due to the non-uniform velocity behind the blade trailing edge [39]. The wake flow downstream of the exit plane eventually becomes uniform due to turbulent mixing, which is accompanied by an associated increase in entropy and hence pressure loss. Fully uniform flow downstream of the rotor cannot be seen in Figure 5.17 as the outlet zone is not long enough axially, but the early stages of this blending of the non-uniform velocity can be seen, particularly in the contours of spans 0.4 to 0.7. This is not visualized the same way in the comparable plots for the star pattern impeller (Figure 4.39) due to the averaging (mixing plane) between the rotor region and the outlet zone where the mesh

switches from tetrahedral elements to hexahedral, but the phenomenon is present regardless as it is for any blade cascade.

Table 5.3 provides a span-wise distribution of de Haller number and Diffusion Factor values for both calculation and simulation.

Table 5.3. FVB de Haller Numbers and Diffusion Factors

Span	<i>dH</i>			<i>DF</i>		
	Calculation	Simulation	Difference	Calculation	Simulation	Difference
Tip (90%)	0.5959	0.2231	62.6%	0.5366	0.9477	-76.6%
80%	0.5857	0.4423	24.5%	0.5628	0.7258	-29.0%
70%	0.5867	0.4965	15.4%	0.5796	0.6668	-15.0%
60%	0.6060	0.7289	-20.3%	0.5791	0.4145	28.4%
50%	0.6525	0.9769	-49.7%	0.5510	0.1614	70.7%
40%	0.7361	1.1119	-51.1%	0.4839	0.0108	97.8%
30%	0.8663	1.0840	-25.1%	0.3672	0.0293	92.0%
20%	1.0535	1.1111	-5.5%	0.1902	-0.0137	107.2%
Hub (10%)	1.3109	1.2812	2.3%	-0.0603	-0.1993	-230.3%
Area-Weighted Average	0.7259	0.7479	-3.0%	0.4627	0.3925	15.2%

Highlighted values in Table 5.3 indicate quantities that exceed recommended limitations. For the full-turning FVB calculations, only spans below approximately 35% are expected to have both the de Haller number and Diffusion Factor in the acceptable range. However, in simulation with reduced blade turning below mid-span ($\max(\beta_{out}) = 90^\circ$), spans up to approximately 70% exhibit *dH* and *DF* values within the recommended limits, and the area-averages of these values conform to the recommended limits as well. This corresponds to reasonably well-behaved core flow as is described throughout this section at a majority of spans.

5.5 Comparison of Free-Vortex Blade Impeller to Star Pattern Impeller

Table 5.4 compares the calculated values to the simulation results of the major performance parameters for the FVB and star pattern impellers found in Tables 5.2 and 4.7, respectively. The simulated results for each impeller are also compared directly in the right-most

column. The percentages of span up to which the de Haller number and Diffusion Factor criteria are satisfied are presented in the last two rows.

The FVB has smaller differences between calculation and simulation than the star pattern in every category in Table 5.4 aside from the error in expected span where DF is within the limit (23% error vs. 25% for the FVB, which of course indicates an improvement in FVB simulated performance in this case).

Table 5.4. FVB and Star Pattern Performance Comparison

		FVB			Star Pattern (SP)			Difference FVB-SP
		Calc	Sim	Diff	Calc	Sim	Diff	
$\tilde{e}$	(kJ/kg)	20.02	12.79	36.1%	19.57	11.15	43.0%	12.8%
$\dot{m}$	(kg/s)	3.822	3.234	15.4%	3.54	2.58	27.3%	20.2%
$\dot{m}_{adj}$	(kg/s)	3.218	3.234	-0.5%	2.73	2.58	5.6%	20.2%
η_{pt}	%	80.0%	74.8%	5.2%	80.0%	63.1%	16.9%	11.70%
Π_t	-	1.197	1.111	7.2%	1.193	1.074	10.0%	3.3%
$\tilde{P}$	(kW)	64.4	41.4	35.8%	53.5	28.7	46.4%	30.6%
$c_{m,avg}$	(m/s)	105.6	97	8.1%	98.7	83.3	15.6%	14.1%
Span $dH \geq 0.72$	%	42%	60%	18%	17%	52%	35%	8%
Span $DF \leq 0.45$	%	37%	62%	25%	30%	53%	23%	9%

Comparing the difference in simulated performance between the FVB and star pattern impellers, the FVB shows improvement in every category when simulating these geometrically similar impellers rotating at the same speed at the same inlet conditions. Direct comparison of the simulated values indicates enhanced performance of the FVB impeller over the star pattern for all quantities considered.

Both impellers see regions of separated boundary layer accompanied by back flow due to excessive blade turning, but the FVB impeller sees total pressure approximately uniformly increase (without seeing a subsequent loss as discussed in Sections 4.5.2 and 5.4) from inlet to outlet for spans up to 85% as opposed to just 50% for the star pattern impeller. This is due to the smaller separated region seen in the FVB impeller. While both impellers see boundary layer growth on the shroud, the star pattern configuration has the unique issue of having two separate

blades emanating from the same slot, and so boundary layers that are more prone to blade separation at higher span due to higher blade curvature are also naturally closer together due to the inherent geometry of the star pattern (effectively infinite blade solidity at the tip). The neighboring boundary layers interact with one another and this creates a very large separated region through which core flow does not move.

Some of these observed phenomena can be attributed to flow incidence, ι , the difference between the blade angle and the relative flow angle at the leading edge:

$$\iota = \beta_{blade,in} - \beta_{flow,in} \quad (5-26)$$

Figure 5.21 plots the incidence angle against span for both the FVB and star pattern impellers.

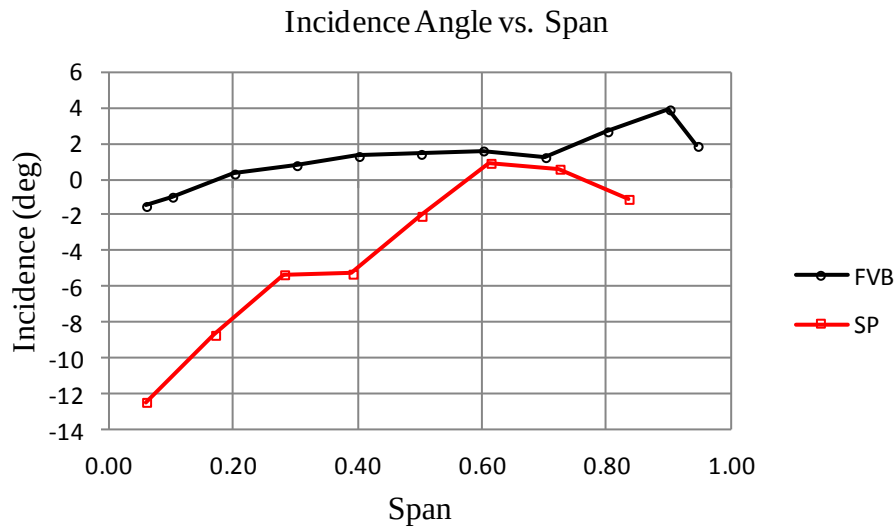


Figure 5.21. Incidence Angle vs. Span for FVB and Star Pattern Impellers

The incidence angle at the largest span is not included for the star pattern impeller to keep Figure 5.21 legible. The blades are close enough together to effectively block the core flow from entering the rotor completely at this span (where $\iota = -193^\circ$). This indicates back flow upstream of

the rotor inlet for high span. The upstream back flow and core flow blockage are further illustrated by the velocity vectors in Figure 5.22 (see also the Span 0.9 pane in Figure 4.39).

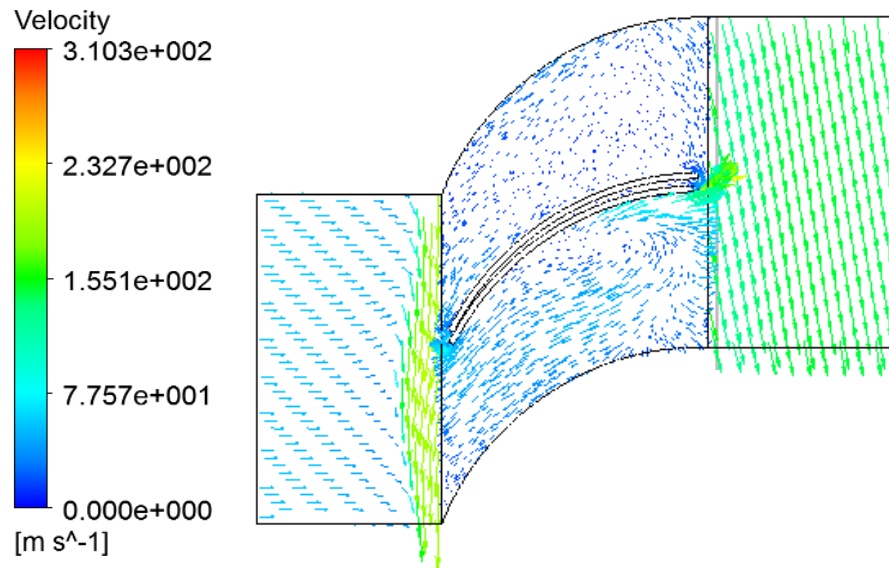


Figure 5.22. Star Pattern Velocity Vectors at Span 0.945

Velocity vectors at high span in the FVB regime also exhibit undesirable behavior, although it is less undesirable than that of the star pattern. Flow is admitted to the rotor region past the blade leading edge before separating. This is shown in Figure 5.23.

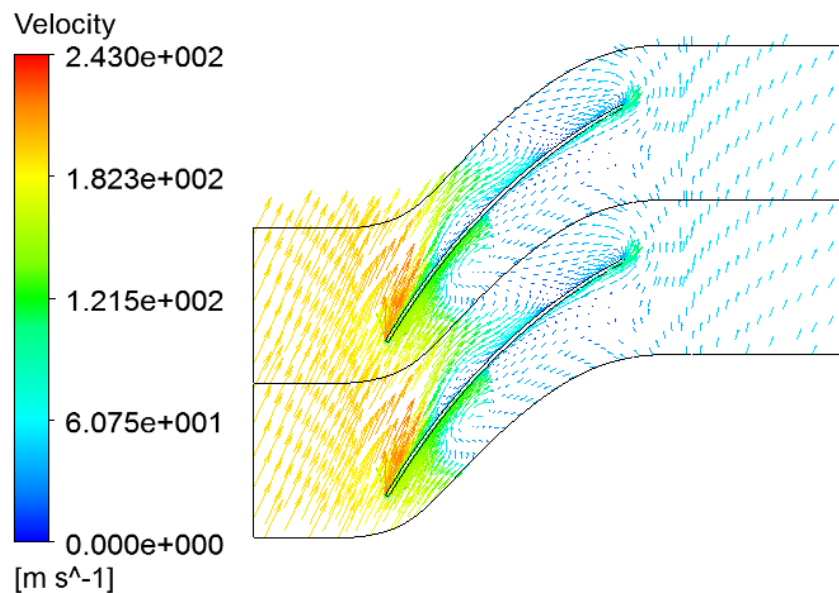


Figure 5.23. FVB Velocity Vectors at Span 0.9

Positive incidence accelerates flow onto suction surface where it then diffuses to match core flow. This local diffusion can increase the boundary layer thickness, and as has been discussed, thick boundary layers on curved surfaces under adverse pressure gradients are prone to separation, creating large losses. Negative incidence causes much of the same phenomena, but it is seen rather on the pressure surface. Smaller incidence angle magnitude is desirable for reducing losses in pressure ratio and efficiency as the blades cause the flow deflection rather than the flow itself correcting around the blade leading edge [54].

The FVB sees incidence angles ranging from -1.5° at low span to 3.9° at high span with an average value of 1.14° , whereas the star pattern impeller incidence ranges from -12.4° at low span to 0.9° at higher span back to -193.1° at very high span. The average incidence of the values plotted in Figure 5.21 is -4.2° . The smaller range of incidence angles (and smaller average incidence magnitude) for the FVB impeller in combination with reduced turning at both high and low spans compared to the star pattern impeller results in less boundary layer separation and overall improvements in total pressure ratio, polytropic efficiency, and mass flow rate. Figures 5.24 and 5.25 show velocity vectors at the more well-behaved mid-span regions for the star pattern and FVB impellers, respectively.

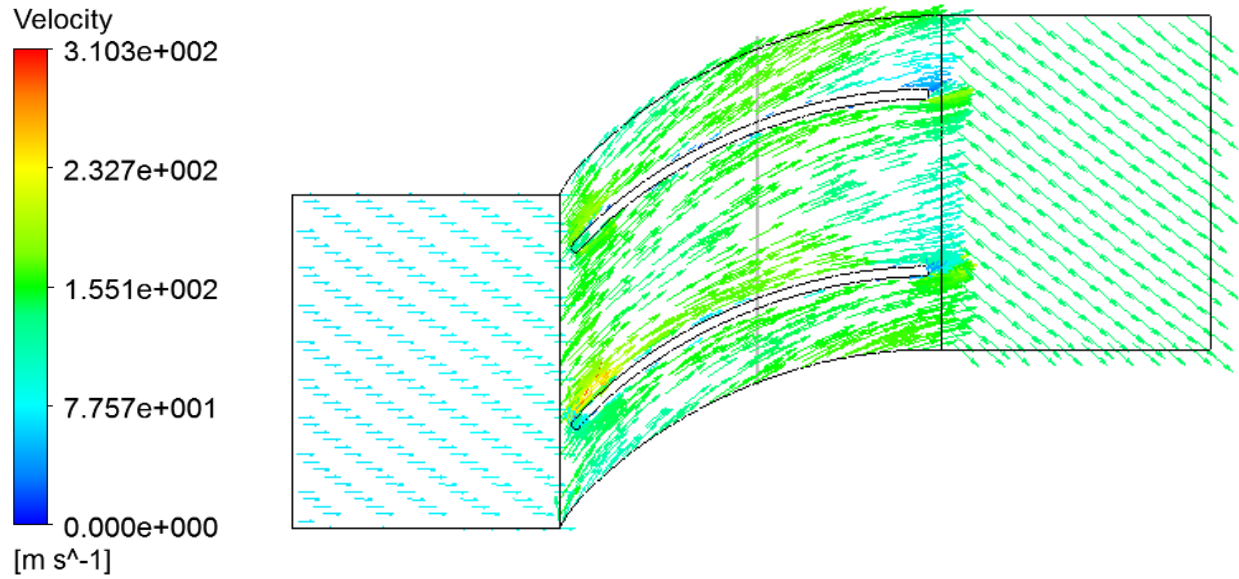


Figure 5.24. Star Pattern Velocity Vectors at Span 0.5

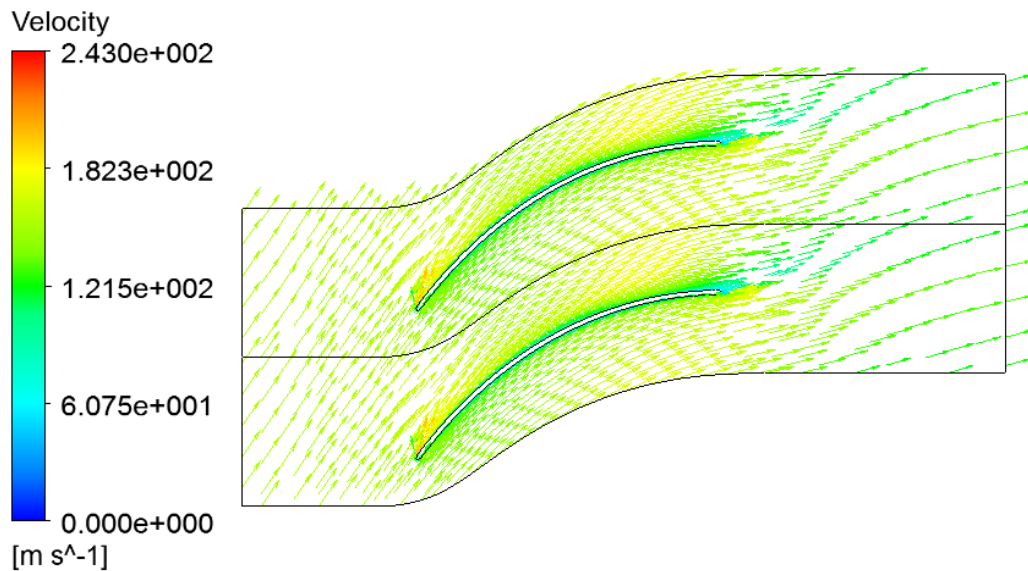


Figure 5.25. FVB Velocity Vectors at Span 0.5

Relatively low incidence at mid-span results in the majority of the core flows seen in both Figures 5.24 and 5.25 following the blades reasonably well until near the rotor outlet where the boundary layer is at its thickest and the onset of separation (particularly at slightly larger spans for the star pattern impeller) is incipient.

Deviation angle is the difference between blade and relative flow angle at the trailing edge:

$$\delta = \beta_{blade,out} - \beta_{flow,out} \quad (5-27)$$

Figure 5.26 plots deviation angle against span for both impeller types. Similar to Figure 5.21, larger spans are not included as the deviation values are very large in separated regions (back flow regions with deviation angles in the neighborhood of -180°). The average deviation angle for the plotted values of the FVB rotor is 3.1° , and the average for the star pattern impeller is 7.6° . Even with the smaller range of spans considered for the star pattern impeller, there is still larger simulated deviation than is seen in the FVB impeller.

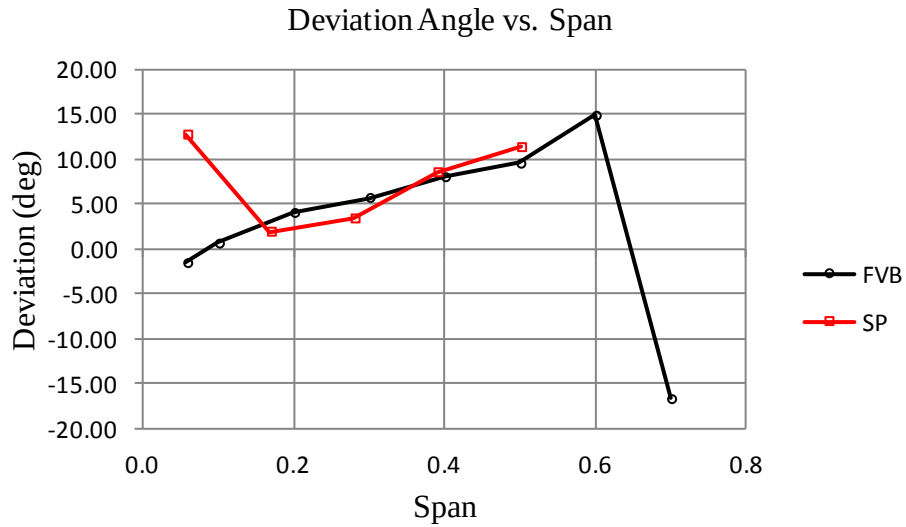


Figure 5.26. Deviation Angle vs. Span for FVB and Star Pattern Impellers

Smaller deviation angles are desirable as the closer the flow follows the blade at the outlet, the more the absolute tangential velocity will be as predicted, and hence predictions of work transfer and total pressure ratio will also be more accurate. This is of course the case when comparing the average deviation values mentioned in the previous paragraph to the error values presented in Table 5.4 (i.e. the FVB has lower average deviation and also less difference between

calculated and simulated quantities). The separated regions in both impeller flows have very large deviation angles, and these regions do not perform useful work.

5.6 Chapter Summary and Conclusions

The geometry of the free-vortex blade impeller analyzed in this chapter was selected to be as similar as possible to the star pattern impeller discussed in Chapter 4 within the confines of the respective flow distributions. Using the same inlet conditions, operating speed, and mean blade turning, the performance of the free-vortex impeller has been predicted analytically and in numerical simulation using ANSYS CFX. The following conclusions can be made:

- It has been found that the FVB impeller shows improvements in average specific work transfer to the fluid and achieves a higher total pressure ratio in both calculation and simulation. Numerical simulation shows that this is achieved while operating at higher polytropic efficiency (see Table 5.4) despite both styles of impeller initially assumed to operate with $\eta_{pt} = 80\%$.
- Lower blade solidity at the FVB tip means that the separated region exhibiting back flow is confined to higher span (less overall area) than the star pattern separated region. The total pressure uniformly increases from inlet to outlet for spans up to than 85% for the FVB impeller as opposed to just 50% for star pattern.
- Absolute and relative Mach numbers are within the recommended limits in simulation indicating that there are no losses due to entropy increase associated with shock formation.
- The de Haller number and Diffusion Factor limitations are satisfied for a wider range of spans for the FVB impeller (from approximately 0% to 60%) than the star pattern impeller (from approximately 0% to 50%).

- The average incidence and deviation angles for the FVB impeller are lower than for the star pattern. The range of incidence and deviation angles is also narrower for the FVB impeller.

Even without carefully controlling blade curvature and rotational speed to remain within the Mach number limits along with the deceleration and diffusion limits set by the de Haller number and Diffusion Factor, respectively, simulation indicates that the FVB rotor will outperform the star pattern rotor under the same conditions with very similar average geometric properties. Since incidence and particularly deviation angles are smaller for the FVB impeller than for the star pattern impeller indicating that the flow more closely follows the blades, the FVB impeller has been selected for further study involving multiple counter-rotating stages. Chapter 6 details this study in full.

CHAPTER 6 OPTIMAL PERFORMANCE OF MULTISTAGE COUNTER-ROTATING AXIAL COMPRESSORS

Euler's turbomachinery equation states $\tilde{e} = u_2 c_{u2} - u_1 c_{u1}$ (see Equations (2-78) and (2-93)) and it is clear that work transfer is large when there is large change in absolute tangential velocity being that there is relatively little change in blade tip speed from inlet to outlet, if any, in an axial impeller. When c_{u1} and c_{u2} have opposite signs, i.e. when the fluid swirl direction at the inlet is opposed to the blade rotation direction at the inlet (and the fluid swirl at the outlet), $\tilde{e}$ can be very large indeed.

The concept of counter-rotating propellers in the aircraft industry has been conceived since the late 1930's where initial wind tunnel tests showed significant increases in effective thrust over a single propeller [57].

In the early 1940's, counter rotating propellers were conceived with the aim of eliminating propulsion losses from the rotational component of the slipstream by imparting equal and opposite torques to two axially aligned propellers acting on the same air [58]. Energy losses in the wake of a propeller (and aircraft) are unavoidable, but reducing the swirl component of the wake will reduce the overall loss [59].

Early efforts at implementation however were not proven to be generally successful. The use of concentric counter-rotating shafts and gear boxes had added significantly to the complexity and weight of aircraft engines resulting in increased cost and maintenance requirements. Operational problems with leakage and gear train failure due to lack of equipment refinement had discouraged the use of counter rotation [60].

However more recent technological developments have led to renewed interest in counter-rotation for aircraft application. For example, ultra-high bypass ratios without large

diameters (and hence weights) are possible with counter-rotating unducted turbofan engines [61]. Another study shows that by decreasing engine weight via the elimination of stators, the improved thrust-to-weight ratio leads to better fuel efficiency (between 9%-17%) than for comparable unidirectional rotation [62].

Studies of counter-rotating axial flow fans and compressors focus primarily on speed ratio of counter-rotating stages and axial spacing, however the investigations tend to expound on very few combinations (typically up to three or four speeds per rotor) and generally for only two counter-rotating stages, e.g. [63]–[66]. It is typically found from experimental testing that the downstream (aft) rotor must rotate faster for best performance as the suction from the aft rotor allows the boundary layer on the upstream (fore) rotor to remain attached. In light of this, studies have focused on methods to maintain boundary layer attachment on blades and end walls by use of complex boundary layer bleed techniques, e.g. [67], [68]. Another study [69] uses essentially identical rotors spinning in opposite directions to investigate the effects of casing treatments (using recessed vanes) on boundary layer development. While studies do exist for higher numbers of stages, e.g. [70] has 14 counter-rotating stages, the rotors and counter rotors are spooled with rotors being driven by a turbine, and counter-rotors driven by another counter-rotating turbine. Hence all rotors spin at the same speed and all counter rotors spin at the same contra-speed. Three speed ratios between these rotor/counter-rotor sets are then investigated with similar findings.

The complete rotational independence of each stage enabled by the integration of permanent magnet motors with the wound composite impeller allows this research to depart from the design limitations found in the literature and widens the design space substantially. The design procedure is as follows.

6.1 Design Procedure

The design methodology detailed in Chapter 2 is used for each stage. The flow chart provided in Figure 6.1 recapitulates the procedure for an individual rotor. Set variable values are in green boxes. Black boxes are either pre-set (left-most column) or are determined by the set and pre-set values.

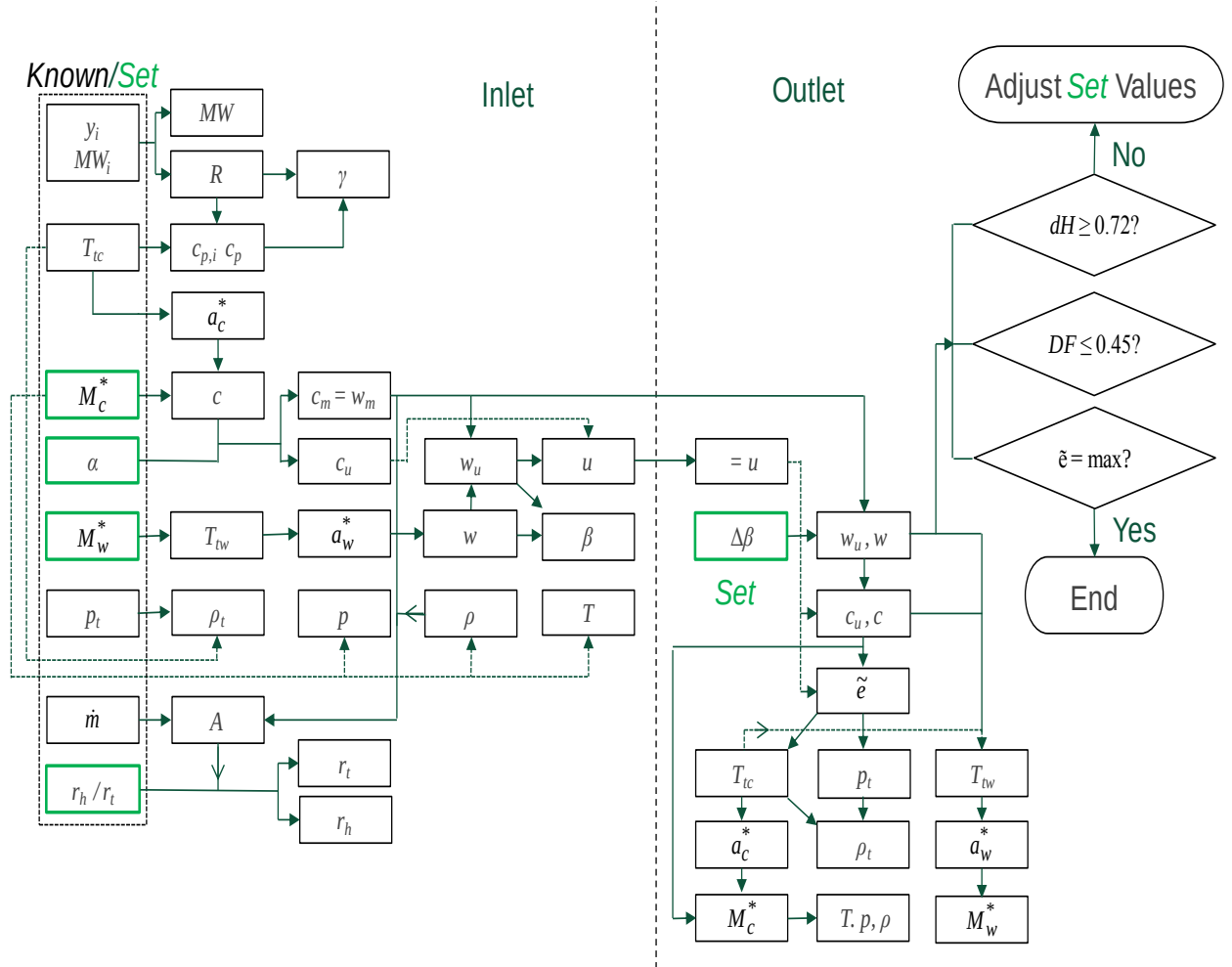


Figure 6.1. Design Process Diagram for a Single Rotor

The inlet of the first rotor is the starting point where the fluid mixture (molecular weight and mole fraction of each component), total temperature, total pressure, and mass flow rate are known quantities. The absolute flow angle here is set using an outlet angle of an inlet guide vane. Continuity is used upstream to set the inlet of the IGV, which is assumed to be swirl free.

Since the outer radius is held constant for ease of construction (winding), all variable quantities are set at the tip to facilitate a non-iterative calculation procedure. This is possible because the tip speed is necessarily constant from inlet to outlet with constant tip radius, and as is shown in Figure 6.1, it is this that bridges the gap from inlet to outlet. The constant tip speed is used in conjunction with constant axial velocity and a choice for blade turning at the tip, and thus the outlet at the tip is fully specified. The free-vortex flow regime sets the spans below the tip radius, and the specific work is calculated from Equation (2-93). The conditions in the absolute reference frame are held constant from the outlet to the inlet of the next downstream rotor. Table 2.4 provides the details of all necessary calculations.

6.1.1 Design Objective

The objective is to impart the maximum amount of work to the fluid for any given number of stages, N_s , i.e.

$$\max \left(\sum_{i=1}^{N_s} \tilde{e}_i \right) \quad (6-1)$$

The analytical nature of this investigation assumes a polytropic efficiency of 80% for each stage (and hence the entire compression process), and so maximizing the combined work transfer of each stage equates to maximizing the overall total pressure ratio. Figure 6.1 must be modified accordingly for multistaging: the maximum work of any individual rotor is not the objective, but the total work of the combined stages as stated in Equation (6-1).

6.1.2 Design Variables

There are global quantities to select at the inlet of the first rotor that will affect the entire machine in terms of both geometry and flow kinematics. Each individual subsequent rotor has its

own variable quantities in addition to these. A summary of the design variables selected for this study in accordance with Figure 6.1 is provided in Table 6.1. All variables are specified at the tip radius when appropriate.

Table 6.1. Summary of Initial Design Variables

Quantity		Location
Hub/Tip Ratio	r_h/r_t	1 st Rotor Inlet
Absolute Mach Number	M_c^*	1 st Rotor Inlet
Absolute Flow Angle	α	1 st Rotor Inlet
Relative Mach Number	M_w^*	Each Rotor Inlet
Blade Turning Angle	$\Delta\beta$	Each Rotor
Number of Blades	N_b	Each Rotor
Aspect Ratio	AR	Each Rotor
Blade Thickness	t_b	Each Rotor

Up to seven counter-rotating stages have been considered. Table 6.2 provides the number of design variables according to the number of stages in consideration.

Table 6.2. Number of Initial Design Variables by Number of Stages Considered

Stages Used	Design Variables
1	8
2	13
3	18
4	23
5	28
6	33
7	38

The inlet blade angle is *not* a design variable. The inlet blade angle of each rotor is determined such that it exactly matches the exiting flow from the preceding rotor (or IGV in the case of the first rotor). This is a major difference from the studies cited at the beginning of this chapter where the blade angles are set and then blade speeds (speed ratios) are varied without regard for the incidence angle increase (or decrease) at the inlet of the downstream rotor.

6.1.3 Design Constraints

While Equation (2-93) may indicate that very large specific work transfer can be obtained with a very large change in absolute tangential velocity, there are practical limits to what can be achieved. This study does not assume an idealized flow situation, but there are many effects that a real flow will experience that are not accounted for. Similar to how the Carnot efficiency is a benchmark for the maximum performance of a thermodynamic cycle operating between two reservoirs [2], this study seeks to determine the conditions at which maximum theoretical performance can be achieved while satisfying the constraints discussed in this section. The constraints described here and in Chapter 2 help to maintain more reasonable expectations on what an actual flow may attain. For convenience they are repeated from Table 2.5:

Table 6.3. Summary of Design Constraints

Quantity	Constraint
Absolute Mach Number	$M_c^* \leq 0.7$
Relative Mach Number	$M_w^* \leq 0.8$
de Haller Deceleration	$dH \geq 0.72$
Diffusion Factor	$DF \leq 0.45$
Hub/Tip Ratio	$r_h/r_t < 1$
Axial Velocity	$c_m = \text{constant}$
Aspect Ratio	$0.5 < AR < 2.0$
Absolute Flow Angle	$15^\circ < \alpha < 165^\circ$
Relative Flow/Blade Angle	$15^\circ < \beta < 165^\circ$

A single constraint violation for a single rotor renders the entire design infeasible. The Mach numbers are selected such that the risk of choking and shock formation, and the associated increase in entropy and pressure loss, is minimized. The de Haller number and Diffusion factor values are limited by the values recommended in [42] and [43], respectively in order to keep boundary layer growth in check and reduce the likelihood of subsequent boundary layer separation. The axial velocity is constant in the span-wise direction according to the free-vortex

condition (itself another constraint), and for simplicity of calculation it is set as constant in the stream-wise direction as well, thereby imposing a fully uniform axial velocity distribution everywhere within the rotor. The hub/tip ratio is limited by its own geometry (i.e. the hub radius must necessarily be less than the tip radius), and aspect ratio selections are limited so that the blades are not exceedingly long (low aspect ratio) or short (high aspect ratio). Absolute and relative flow angles are limited such that the axial velocity component is not negligible relative to the tangential component in either the inertial or rotating reference frame.

6.2 Initial Design Exploration

Early attempts at imparting maximum work while satisfying all constraints were made using a non-linear program solver in Matlab (*fmincon*) and Microsoft Excel (GRG non-linear), and using the genetic algorithm technique both Excel and in the HEEDS Multidisciplinary Design Optimization environment. However the results (if any) were not deemed to be reliable as it was possible on numerous occasions to achieve a feasible result with larger work transfer by simply adjusting an input variable by hand.

This lead to performing a preliminary exhaustive search of the design space for a two stage counter-rotating machine. Figures 6.2 and 6.3 show specific work transfer for the first and second rotor, respectively, and Figure 6.4 shows the total combined specific work of both stages. These are plotted against the global inlet variables M_c^* and α . The contours on the left show the unconstrained design space, and contours on the right show the constrained space. The star indicates the values corresponding to maximum feasible combined work transfer.

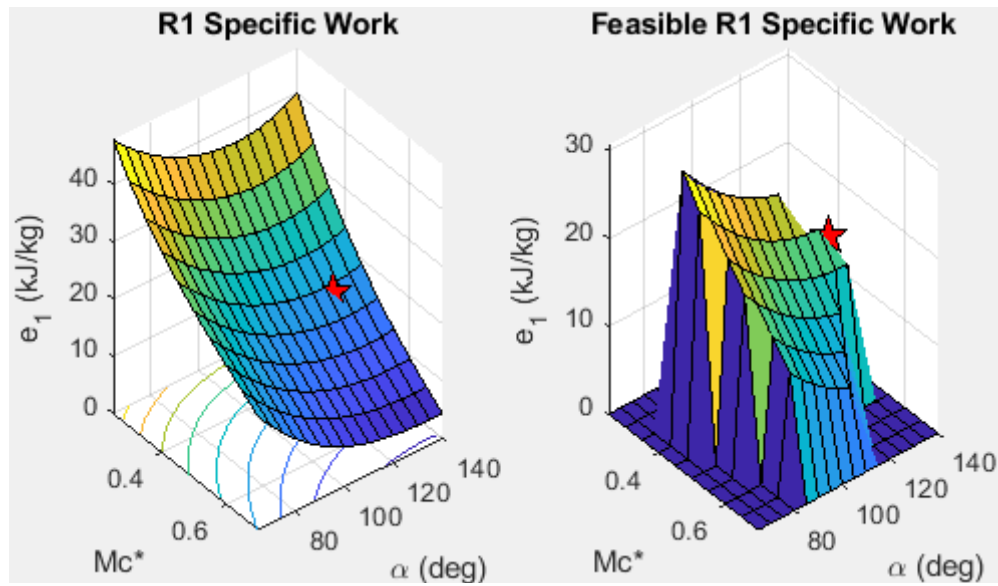


Figure 6.2. Rotor 1 Work Transfer vs. Inlet Flow Angle and Mach Number

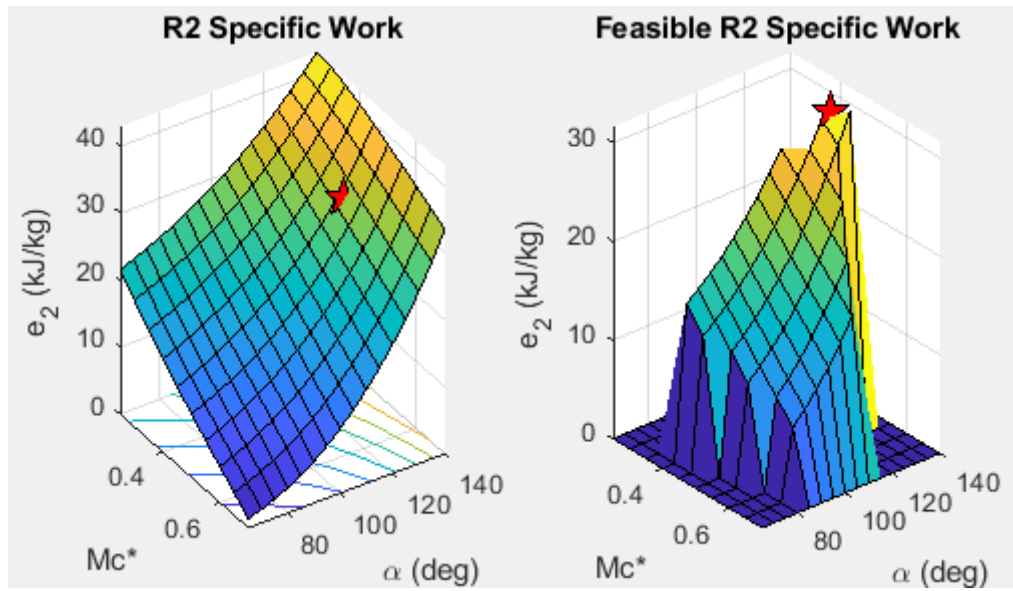


Figure 6.3. Rotor 2 Work Transfer vs. Inlet Flow Angle and Mach Number

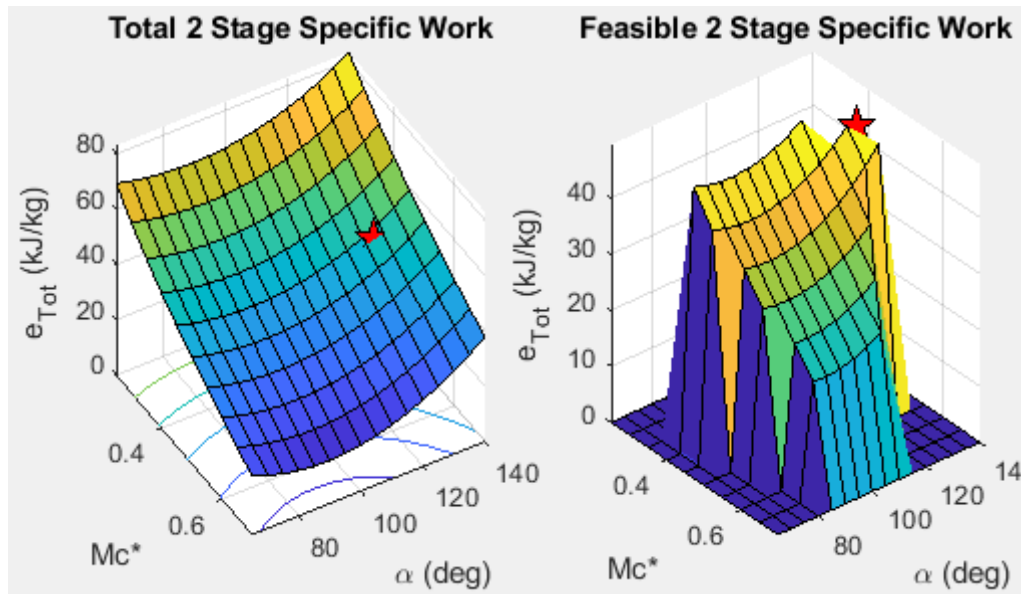


Figure 6.4. Total Two Stage Work Transfer vs. Inlet Flow Angle and Mach Number

It was originally hypothesized that the maximum feasible work for each rotor would combine to produce the maximum overall work; however as can be seen in Figure 6.2, the maximum work imparted by the first rotor does not correspond to the maximum combined feasible work transfer (although it does for the second rotor). With the hypothesis rejected for the simplest counter-rotating case, the design space for a larger number of stages was searched. As an example, Figure 6.5 shows the surface and contours of feasible combined work for a four stage counter-rotating machine plotted again against the global inlet variables critical Mach number and absolute flow angle.

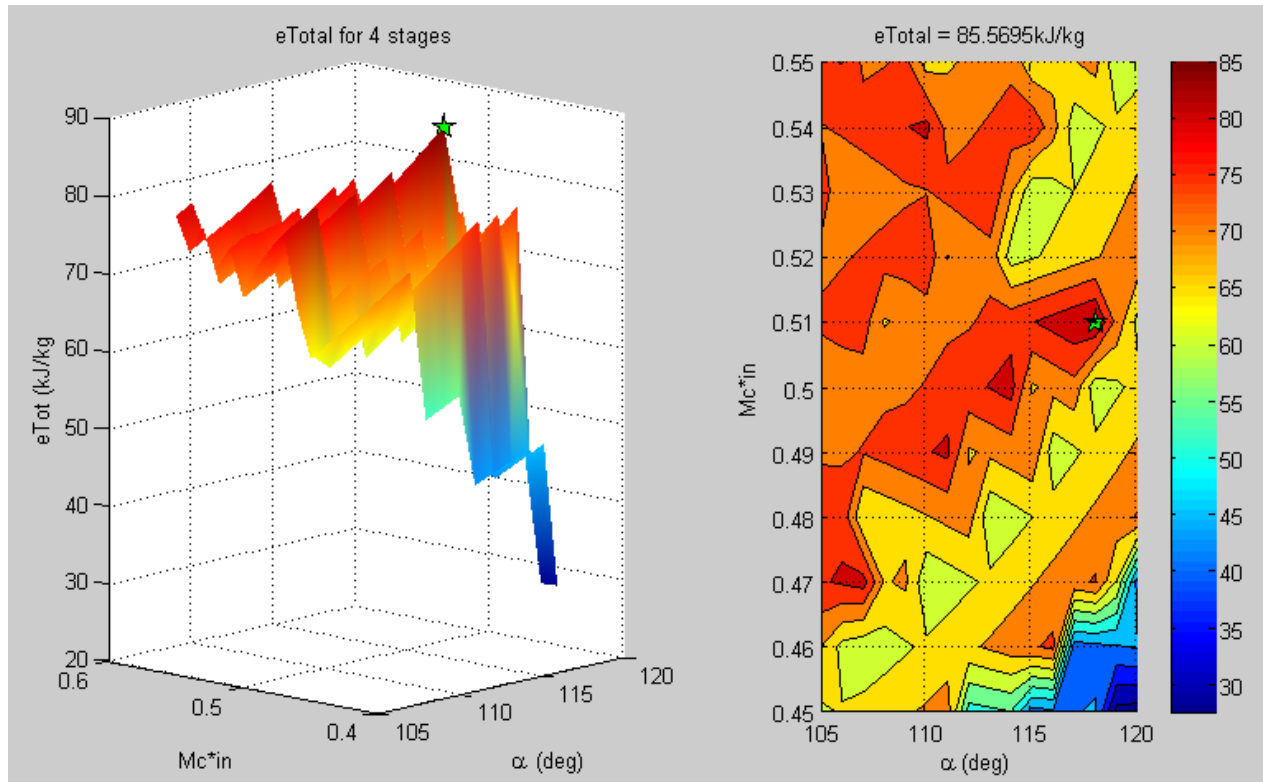


Figure 6.5. Total Four Stage Feasible Work Transfer vs. Inlet Flow Angle and Mach Number

The highly irregular surface of the feasible design space helps to explain why the non-exhaustive search methods failed to find the global maximum as there are local maxima at approximately every other variable value. In light of this, a full exhaustive search of the design space for all numbers of stages used is made.

However, there are a significantly large number of design options. Table 6.2 indicates that there are 38 variables for a 7 stage machine. With even just two options per variable, that results in 2^{38} (approximately 275 billion) possible design combinations, and this is an extremely coarse search. Thus, a significant reduction in variables is necessary in order to perform a reasonably fine search in a reasonable amount of time (in the order of days rather than months or years).

6.2.1 Design Variable Reduction

The simplest variable to fix is the blade thickness. Each rotor is set to have 2mm blade thickness at all spans since this has been approximately the resulting thickness after various winding tests. This eliminates seven variables (new total: 31).

The number of blades for each rotor is fixed. Table 6.4 provides the selected values. The blade number for each rotor is not the same in order to reduce the possibility of encountering resonance effects on the shared shaft due to natural vibration frequencies of one rotor exciting those of another [71]. The blade counts are however close to each other so that the pitch ratio ($N_{b,i+1}/N_{b,i}$) is near unity such that the flow does not experience drastic cross-sectional area change transitioning between rotors. This eliminates seven variables (new total: 24)

Table 6.4. Blade Number by Stage	
Stage	Blade Number
IGV	12
1	11
2	13
3	19
4	20
5	16
6	15
7	18

The aspect ratio as described in Equation (2-127) is removed as a variable by setting it according to unity mean solidity. Having high blade solidity results in a large wetted area which will result in friction loss. Reducing the solidity reduces friction loss but increases separation loss. Unity solidity best balances the friction and separation losses resulting in the lowest overall profile loss for a large number of situations [39].

The blade count in conjunction with rotor radii determine the blade stagger in Equation (2-132), so unity mean solidity sets the mean chord length as per Equation (2-131). Equation (2-102) describing blade radius as a function of blade turning and axial length is substituted into the

general expression for chord length in Equation (2-95), which is in turn substituted into (2-131).

A final substitution of (2-127) gives the aspect ratio for unity mean blade solidity as a function of blade number, turning, and radii as

$$AR = \frac{N_b(r_t - r_h) \sin\left(\frac{\beta_{out} - \beta_{in}}{2}\right)}{\pi r_m (\cos \beta_{in} - \cos \beta_{out})} \quad (6-2)$$

which has not been found in the literature. This eliminates seven variables (new total: 17)

The last group of variables to eliminate is the relative inlet Mach number at each rotor. Starting from any set of variables found to impart maximum total work transfer as part of an exhaustive search and holding these constant, varying relative inlet Mach number for each rotor reveals that in all cases that the maximum allowable relative inlet Mach number to each stage corresponds to maximum combined work transfer. Figures 6.6–6.8 plot the individual stage work and combined work vs. $M_{w,in1}^*$, $M_{w,in2}^*$, and $M_{w,in3}^*$, respectively, for an example 3 stage case.

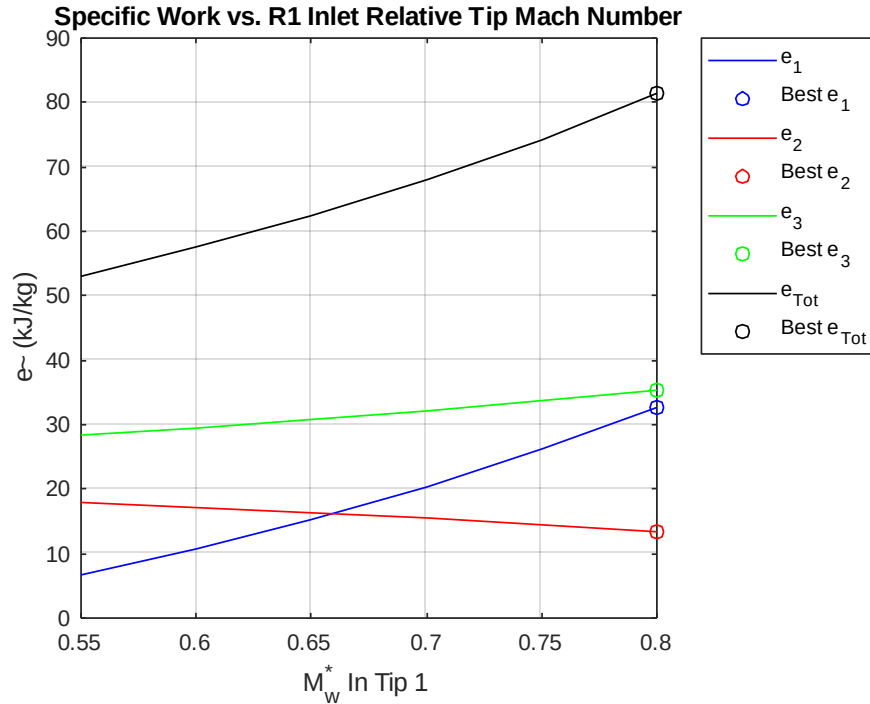


Figure 6.6. Work Transfer vs. Inlet Mach Number of First Rotor around Best-Point

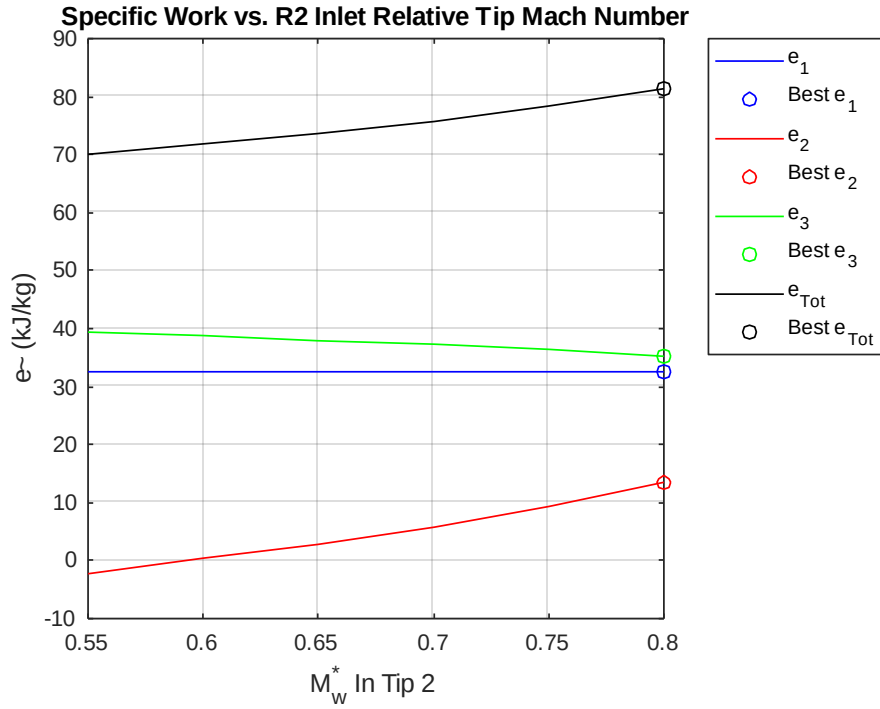


Figure 6.7. Work Transfer vs. Inlet Mach Number of Second Rotor around Best-Point

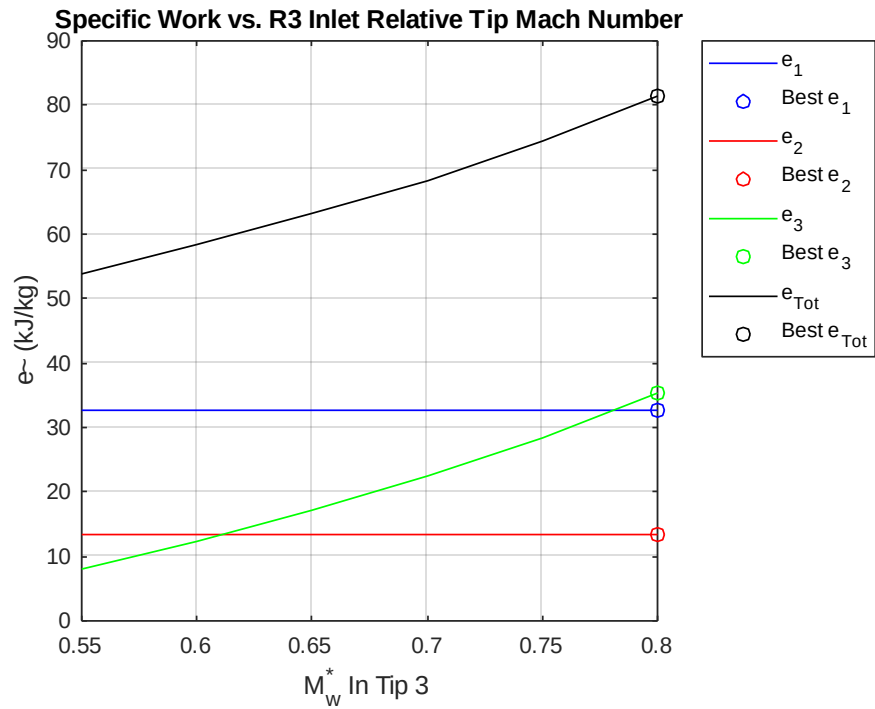


Figure 6.8. Work Transfer vs. Inlet Mach Number of Third Rotor around Best-Point

Selecting the maximum value for each allows the elimination of seven variables (new total: 10). Table 6.5 summarizes the remaining variables from Table 6.1 for the multistage exhaustive search.

Table 6.5. Reduced Design Variable Set		
Stages Used	Variable	Number Design Variables
1-7	r_h/r_t	
1-7	M_c^*	
1-7	α	
1-7	$\Delta\beta_1$	4
2-7	$\Delta\beta_2$	5
3-7	$\Delta\beta_3$	6
4-7	$\Delta\beta_4$	7
5-7	$\Delta\beta_5$	8
6, 7	$\Delta\beta_6$	9
7	$\Delta\beta_7$	10

6.2.2 Design Constraint Evolution

There are additional constraints that have evolved with the study itself. It was originally hypothesized that the blade turning should be such that the absolute flow angle should change evenly on either side of $\alpha = 90^\circ$ from one rotor to the next counter-rotor, and so on downstream to the exit (e.g. $\alpha_{1,out} = \alpha_{1,in} + \Delta\alpha$ and $\alpha_{2,out} = \alpha_{1,out} - 2\Delta\alpha = \alpha_{1,in}$). This would ensure behavior similar to a traditional unidirectional rotor-stator multistage machine: the rotor imparts swirl to the flow and the stator (counter-rotor here) serves to change the swirl back to its original upstream-of-the-rotor state so that it may then enter the next rotor with the same swirl distribution. This pattern repeats until the flow exits the final stage. With reference to Figure 2.9, this also ensures that axial velocities do not diminish while maintaining tip speeds. The ultimate goal of imposing this was also eliminating $\Delta\beta$ as a variable and making any machine fully dependent on the conditions at the inlet of the first rotor.

However it was determined that imposing even $\Delta\alpha$ will not achieve maximum work transfer. As total pressure increases due to compression, temperature necessarily increases as well (see Equation (2-32)). Maintaining the same absolute portion of the velocity diagram at the inlet of e.g. the first and third rotors necessitates the relative components also be the same (see Figure 2.11 for schematic representation). The temperature is higher, and hence higher speed of sound, but with the same relative flow speed there is lower relative Mach number (which was determined to require maximization in Section 6.2.1 in order to impart maximum combined work).

It was subsequently hypothesized that even $\Delta\alpha$ angles will produce maximum work if the axial velocity component is increased thereby allowing for the increase of u (and hence w_u and M_w^*) to maintain similarity of the triangles in the velocity diagram. This was accomplished by keeping the axial component Mach number M_{cm}^* (see Equation (2-125)) constant for ease of calculation such that the increased sound speed results in increased c_m for the same M_{cm}^* (axial velocity is still held constant in the span-wise direction per free-vortex requirements). This scheme allowed for greater combined work transfer than with constant c_m , but still did not ultimately allow for maximization of M_w^* . The resulting work for each stage was very similar (within 1%) as downstream stages were required to do less work than they were capable of achieving (as an example, Figures 6.6–6.8 show values of work to be different by 143% between stages 1 and 2, and 62% different between stages 2 and 3 for maximum work transfer). As such, the even $\Delta\alpha$ constraint was dropped for the exhaustive search (keeping $\Delta\beta$ as a variable as per Table 6.5), but the constant M_{cm}^* constraint was kept, as improvements in combined work transfer were found to be approximately 5% on average for otherwise unchanged conditions.

As a supplemental advantage, allowing the axial velocity to increase in the axial direction has the effect of reducing boundary layer growth for subsonic flow relative to what would otherwise be expected from constant axial velocity from inlet to outlet [72].

6.3 Design Exploration

The full design exploration is set as follows. The reduced set of design variables is given in Table 6.5, and the constraints for determining a feasible design are set in Table 6.3 with the exception of M_{cm}^* being held constant rather than c_m .

Two different exhaustive searches of the design space are performed. First, a coarse global search using a wide range for each variable is performed. Upon determination of the variable combination for imparting maximum overall work, the range of each variable is refined in the neighborhood of the coarse search best-point. A fine exhaustive search of the narrowed local space is then performed. Table 6.6 provides the variable ranges and number of discrete points (resolution) for each variable in the global search of the design space along with the resulting variable increment.

Table 6.6. Variable Range and Resolution for Global Design Space Search

Coarse/Global Variable Range	Resolution	Increment
$70^\circ \leq \alpha \leq 140^\circ$	14	5°
$0.4 \leq M_c^* \leq 0.7$	7	0.05
$0.3 \leq r_h / r_t \leq 0.95$	9	0.05 or 0.1
$5^\circ \leq \Delta\beta_1 \leq 20^\circ$	5	4°
$5^\circ \leq \Delta\beta_2 \leq 20^\circ$	5	4°
$5^\circ \leq \Delta\beta_3 \leq 20^\circ$	5	4°
$5^\circ \leq \Delta\beta_4 \leq 20^\circ$	5	4°
$5^\circ \leq \Delta\beta_5 \leq 20^\circ$	5	4°
$5^\circ \leq \Delta\beta_6 \leq 20^\circ$	5	4°
$5^\circ \leq \Delta\beta_7 \leq 20^\circ$	5	4°

Note: hub/tip ratio switches increments in Table 6.6 from 0.1 to 0.05 at $r_h/r_t = 0.8$.

Table 6.7 provides the variable ranges and resolution changes depending on the number of counter-rotating stages used. The ranges themselves of course bracket the individual variable value corresponding to maximum work transfer as determined by the coarse global search. Additionally, the method for determining the blade turning is modified for the refined search. Rather than refining $\Delta\beta$, the local search uses the outlet blade angle determined from the coarse search and moves in both directions. This evaluates designs with both more and less blade turning as opposed to strictly increasing blade turning from the coarse result in order to perform a more complete search of the local design space.

Table 6.7. Variable Range and Resolution for Local Design Space Search							
Fine/Local Variable Range	Resolution (Increment)						
$\alpha \pm 5^\circ$	10 (1.1°)						
$M_c^* \pm 0.05$	10 (0.11)						
$r_h / r_t \pm 0.05$	6 (0.18)						
	Number of Stages Used						
	1	2	3	4	5	6	7
$\beta_{1,out} \pm 4^\circ$	7 (1.3°)						
$\beta_{2,out} \pm 4^\circ$	7 (1.3°)	7					
$\beta_{3,out} \pm 4^\circ$	7 (1.3°)	7	6				
$\beta_{4,out} \pm 4^\circ$	7 (1.3°)	7	6	6			
$\beta_{5,out} \pm 4^\circ$	7 (1.3°)	7	6	6	6		
$\beta_{6,out} \pm 4^\circ$	6 (1.5°)	6	6	6	6	6	
$\beta_{7,out} \pm 4^\circ$	5 (1.8°)	5	5	5	5	5	5

Table 6.8 provides the number of individual designs evaluated for any single set of inlet conditions (total temperature, pressure, mass flow rate). Since all seven configurations are evaluated, the grand total is also provided.

Table 6.8. Design Evaluations for One Set of Inlet Conditions			
Stages Used	Coarse/Global	Fine/Local	Total Designs
1	4,410	4,200	8,610
2	22,050	29,400	51,450
3	110,250	176,400	286,650
4	551,250	1,058,400	1,609,650
5	2,756,250	6,350,400	9,106,650
6	13,781,250	27,993,600	41,774,850
7	68,906,250	46,875,000	115,781,250
Total	86,131,710	82,487,400	168,619,110

The designs are evaluated using Matlab. A script file was written to define the inlet conditions and the vectors to store the coarse values of α , M_c^* , r_h / r_t , and $\Delta\beta$. These values are individually input to a user-defined function (UDF) that executes the process shown in Figure 6.1 using a series of nested loops to evaluate all possible combinations of the inputs. To save computational uptime, the process of evaluating a 7 stage configuration includes all lower numbered stage configurations for the coarse search. The input values corresponding to maximum combined specific work for each number of stages used are then refined and sent to separate individual UDFs to perform each local search. When each of the nearly 170 million designs is evaluated, the best combination of input values for each number of stages used is determined.

It must be noted that the elimination of each M_w^* as a variable in combination with the very large number of design evaluations as per Table 6.8 is what makes Equation (2-114), not found elsewhere in the literature, so critical. Since this study has found that M_w^* should always be maximized, the ability to directly specify the value of M_w^* saves a significant amount of computational cost as the need to iterate is removed. Knowledge of the absolute velocity vector in conjunction with the relative Mach number fixes the blade tip speed through the velocity diagram. In single stage compressor design, it is common to set the absolute Mach number and

tip speed which then sets the relative velocity and relative Mach number. Iteration of the tip speed setting would be needed in order to achieve any particular value of M_w^* . As was mentioned in the beginning of this chapter, typical counter-rotation studies vary the tip speed ratio between stages. This involves selecting both a speed for the first rotor and a tip speed ratio (to set the speed for the second rotor). For a two stage case, this adds two additional variables to the design. For a seven stage case, this adds seven additional variables. Hypothetically, if this study were to operate this way using the coarse resolution of 5, the coarse design space alone would have 270 billion options, which would take approximately two years to run on a PC. Taking advantage of Equation (2-114) allows the reduced study to complete the coarse and refined searches of the design space in less than 13 hours for a single set of inlet conditions.

6.3.1 Scope of Design Exploration

The areas of application envisioned for the wound impeller are detailed in Chapter 1. With the exception of the waste water aeration application, the others all use water vapor under vacuum pressure as the working fluid, at least in part. As such, the design exploration is performed for saturated water vapor between 5°C and 100° in increments of 5°C with two additional temperatures included to accommodate case studies, the particular details of which will not be discussed here. In this study the value of the mass flow rate itself only serves to affect the power consumption and physical dimensions of a machine, but never the less a value must be selected so that continuity may be enforced and the dimensionless geometric ratios (AR , r_h/r_t) applied appropriately. The value 0.458kg/s is chosen to match that of a small scale mechanical vapor compression process.

Additional gas mixtures are investigated at different temperatures to determine if similar trends in optimal behavior exist. These include air, methane, carbon dioxide, helium, hydrogen, a

carbon dioxide/water vapor mixture, and a gas mixture of water vapor and non-condensable gases with air representative of the contents of the condenser in a geothermal power plant. All gas mixtures are evaluated for up to seven counter rotating stages. Table 6.9 provides a summary of the designs evaluated.

Table 6.9. Designs Evaluated in Full Scale Exploration		
Gas Mixture	Number of Inlet Conditions	Designs Evaluated
Saturation H ₂ O Vapor	22	3.71×10^9
Ambient Air	13	2.19×10^9
H ₂ O CO ₂ Mixture	12	2.02×10^9
CH ₄	9	1.52×10^9
CO ₂	12	2.02×10^9
H ₂	11	1.85×10^9
He	9	1.52×10^9
NCG	10	1.69×10^9
Total	98	16.52×10^9

As the majority of the applications involve water vapor, the water vapor design exploration will be the primary focus of this chapter.

6.3.2 Objective and Constraint Sensitivity

Similar to Figures 6.6–6.8, the objective and constraint sensitivity study will be conducted from the standpoint of starting from an optimal design and changing an individual variable while holding the others constant to see its effect on maximum combined work transfer as well as the individual work performed by each rotor. The constraints will also be examined in this manner.

Figures 6.6–6.8 show work transfer as a function of M_1^* , M_2^* , and M_3^* , respectively for a three stage machine compressing initially saturated water vapor at 40°C. These conditions will

accordingly be used as a representative discussion for all cases as there are far too many to discuss here individually. As has already been discussed at length, each relative Mach number attaining its maximum permissible value corresponds to maximum combined work transfer thereby allowing them to be eliminated as variables. It should be noted that changing M_1^* affects the first rotor and the two downstream rotors. Variation of M_2^* affects rotors 2 and 3 with no effect on rotor 1, and changing M_3^* only affects rotor 3. As these were set for individual rotors (as opposed to only being set at the inlet of the first rotor), it is expected that the blade turning for each rotor will only affect itself and downstream rotors. Figures 6.9–6.11 show work transfer as a function of $\Delta\beta_1$, $\Delta\beta_2$, and $\Delta\beta_3$, respectively.

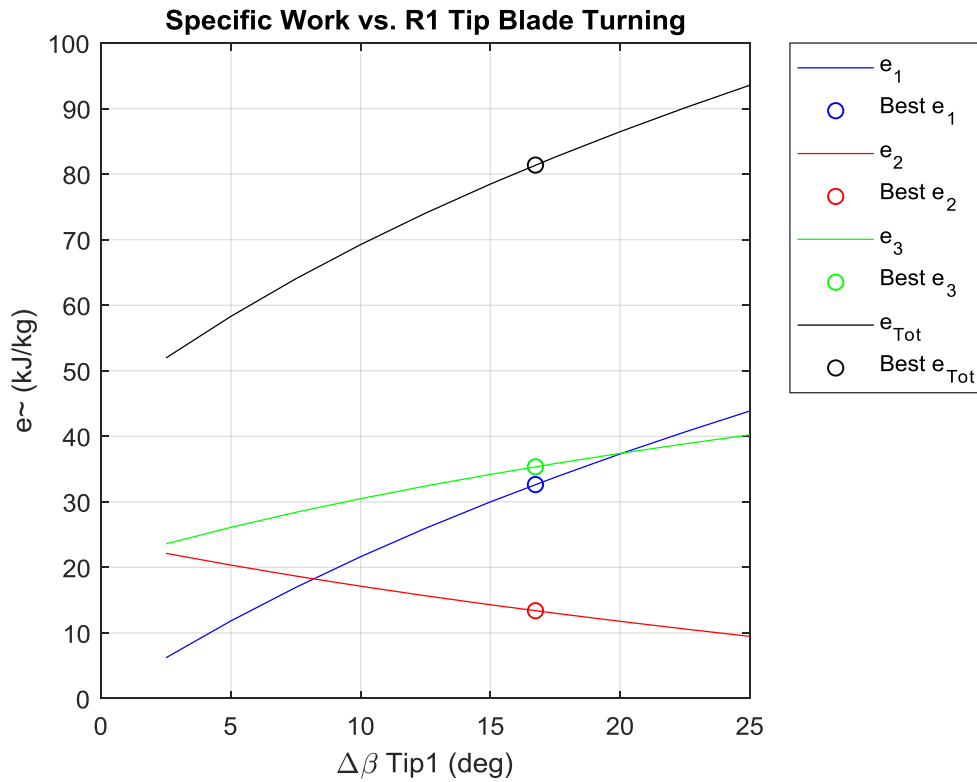


Figure 6.9. Specific Work Transfer vs. First Rotor Blade Turning

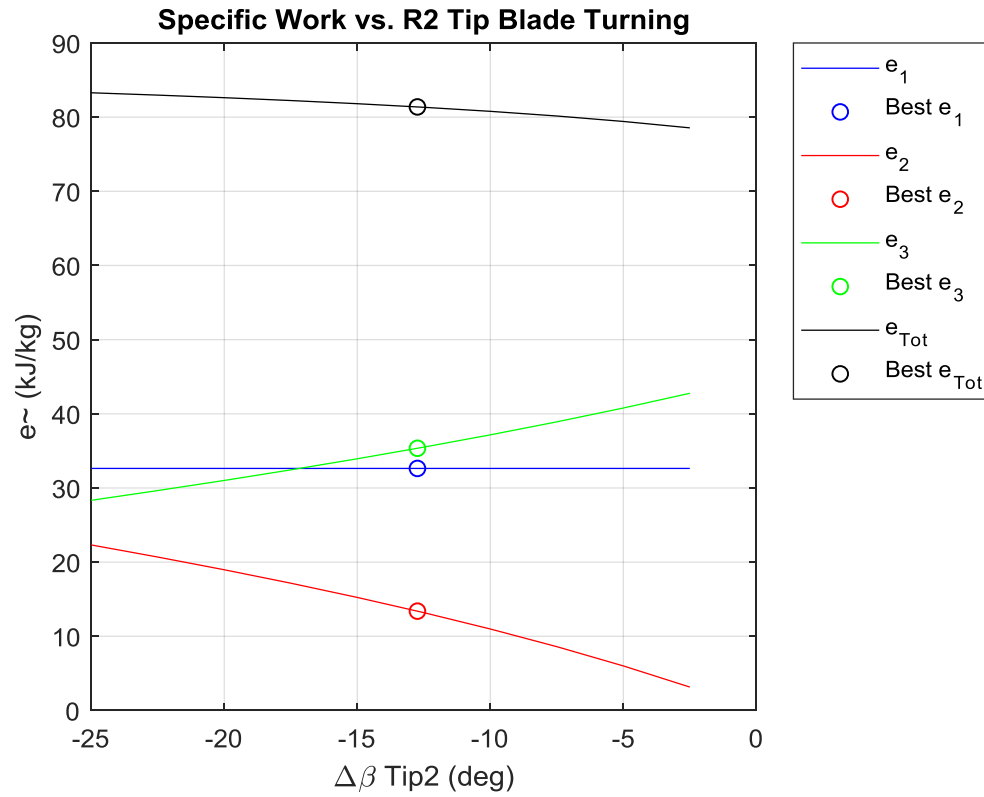


Figure 6.10. Specific Work Transfer vs. Second Rotor Blade Turning

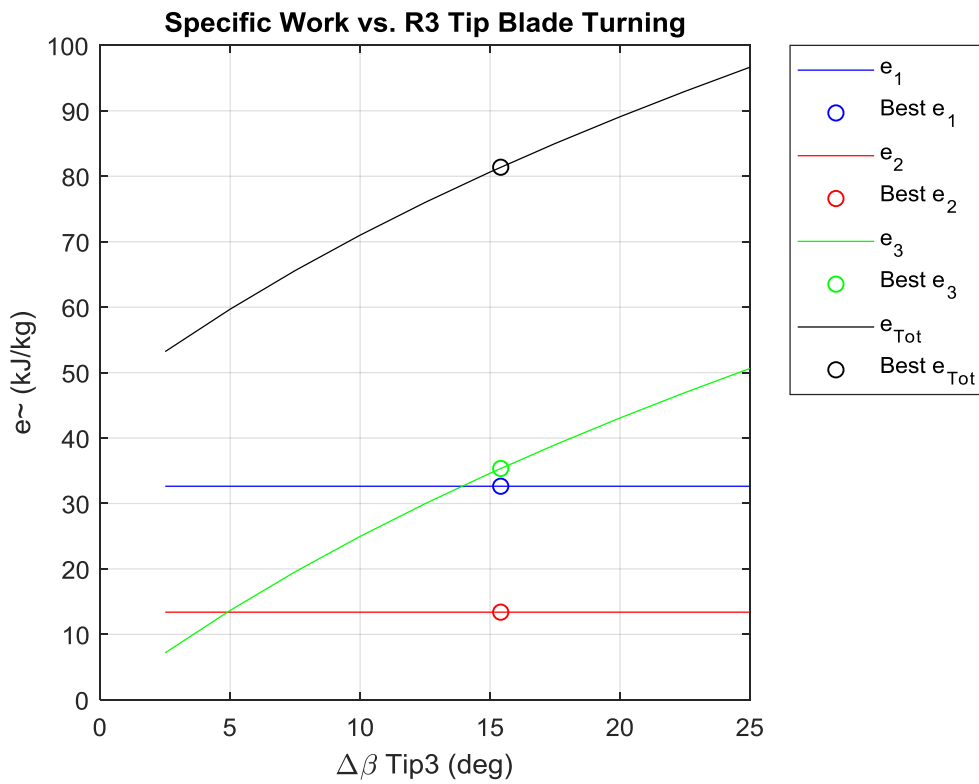


Figure 6.11. Specific Work Transfer vs. Third Rotor Blade Turning

As expected, the blade turning of a rotor has an effect on its own work transmission and an effect on the downstream rotors since it alters the tangential velocity component of the flow, but it does not have an effect on upstream rotors in terms of individual work transmission. This is also true for the constraining de Haller numbers and Diffusion Factors shown in Figures 6.12–6.17.

The limiting constraints on combined work transfer are the mean (and hub) diffusion in the first rotor, the de Haller number at the hub in the second rotor, and the hub diffusion in the third rotor. The constraints at the mean line are slightly relaxed in order not to reject a slightly infeasible design as the simulation results from Chapters 4 and 5 indicated that flow at the mean line is typically healthy even if the calculated de Haller number and Diffusion Factor are well beyond their respective limits. In this case the mean Diffusion Factor in the first rotor is just 3.3% beyond the limit and it was the hub diffusion attaining the limiting value that set this particular design's best-point.

It must be noted that even-numbered stages are given a negative turning angle in order to distinguish rotation and swirl direction between rotors and counter-rotors in the Matlab code. In all cases, the de Haller numbers decrease and the Diffusion Factors increase with increasing blade turning magnitude.

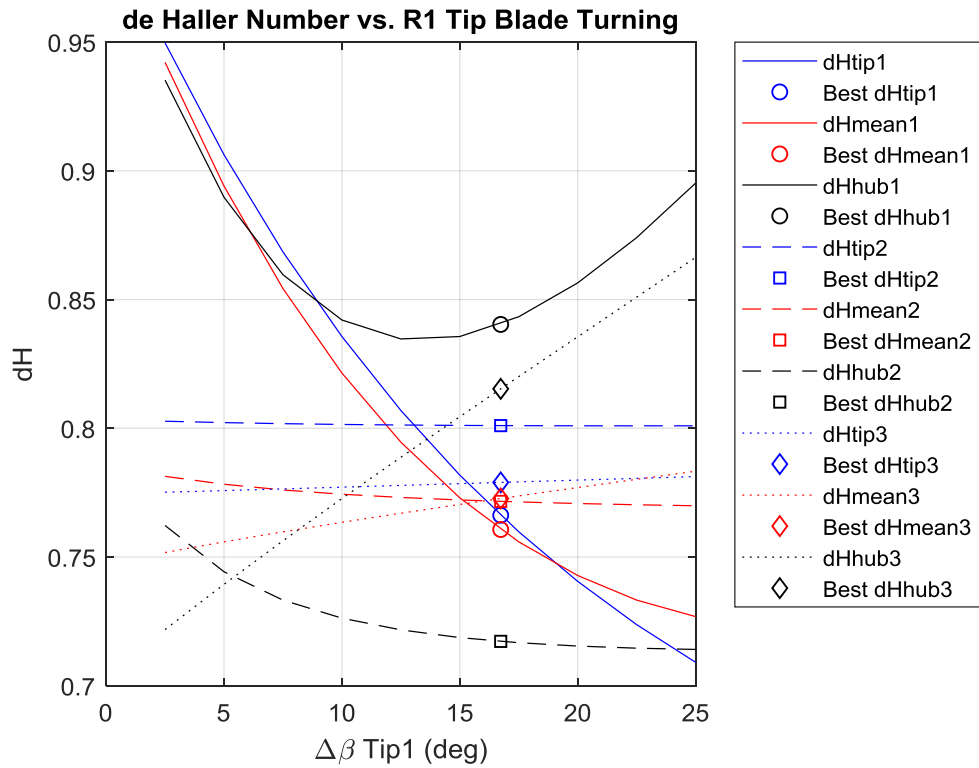


Figure 6.12. de Haller Numbers vs. First Rotor Blade Turning

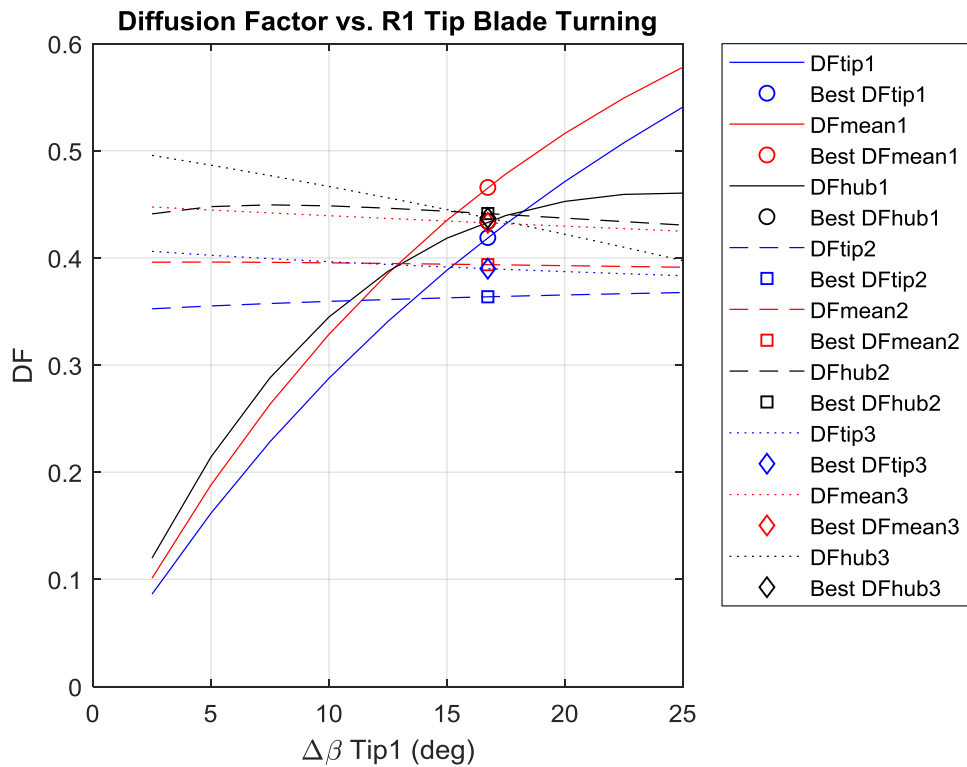


Figure 6.13. Diffusion Factors vs. First Rotor Blade Turning

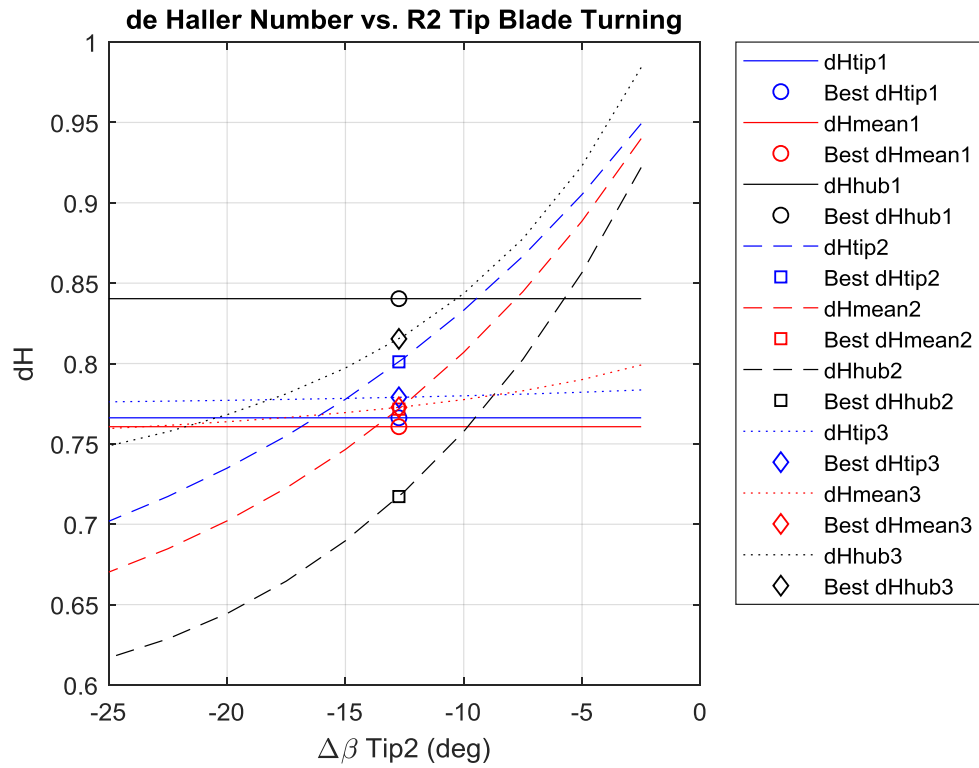


Figure 6.14. de Haller Numbers vs. Second Rotor Blade Turning

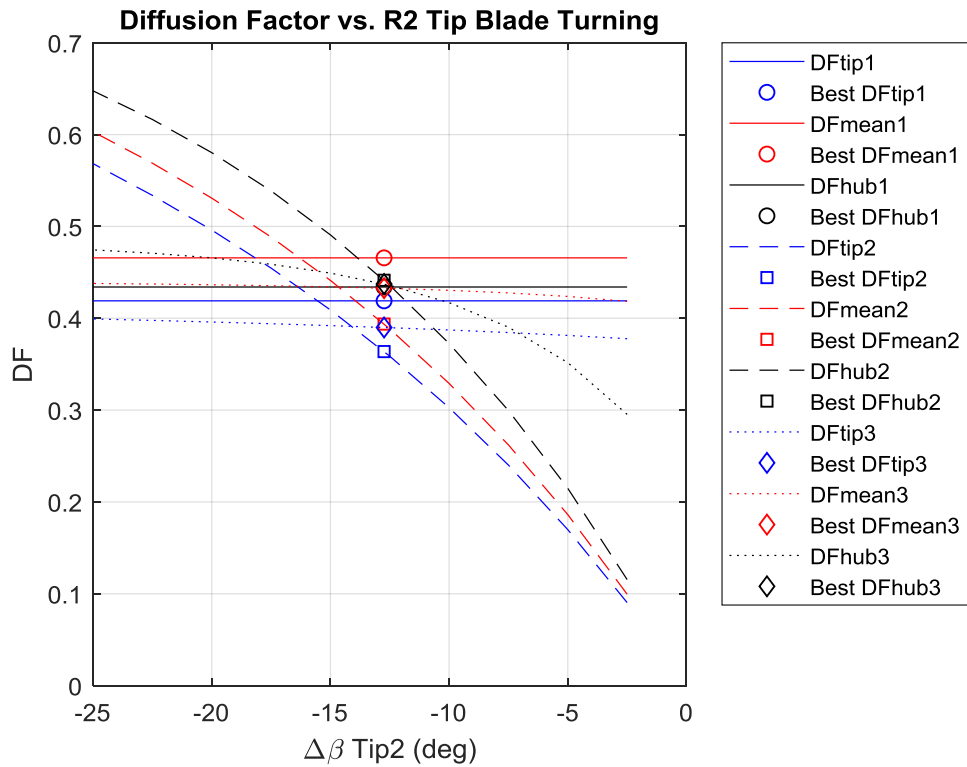


Figure 6.15. Diffusion Factors vs. Second Rotor Blade Turning

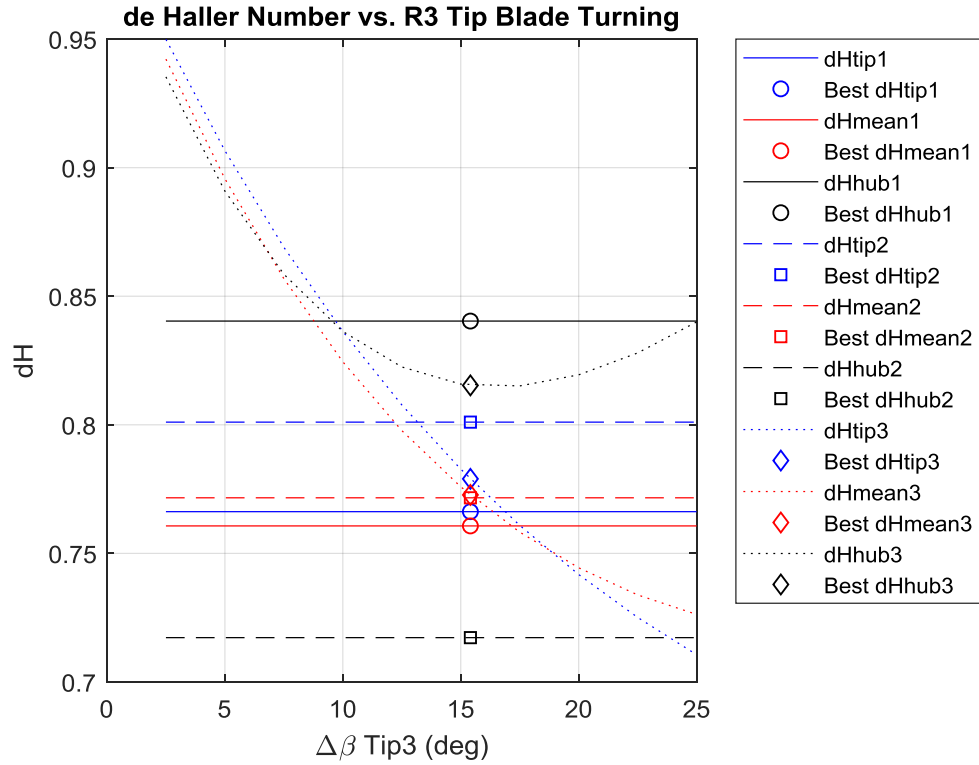


Figure 6.16. de Haller Numbers vs. Second Rotor Blade Turning

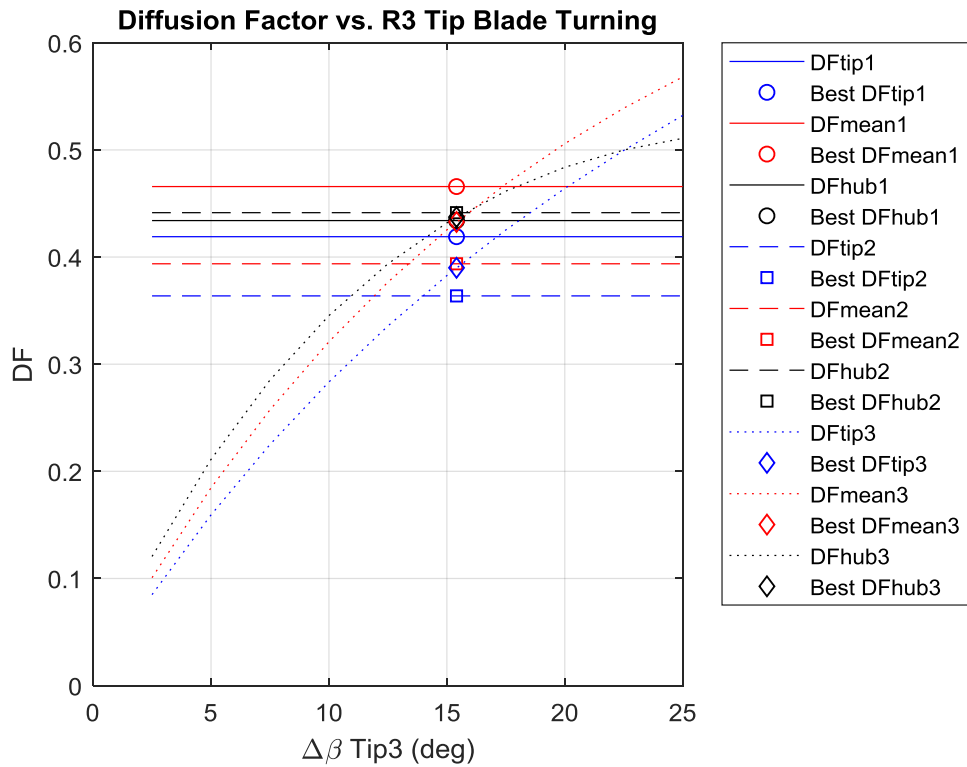


Figure 6.17. Diffusion Factors vs. Second Rotor Blade Turning

Relative Mach at hub and mean do change with varying blade turning, but the hub relative Mach number at the inlet is generally lower than at the tip, which is of course maximized. Tip, mean, and hub relative Mach numbers all decrease across each rotor towards the outlet as the flow follows the blades.

Absolute inlet Mach numbers are unaffected by their own blade turning as the turning happens downstream from the inlet. However subsequent downstream absolute inlet Mach numbers are affected by upstream blade turning as they are by definition considered to be the same as the outlet of the previous rotor (no change of reference frame in the absolute system). Outlet absolute Mach numbers at the hub are very near their limiting value in both the first and third stages, coinciding with Diffusion Factors at the hub of both of these rotors attaining their limiting value. Figures 6.18–6.20 show absolute Mach numbers plotted against blade turning within the same rotor. The behavior of in the first and third rotors is very similar.

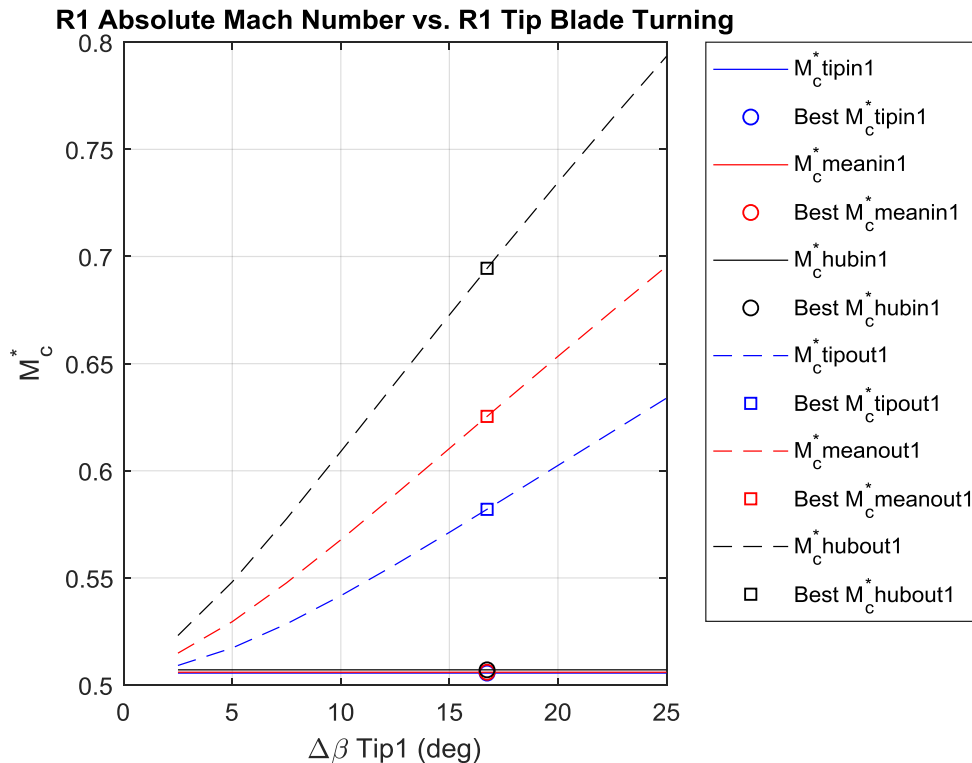


Figure 6.18. Absolute Mach Number vs. Blade Turning in the First Rotor

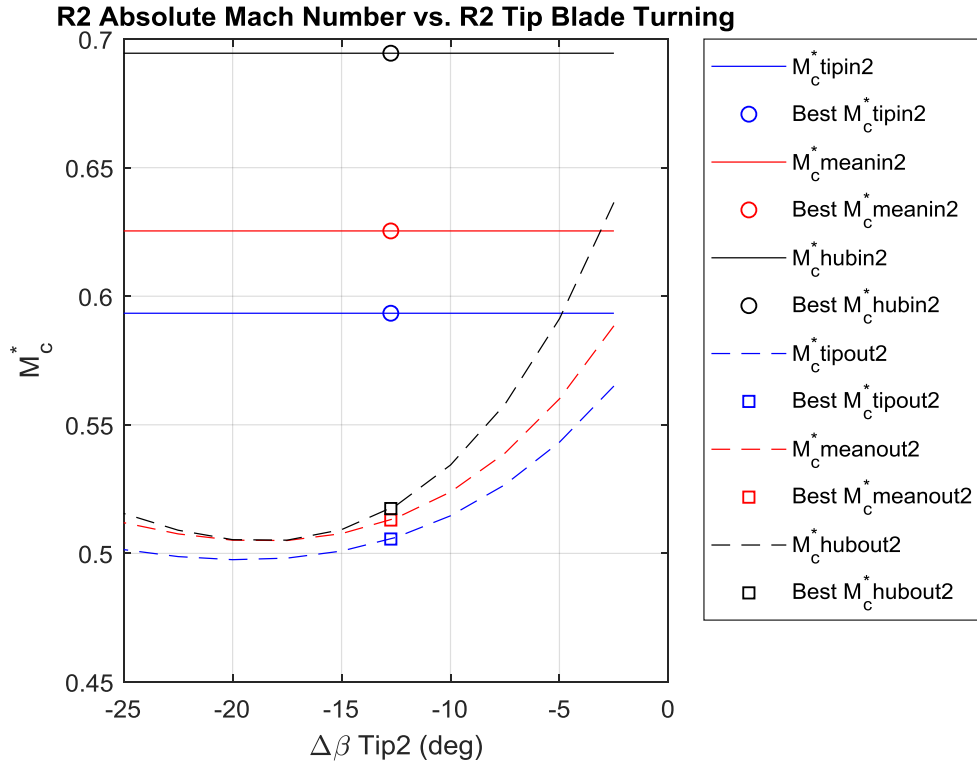


Figure 6.19. Absolute Mach Number vs. Blade Turning in the Second Rotor

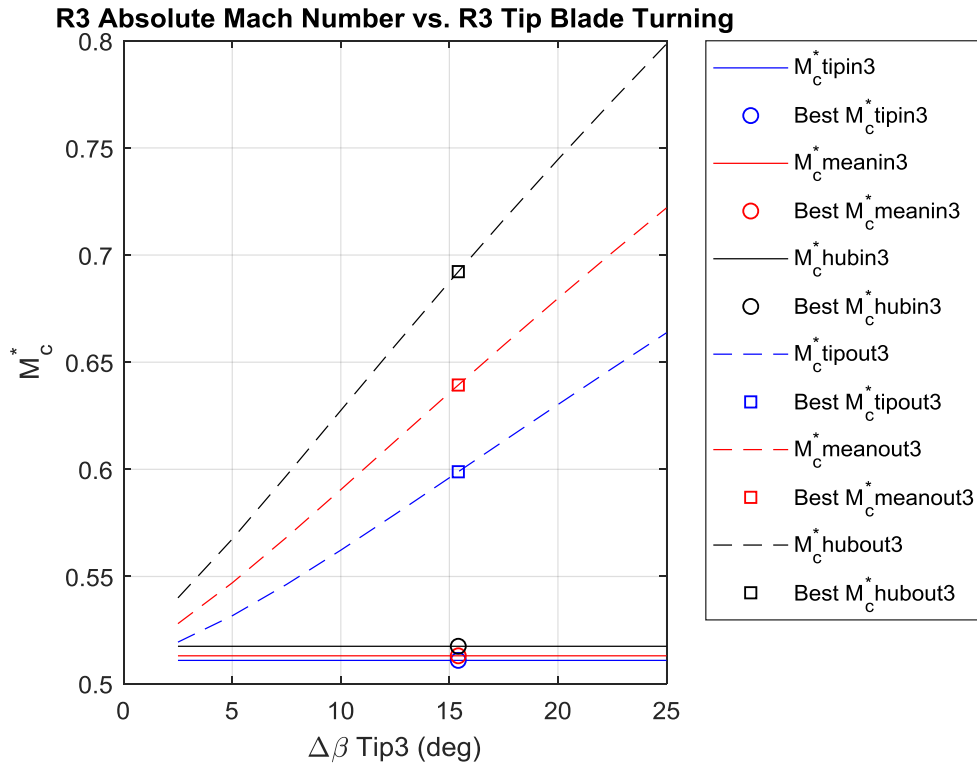


Figure 6.20. Absolute Mach Number vs. Blade Turning in the Third Rotor

Figures 6.21–6.23 plot work transmission against the primary inlet variables α , M_c^* , and r_h/r_t , respectively.

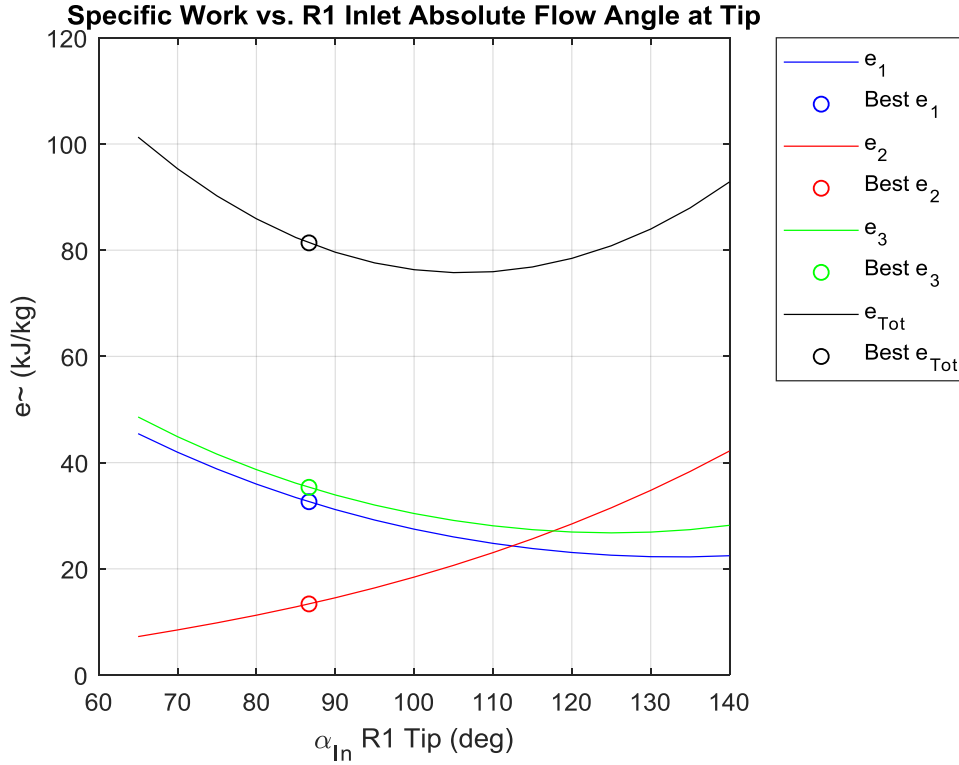


Figure 6.21. Work Transfer vs. Absolute Inlet Flow Angle

Around the best combined work transmission, the first and third rotor contribution decrease with increasing α where as the second rotor behaves in opposite fashion. All rotors exhibit decreased work transfer with increasing inlet Mach number. Since the relative inlet Mach number is fixed at the maximum value of 0.8, the relative velocity magnitude is fixed

The hub/tip ratio does not affect the work transfer of any rotor. It does however have a large influence on the constraint values since the tangential velocity components vary with radius. The de Haller number and Diffusion Factor are plotted against hub/tip ratio in Figures 6.24 and 6.25, respectively. Additional plots of constraint quantities against first rotor inlet variables can be found in Appendix A3.

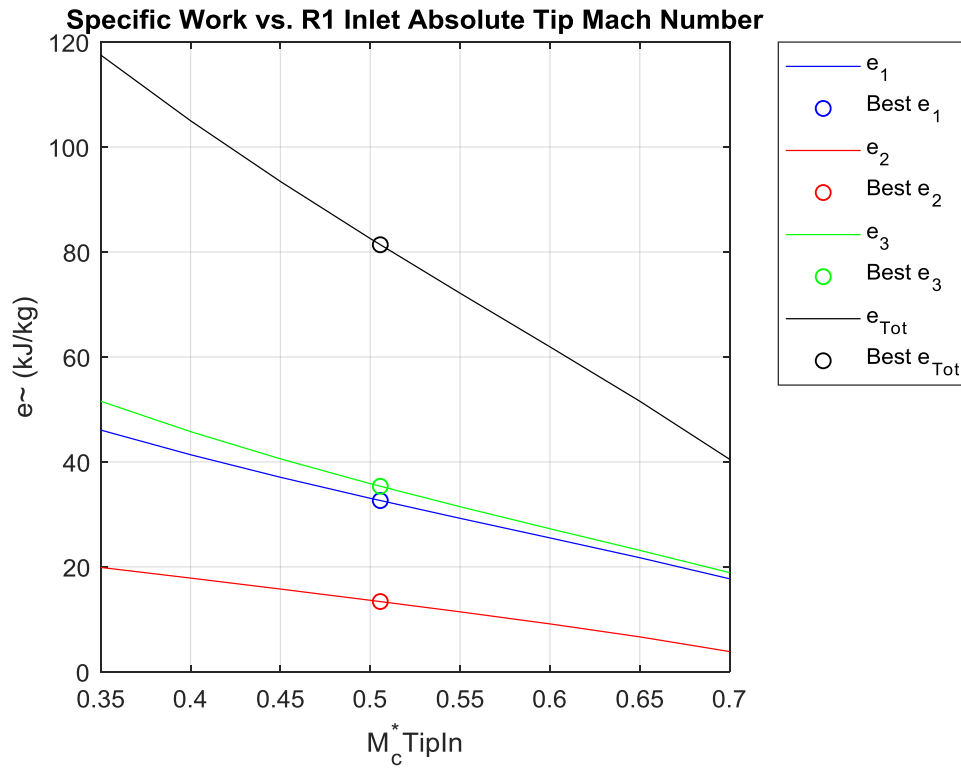


Figure 6.22. Work Transfer vs. Absolute Inlet Critical Mach Number

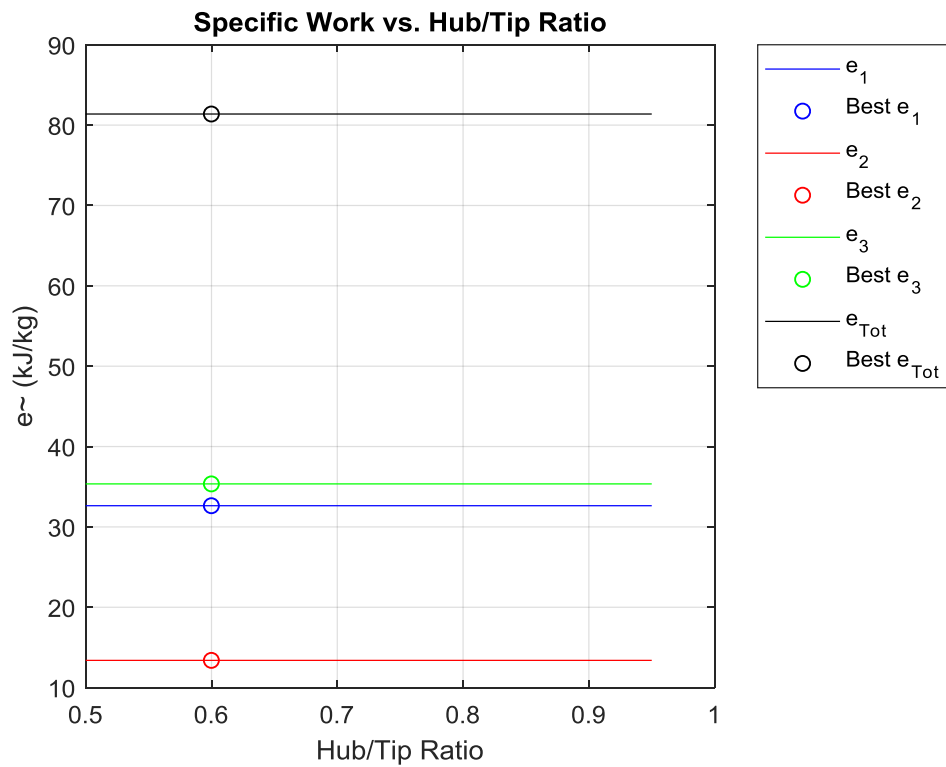


Figure 6.23. Work Transfer vs. Inlet Hub/Tip Ratio

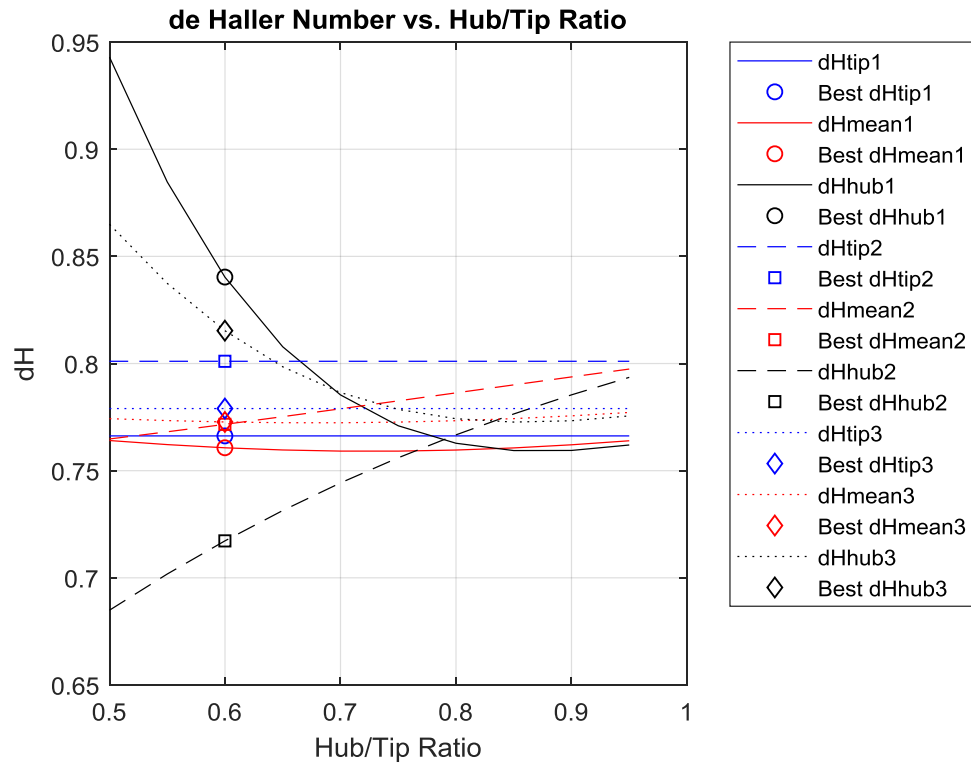


Figure 6.24. de Haller Number vs. Inlet Hub/Tip Ratio

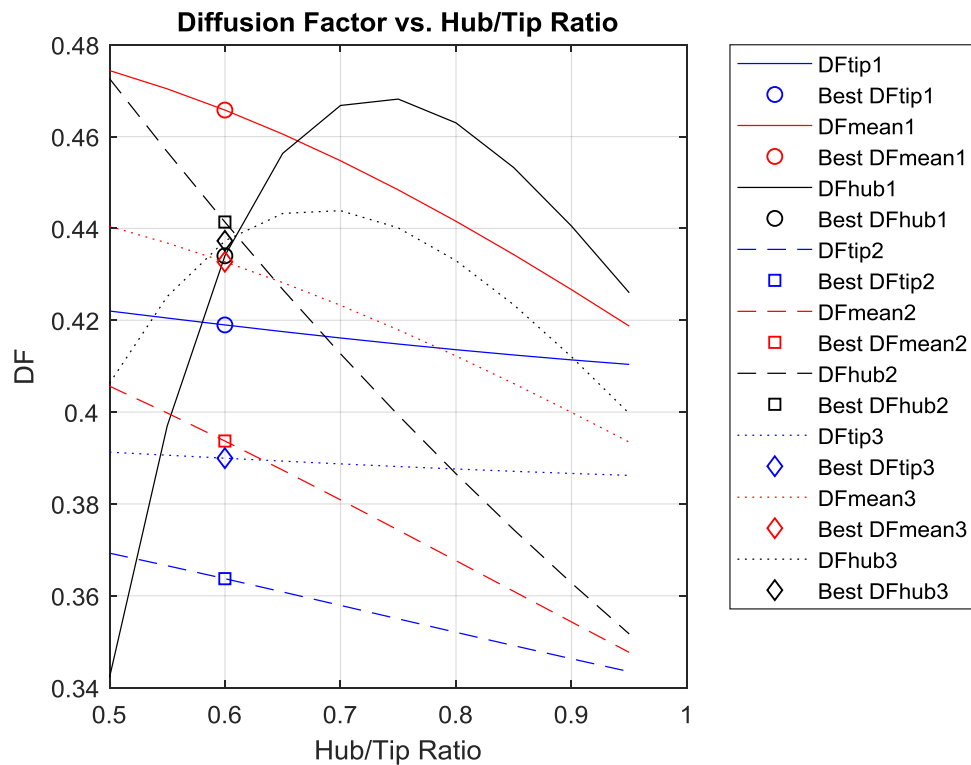


Figure 6.25. Diffusion Factor vs. Inlet Hub/Tip Ratio

6.4 Best Design Conditions

Each best-point configuration depending on the number of stages used and the saturation temperature at the inlet to the first rotor show the same sort of sensitivity to design variables as discussed in Section 6.3.2. However, the number of stages used, particularly if it is an odd-numbered configuration vs. an even-numbered configuration, as well as where the saturation temperature lies within the range of vacuum pressures studied (below vs. above approximately 55°C) has a large effect on the exact nature of the sensitivity. There are far too many cases to discuss individually, and so this section presents the behavior the best-points relative to one another.

The work transfer from each rotor is normalized by the average of the combined work transfer for the number of stages used, i.e.

$$\tilde{e}_{i,norm} = \frac{\tilde{e}_i}{\frac{1}{N_s} \sum_{i=1}^{N_s} \tilde{e}_i} \quad (6-3)$$

in order to visualize the contribution each stage makes to the combined maximum work transfer at a particular set of inlet conditions. The following figures plot actual specific work transfer, normalized specific work transfer, the corresponding design input variable values, and the constraint values against saturation temperature at the first rotor inlet. The plots of relative Mach numbers at the hub are not included as they do not reach their limiting values other than when hub/tip ratio is near 1 at higher temperature as there is little difference between tip (maximized) and hub velocities.

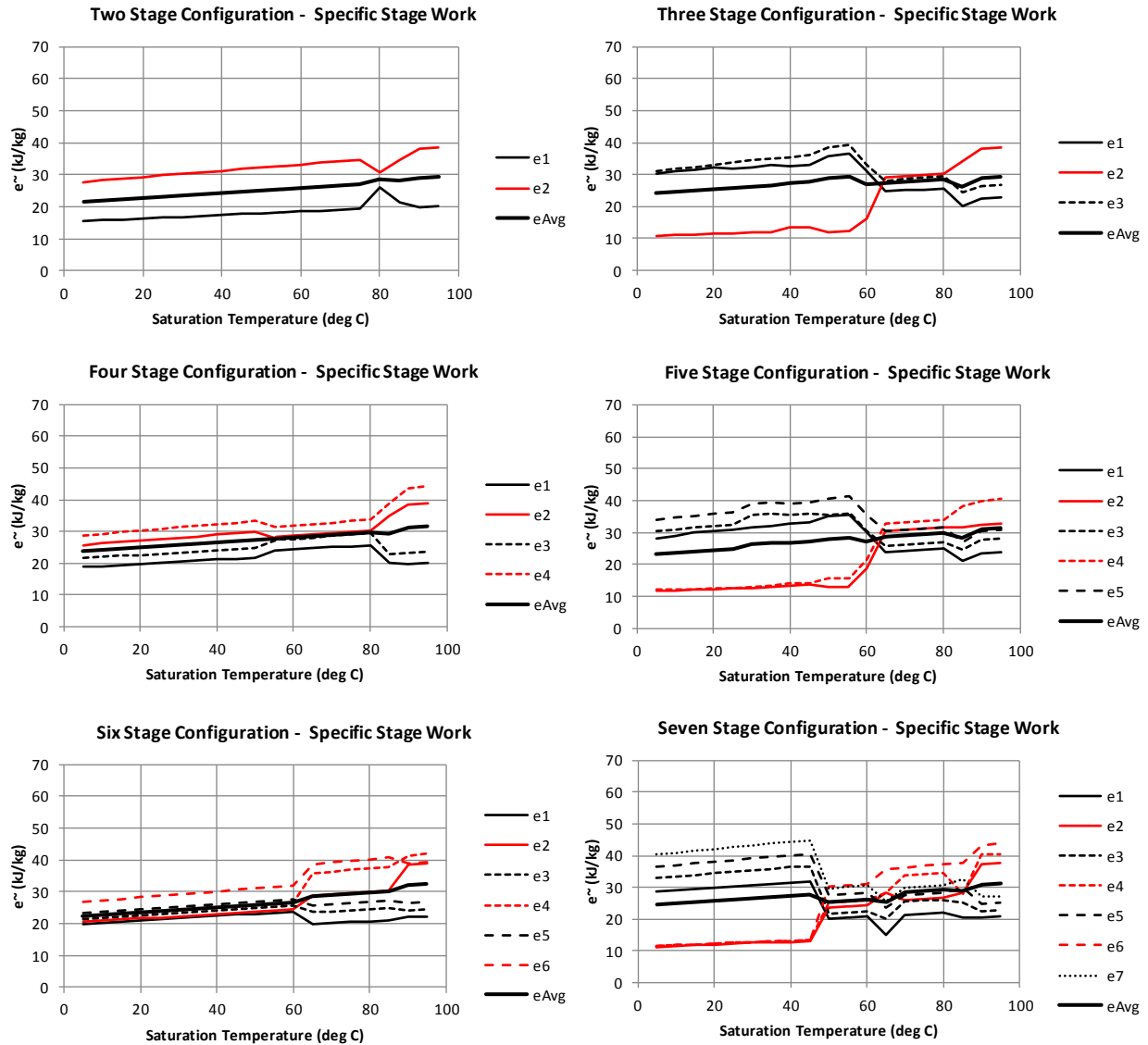


Figure 6.26. Stage Work Transfer vs. Saturation Temperature at First Rotor Inlet

As is to be expected, the maximum combined work transfer generally increases with saturation temperature as work transfer is proportional to inlet temperature per Equation (2-93).

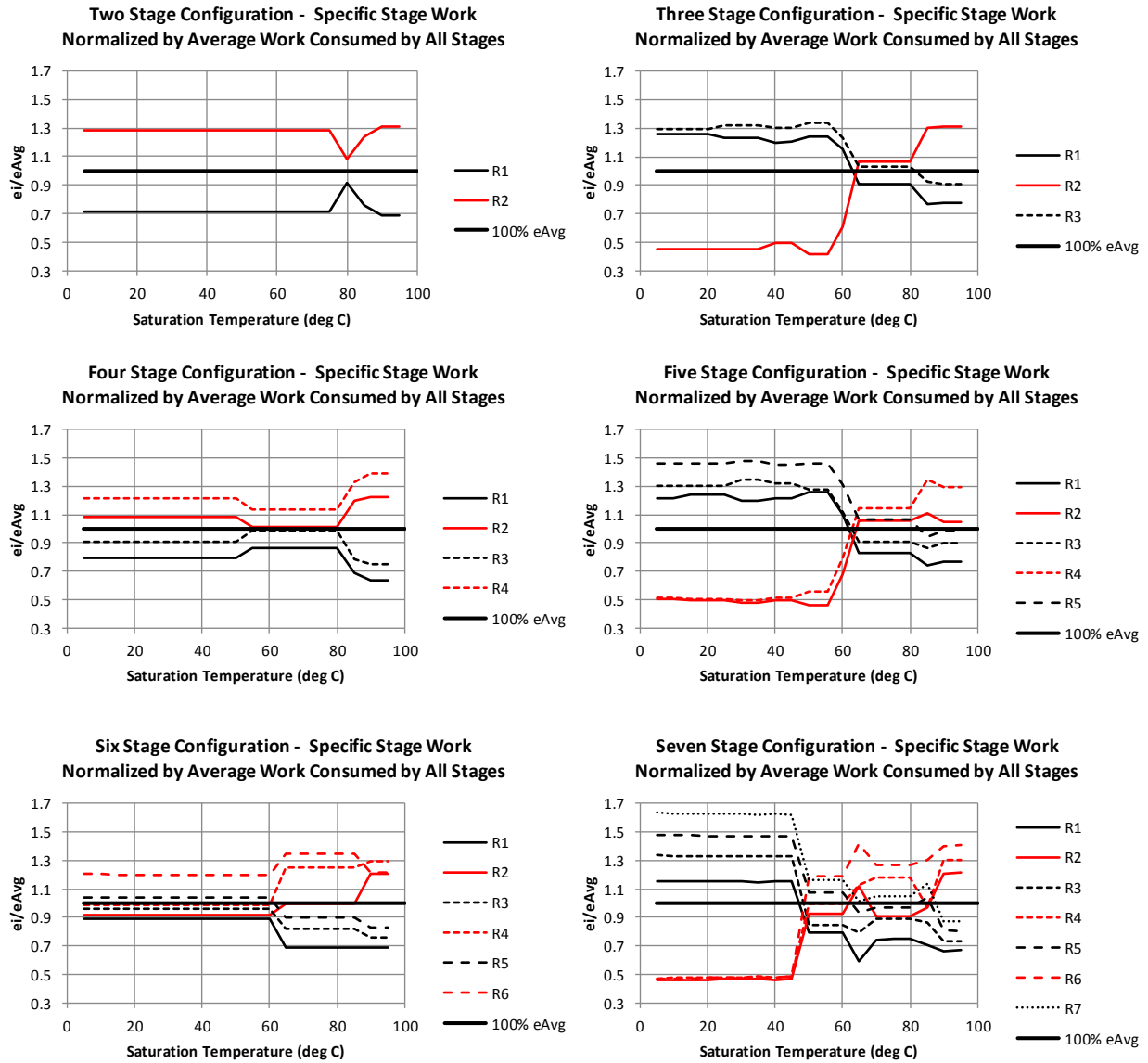


Figure 6.27. Normalized Stage Work Transfer vs. Saturation Temperature at First Rotor Inlet

Figure 6.27 indicates that even N_s show similar behavior to one another, as do odd N_s when transferring maximum combined work. For an even N_s , the even-numbered rotors contribute the majority of the total work (aside from the second and fourth stages in a 6 stage machine at saturation temperatures below 60°C). For odd N_s , the odd-numbered rotors are responsible for transferring the majority of the combined total up to saturation temperatures of 65°C, at which point the even-numbered rotors are found to contribute the majority of work transfer. However for the 7 stage configuration, this switch occurs at 50°C. This behavior is

closely mimicked by the corresponding tip speed ratios for all cases in that a higher tip speed ratio corresponds to higher work transfer in the downstream rotor of the pair in question (e.g. $u_3/u_2 > 1$ corresponds to $\tilde{e}_3 > \tilde{e}_{avg}$ and $\tilde{e}_2 < \tilde{e}_{avg}$).

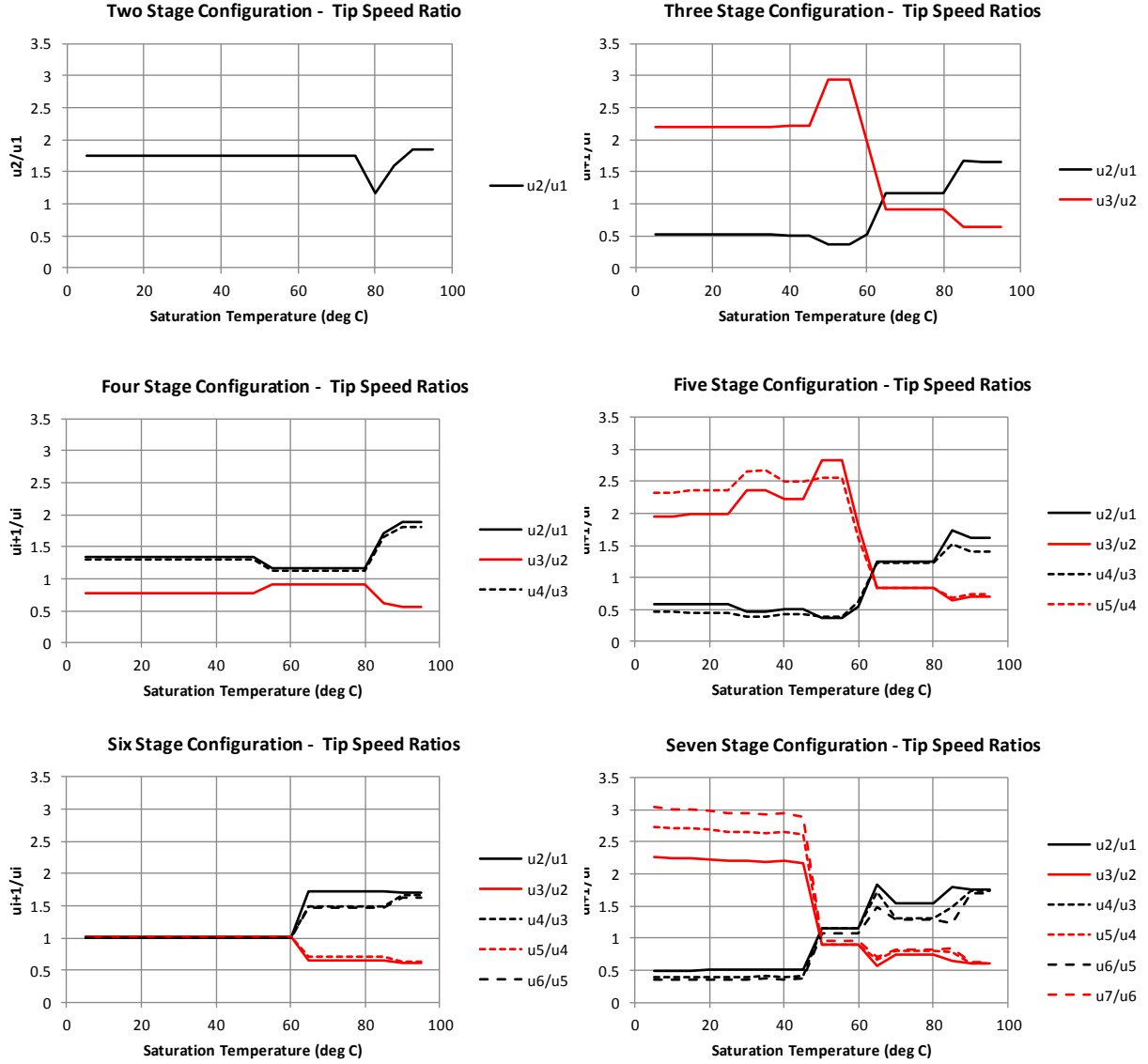


Figure 6.28. Tip Speed Ratio vs. Saturation Temperature at First Rotor Inlet

The absolute inlet flow angle at the first rotor inlet is greater than 90° for all even N_s at all temperatures, indicating that the IGV is providing counter-swirl at the inlet to the first rotor. Odd N_s also exhibit this behavior, but only for inlet saturation temperatures greater than 60°C . Below

60°C, the machines with odd N_s are provided with pre-swirl (absolute tangential velocity component in the same rotation direction as the first rotor).

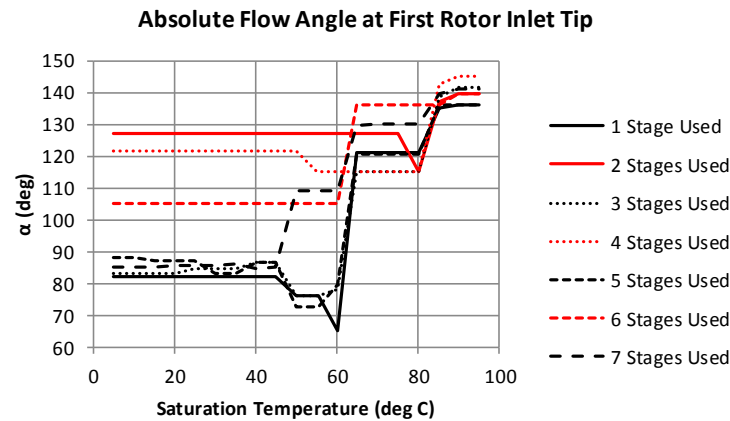


Figure 6.29. Absolute First Rotor Inlet Flow Angle vs. Saturation Temperature at First Rotor Inlet

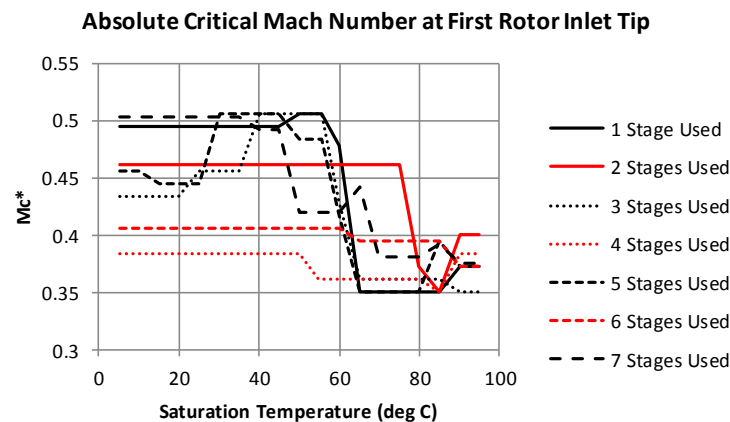


Figure 6.30. First Rotor Absolute Critical Mach Number vs. Saturation Temperature at First Rotor Inlet

Trends of absolute critical Mach number at the inlet are difficult to discern from Figure 6.30, although it can be said that even N_s generally sees decreasing Mach number with increasing saturation temperature, while odd N_s sees both increases and decreases in Mach number with increasing saturation temperature. In all cases, the Mach numbers are higher at the low end of the saturation temperature range studied, and lower at the high end of the temperature range.

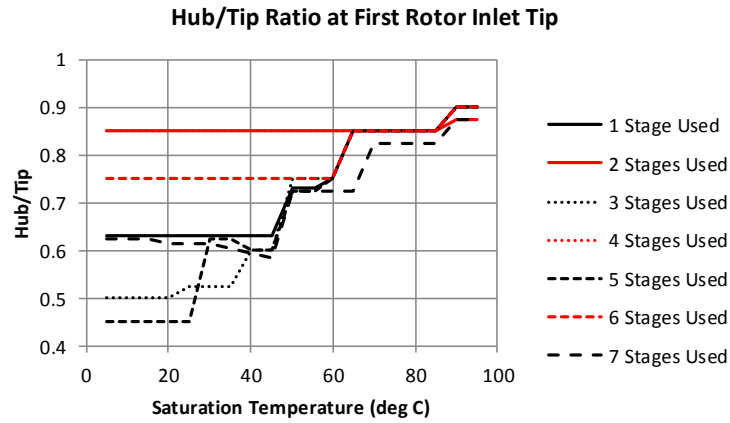


Figure 6.31. Hub/Tip Ratio at First Rotor Inlet vs. Saturation Temperature at First Rotor Inlet

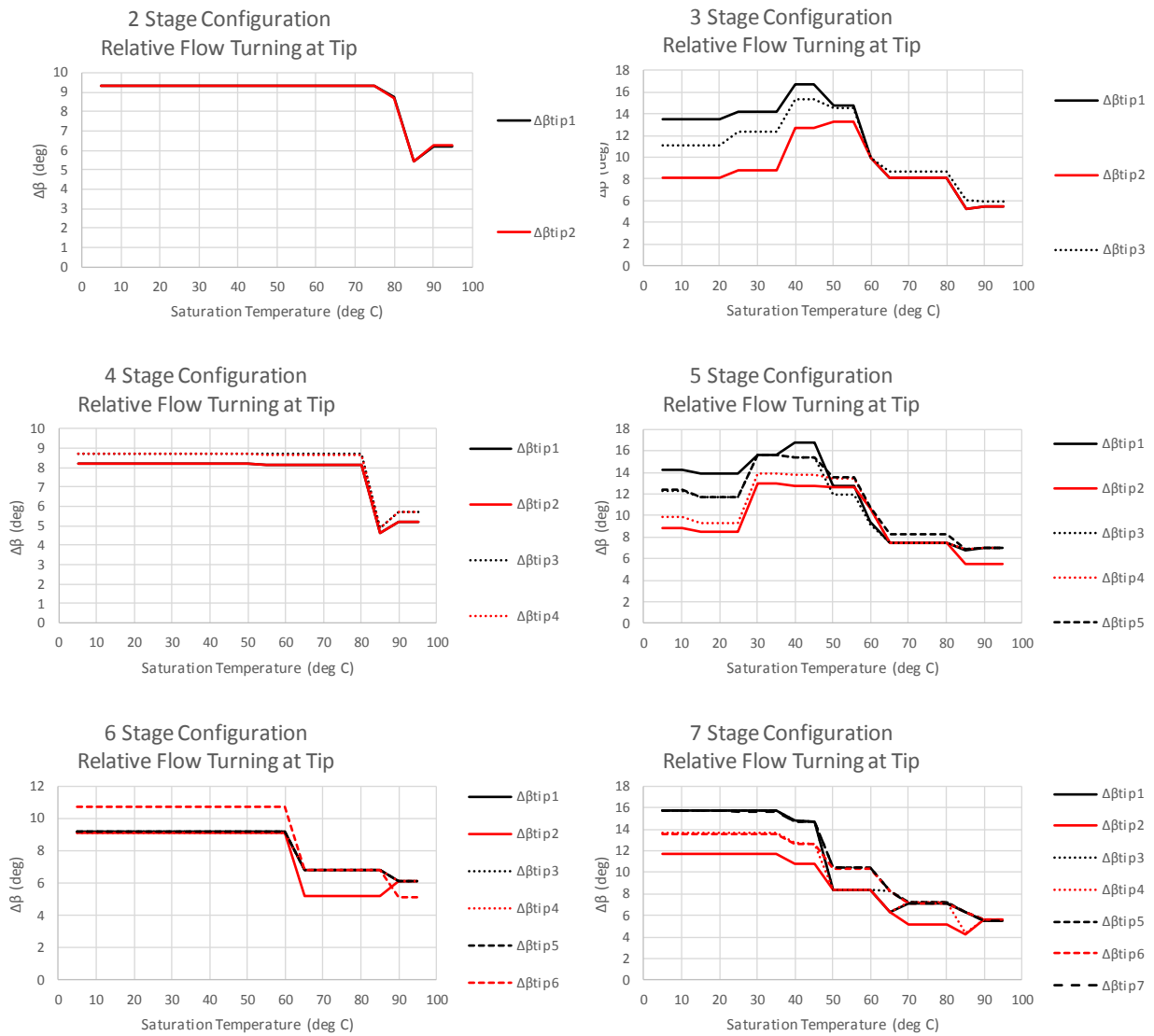


Figure 6.32. Blade Turning Angle of Each Rotor vs. Saturation Temperature at First Rotor Inlet

For all N_s , the hub/tip ratio is initially constant with saturation temperature, although this is true over a wider range of temperatures for even N_s . Hub/tip ratio is constant for $N_s = 2$ and 4 between 5°C and 85°C. $N_s = 6$ machines see increased hub/tip ratio beyond 60°C. Odd-numbered N_s see an increase at temperatures as low as 20°C for $N_s = 3$. In all cases, the hub/tip ratio ultimately increases with saturation temperature resulting in ratios between 0.87 and 0.90 for all machines regardless of the number of stages used. This is to be expected in the context of this study as the mass flow rate is held constant. As the saturation temperature increases, the saturation pressure increases as shown in Figure 6.33.

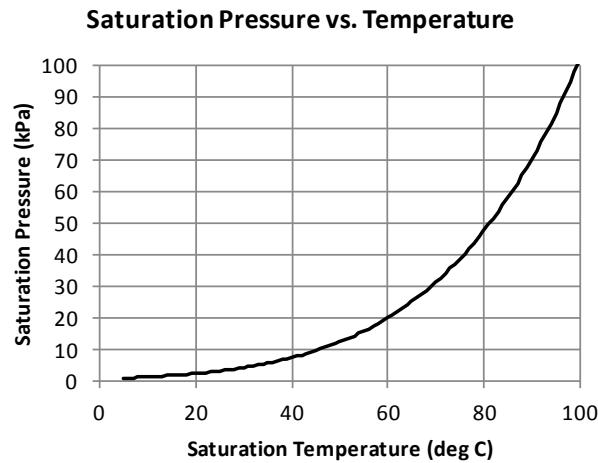


Figure 6.33. Saturation Pressure vs. Saturation Temperature

The combination of increasing saturation temperature and pressure necessitates an increase in density. This requires reduced cross sectional area, which requires larger hub/tip ratio. With this in mind, it is unexpected that the maximum combined work transfer for $N_s = 2$ and $N_s = 4$ machines have the same inlet hub/tip ratio over such a wide range of saturation temperatures.

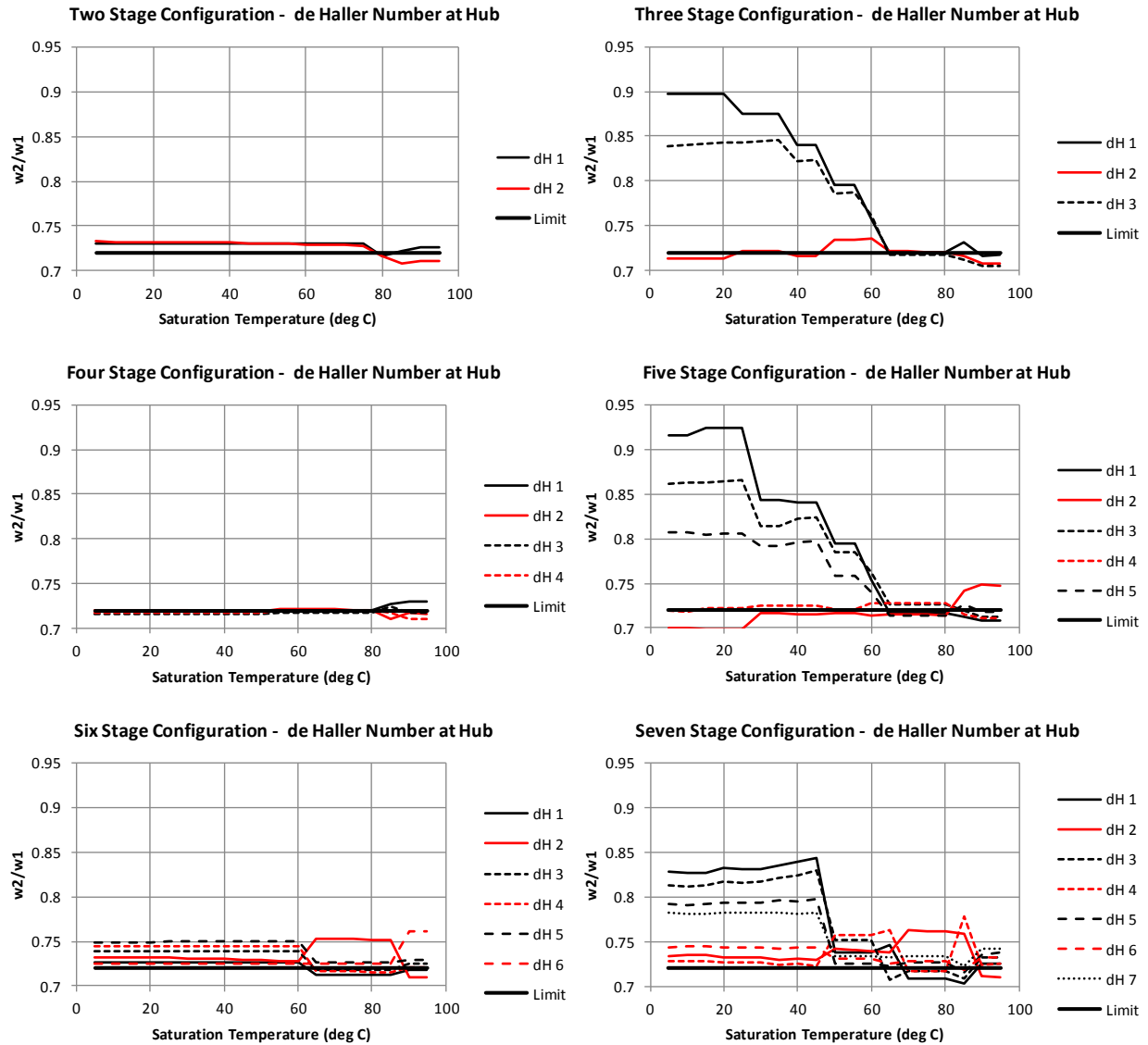


Figure 6.34. de Haller Number for Each Rotor vs. Saturation Temperature at First Rotor Inlet

The de Haller numbers at the hub are near their lowest permissible value for all N_s , particularly in the 2 and 4 stage cases for all temperatures. For odd N_s , only the even-numbered rotors are near the limit for all temperatures. The de Haller numbers for odd-numbered rotors only approach the limiting value for saturation temperatures beyond 65°C for 3 and 5 stages, and beyond 50°C for 7 stages. The de Haller numbers at the tip radius do not approach the limiting value and so are not shown here. FVB rotors are generally limited by conditions at the hub [73].

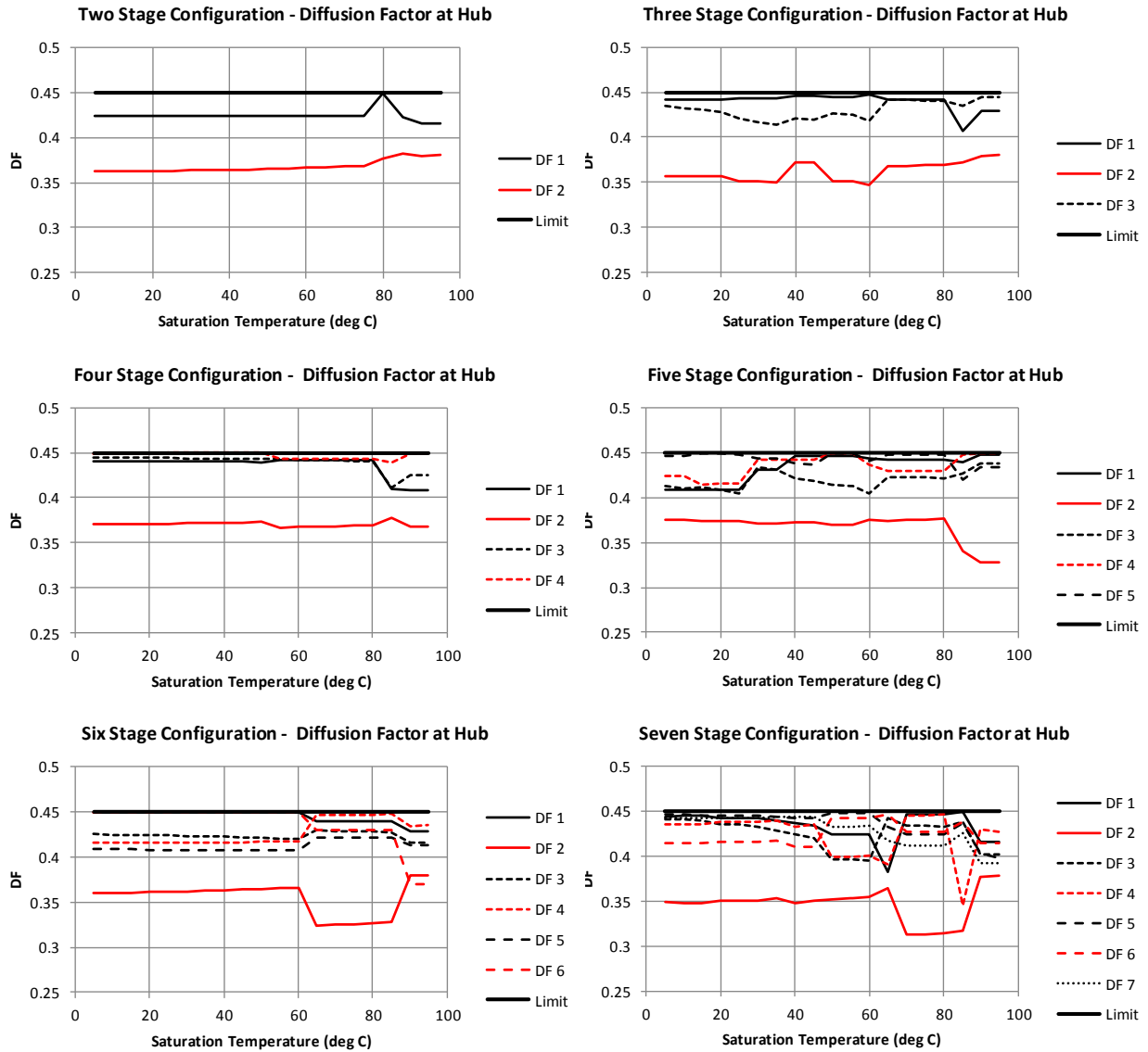


Figure 6.35. Diffusion Factor for Each Rotor vs. Saturation Temperature at First Rotor Inlet

Figure 6.35 indicates that for machines with any number of stages, the Diffusion Factor at the hub of each rotor will be a limiting constraint at multiple saturation temperatures with the lone exception of the second stage, which stays relatively constant for saturation temperatures below 60°C in all configurations.

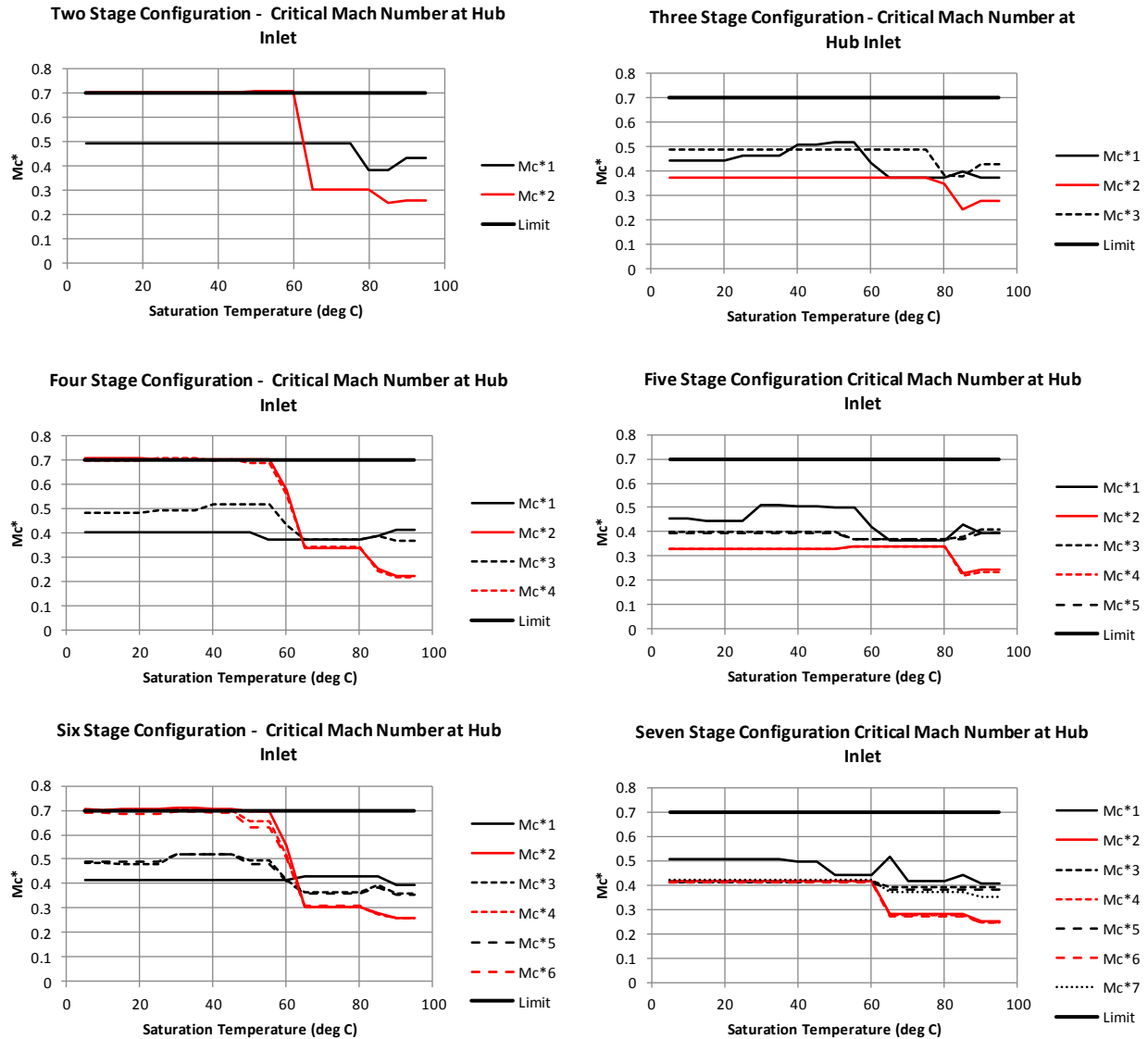


Figure 6.36. Critical Absolute Mach Number at the Hub Inlet for Each Rotor vs. Saturation Temperature at First Rotor Inlet

For saturation temperatures below 60°C, Figure 6.36 shows that for even N_s , even-numbered rotors are limited absolute critical Mach number at the hub inlet. Conversely, Figure 6.37 indicates that for the same temperature range odd-numbered rotors in odd N_s configurations are limited by the absolute Mach number at the hub outlet. For odd N_s , the outlet absolute hub Mach number in odd-numbered rotors moves away from the limit at the same temperature that the hub de Haller numbers for odd numbered rotors reach their respective limit.

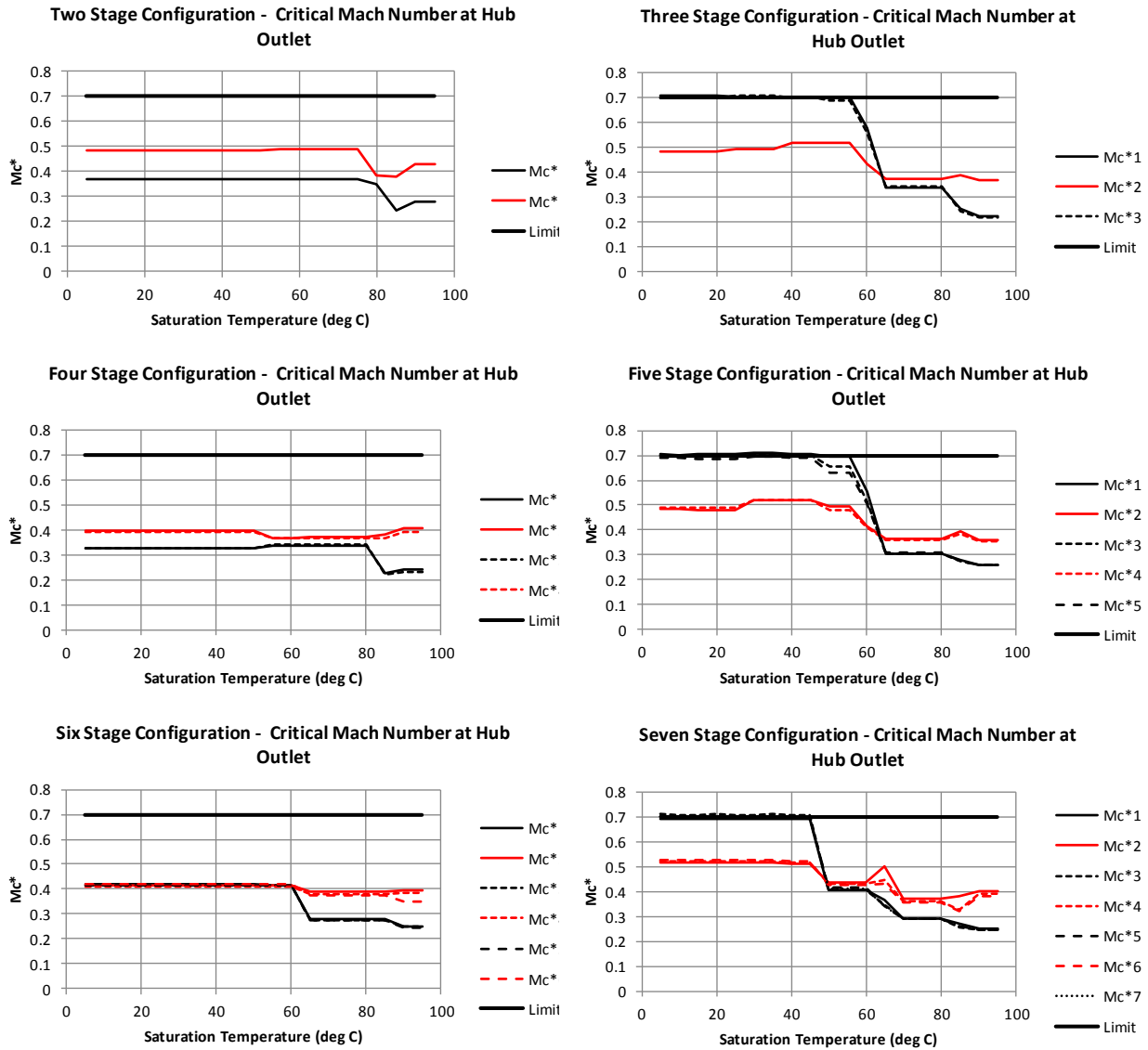


Figure 6.37. Critical Absolute Mach Number at the Hub Outlet for Each Rotor vs. Saturation Temperature at First Rotor Inlet

There is no equally clear changeover of reached constraint limits for even N_s , although for $N_s = 7$, all hub Diffusion Factors (aside from rotor 2) move closer to the limiting value at the same temperature the hub inlet Mach number moves away from its limit for even-numbered rotors. Aside from the $N_s = 2$ case, there is at least one rotor with the hub Diffusion Factor at its limiting value for all saturation temperatures.

It is typical to discuss axial compressors in terms of the dimensionless flow coefficient, ϕ , the ratio of axial velocity to tip speed, and the blade loading coefficient, ψ , defined as the ratio of specific work to the tip speed squared (a measure of kinetic energy available from the blades purely from tangential velocity) defined as

$$\phi = \frac{c_m}{u} \quad (6-4)$$

and

$$\psi = \frac{\tilde{e}}{u^2} \quad (6-5)$$

Equation (6-4) uses axial velocity at the rotor inlet. Typical flow coefficient range is between 0.3 and 0.9 for axial compressors [72], and typical blade loading coefficients for axial compressors are limited to 0.4 [54]. Figure 6.38 plots the flow coefficients corresponding to maximum combined work transfer, and Figure 6.39 plots the corresponding blade loading coefficients.

Even N_s have ϕ within the usual range at all saturation temperatures while odd N_s see even-numbered rotors with flow coefficients well above the usual range for saturation temperatures in the lower half to two-thirds of the saturation temperature range. The same is true of the blade loading coefficient.

These trends also closely resemble those of the tip speed ratios in Figure 6.28. The largest values in the tip speed ratio plots for odd N_s correspond to downstream odd-numbered rotors rotating from between 2 to 3 times the tip speed magnitude of the upstream even-numbered rotors. This then corresponds to the very large flow and blade loading coefficients of even-numbered rotors (e.g. large u_3/u_2 sees large ϕ_2 and ψ_2).

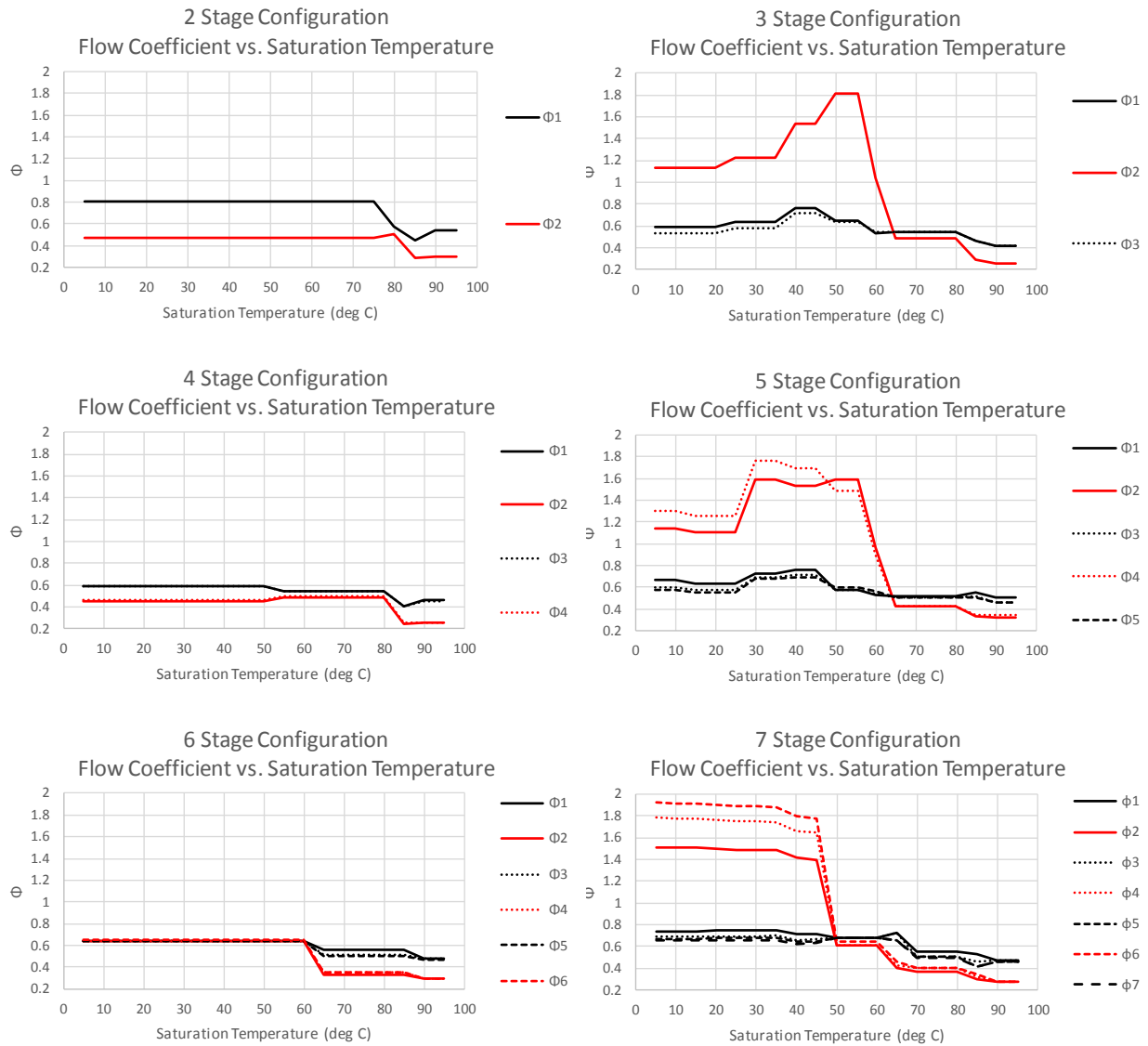


Figure 6.38. Flow Coefficient vs. Saturation Temperature at First Rotor Inlet

The even-numbered rotors in odd N_s machines exhibiting larger than usual flow and blade loading coefficients is to be expected for at least some of the best-point counter-rotating designs. When viewed from the conventional multistage axial compressor context of rotor-stator pairs with unidirectional rotation, the even-numbered rotors in the odd N_s machines are where the stators would otherwise be. Considering a hypothetical stator when using Equation (6-4), this amounts to an infinite flow coefficient as there is zero tip speed in a stationary blade set.

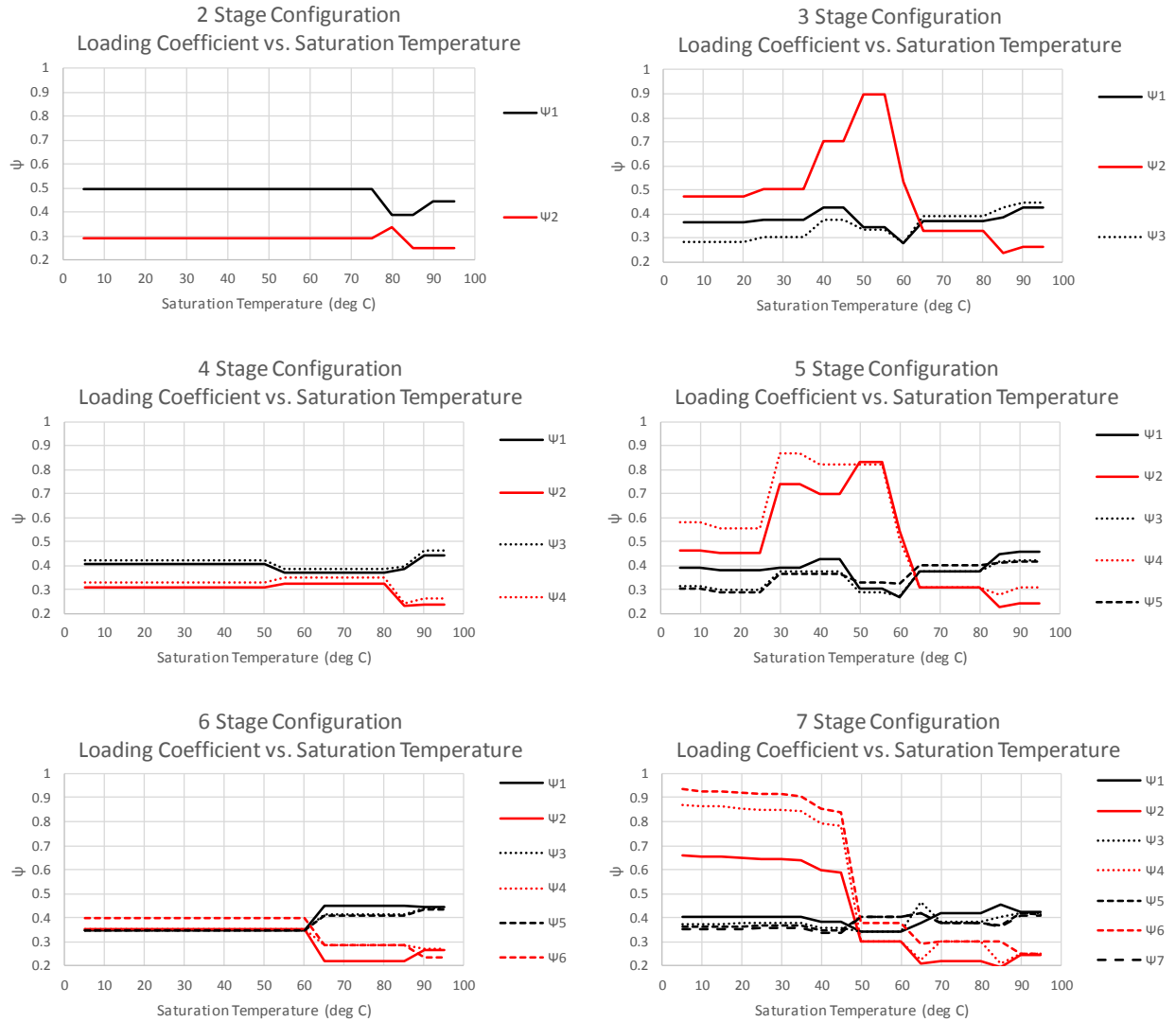


Figure 6.39. Blade Loading Coefficient vs. Saturation Temperature at First Rotor Inlet

With regard to Equation (6-5), the hypothetical stator is meaningless as a zero-divided-by-zero situation arises since there is again zero tip speed for a stator, and as such, stators are unable to perform work and create non-zero $\tilde{e}$.

However, when viewed from the counter-rotor standpoint, flow coefficients approaching 2 and blade loading coefficients up to above 0.9 appear modest relative to an infinity or undefined situation, respectively. Even though these counter-rotors are rotating relatively slowly (large tip speed ratio indicating odd-numbered rotors are spinning faster than even-numbered

counter-rotors), the large change in tangential velocity they encounter (due to maximized M_w^*) results in very large work transfer relative to the rotational speed. Referring to the corresponding normalized work transfer plots in Figure 6.27, these even-numbered counter-rotors achieve this while still only producing under half of the average stage work transfer of the entire machine. However performing around 50% of the average work of the entire machine is of course infinitely more than producing 0% as a stator, and again this is achieved without violating any of the constraints.

6.4.1 Comparison to Traditional Rotor-Stator Case

As an example case study, a best-point design from this study with very high tip speed ratio (and hence high flow and blade loading coefficients) and another with more modest tip speed ratio (hence flow and blade loading coefficients that fall within the usual range) are compared to a traditional rotor-stator multistage machine with the same number of rotating stages at the same inlet conditions. The same relative and absolute Mach number constraints are enforced along with the limits on de Haller number and Diffusion Factor.

Traditional machines commonly have all rotors mounted on the same shaft and so all rotors are required to rotate at the same speed. Many traditional axial compressors are designed with constant mean radius (rather than constant tip radius for the wound impellers) and similar velocity diagrams for each rotor. This is achieved by enforcing a 50% reaction, R , [54] in a rotor-stator pair, where reaction is defined as the ratio of the static enthalpy rise in the rotor to the static enthalpy rise across the rotor-stator pair, with the stator downstream of the rotor:

$$R = \frac{h_{rotor,out} - h_{rotor,in}}{h_{stator,out} - h_{rotor,in}} \quad (6-6)$$

which can be expressed in terms of static temperature difference and specific heats.

The Solver analysis tool in Microsoft Excel was used to obtain maximum work transfer while satisfying all constraints for the rotor-stator calculations that include 50% reaction and a maximum blade loading coefficient of 0.4 in addition to the constraints used in the counter-rotating study. There are only five independent design variables: hub/tip ratio, absolute and relative Mach numbers at the rotor inlet, absolute flow angle at the inlet, and blade turning which is the same for all rotors. Table 6.10 provides the results of the case study.

Table 6.10. Case Study Comparing Counter-Rotor to Rotor Stator at Same Inlet Conditions

Maximum Tip Speed Ratio $u_3/u_2=2.94$	3 Stages Counter-Rotating	Location	3 Pairs Rotor-Stator	Location	Difference
$T_{sat,in}$ °C	50		50		
$p_{sat,in}$ kPa	12.35		12.35		
$\tilde{e}_{Tot}$ kJ/kg	86.5		55.16		36.2%
Π_t	1.54		1.327		13.8%
$M_{c,max}^*$	0.70	$R_{1,hub,out}$	0.60	$R_{1,hub,out}$	
$M_{w,max}^*$	0.80	All Rotor Tip Inlets	0.80	$R_{1,tip,in}$	
dH_{min}	0.735	$R_{2,hub}$	0.72	All Rotors (mean)	
DF_{max}	0.455	$R_{1,hub}$	0.450	$R_{1,hub}$	
ϕ_{max}	1.815	R_2	0.508	R_3	72.0%
ψ_{max}	0.897	R_2	0.400	All Rotors	55.4%
Maximum Tip Speed Ratio $u_2/u_1=1.34$	4 Stages Counter-Rotating	Location	4 Pairs Rotor-Stator	Location	Difference
$T_{sat,in}$ °C	40		40		
$p_{sat,in}$ kPa	7.38		7.38		
$\tilde{e}_{Tot}$ kJ/kg	106.4		71.13		33.1%
Π_t	1.719		1.450		15.6
$M_{c,max}^*$	0.40	$R_{1,hub,in}$	0.604	$R_{1,hub,out}$	
$M_{w,max}^*$	0.80	All Rotor Tip Inlets	0.80	$R_{1,tip,in}$	
dH_{min}	0.720	$R_{2,hub}$	0.720	All Rotors (mean)	
DF_{max}	0.450	$R_{4,hub}$	0.450	$R_{1,hub}$	
ϕ_{max}	0.591	R_1	0.513	R_4	13.2%
ψ_{max}	0.423	R_3	0.400	All Rotors	5.4%

The counter-rotating cases achieve 36.2% and 33.1% higher work transfer for the 3 and 4 stage cases, respectively. The 3 stage counter-rotating case with high values of tip speed ratio,

flow coefficient, and blade loading coefficient was able to achieve a 3.1% larger difference in work transfer (compared to the 3 stage rotor-stator) than the 4 stage case comparing values of flow and blade loading coefficient in the usual range. Of course in both comparisons, the same number of rotors is present, but the rotor-stator cases require 7 and 9 impellers for 3 and 4 stages, respectively, including an IGV (necessary for control of inlet absolute flow angle).

Reaction in a rotor-stator pair indicates how much of the flow deceleration and hence generation of an adverse static pressure gradient is done by the rotor ($1-R$ is the stator contribution) It is possible to apply the same principle to neighboring rotors and counter-rotors. Moving from inlet to outlet, a rotor and counter-rotor can be considered, where the next consideration point would consider the same counter-rotor and the next downstream rotor such that

$$R_{rot-crot} = \frac{h_{rot,out} - h_{rot,in}}{h_{crot,out} - h_{rot,in}} \quad (6-7)$$

The subsequent downstream reaction is then

$$R_{crot-rot} = \frac{h_{crot,out} - h_{crot,in}}{h_{rot,out} - h_{crot,in}} \quad (6-8)$$

where the subscripts *rot* and *crot* refer to rotors and counter-rotors , respectively. As an example, Equation (6-7) would consider the first and second stage in the machine, and the subsequent reaction in Equation (6-8) would consider the second and third stage in the machine. Reactions using the definitions in Equations (6-7) and (6-8) are plotted for all the best-point designs in Figure 6.40.



Figure 6.40. Counter-Rotating Reaction Pairs vs. Saturation Temperature at First Rotor Inlet

The trends of the reaction pairs are somewhat similar to those of the tip speed in Figure 6.28 (although note that here the reaction references the upstream rotor of the two rotors in question, whereas the tip speed ratio uses the quotient of downstream rotor tip speed to upstream rotor tip speed of the two rotors in question). Higher temperatures for odd N_s see reactions nearer to 50% just as tip speed ratios are nearer to 1.

While reaction in Figure 6.40 and work transfer normalized by the average work transfer of the machine in question indicate how much of a contribution each rotor makes to the

combined work transfer of all the various best-point designs found in this study, it is perhaps most clear when viewed directly as $\tilde{e}_i/\tilde{e}_{\text{Tot}}$ shown in Figure 6.41.

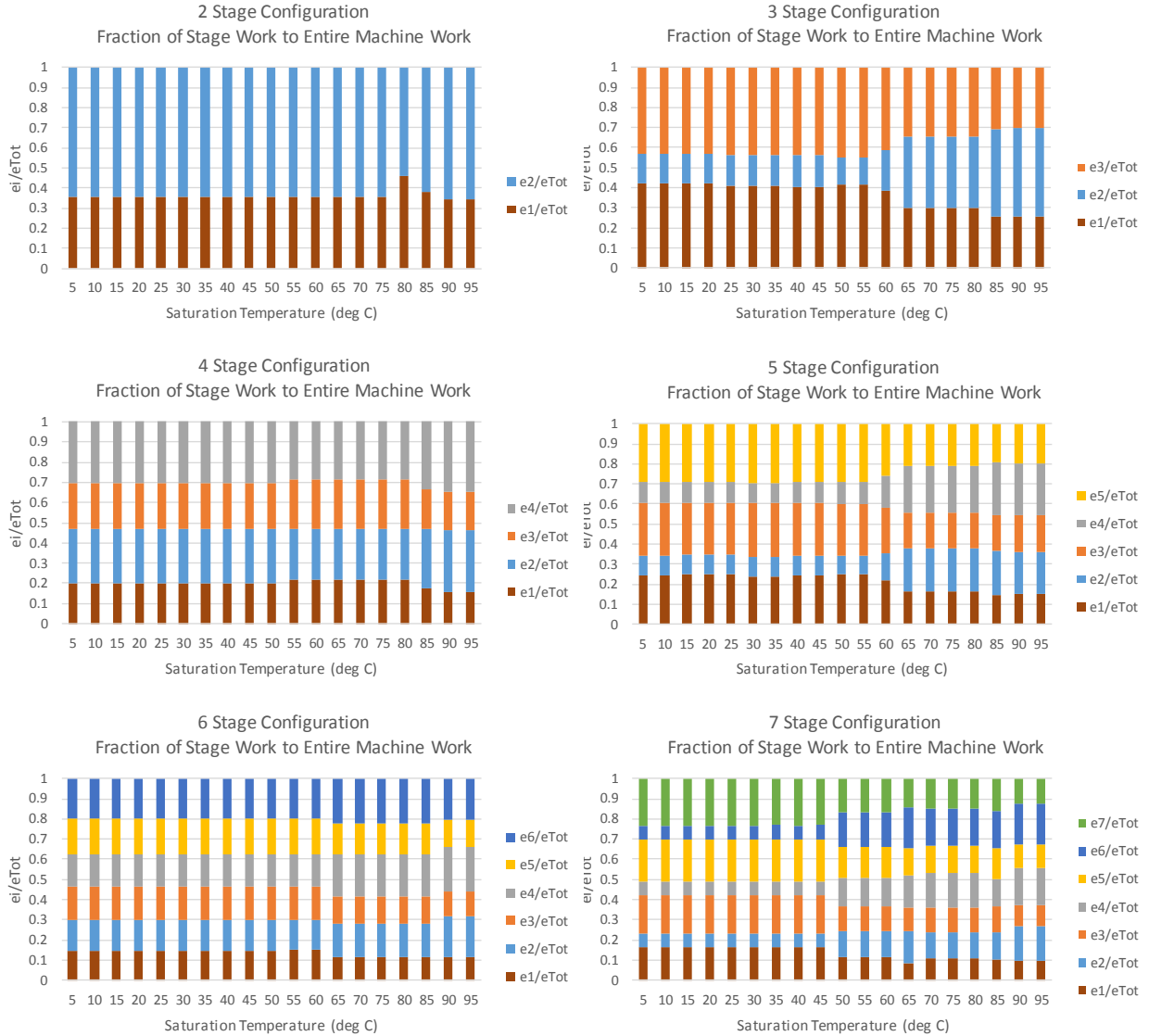


Figure 6.41. Rotor Work Fraction of Combined Work Total vs. Saturation Temperature at First Rotor Inlet

In the vast majority of cases, even-numbered rotors in even N_s machines combine to produce the majority of the overall work transfer of the machine, and the same is true of odd-numbered rotors in odd N_s machines. In odd-numbered machines, the majority fraction is strongly dependent on saturation temperature, and in the $N_s = 7$ cases for temperatures above

90°C, the even-numbered stages account for over 50% of the combined work transfer. The contributions are plotted in Figures 6.42 and 6.43.

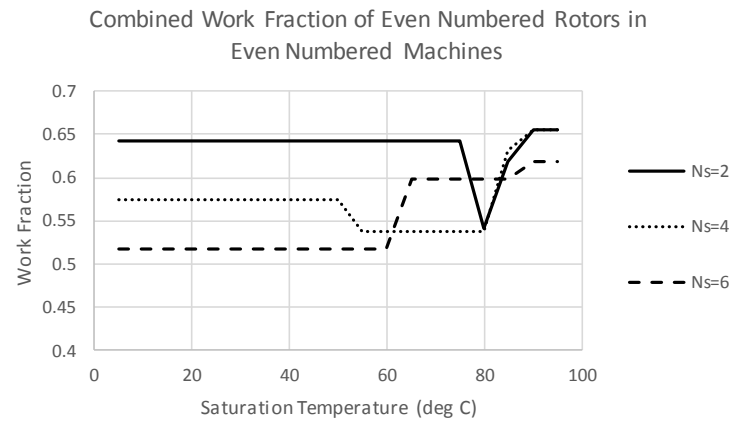


Figure 6.42. Work Contribution Fraction of Even-Numbered Stages in Machines with Even Number Total Stages vs. Saturation Temperature at First Rotor Inlet

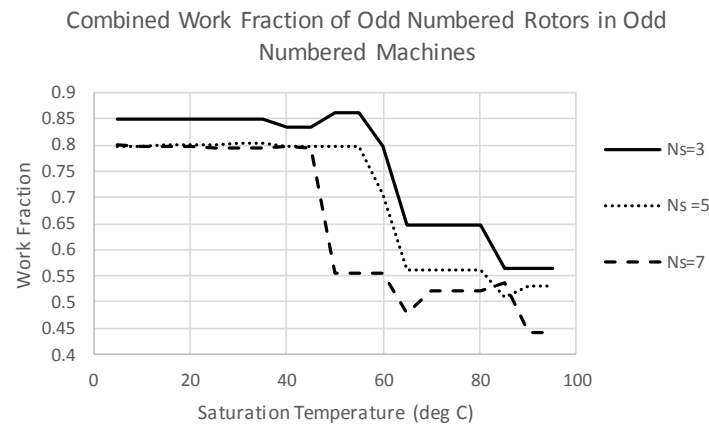


Figure 6.43. Work Contribution Fraction of Odd-Numbered Stages in Machines with Odd Number Total Stages vs. Saturation Temperature at First Rotor Inlet

Even rotors in even N_s machines account for between 52% and 66% of overall work. Odd-numbered stages account for between 80% and 85% in machines with odd N_s in the lower end of the saturation temperature range. This is more pronounced for odd N_s because odd N_s machines inherently have an additional odd-numbered rotor present. For temperatures larger than 50°C, the even-numbered stages account for a larger fraction of the combined work, and this increases as temperature increases. Notably, the pre-swirl imparted by the IGV at lower temperatures switches to counter-swirl as the even stages contribute more at higher temperature.

This corresponds to decreases in odd-to-even tip speed ratios and in increase in even-to-odd tip speed ratios in this same range. Flow and blade loading coefficients correspond accordingly as discussed previously in this section.

Specific speed, σ , is used to classify the type of turbomachine best suited to a given situation. Radial machines have low specific speeds as they impart relatively high pressure ratios while moving relatively low mass flow rates. Axial machines have higher specific speeds and typically impart lower pressure ratio to higher mass flow rates [54]. The range for each varies depending on the definition used. The low end of the axial compressor range for

$$\sigma = 2^{1/4} \sqrt{\pi} \frac{n \sqrt{\dot{V}}}{\dot{c}^{3/4}} \quad (6-9)$$

is around 0.8 and n is the rotational speed in revolutions per second. Figure 6.44 shows plots of specific speed against saturation temperature for all the best-point designs.

All specific speeds decrease with increasing saturation temperature. Specific speeds are highest for odd N_s machines at lower temperature where the tip speed ratios are very high. This is the situation that more closely resembles a traditional axial machine in that the tip speed ratio when used in this sense for a rotor-stator pair is infinite. As the tip speed ratios decrease, the specific speed of each rotor decreases as well. Specific speeds seen here and in are more in the range of a mixed flow compressor because counter-rotation allows rotors to transfer much higher amounts of work (and produce much higher pressure ratios) at relatively lower rotational speeds. The counter-swirl of the flow at the inlet to each rotor enables very large change in tangential velocity without the rotor itself needing to attain the tip speed of the magnitude of the tangential velocity difference, thus work transfer is high for the tip speed and mass flow rate, and specific speed for a counter-rotating axial machine is lower than for a traditional axial machine. This is all accomplished with no constraint violations.

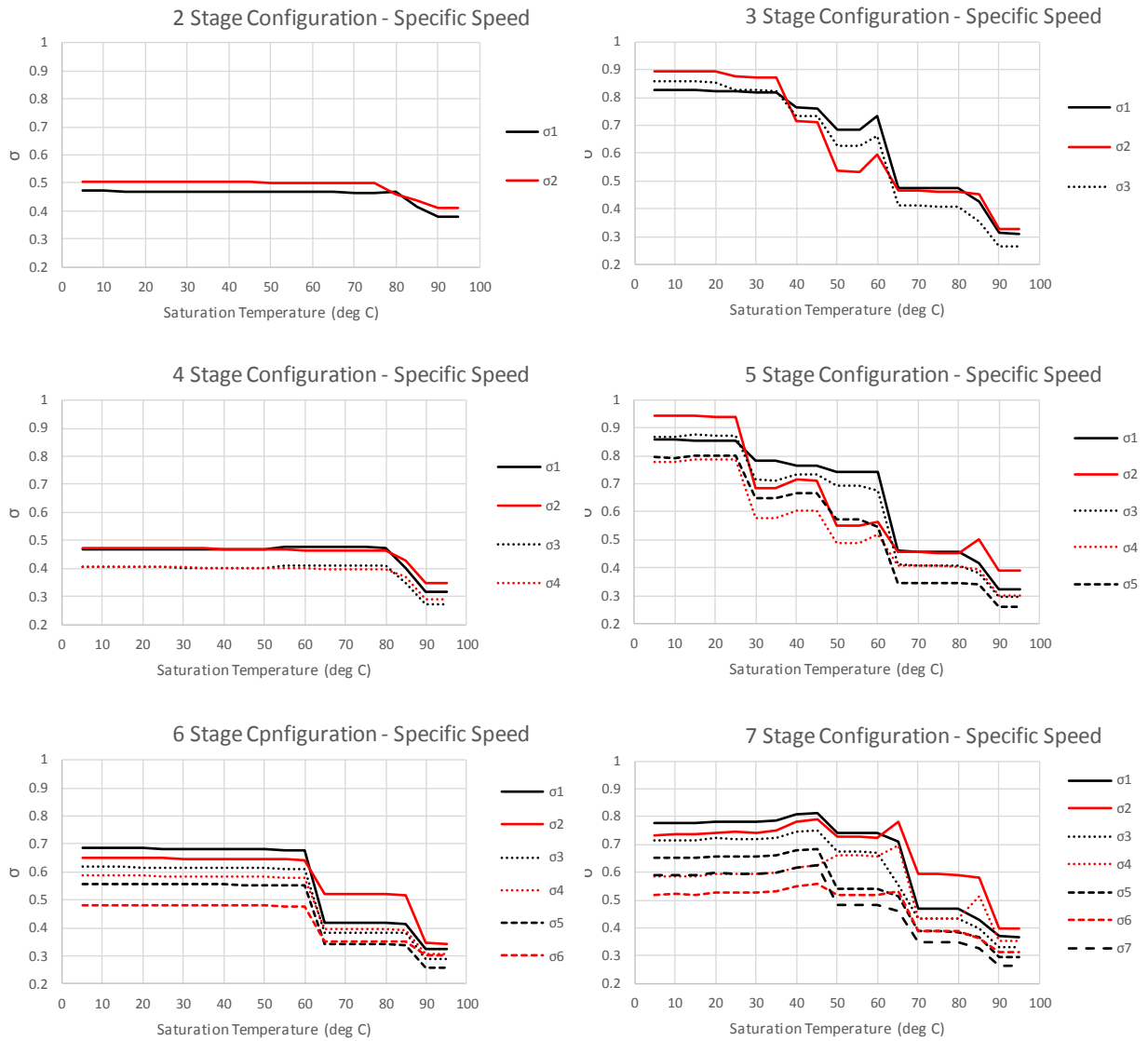


Figure 6.44. Specific Speed vs. Saturation Temperature at First Rotor Inlet

To get a sense of the magnitude of pressure ratios produced, Figure 6.45 plots the best-point pressure ratios, which display very similar trends to normalized work transfer. As temperature increases and more work transfer is permitted creating higher outlet pressure, the inlet pressure has also increased, thus total pressure ratio is in a sense self-normalizing.



Figure 6.45. Total Pressure Ratio vs. Saturation Temperature at First Rotor Inlet

6.5 Numerical Simulation of Select Cases

Numerical simulations have been performed using ANSYS CFX of individual best-point designs for 2, 3, and 4 counter-rotating stages at saturated inlet temperatures of 5°C, 20°C, 40°C, 60°C, 80°C, and 95°C, yielding 18 unique scenarios.

As an aside, a 1 stage machine was simulated at the conditions in Chapters 4 and 5 upon determining the best-point design variables using the techniques described in this chapter. The exhaustive search determined $\tilde{e} = 23.47\text{kJ/kg}$ (an increase of 20.0% from the star pattern in

Chapter 4 and an increase of 16.1% from the FVB impeller without an IGV in Chapter 5), and found $\tilde{e} = 23.67\text{kJ/kg}$ in numerical simulation (and increase of 112.3% and 85.1% compared to simulation results in Chapter 4 and Chapter 5, respectively). This corresponds to $\Pi_t = 1.234$ analytically ($\Pi_t = 1.266$ in simulation) while satisfying all constraints, which is possible due to the inclusion of an IGV in the search of the design space. The exhaustive search of the 1 stage design space determined that the optimal rotational speed is 7640rpm with 21.0° mean blade turning (star pattern and FVB impellers rotate at 7500rpm with 52.9° mean blade turning). Numerical simulation of the IGV and Rotor attained $\eta_{pt} = 87.33\%$, which is above the estimated 80% used in calculation. Simulation results yielded higher work transfer, and pressure ratio than analytical calculation due to simulation producing a larger absolute tangential velocity difference than was predicted. Contour plots visualizing the flow field are provided in Appendix A4.

For each of the 18 scenarios ($N_s = 2, 3, 4$ at six different temperatures), total temperature and total pressure boundary conditions were set at the IGV inlet. The axial spacing between stages was selected to be the 50% of the average axial length of the neighboring rotors as 50% axial spacing is found to provide highest total pressure ratio and work transfer [63].

As discussed in Chapters 4 and 5, the mass flow rate in simulation will be lower than used in the analytical calculations. Estimating boundary layer thickness to approximate the new mass flow rate in simulation in a multistage machine is much more challenging than for a single stage as the question becomes how much of the boundary layer growth from the upstream stage to include in the downstream stage. As such, a static pressure boundary was used at the outlet of the final rotor that determines the mass flow rate. It is recognized that as the mass flow rate in calculation will be different in simulation, so too will the static pressure. As such, the static pressure was varied from 75% to 100% of calculation for each individual case. The result is a set

of compressor performance plots. These are included here for the 2, 3, and 4 stage best-point design cases at 60°C. The performance plots for the other temperatures can be found in Appendix A5.

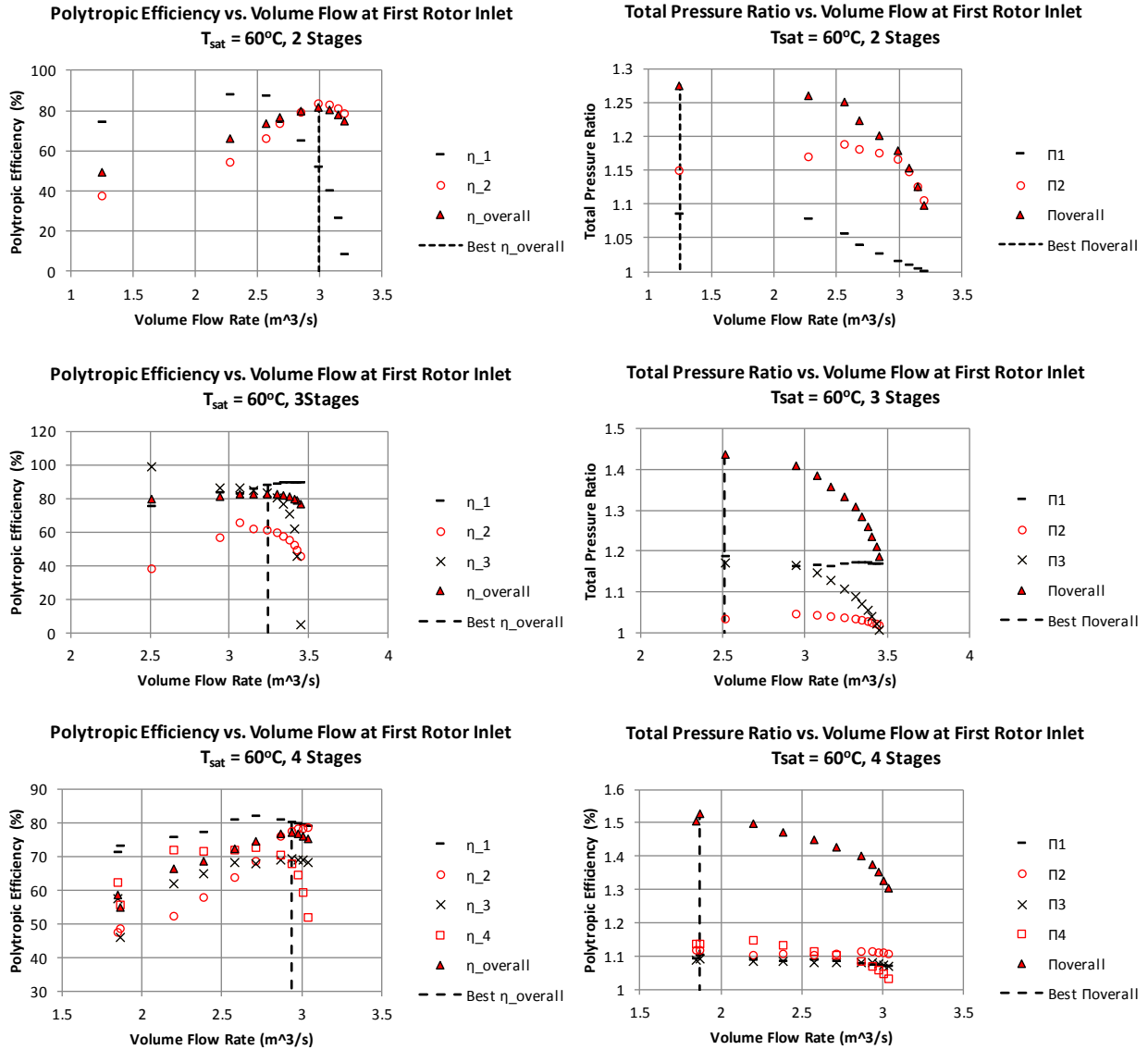


Figure 6.46. Performance Charts for 2, 3, and 4 Stages Used at 60°C

All cases see the highest overall total pressure ratio achieved (highest combined work transfer) at the lowest mass flow rate (lowest static pressure outlet boundary condition). The highest overall polytropic efficiency is achieved at or near the high end of the resulting mass flow rate.

Figure 6.47 provides normalized work transfer for each simulation for highest overall polytropic efficiency and largest combined work transfer.

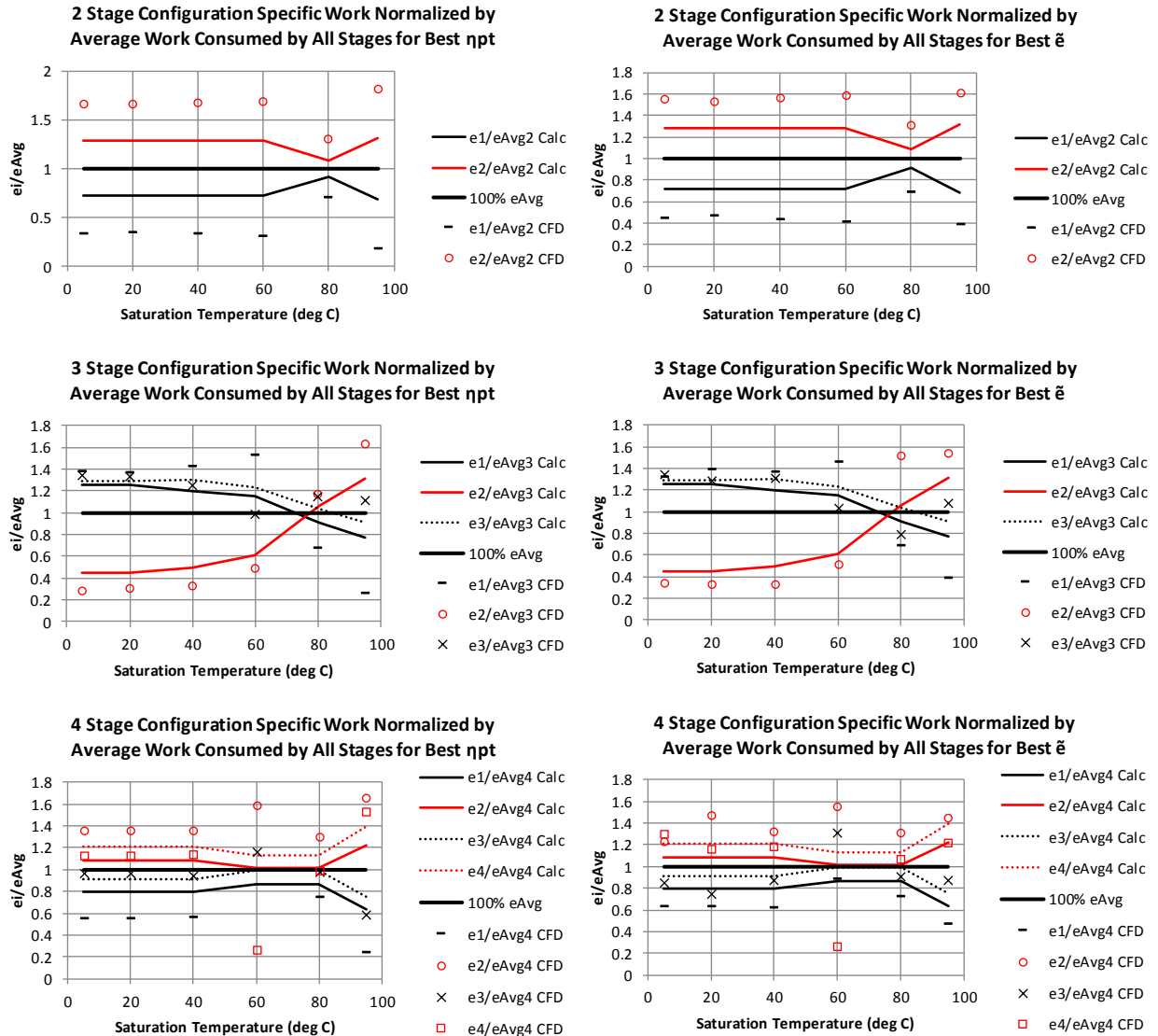


Figure 6.47. Normalized Combined Work Transfer for Highest Polytropic Efficiency and Largest Combined Work Transfer vs. Saturation Temperature at First Rotor Inlet

With the exception of a few outliers, it is generally seen that the rotors predicted to transfer the majority of the work see their contribution increased in simulation with the converse true for rotors predicted to make a lesser contribution to the total. The work fractions determined in simulation are shown in Figure 6.48 for the cases of best overall polytropic efficiency, and in Figure 6.49 for cases of best combined work transfer.

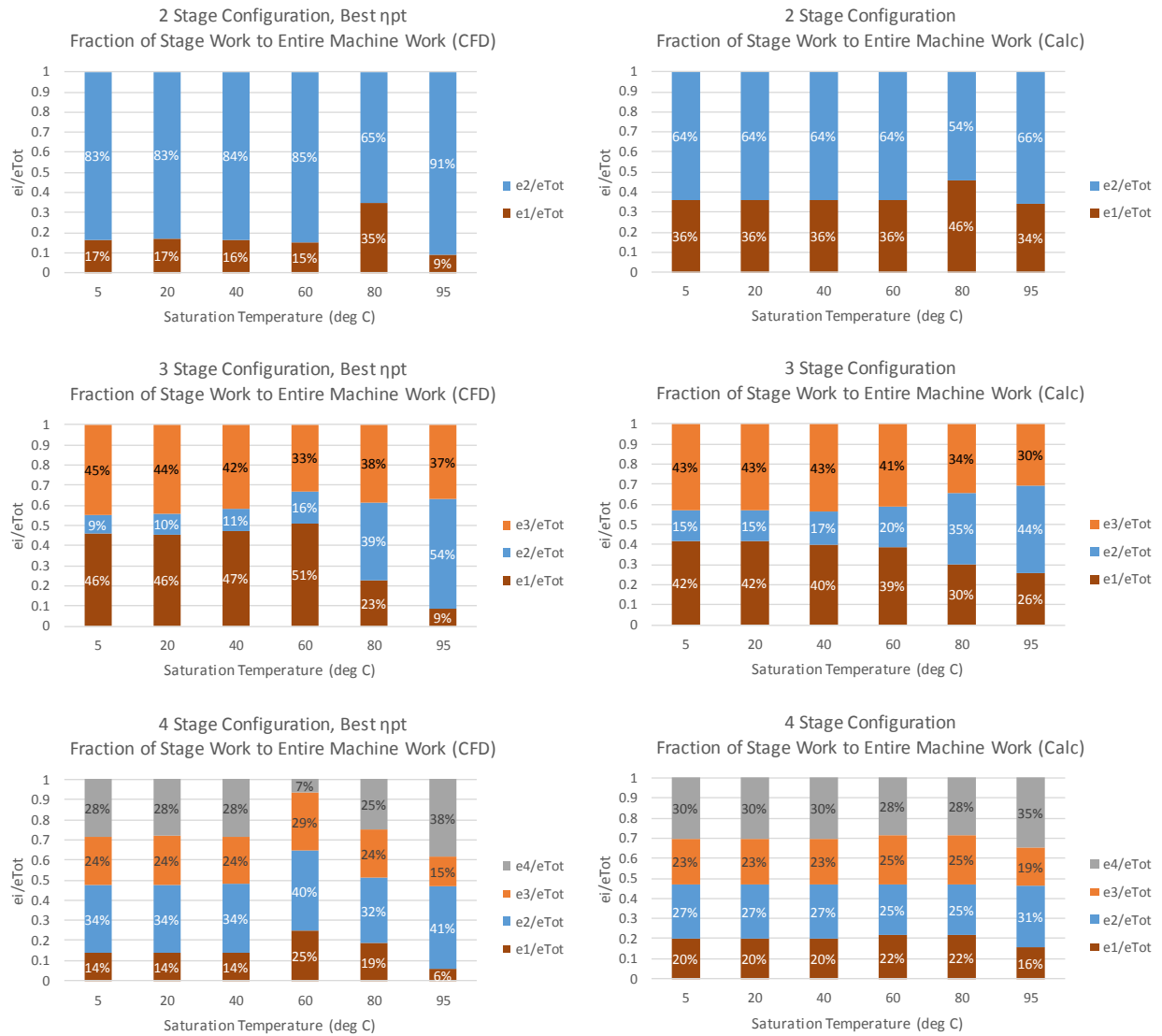


Figure 6.48. Rotor Work Fraction of Combined Work Total vs. Saturation Temperature at First Rotor Inlet for Cases of Highest Overall Polytropic Efficiency in Simulation Compared to Calculated Work Fraction

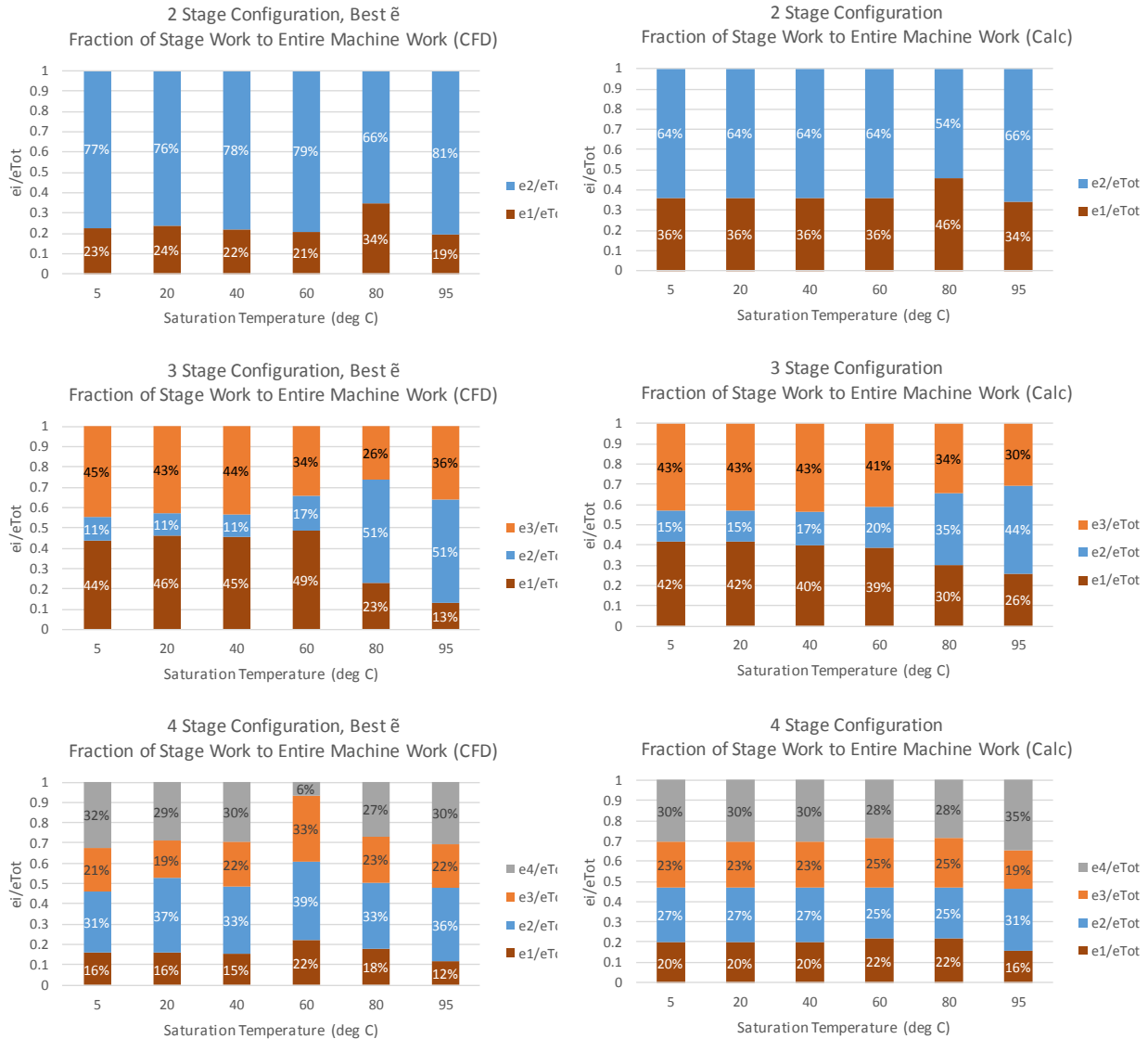


Figure 6.49. Rotor Work Fraction of Combined Work Total vs. Saturation Temperature at First Rotor Inlet for Cases of Largest Combined Work Transfer in Simulation Compared to Calculated Work Fraction

The fraction of the combined work performed by each rotor for the cases of highest overall polytropic efficiency and largest combined work transfer are more similar to each other than they are to the expected work fractions determined in calculation. However the increasing work fraction with temperature of the second rotor in the 3 stage machine is clearly observed in all cases.

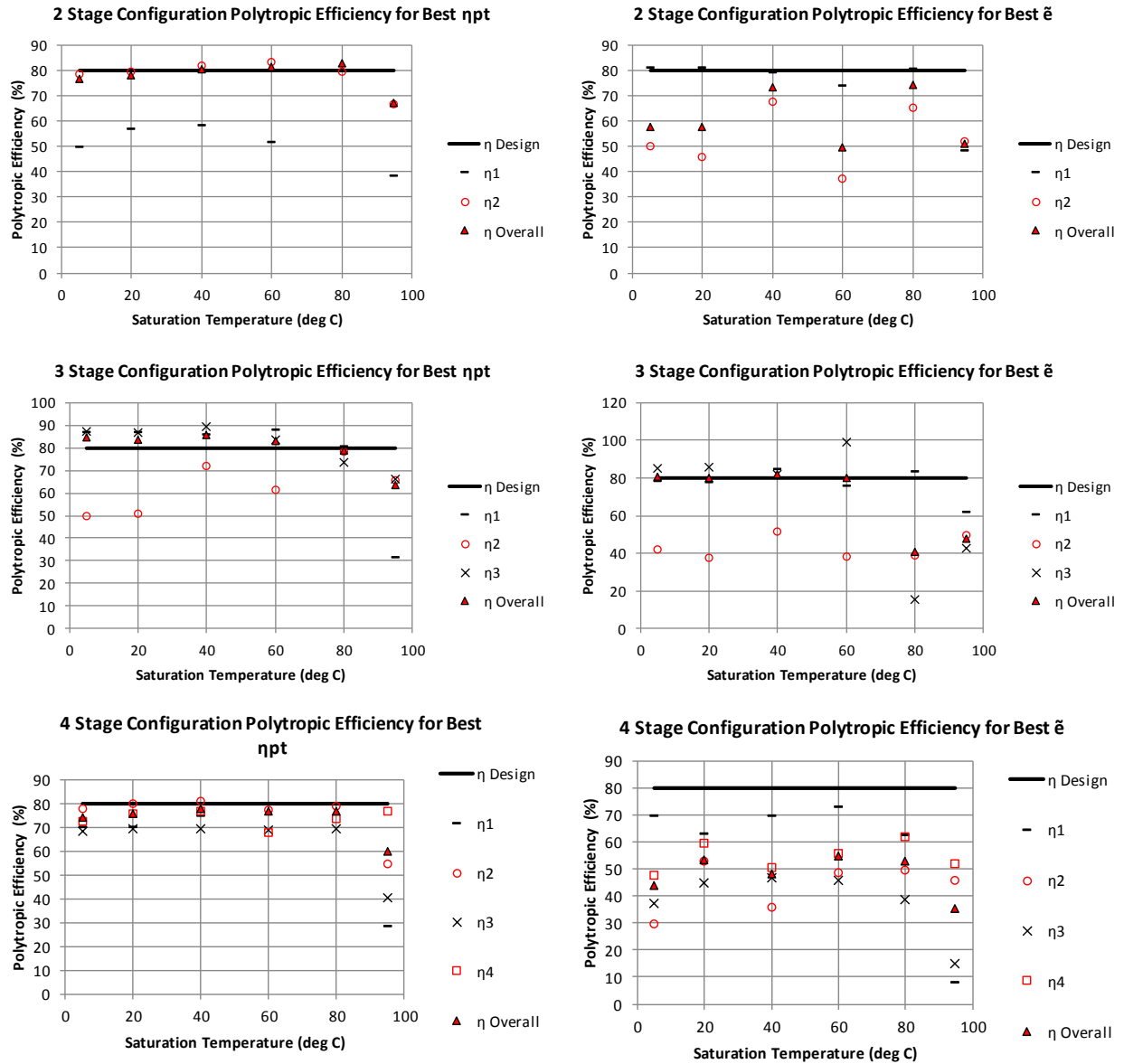


Figure 6.50. Polytropic Efficiency vs. Saturation Temperature at First Rotor Inlet

Rotors making large contributions to the overall work also individually have high polytropic efficiency. This large contribution at high efficiency outweighs low efficiency lower contributions, which results in high overall polytropic efficiency. All machines see overall polytropic efficiency greater than the assumed 80% for at least one saturated inlet temperature. The high efficiency points for individual rotors making a large work contribution correspond to lower incidence and deviation angles.

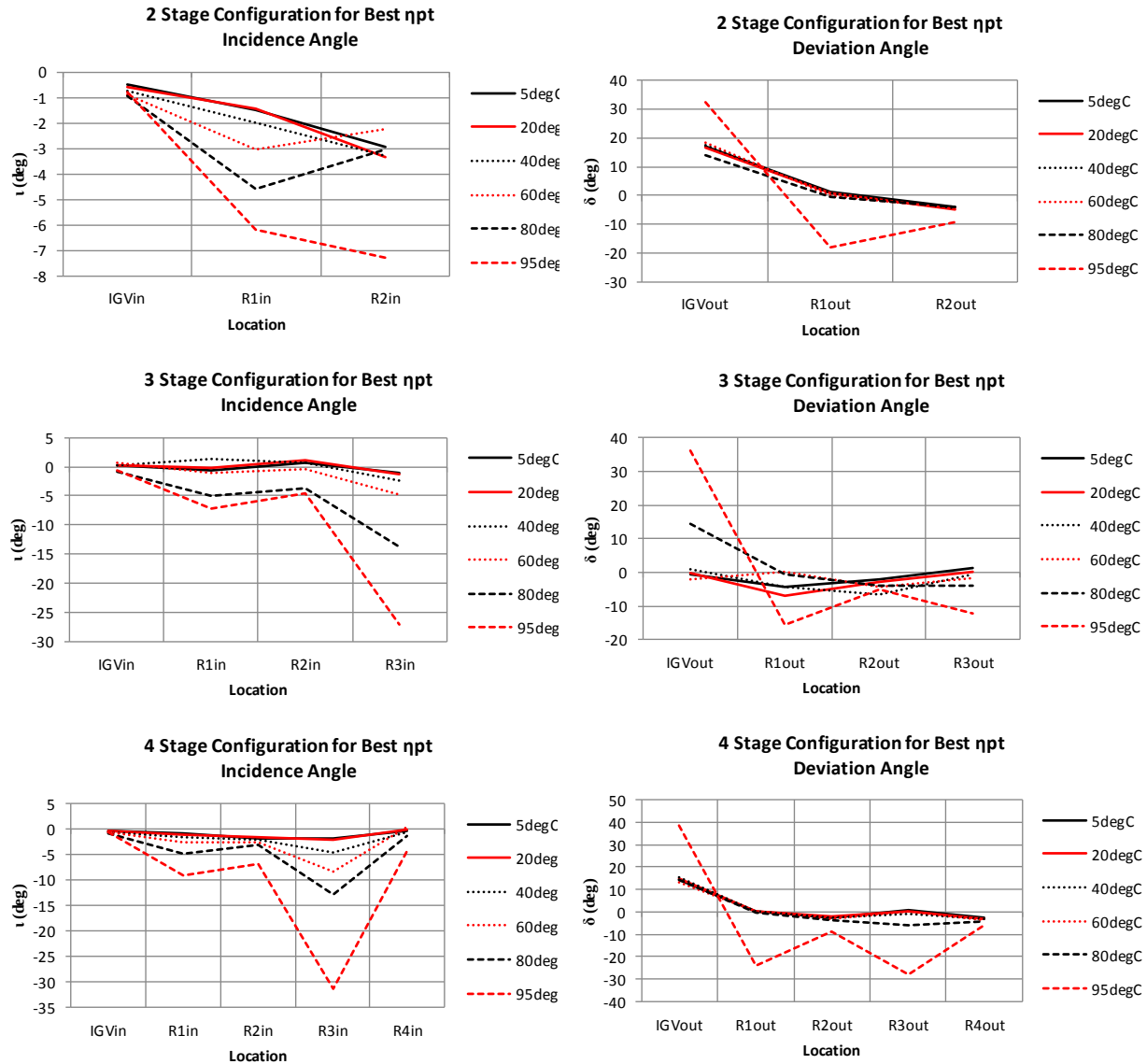


Figure 6.51. Incidence and Deviation Angles vs. Axial Location for Highest Polytropic Efficiency Cases

Both incidence and deviation increase in magnitude overall with increasing stream-wise location, and this trend increases with increasing temperature. The highest temperatures are where the lowest individual rotor efficiency and lowest overall efficiencies are found. Figure 6.50 provides the absolute flow angles along with relative flow and blade angles as a function of their axial location to give a sense of how the fluid moves downstream from inlet to outlet.

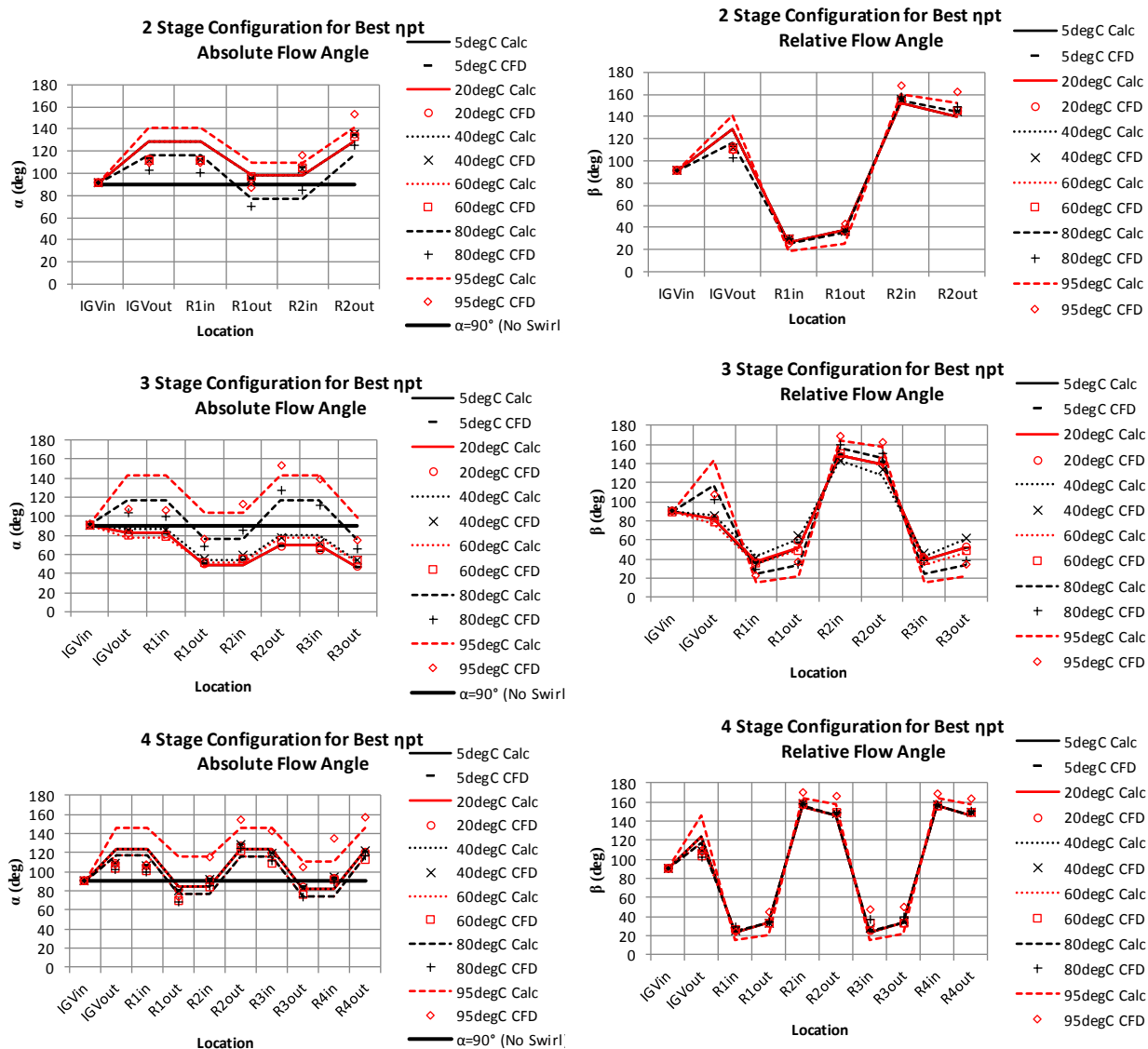


Figure 6.52. Absolute and Relative Flow Angles vs. Axial Location for Highest Polytropic Efficiency Cases

Despite cases of large incidence and deviation indicated in Figures 6.51, Figures 6.52 shows that the flow in general tends to follow the intended path in both the absolute and relative reference frames. Figure 6.53 provides incidence and deviation angles, and Figure 6.54 shows absolute and relative flow angles as a function of temperature for the simulated cases resulting in maximum combined work transfer.

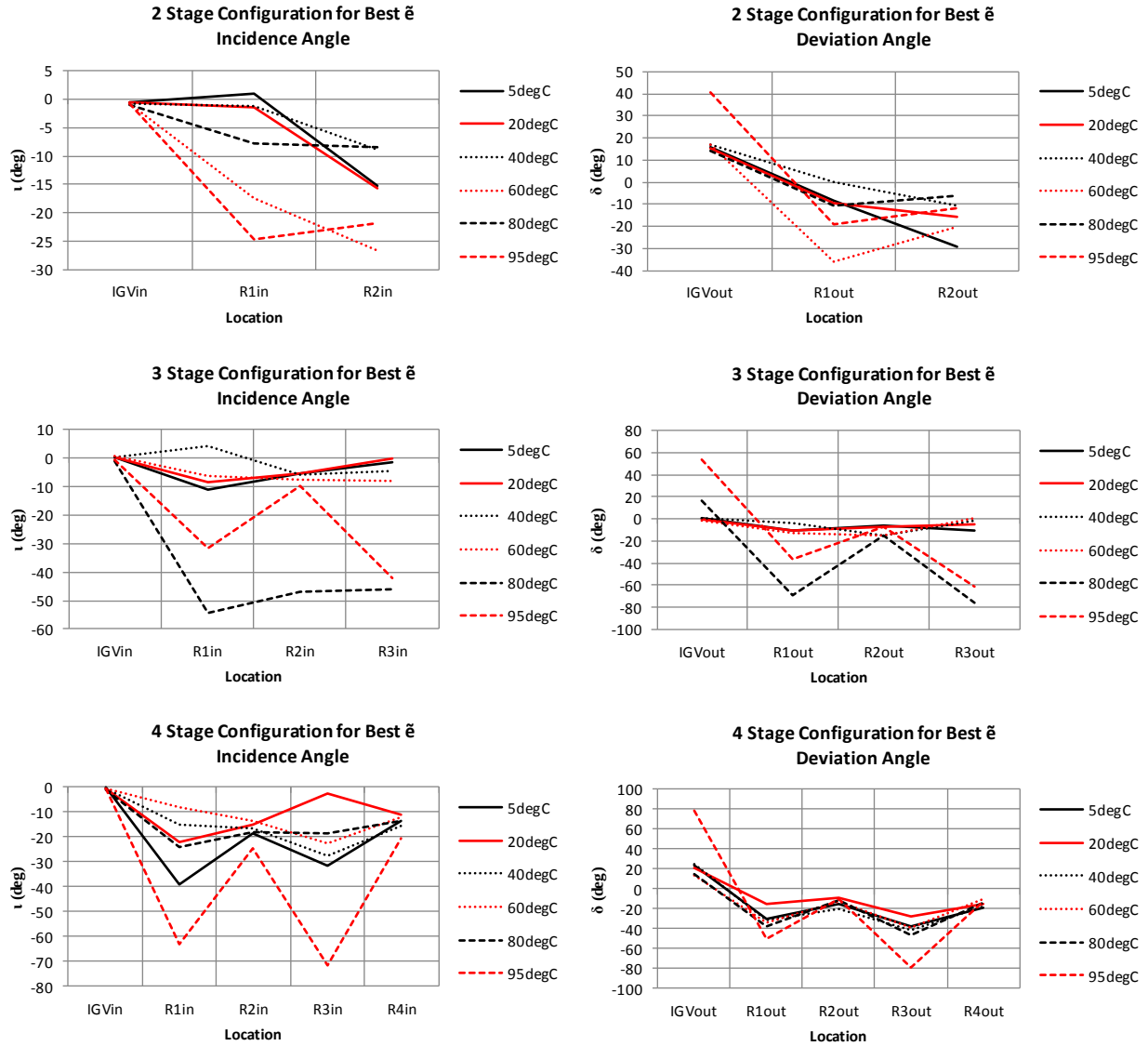


Figure 6.53. Incidence and Deviation Angles vs. Axial Location for Largest Combined Work Transfer Cases

Incidence and deviation angles are generally larger for the cases of maximum combined work transfer than for cases of best overall polytropic efficiency. Higher efficiency of operation is expected when the flow angles more closely match the blade angles.

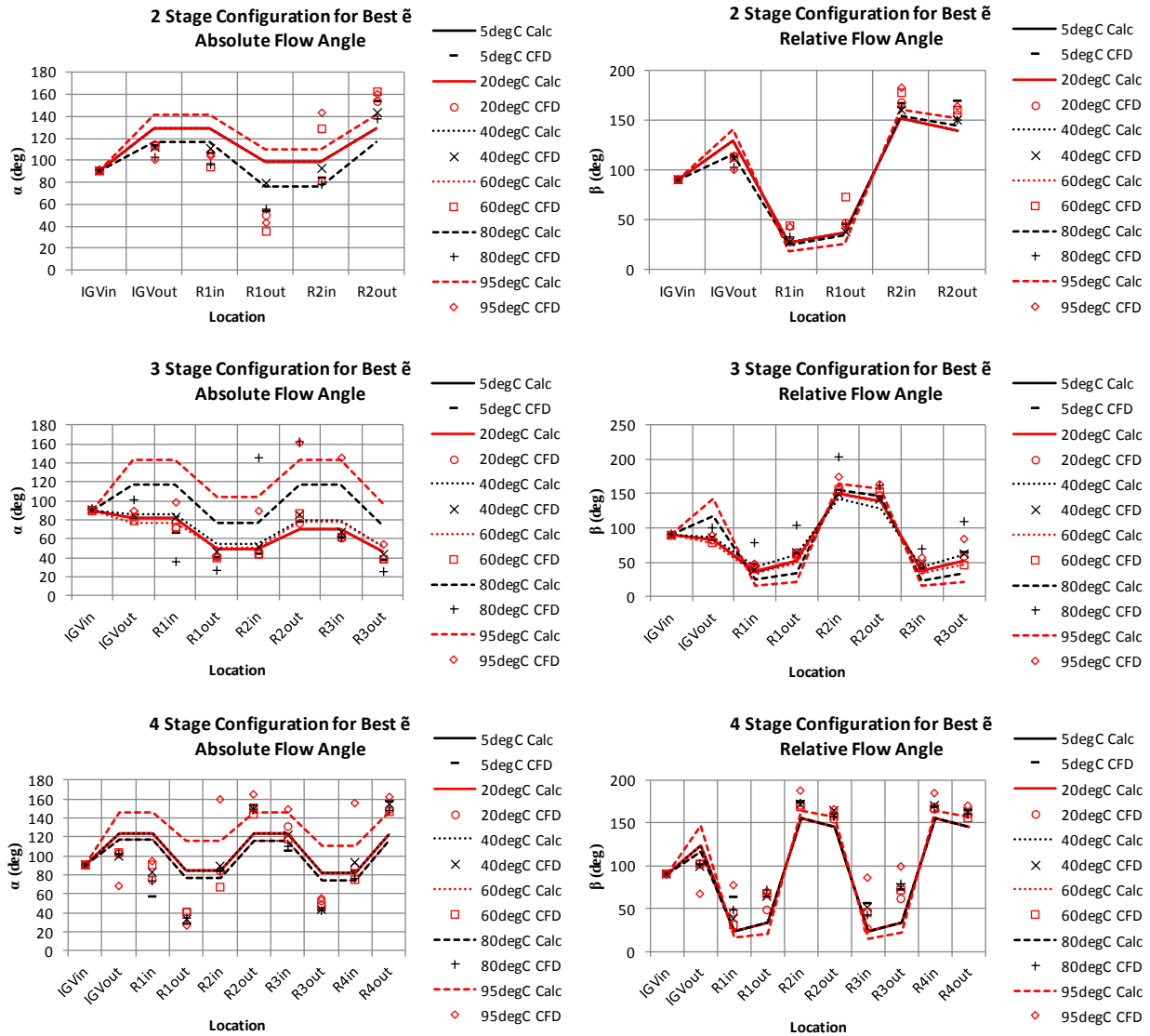


Figure 6.54. Absolute and Relative Flow Angles vs. Axial Location for Largest Combined Work Transfer Cases

The relatively high values of polytropic efficiency attained in cases of maximum overall polytropic efficiency (as well as many of the individual values for cases of maximum combined work transfer), while exhibiting the relatively large incidence and deviation angles suggest that counter-rotating axial compressors have a wide operating range for achieving high efficiency and combined work transmission.

Relative Mach number contours are shown in the meridional plane in Figure 6.55 for the two simulation cases that achieve highest overall polytropic efficiency and largest overall

combined work transfer for the 4 stage configuration with 60°C saturated water vapor at the inlet to the first rotor.

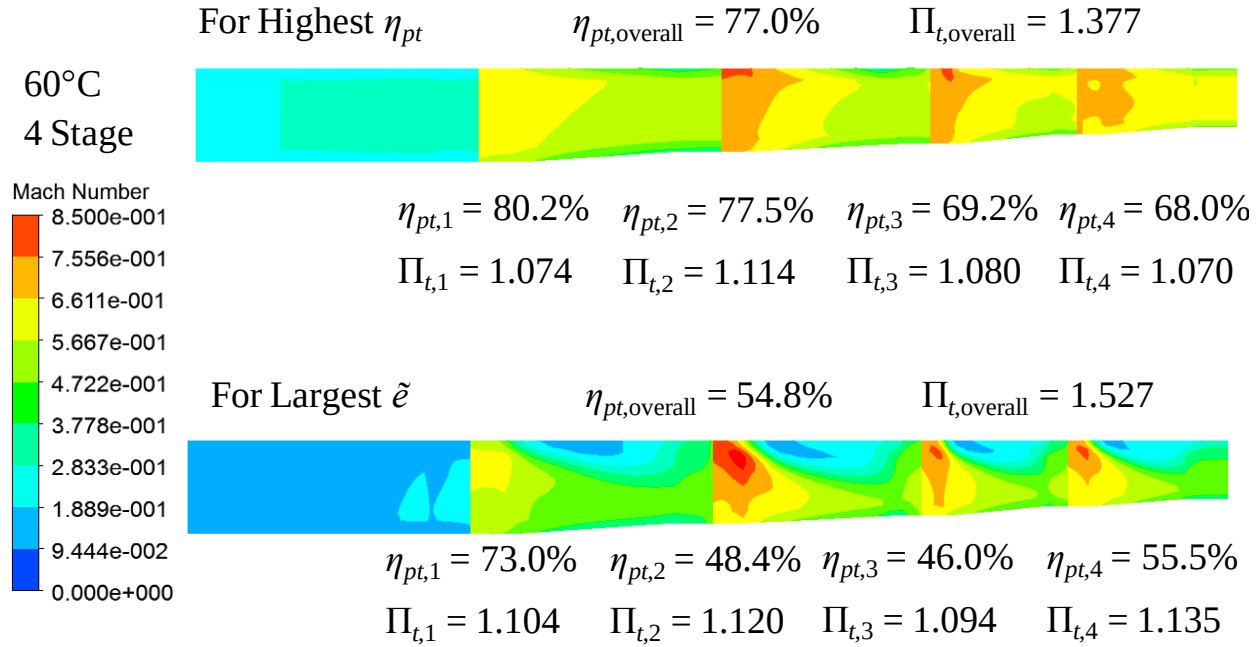


Figure 6.55. Meridional View of Relative Mach Number Contours for 4 Counter-Rotating Stages with Saturated Vapor Inlet at 60°C

For the case of highest polytropic efficiency, the circumferential average of inlet tip Mach number for each rotor is very near 0.8 as was set in the design procedure. In this case, the Mach number decreases from inlet to outlet for all spans fairly uniformly, indicating that the flow is following the blades well on average. The high efficiency achieved by each individual rotor as well as the large overall efficiency (just 3% lower than the assumed 80%) confirm this. However, for the case with largest work transfer, the relative Mach numbers at the inlet of all rotors downstream of the first rotor is closer to 0.85 (circumferential average), followed by regions of very low Mach number at high span, indicating that there may be separation from the shroud and blades in this region. Figure 6.56 shows relative Mach number contours in the blade-to-blade view showing Mach numbers between blades without circumferential averaging for low (0.1) mean (0.5), and high (0.9) span.

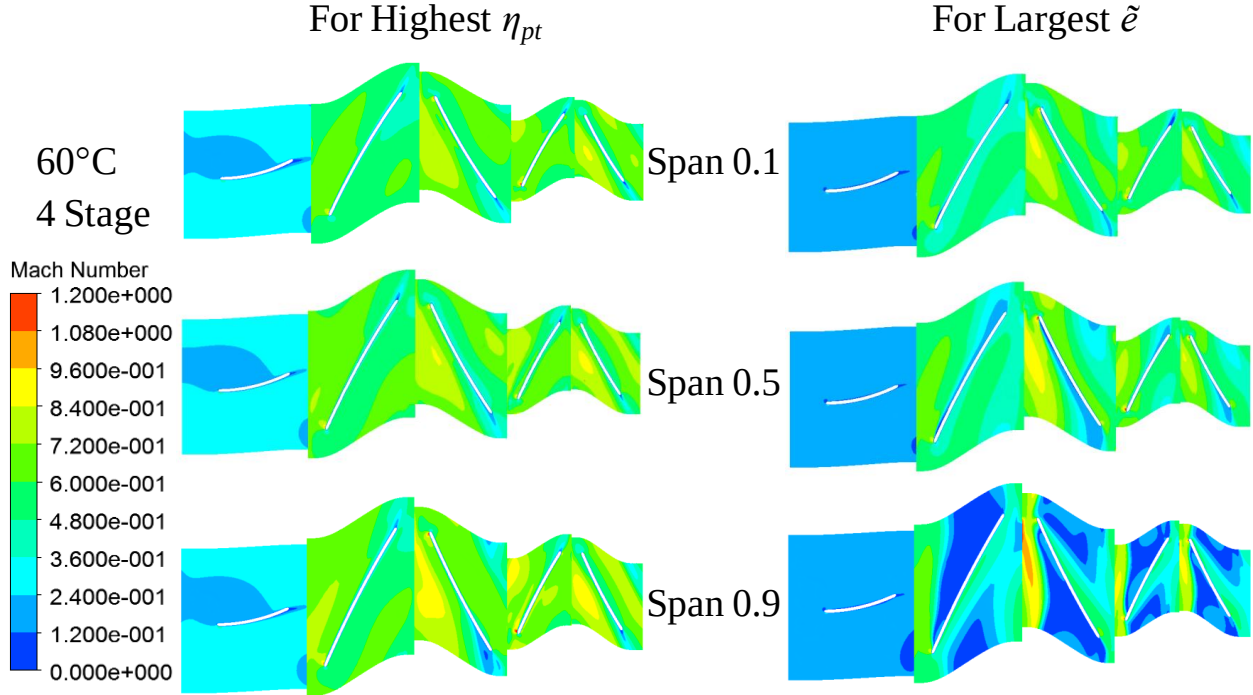


Figure 6.56. Blade-to-Blade View of Relative Mach Number Contours for 4 Counter-Rotating Stages with Saturated Vapor Inlet at 60°C

The highest efficiency case shows that the flow is locally supersonic in small regions on the suction surface of the leading edge of each rotor. That the individual rotors and the overall machine are able to achieve high polytropic in spite of this indicates that the selection of $M_w^* = 0.8$ at each rotor inlet is as high as should be allowed in the design space. Larger values will likely lead to larger supersonic regions that can potentially lead to a choked condition at the inlet. This is the case for the largest combined work transfer simulation. The inlet to the second, third, and fourth rotors all appear to be choked at high span which leads to shock formation, rapid flow deceleration across the shock, and large decreases in efficiency due to entropy generation. Large work transfer is achieved with the large deceleration, but comes with a steep reduction in individual rotor and overall operating efficiency. The choking at high span downstream of the first rotor is common for the cases with largest combined work transfer.

Choking is not seen for cases with highest overall polytropic efficiency. Mach number contours for all other simulated cases are presented in this manner in Appendix A5

6.6 Chapter Summary and Conclusions

A set of independent design variables for a counter-rotating axial compressor have been identified (38 for 7 stages) and reduced to a manageable size (10 for 7 stages) by utilizing Equation (2-114) and Equation (6-2) in conjunction with design procedure outlined in Chapter 2 and in Figure 6.1. An exhaustive search of each design space has been performed to determine the values of the input variables corresponding to maximum combined work transfer. In total, 154 best-point designs have been found (1 through 7 stages employed at 22 different saturated water vapor temperatures under vacuum pressure) of a possible 3.7 billion design options.

It has been found that the design variables and constraints are all dependent on inlet saturation temperature. Even-numbered rotors in machines with an even total number of stages contribute the majority of the combined work transfer. The same is true for odd-numbered rotors in machines with an odd total number of stages. Most variables and constraints tend to show similarity in this manner (even rotors in even total stage configurations, odd rotors in odd total stage configurations)

For large tip speed ratio, the slower rotor of the pair has high flow and blade loading coefficients as well as low specific speed compared to traditional axial compressors. This is due to very large work transfer for a relatively low rotational speed. The large change in tangential velocity associated with large work transfer is able to be achieved without the rotor itself having to attain this same blade speed. This is because the flow enters the rotor with tangential velocity in the opposite direction (counter-swirl), and this is a feature of counter-rotation: high work

transfer with relatively low rotational speeds in an axial configuration. This is achieved without violating any of the imposed Mach number, de Haller number, and Diffusion Factor constraints.

Simulation of 2, 3, and 4 stages at inlet saturation temperatures of 5°C, 20°C, 40°C, 60°C, 80°C, and 95°C indicate that rotors predicted to contribute the majority of the combined work (even rotors for even number of stages, odd rotors for odd number of stages) contribute more than expected, and generally do so at high efficiency. This makes up for under-contributing, more slowly spinning rotors that operate with lower efficiency, and this results in overall efficiencies near (and in many cases above) the assumed 80%.

The focus of Chapter 6 is on compressing saturated water vapor under vacuum pressure at the inlet using counter-rotating axial stages. However, exhaustive searches of the design space for seven other gas mixtures have also been performed. The resulting best-point designs for these cases (1 to 7 stages at varying temperature) are located in Appendix A6.

CHAPTER 7 CONCLUSIONS

7.1 Summary of Conclusions

7.1.1 Chapter 4

A full 3D characterization of the star pattern geometry has been performed. Methods to predict flow behavior and performance characteristics are also developed and described. It has been verified that the star pattern impeller can in general be reasonably described by the forced-vortex condition.

Simulated performance using ANSYS CFX and experiments performed under the same conditions resulting in the in following conclusions:

- Although the calculations predicted excessive de Haller numbers and Diffusion Factors for spans greater than 17% and 28%, respectively, the simulations show that this is in fact only true for spans greater than 50%.
- This implies that there is excessive blade turning at spans greater than 50%.
- At spans greater than 50%, there are large regions of back flow due to boundary layer separation. The boundary layer separation is a result of excessive flow turning/blade surface curvature in an adverse pressure gradient.
- Taking the area average of the calculated specific work transfer for spans less than 50% provides very good agreement with the simulated results (within 1.5%).
- The 1.5% accuracy with the simulated results leads to the supposition that the experiments are providing similar work transfer to the fluid. However, for the measured pressure ratio, this corresponds to a polytropic efficiency of 26.1% for the experimental prototype.

- Reducing tip blade turning will very likely result in an increase of the measured pressure ratio as the spans exhibiting healthy core flow will be larger, which correspond to larger potential work transfer.

7.1.2 Chapter 5

The geometry of the analyzed free-vortex blade impeller was selected to be as similar as possible to the star pattern impeller discussed in Chapter 4 within the confines of the respective flow distributions. With the same inlet conditions, operating speed, and mean blade turning, the performance of the free-vortex impeller was predicted both analytically and in numerical simulation using ANSYS CFX.

- It has been found that the FVB impeller shows improvements in average specific work transfer to the fluid and achieves a higher total pressure ratio in both calculation and simulation. Numerical simulation shows that this is achieved while operating at higher polytropic efficiency (74.8% vs. 63.1%).
- Lower blade solidity at the FVB tip means that the separated region exhibiting back flow is reduced in comparison to the star pattern. The total pressure uniformly increases from inlet to outlet for spans up to than 85% for the FVB impeller as opposed to just 50% for star pattern.
- The de Haller number and Diffusion Factor limitations are satisfied for a wider range of spans for the FVB impeller (from approximately 0% to 60%) than the star pattern impeller (from approximately 0% to 50%).
- The average incidence and deviation angles for the FVB impeller are lower than for the star pattern. The range of incidence and deviation angles is also narrower for the FVB impeller.

7.1.3 Chapter 6

A set of independent design variables for a counter-rotating axial compressor have been identified. For 7 stages, an intractable 38 variables has been reduced to a manageable 10 variables for an individual set of inlet conditions. This is made possible by employing Equation (2-114) and Equation (6-2), both derived in this work, in conjunction with design procedure outlined in Chapter 2 (depicted in Figure 6.1). Relative Mach numbers are set rather than iterated for at each rotor inlet. Variable sensitivity analysis determined that the relative Mach number should always be maximized at the tip of the inlet to each rotor.

An exhaustive search of the design space for 22 saturated water vapor inlet temperatures has been performed to determine the values of the input variables corresponding to maximum combined work transfer depending on the number of stages used. 154 best designs have been found from 3.7 billion possible design variable combinations.

- It has been found that the reduced number of design variables, constraints, and the objective are all dependent on inlet saturation temperature.
- Machines with an even number of total stages see the even-numbered rotors contribute to the majority of the combined work. This is also true for odd-numbered rotors in machines with an odd total number of stages.
- Most variables and constraints tend to show similarity in this manner (even rotors in even total stage configurations, odd rotors in odd total stage configurations), all of which are temperature-dependent.
- Inspection of the best-point tip speed ratios indicate that the slower rotor in a pair has high flow and blade loading coefficients as well as low specific speed compared to traditional axial compressors. This is a result of the work transfer being very large

relative to the low rotational speed. Large tangential velocity change associated with large work transfer is attained without the rotor itself having to reach this same blade speed, which is a feature of the counter-rotating configuration in general.

- Large flow and blade loading coefficients are achieved without violating any of the imposed Mach number, de Haller number, and Diffusion Factor constraints.
- Simulation of 2, 3, and 4 stage configurations at inlet saturation temperatures of 5°C, 20°C, 40°C, 60°C, 80°C, and 95°C indicate that rotors predicted to contribute the majority of the combined work (even rotors for an even number of total stages, odd rotors for an odd number of total stages) contribute more to the combined work than indicated by analytical calculation, and generally do so at a high efficiency. This counteracts the low efficiency achieved by the slower spinning rotors that contribute less than expected. This results in overall polytropic efficiencies near (and in many cases above) the assumed 80% for each stage (and hence overall).

7.2 Contribution

Research contributions include the following:

- The star pattern impeller is a unique design in the field of turbomachinery, and so a full 3D generalized characterization of the geometry and analytical flow field has been accomplished while ensuring circular fidelity of the blade slots machined into the mandrel. This characterization applies to the 8B pattern described in detail in Chapter 4, and in general also for all other star patterns of any diameter with any amount of tip blade turning. For the geometry created by winding continuous fiber, which is fixed for a first-generation mandrel unaccompanied by a hub guide, an iterative method to then predict flow behavior

and performance characteristics are also developed and described. It has been verified that the star pattern impeller can in general be reasonably described by the forced-vortex condition.

- The use of individually controlled counter-rotating stages in an axial compressor significantly increases the number of design input variables. This work describes how to effectively reduce the number of variables from 38 to 10 (from upwards of 270 billion designs down to 168 million for a 7 stage case at a single set of inlet conditions), eliminating iterations while still fully describing the design space. In particular, it is found that the relative critical Mach number at the tip of each rotor must be set to its maximum permissible limit in order to achieve the maximum combined work transfer for a multistage counter-rotating compressor. The derivation of Equation (2-114) is what allows the relative critical Mach number to be specified at each rotor inlet rather than determined iteratively through the use of another variable, i.e. tip speed ratio. Additionally, the derivation of the expression in Equation (6-2) allows the rotor aspect ratio to be determined as a function of rotor radii, blade angles, and blade number rather than as an input variable itself.
- The exhaustive search of the reduced design space (made possible by the above) for independently controlled counter-rotating compressor stages provides the values of absolute flow angle, critical absolute Mach number, and hub/tip ratio at the inlet to the first rotor as well as the blade turning for each rotor as a function of saturated water vapor temperature and the number of stages used. These are found such that the compressor will impart the maximum combined work transfer

for up to seven stages. The corresponding rotor tip speeds and tip speed ratios are also determined along with the blade angles required to exactly match the incoming flow.

- The exhaustive search has also been performed for seven other gas mixtures with the same information provided as listed above. These gases are air, methane, carbon dioxide, helium, hydrogen, a carbon dioxide/water vapor mixture, and a gas mixture of water vapor and non-condensable gases representative of the contents of the condenser in a geothermal power plant. This and the above can be used as a reference for designing subsonic counter-rotating stages with constant tip radius and free-vortex blades.

7.3 Recommendations for Future Work

Based on the scope of this work, there is vast opportunity to both expand and narrow the inquiry into counter-rotating axial compressors. Since the rotors can be individually controlled, it is theoretically possible that a single fixed set of blade geometries can have a wide operating range with high efficiency and useful work transfer as speeds can be adjusted to minimize incidence into a downstream stage when accounting for deviation from an upstream stage. The results from the exhaustive search of the design space can be used as the starting point for such an investigation.

With the blade turning angles determined for each rotor at all of the inlet conditions studied, an investigation into blade thickness as a function of span-wise and stream-wise location for each rotor should be undertaken to improve polytropic efficiency and pressure ratio as determined by simulation. This should be performed in conjunction with full-wheel simulations

(i.e. no use of periodic surfaces) to give insight into the full flow field and validate the simulation results obtained with periodic surfaces.

Conducting experiments at simulated conditions of a best-point design from an exhaustive search would serve to validate the predictions. The best-point single stage with IGV case compressing air under vacuum (Appendix A4) matching the conditions studied in Chapters 4 and 5 is the logical choice for the next test as a single rotating stage is simplest to implement in a laboratory setting, and there is pre-existing experimental data for comparison. Fluid dynamic similarity laws can be used to adjust for any sizing mismatch that may exist between test apparatus and best-point findings.

A test loop developed by Pohl at MSU [53] shown schematically in Figure 7.1 is ready for use. A vacuum pump is used to control the pressure in the loop. Additionally, it includes a water reservoir that will evaporate to fill the loop with vapor when sufficient air has been evacuated. This enables experiments with water vapor as the working fluid.

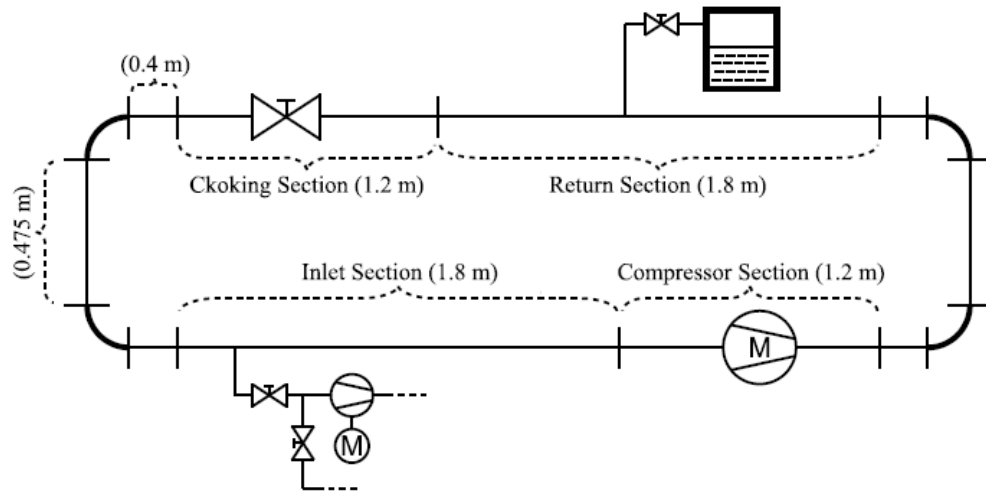


Figure 7.1. Test Loop Schematic Including Vacuum Pump and Water Reservoir for Low Pressure Evaporation [53]

However, the compressor section shown in Figure 7.2 requires modification to accommodate more than one rotor.

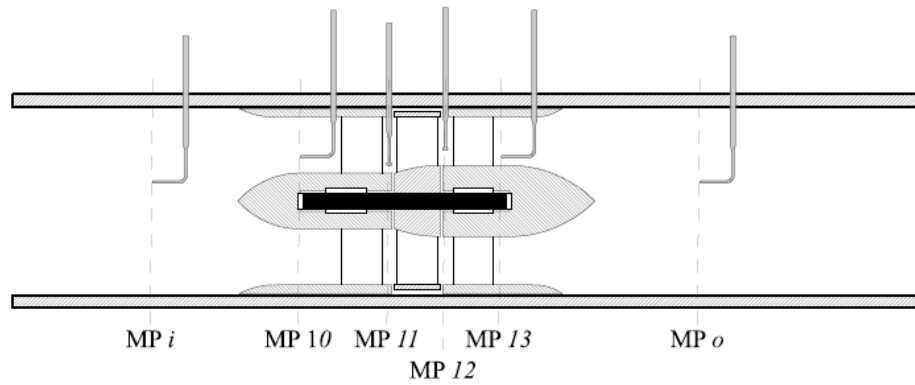


Figure 7.2. Compressor Section Schematic [53]

Additionally, rotors with geometry as determined by the exhaustive search should be manufactured, and the additional VFDs necessary to control the permanent magnet motors driving multistage machines should be purchased.

APPENDICES

APPENDIX 1 Mesh Independence Study for Star Pattern Impeller

Table A1.1. Mesh Independence Study Parameters

		Very Coarse	Coarse	Medium	Medium/Fine	Fine
mm	Max Face					
	Size	12	6	3	2.25	1.5
	Nodes	150,794	335,119	541,671	778,676	1,817,905
	Elements	724,275	1,496,893	1,957,340	2,229,748	3,904,114
kg/s	$\dot{m}$	2.5751	2.5779	2.5748	2.578	2.5672
m ³ /s	$\dot{V}$	9.7853	9.7958	9.7842	9.7965	9.7557
	Π_t	1.0971	1.0708	1.0742	1.0749	1.0712
	Γ_t	1.0416	1.0357	1.0382	1.0371	1.0371
%	η_{is}	71.9704	62.366	62.7315	62.6022	63.2156
%	η_{pt}	72.3809	62.6935	63.1322	62.9516	63.5664

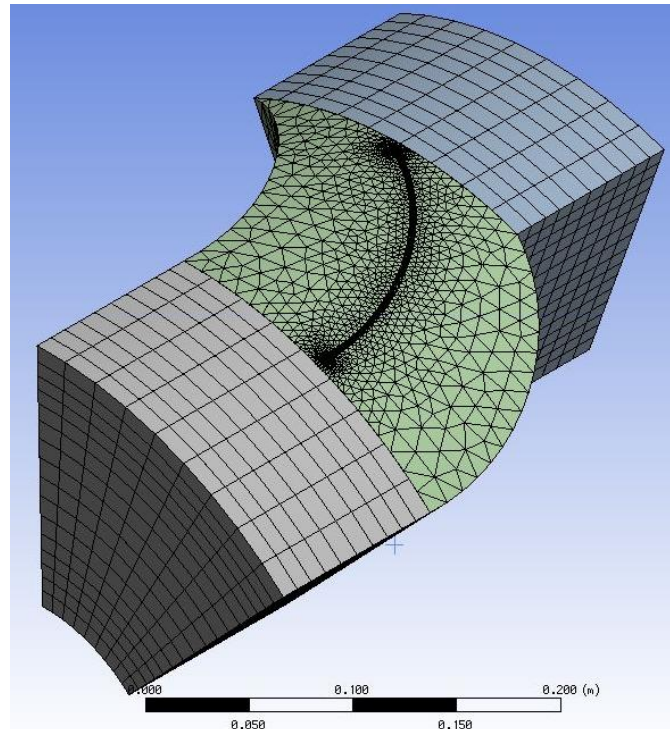


Figure A1.1. Very Coarse Mesh

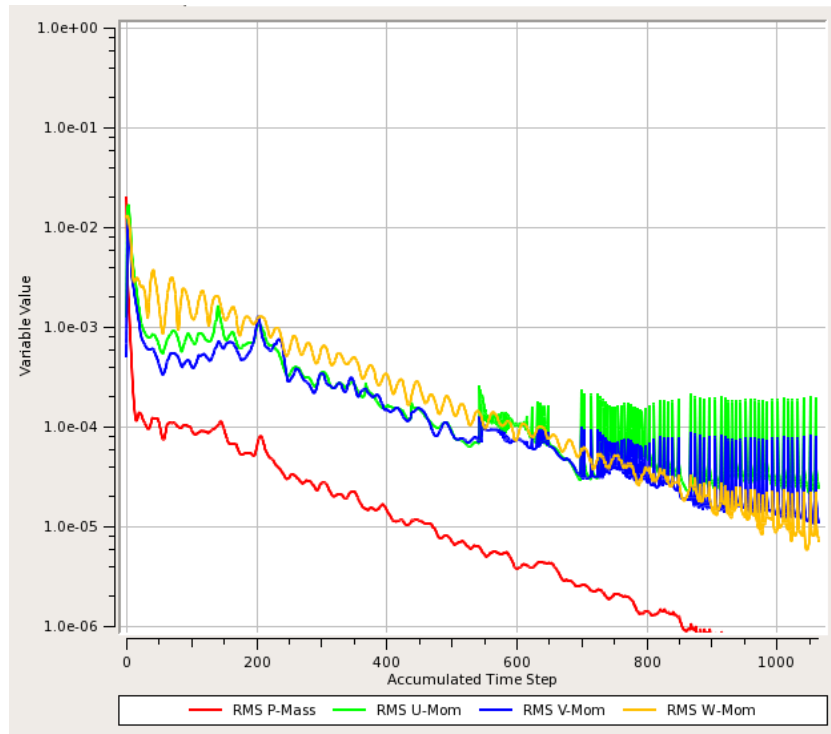


Figure A1.2. Very Coarse Mesh RMS Residuals

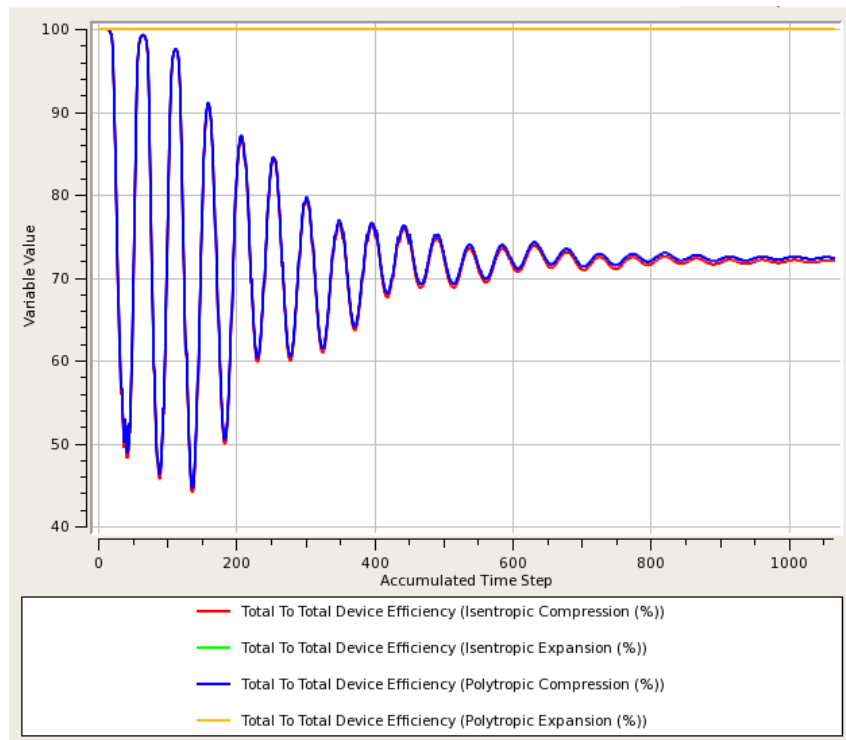


Figure A1.3. Very Coarse Mesh Efficiency Monitor History

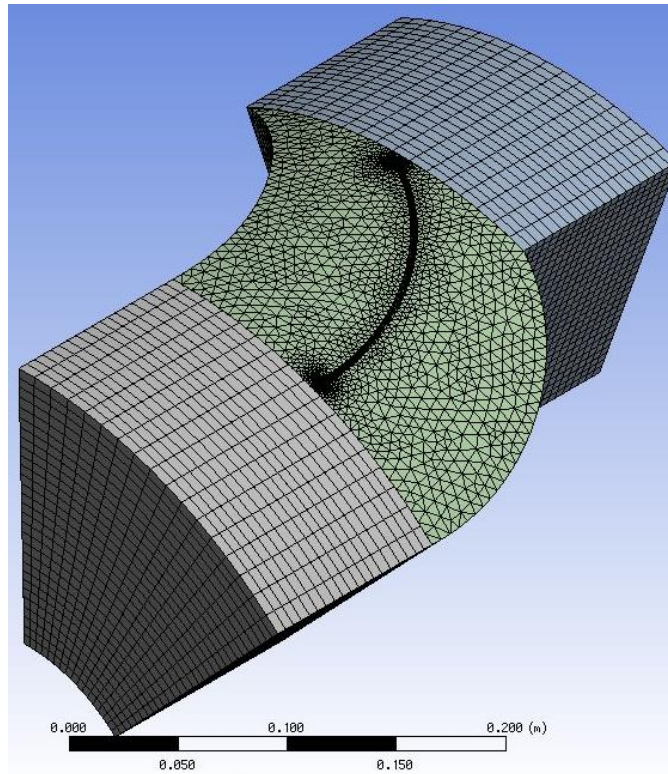


Figure A1.4. Coarse Mesh

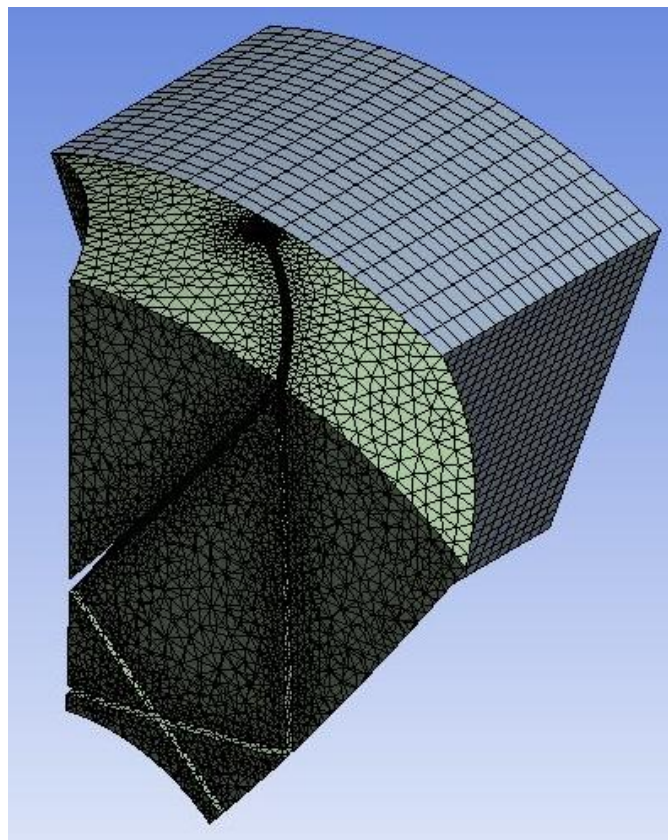


Figure A1.5. Coarse Mesh Section View

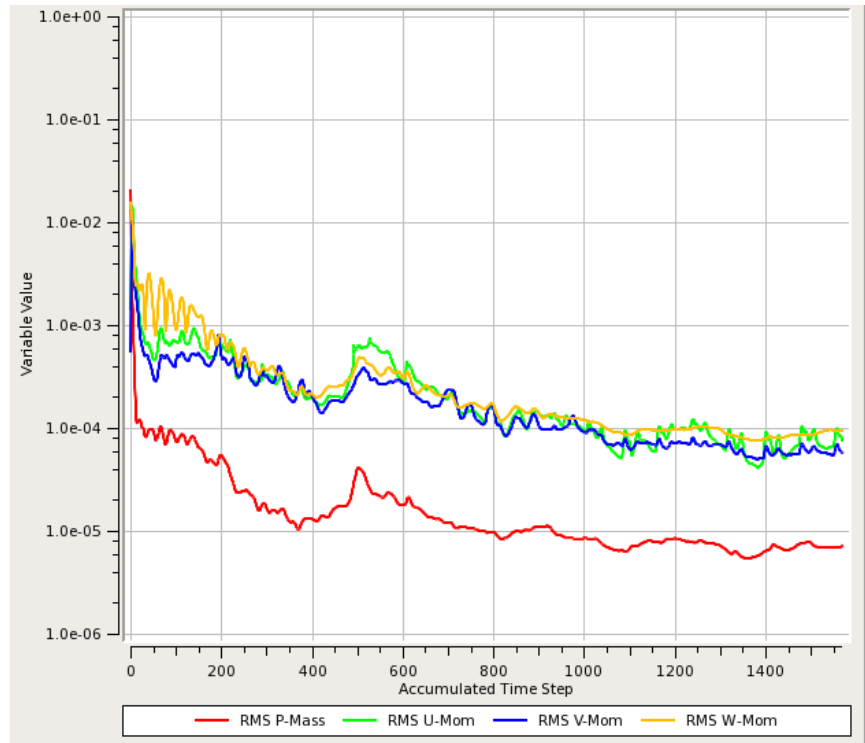


Figure A1.6. Coarse Mesh RMS Residuals

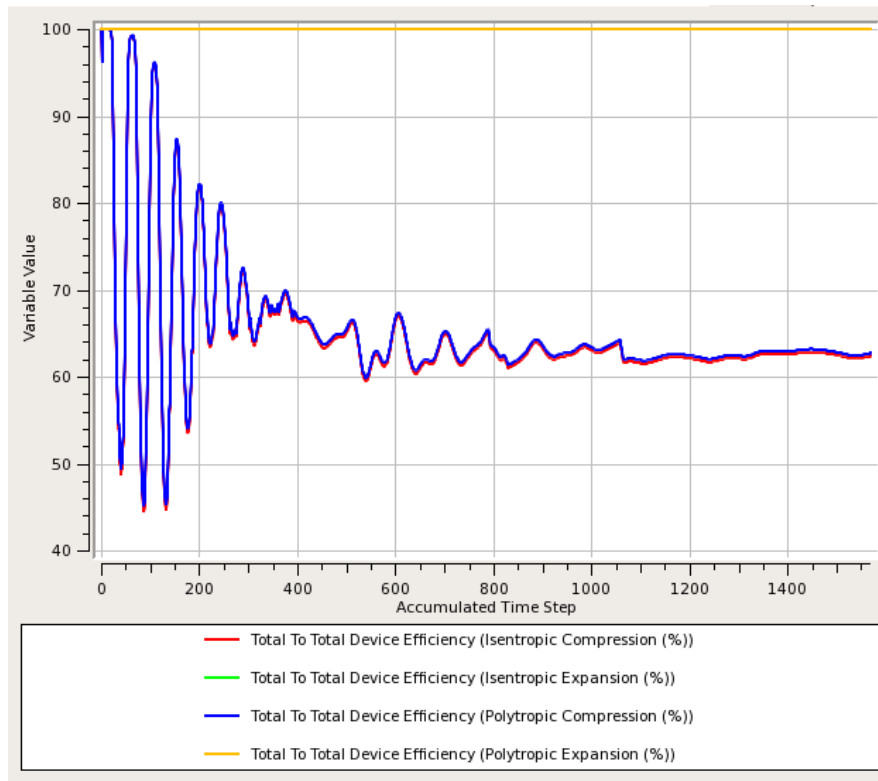


Figure A1.7. Coarse Mesh Efficiency Monitor History

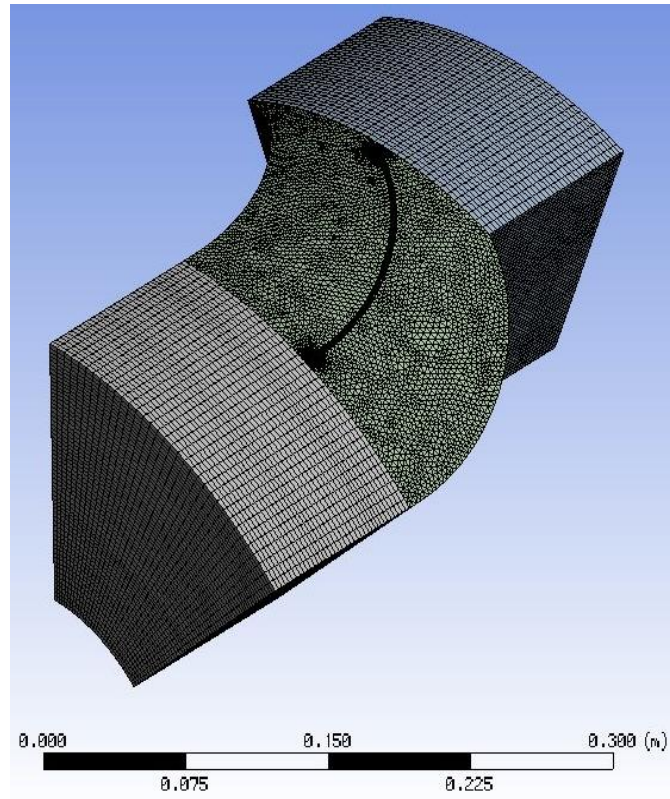


Figure A1.8. Medium Mesh

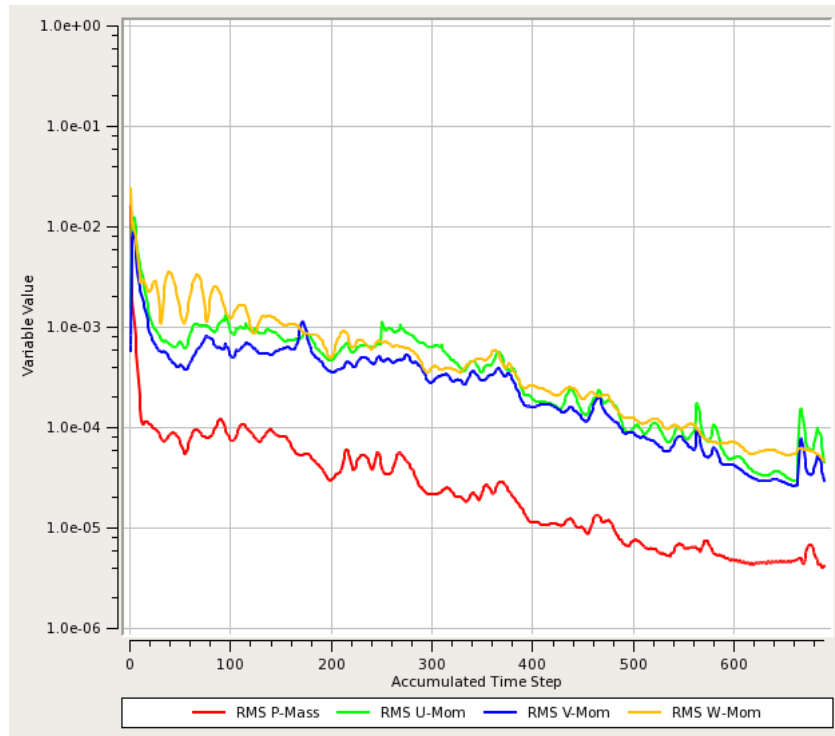


Figure A1.9. Medium Mesh RMS Residuals

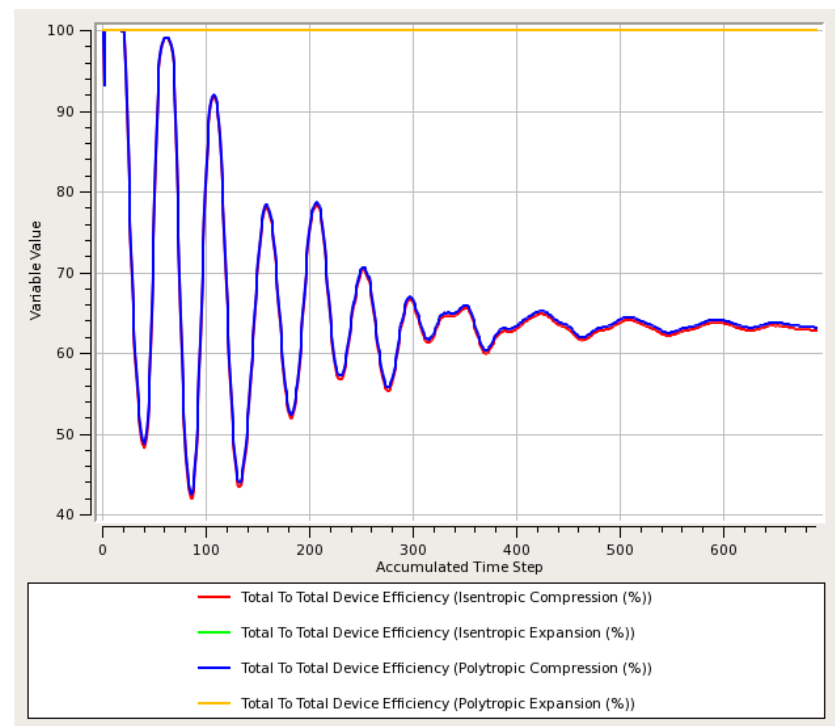


Figure A1.10. Medium Mesh Efficiency Monitor History

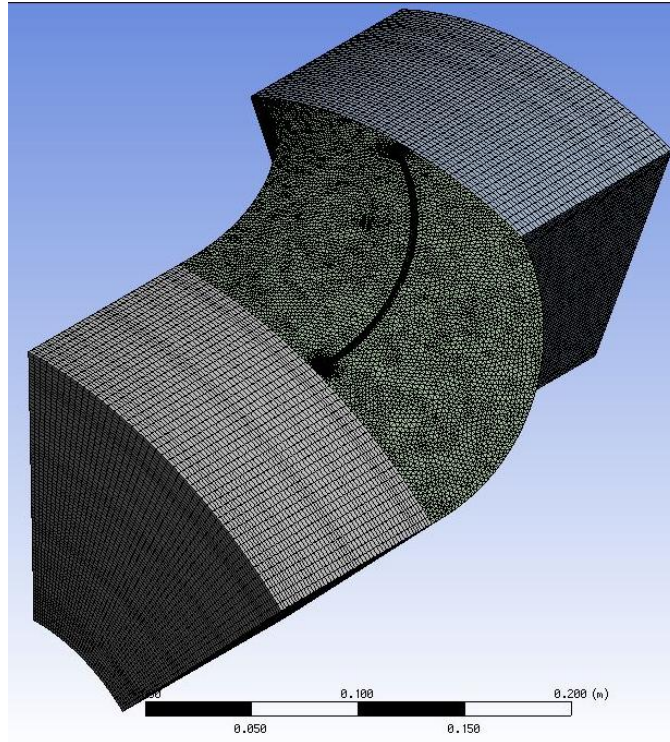


Figure A1.11. Fine Mesh

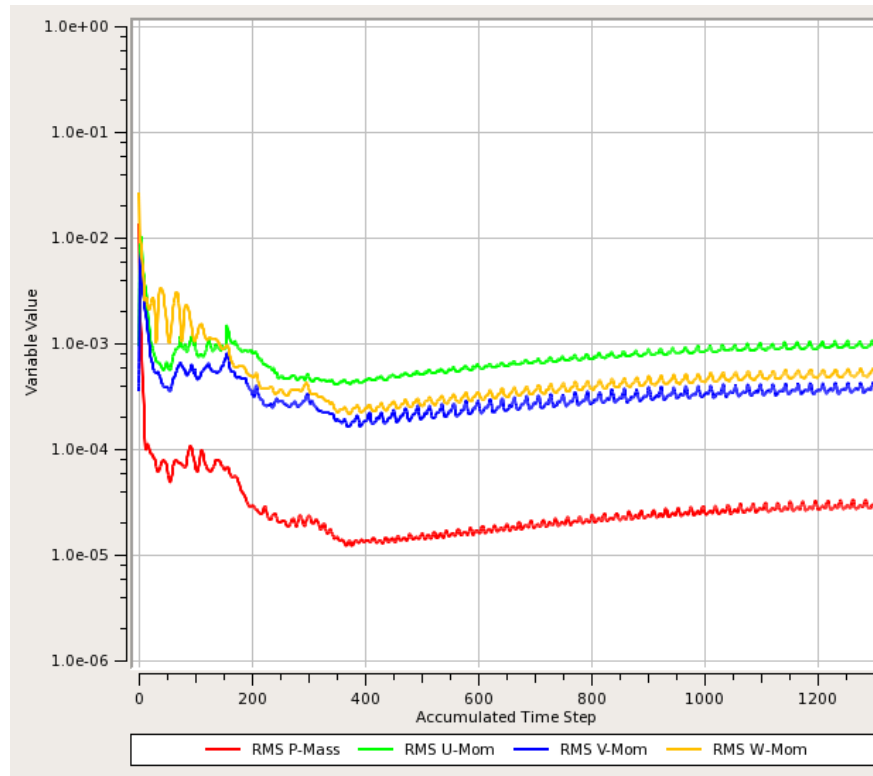


Figure A1.12. Fine Mesh RMS Residuals

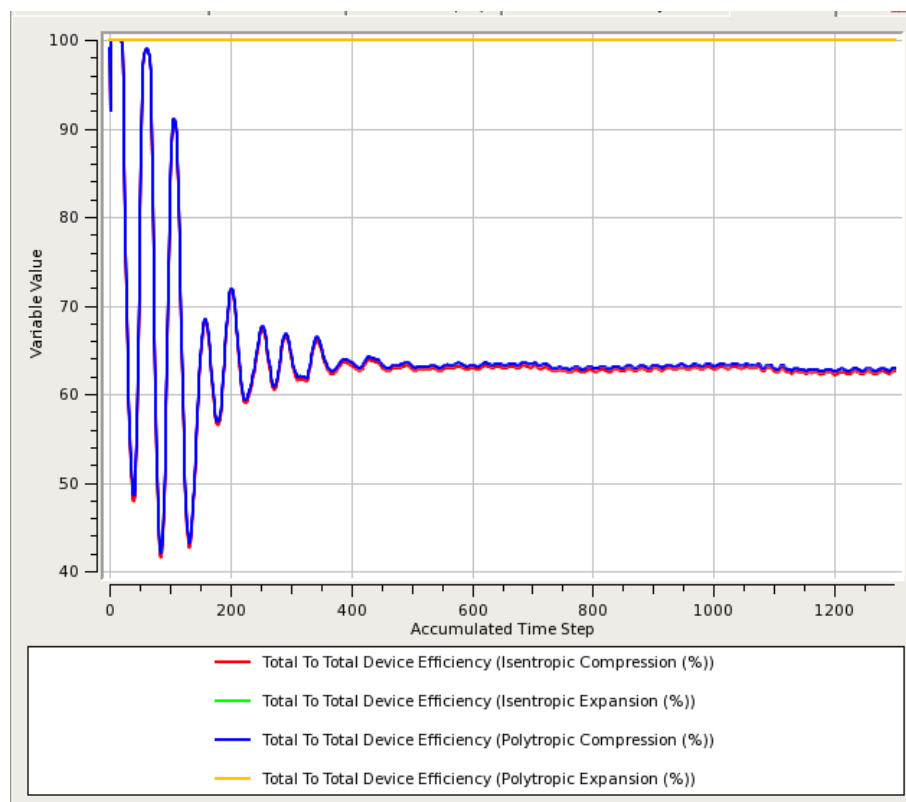


Figure A1.13. Fine Mesh Efficiency Monitor History

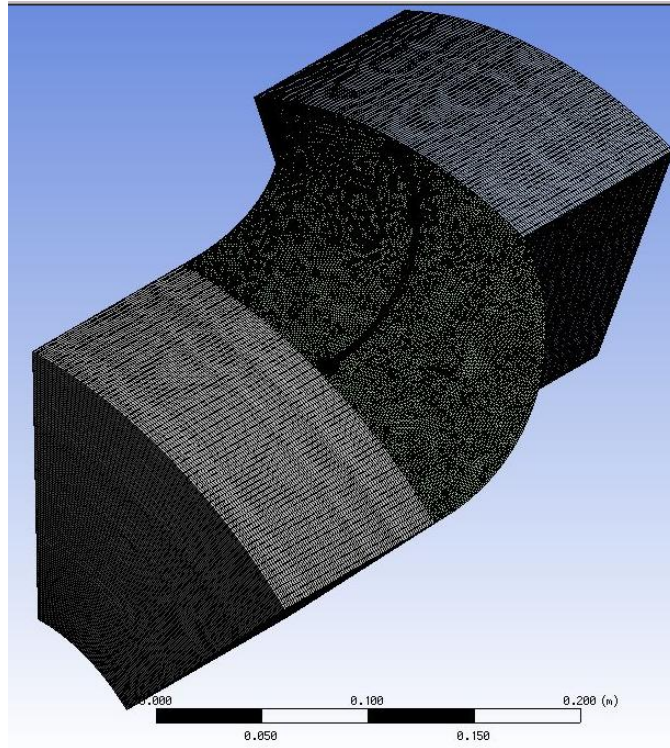


Figure A1.14. Very Fine Mesh

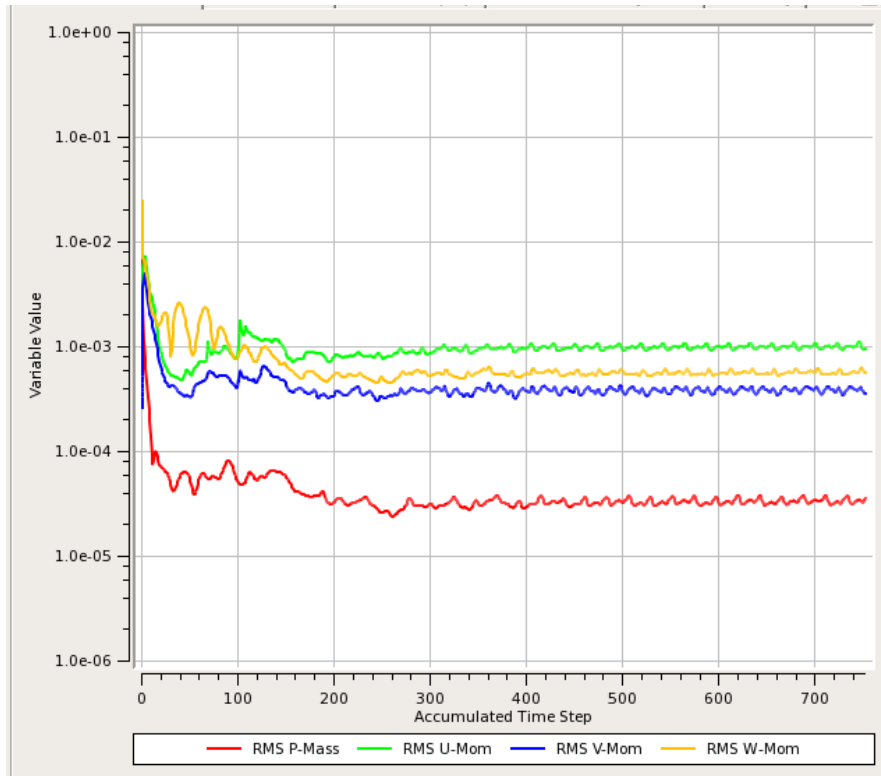


Figure A1.15. Very Fine Mesh RMS Residuals

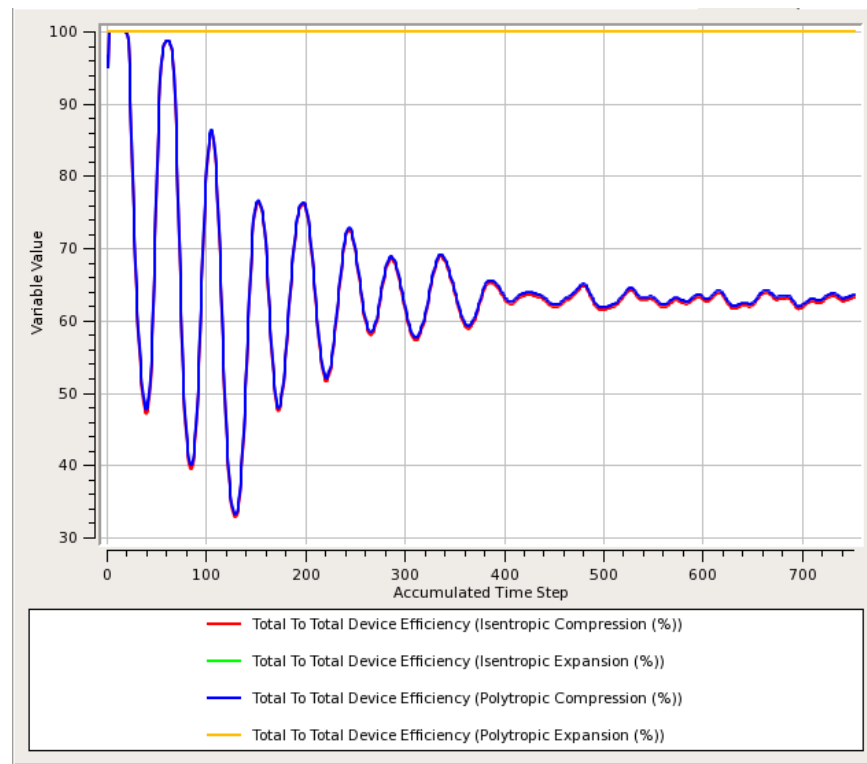


Figure A1.16. Very Fine Mesh Efficiency Monitor History

APPENDIX 2 Polytropic Efficiency

The isentropic efficiency of a turbomachine has general validity, but using it to compare efficiencies of turbomachinery with different pressure ratios can be deceptive [54]. If a turbomachine is considered to have a very large number of very small stages, with *each stage having the same efficiency*, the isentropic efficiency of the entire machine will be less than that of an individual small stage. The following development is based on [54].

The Mollier diagram in Figure A2-1 shows a compression process from p_1 to p_2 , and the corresponding reversible process is illustrated using the isentropic line 1-2s. The compressor stage used to bring the fluid from p_1 to p_2 is divided into a large number of very small stages, each having an equal efficiency of η_{pt} , the polytropic efficiency. For each small stage, the polytropic efficiency is the ratio of the minimum possible (isentropic) work input, $\delta W_{\min}$, to the actual work input, δW . Using the terminology from Figure A2-1, the polytropic efficiency can be written

$$\eta_{pt} = \frac{\delta W_{\min}}{\delta W} = \frac{h_{xs} - h_1}{h_x - h_1} = \frac{h_{ys} - h_x}{h_y - h_x} = \frac{h_{zs} - h_y}{h_z - h_y} = \dots \quad (\text{A2-1})$$

Since each small stage by definition has the same efficiency, it is also true that

$$\eta_{pt} = \frac{\sum \delta W_{\min}}{\sum \delta W} \quad (\text{A2-2})$$

Equation (2-24) for a constant pressure process (i.e. $dp = 0$) can be written

$$Tds = dh - vdp \quad (2-24)$$

$$Tds = dh \quad (\text{A2-3})$$

$$\left. \frac{\partial h}{\partial s} \right|_p = T \quad (\text{A2-4})$$

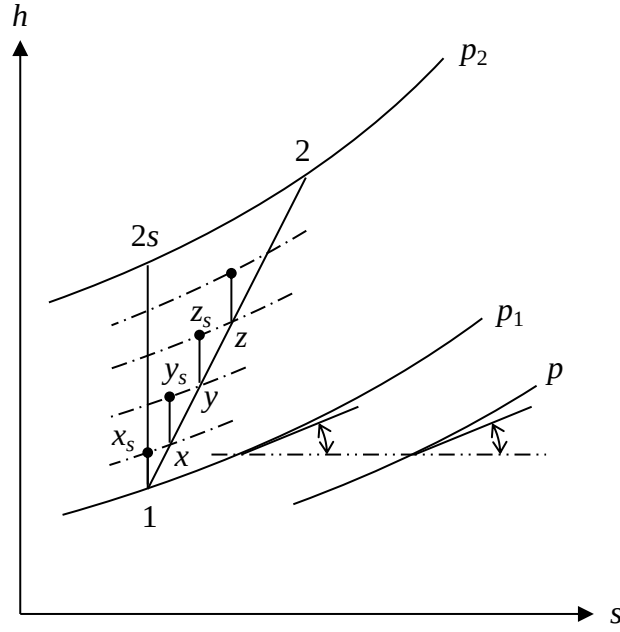


Figure A2.1. Mollier Diagram of Small-Stage Compression Processes. Adapted from [54]

and hence for a greater fluid temperature, the constant pressure lines on the Mollier diagram have a greater slope. For a gas with $h = h(T)$, the slope of the constant pressure line p_2 is greater than the slope of constant pressure line p_1 at a given entropy. Additionally, as Equation (A2-4) indicates and as is depicted in Figure A2-1, for a given temperature, the constant pressure lines have equal slope.

For the more general case, the denominator of Equation (A2-2) can be expanded as

$$\Sigma \delta W = W = (h_x - h_1) + (h_y - h_x) + (h_z - h_y) + \dots = (h_2 - h_1) \quad (\text{A2-5})$$

and so

$$\eta_{pt} = \frac{(h_{xs} - h_1) + (h_{ys} - h_x) + (h_{zs} - h_y) + \dots}{(h_2 - h_1)} \quad (\text{A2-6})$$

The isentropic efficiency of the entire process is

$$\eta_{is} = \frac{(h_{2s} - h_1)}{(h_2 - h_1)} = \frac{W_{\min}}{W} \quad (\text{A2-7})$$

Since the constant pressure lines diverge, it is clear that

$$(h_{xs} - h_1) + (h_{ys} - h_x) + (h_{zs} - h_y) + \dots > (h_{2s} - h_1) \quad (\text{A2-8})$$

and so

$$\frac{\Sigma \delta W_{\min}}{W} > \frac{W_{\min}}{W} \quad (\text{A2-9})$$

hence

$$\eta_{pt} > \eta_{is} \quad (\text{A2-10})$$

thereby showing that the isentropic efficiency of a compressor is less than that of the small stage i.e. polytropic efficiency, this being attributable to the divergence of the constant pressure lines. This also applies to stagnation properties if kinetic energies are equal at the compressor inlet and outlet.

The change in enthalpy of an ideal gas is related to a change in temperature proportional to the specific heat:

$$dh = c_p dT \quad (2-25)$$

For a *single* small stage, the polytropic efficiency is by definition

$$\eta_{pt} = \frac{dh_{is}}{dh} \quad (\text{A2-11})$$

as shown in Figure A2-2

When the relation

$$Tds = dh - vdp \quad (2-24)$$

is considered for the isentropic case (i.e. $ds = 0$), (2-24) becomes

$$dh_{is} = vdp \quad (\text{A2-12})$$

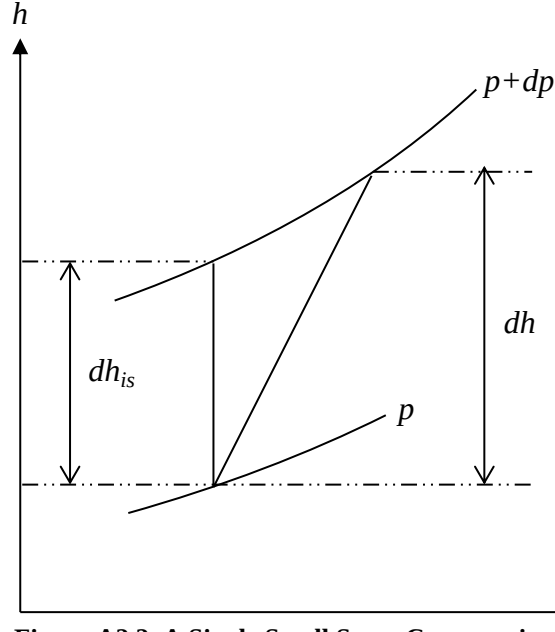


Figure A2.2. A Single Small Stage Compression Process

Substituting Equations (2-25) and (A2-12) into the numerator and denominator of (A2-11), respectively, gives

$$\eta_{pt} = \frac{v dp}{c_p dT} \quad (\text{A2-13})$$

The ideal gas law given in (2-27) can be written in terms of specific volume as

$$v = \frac{RT}{p} \quad (\text{A2-14})$$

Substituting Equation (A2-14) into (A2-13) yields

$$\eta_{pt} = \frac{RT dp}{c_p p dT} \quad (\text{A2-15})$$

Further substitution of (2-15) for R/c_p into (A2-15) after some rearrangement gives

$$\frac{dT}{T} = \frac{\gamma - 1}{\gamma \eta_{pt}} \frac{dp}{p} \quad (\text{A2-16})$$

Integrating Equation (A2-16) across the entire compressor and maintaining equal polytropic efficiency for each of the small stages gives

$$\int_1^2 \frac{dT}{T} = \int_1^2 \frac{\gamma-1}{\gamma \eta_{pt}} \frac{dp}{p} \quad (\text{A2-17})$$

resulting in

$$\frac{T_2}{T_1} = \left(\frac{p_2}{p_1} \right)^{\frac{\gamma-1}{\gamma \eta_{pt}}} \quad (\text{A2-18})$$

The isentropic efficiency for the entire compressor is

$$\eta_{is} = \frac{h_{2s} - h_1}{h_2 - h_1} = \frac{c_p (T_{2s} - T_1)}{c_p (T_2 - T_1)} \quad (\text{A2-19})$$

For the ideal process (i.e. $\eta_{pt} = 1$), Equation (A2-18) becomes

$$\frac{T_{2s}}{T_1} = \left(\frac{p_2}{p_1} \right)^{\frac{\gamma-1}{\gamma}} \quad (\text{A2-20})$$

Equation (A2-19) can be rewritten as

$$\eta_{is} = \frac{T_1 \left(\frac{T_{2s}}{T_1} - 1 \right)}{T_1 \left(\frac{T_2}{T_1} - 1 \right)} \quad (\text{A2-21})$$

Appropriately substituting Equations (A2-18) and (A2-20) into (A2-21) allows the isentropic efficiency to be expressed in terms of pressure ratio and polytropic efficiency:

$$\eta_{is} = \frac{\left(\frac{p_2}{p_1} \right)^{\frac{\gamma-1}{\gamma}} - 1}{\left(\frac{p_2}{p_1} \right)^{\frac{\gamma-1}{\gamma \eta_{pt}}} - 1} \quad (\text{A2-22})$$

Conversely, the polytropic efficiency can be expressed in terms of pressure ratio and isentropic efficiency through manipulation of (A2-22):

$$\eta_{pt} = \frac{\ln\left(\frac{p_2}{p_1}\right)^{\frac{\gamma-1}{\gamma}}}{\ln\left\{\frac{\left(\frac{p_2}{p_1}\right)^{\frac{\gamma-1}{\gamma}} - 1}{\eta_{is}} + 1\right\}} \quad (\text{A2-23})$$

Similar to Equation (2-92), substituting (A2-18) into (2-85) and assuming negligible changes in kinetic energy from inlet to outlet gives the specific work done by the compressor in terms of stagnation pressure ratio and polytropic efficiency:

$$\tilde{e} = c_p T_{t1} \left(\left(\frac{p_{t2}}{p_{t1}} \right)^{\frac{\gamma-1}{\gamma \eta_{pt}}} - 1 \right) \quad (\text{A2-24})$$

Figure A2-3 plots isentropic efficiency vs. pressure ratio using polytropic efficiency as a parameter using the result from Equation (A2-22). This serves to further the case that compressors with equal polytropic efficiency will exhibit lower isentropic efficiency at higher pressure ratios.

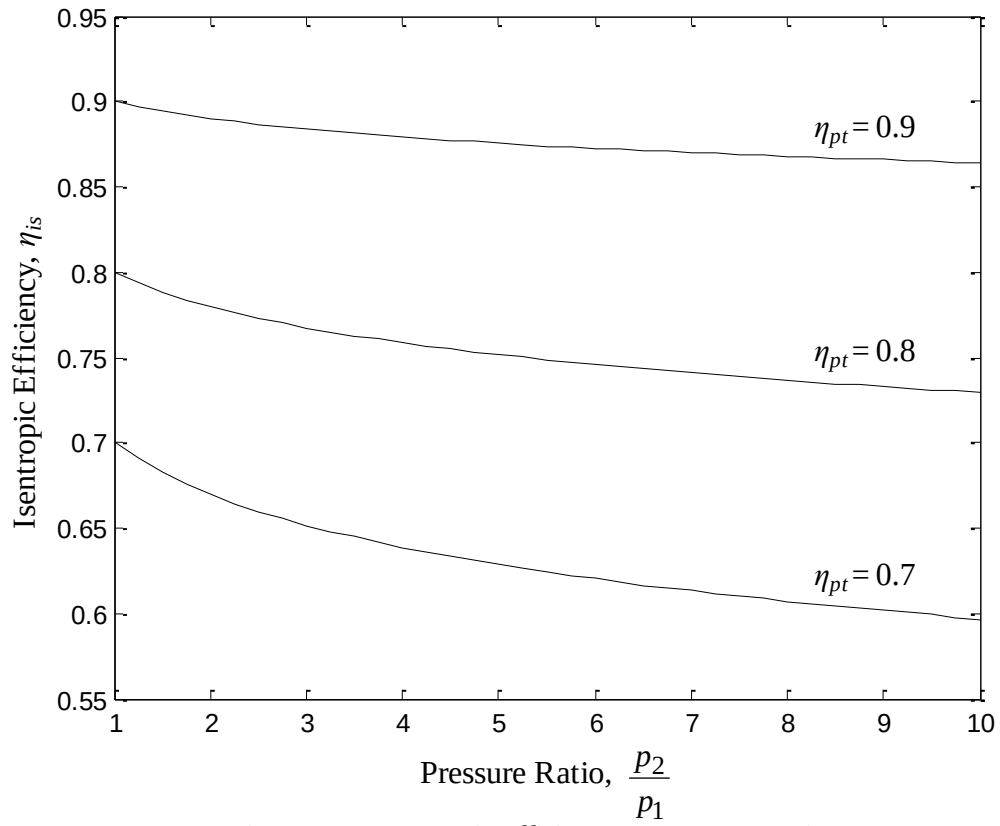


Figure A2.3. Isentropic Efficiency vs. Pressure Ratio

APPENDIX 3 Supplemental Objective and Constraint Sensitivity Plots

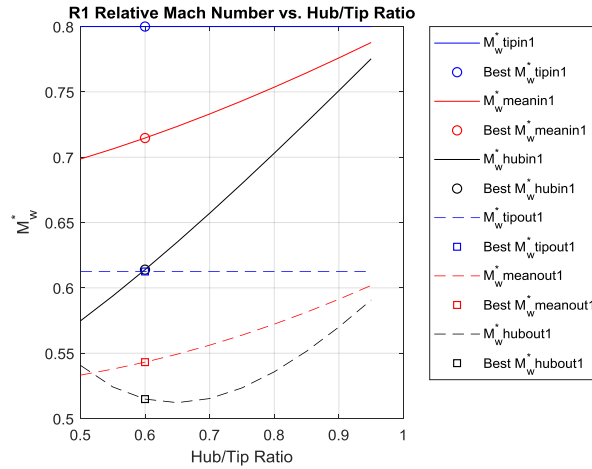


Figure A3.1. First Rotor Relative Mach Number vs. Hub/Tip Ratio

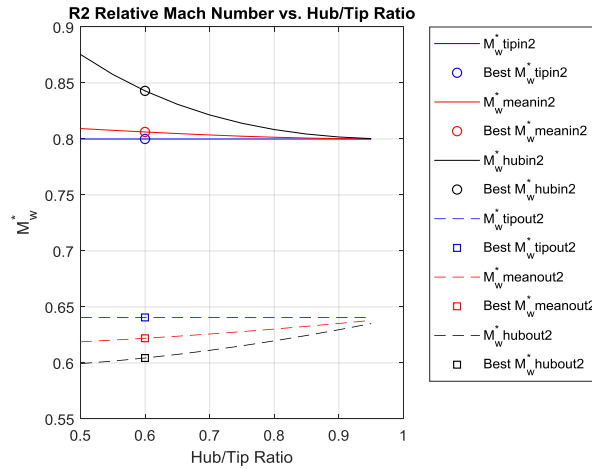


Figure A3.2. Second Rotor Relative Mach Number vs. Hub/Tip Ratio

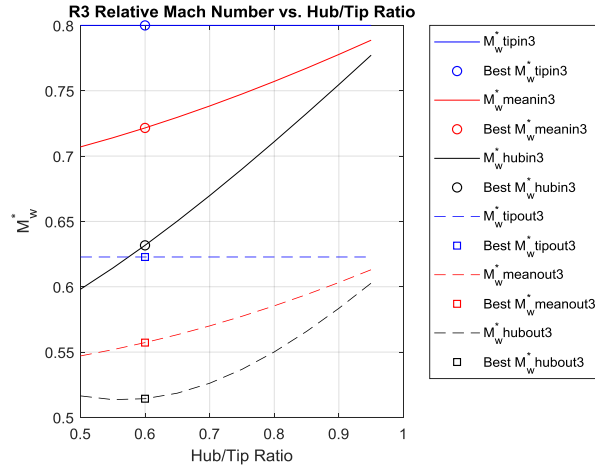


Figure A3.3. Third Rotor Relative Mach Number vs. Hub/Tip Ratio

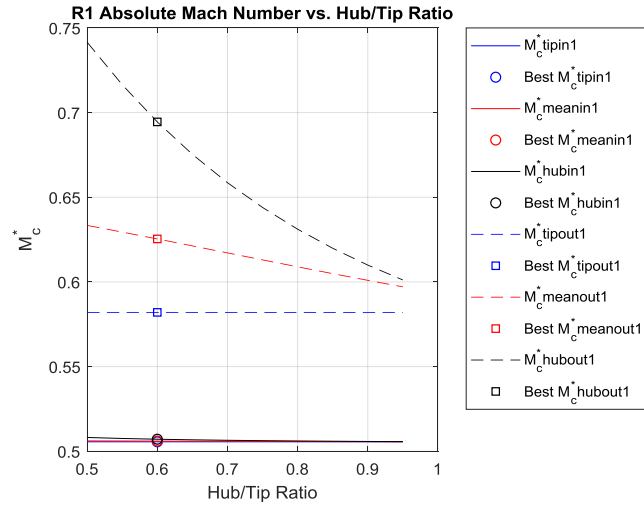


Figure A3.4. First Rotor Absolute Mach Number vs. Hub/Tip Ratio

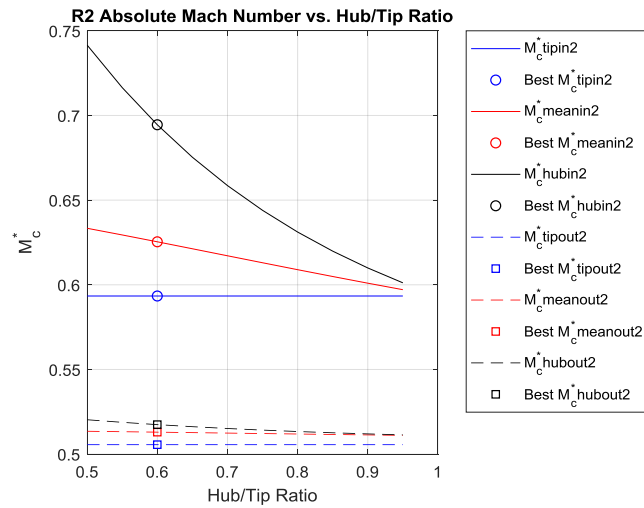


Figure A3.5. Second Rotor Absolute Mach Number vs. Hub/Tip Ratio

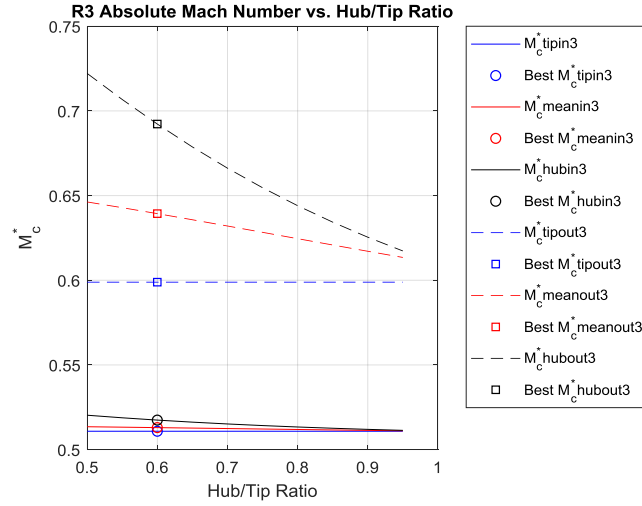


Figure A3.6. Third Rotor Absolute Mach Number vs. Hub/Tip Ratio

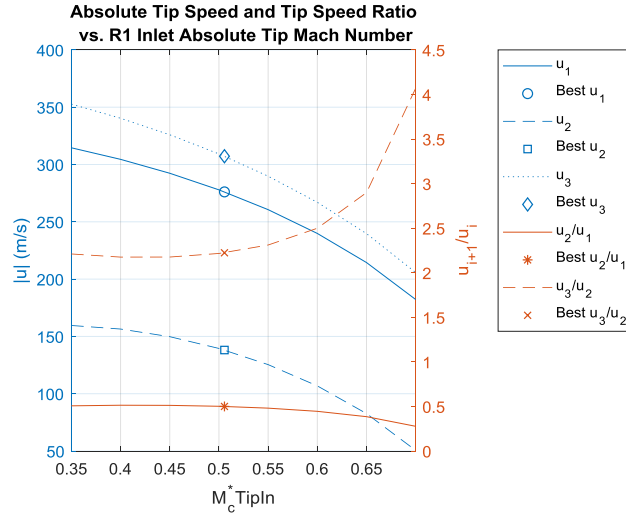


Figure A3.7. Tip Speed and Tip Speed Ratio vs. Absolute Mach Number at First Rotor Inlet

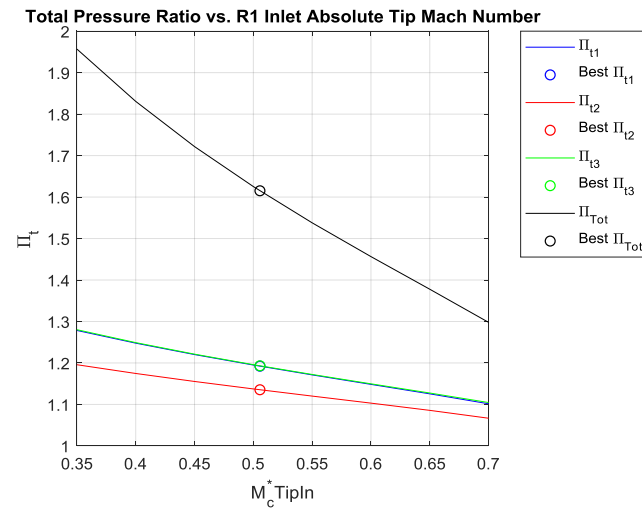


Figure A3.8. Total Pressure Ratio vs. Absolute Mach Number at First Rotor Inlet

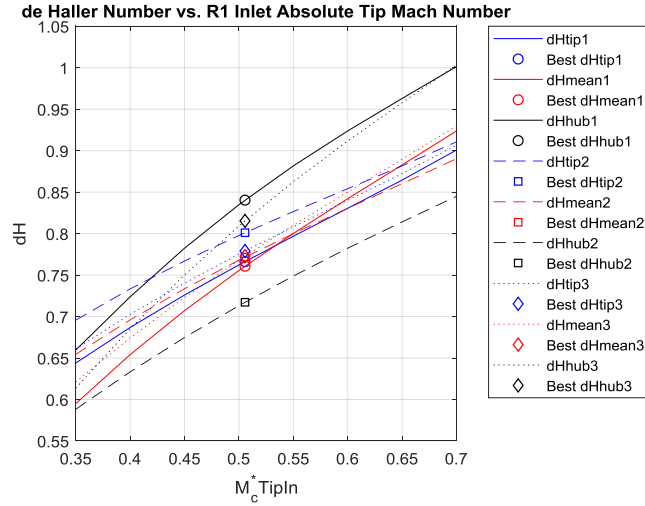


Figure A3.9. de Haller Number vs. Absolute Mach Number at First Rotor Inlet

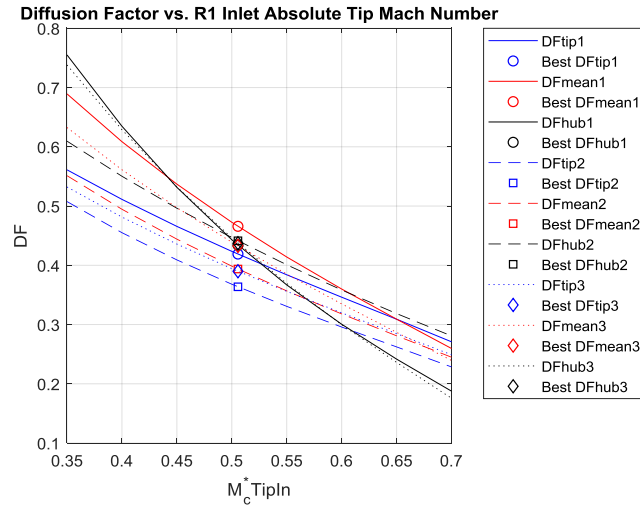


Figure A3.10. Diffusion Factor vs. Absolute Mach Number at First Rotor Inlet

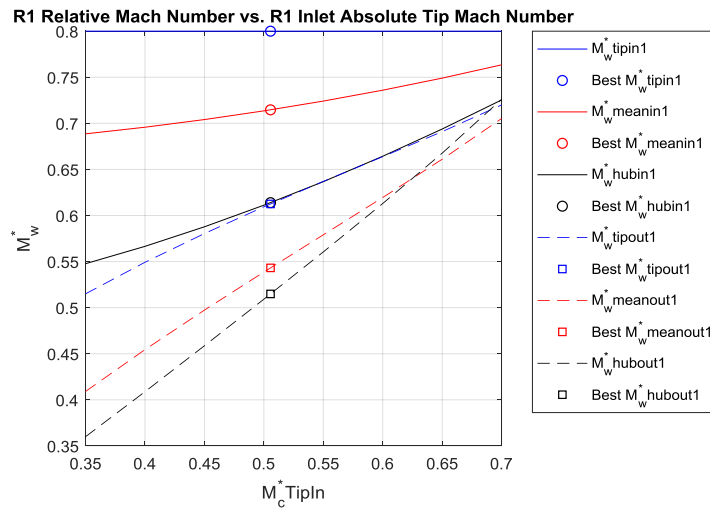


Figure A3.11. First Rotor Relative Mach Number vs. Absolute Mach Number at First Rotor Inlet

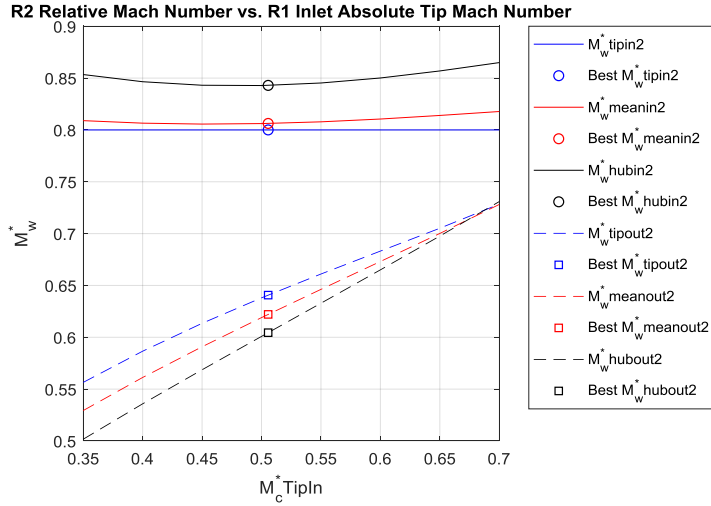


Figure A3.12. Second Rotor Relative Mach Number vs. Absolute Mach Number at First Rotor Inlet

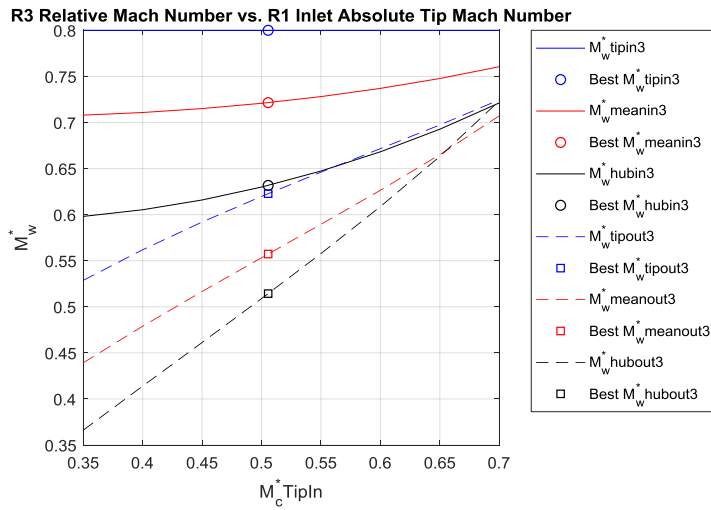


Figure A3.13. Third Rotor Relative Mach Number vs. Absolute Mach Number at First Rotor Inlet

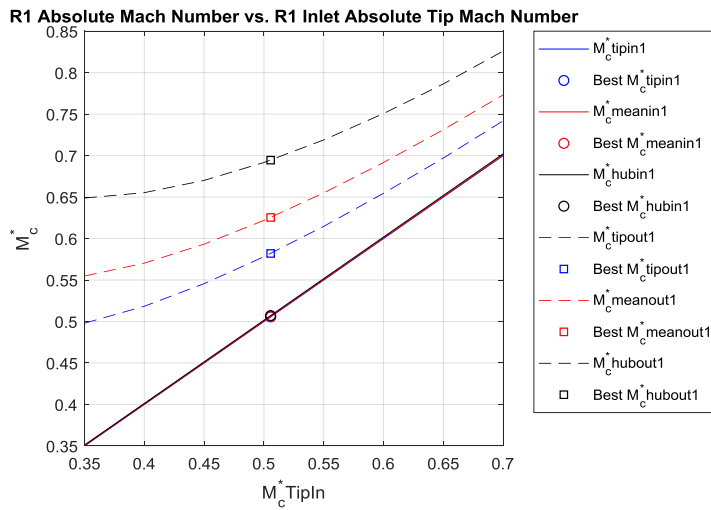


Figure A3.14. First Rotor Absolute Mach Number vs. Absolute Mach Number at First Rotor Inlet

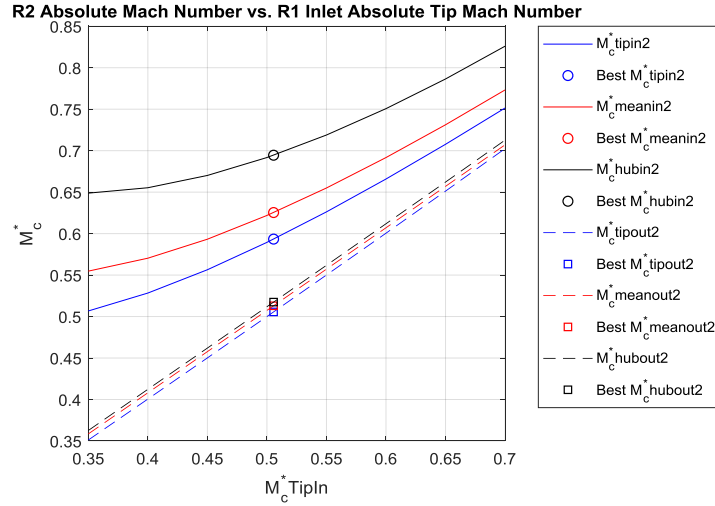


Figure A3.15. Second Rotor Absolute Mach Number vs. Absolute Mach Number at First Rotor Inlet

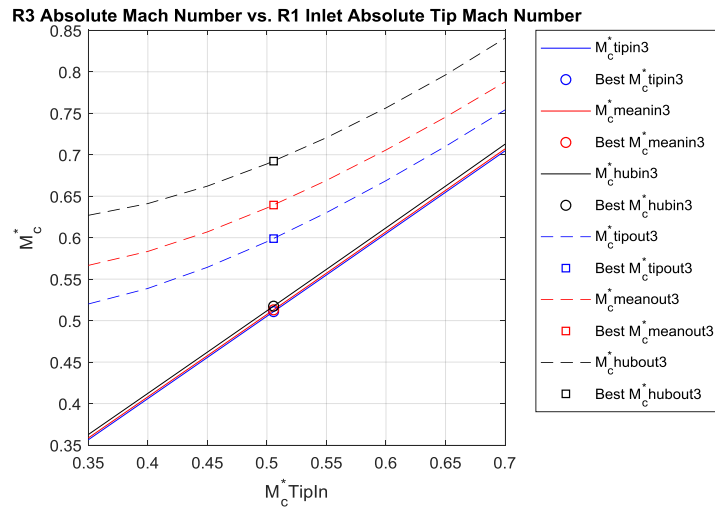


Figure A3.16. Third Rotor Absolute Mach Number vs. Absolute Mach Number at First Rotor Inlet

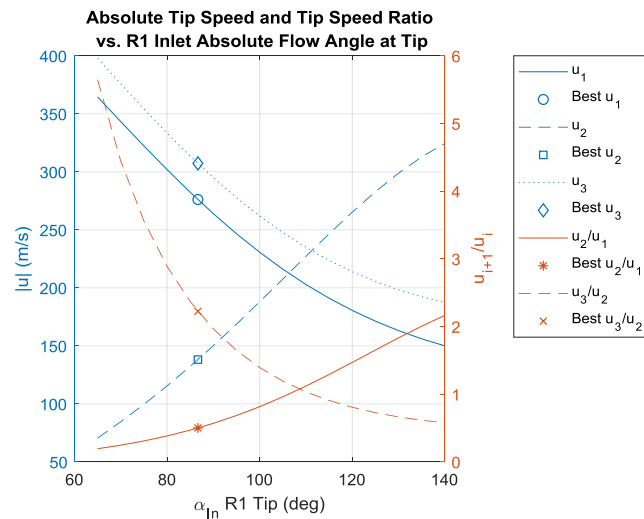


Figure A3.17. Tip Speed and Tip Speed Ratio vs. Absolute Flow Angle at First Rotor Inlet

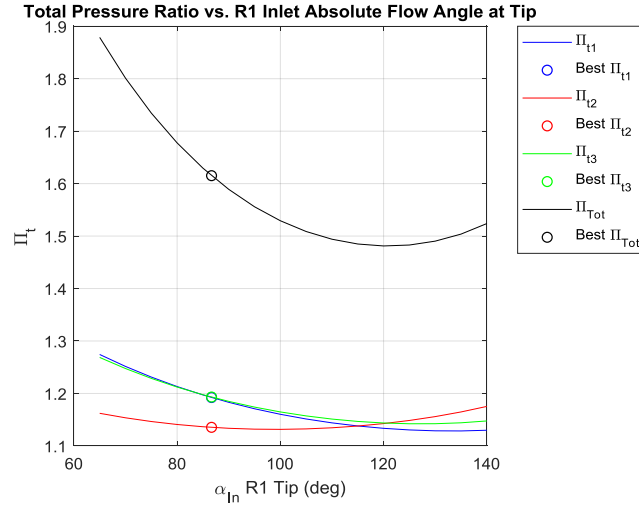


Figure A3.18. Total Pressure Ratio vs. Absolute Flow Angle at First Rotor Inlet

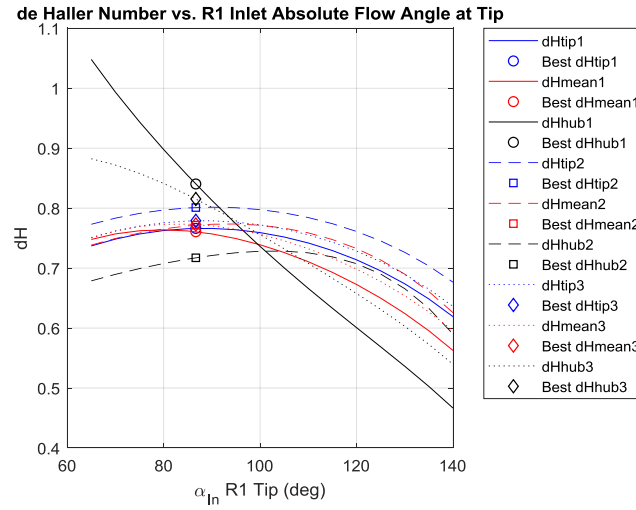


Figure A3.19. de Haller Number vs. Absolute Flow Angle at First Rotor Inlet

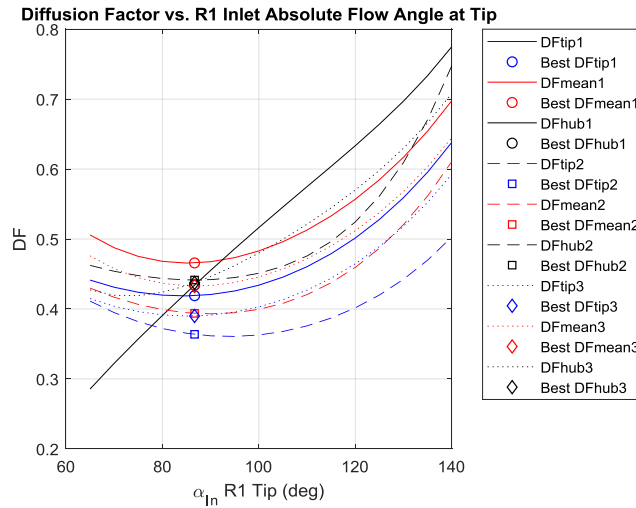


Figure A3.20. Diffusion Factor vs. Absolute Flow Angle at First Rotor Inlet

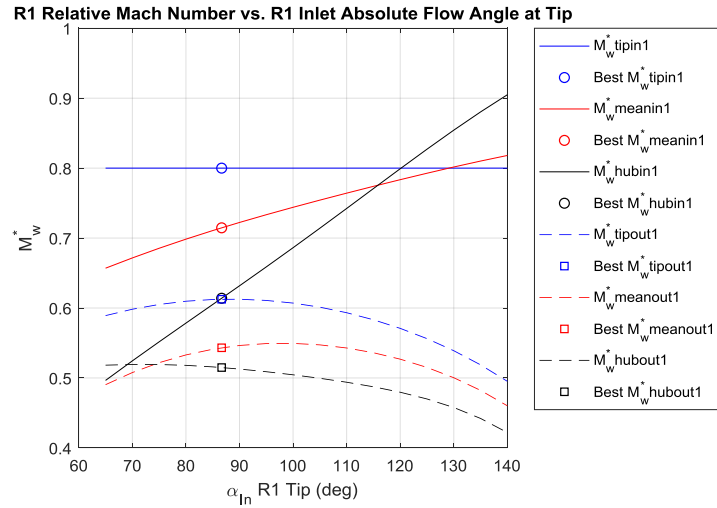


Figure A3.21. First Rotor Relative Mach Number vs. Absolute Flow Angle at First Rotor Inlet

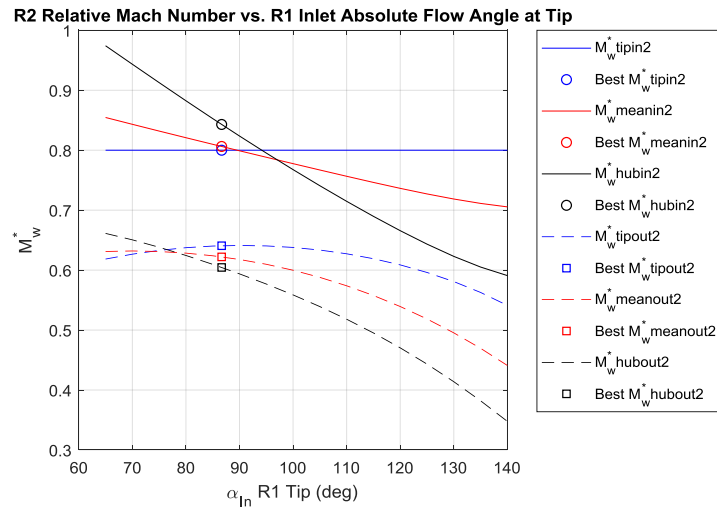


Figure A3.22. Second Rotor Relative Mach Number vs. Absolute Flow Angle at First Rotor Inlet

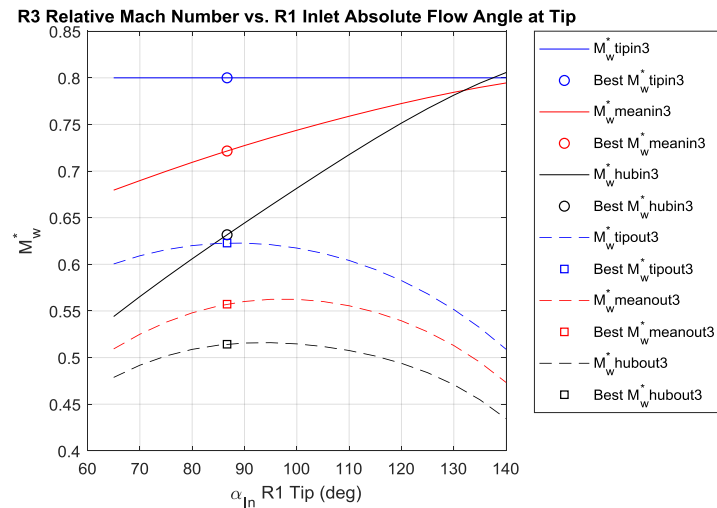


Figure A3.23. Third Rotor Relative Mach Number vs. Absolute Flow Angle at First Rotor Inlet

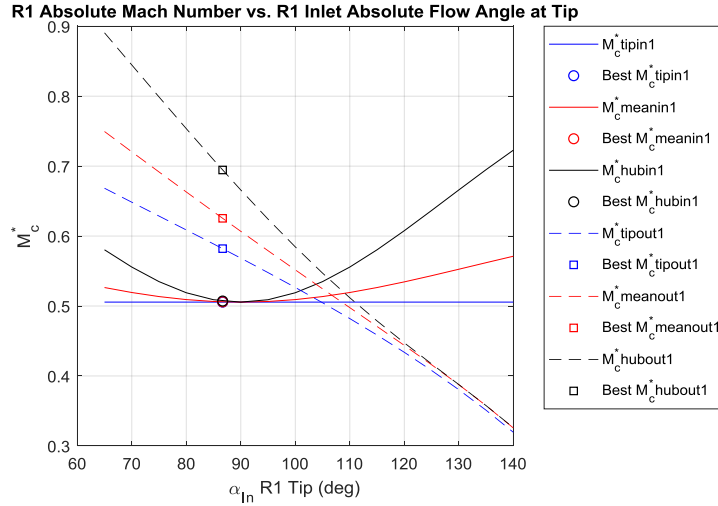


Figure A3.24. First Rotor Absolute Mach Number vs. Absolute Flow Angle at First Rotor Inlet

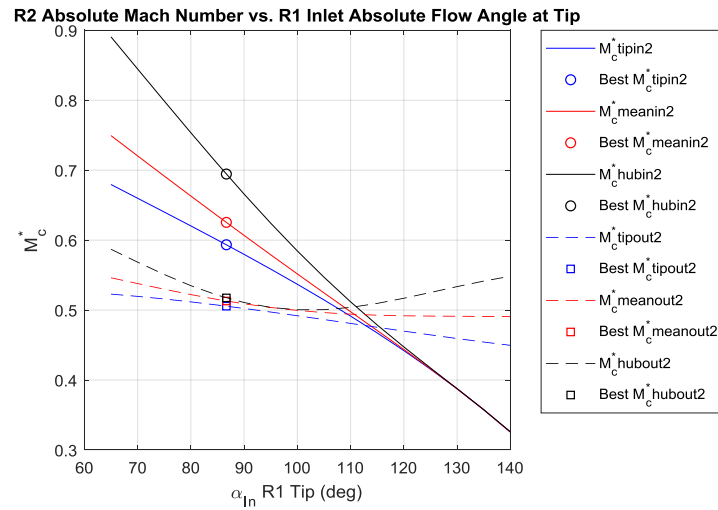


Figure A3.25. Second Rotor Absolute Mach Number vs. Absolute Flow Angle at First Rotor Inlet

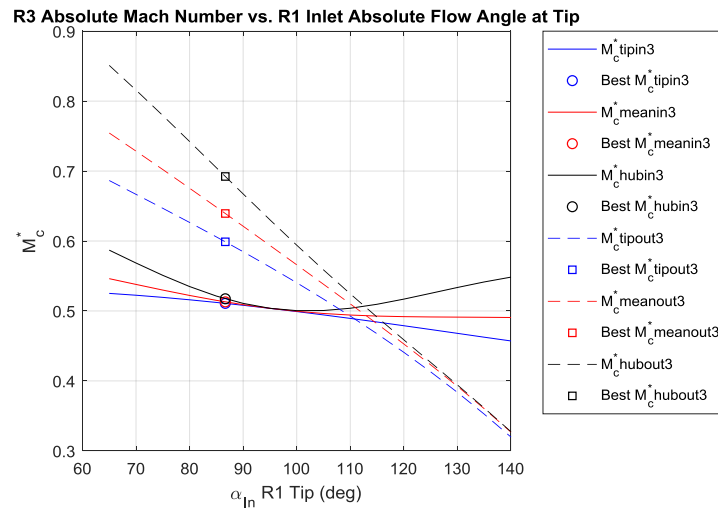


Figure A3.26. Third Rotor Absolute Mach Number vs. Absolute Flow Angle at First Rotor Inlet

APPENDIX 4 Simulation Results for Best-Point Design of Single Stage with IGV Compressing Air Under Vacuum Pressure

The maximum work transfer for a single rotor with an IGV was determined using the techniques detailed in Chapter 6. The IGV inlet conditions ($p_t = 22.75\text{kPa}$, $T_t = 301.15\text{K}$) matched the rotor inlet conditions of the investigations regarding the star pattern impeller in Chapter 4 and the FVB impeller in Chapter 5 (no IGV used in Chapters 4 and 5). The rotational speed determined by the exhaustive search is 7640rpm (star pattern and FVB impellers rotated at 7500rpm). Near uniform total pressure increase is seen from inlet to outlet. Radial velocities are small, and negative radial velocity magnitude is below 1m/s. Boundary layer growth is present on the hub, shroud, and blades. There is healthy flow at nearly all spans. Spans 0.9 and larger see the onset of boundary layer separation, but there are no negative axial velocities present.

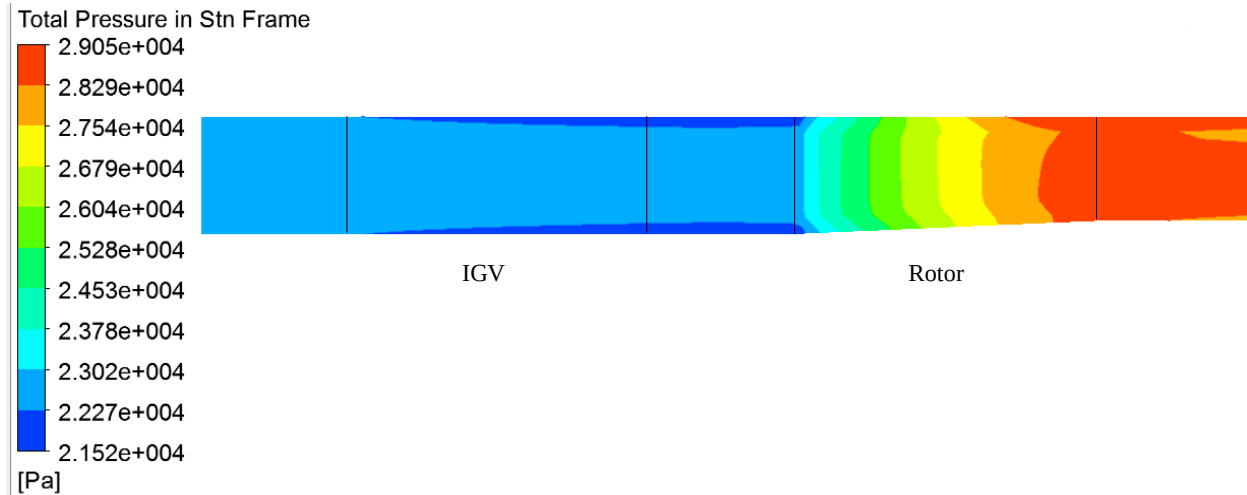


Figure A4.1. Contours of Total Pressure in Meridional Plane

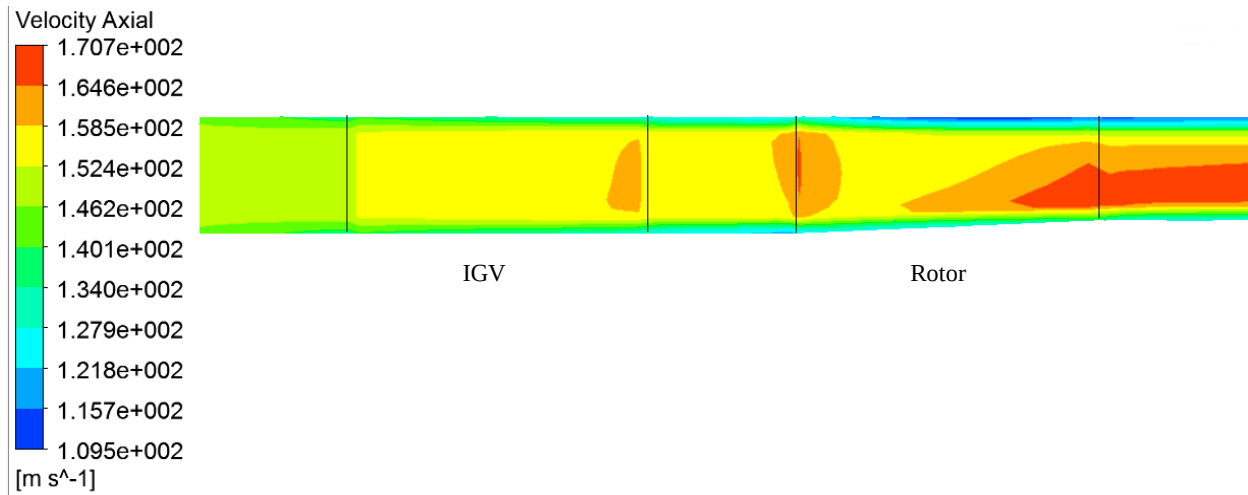


Figure A4.2. Contours of Axial Velocity in Meridional Plane

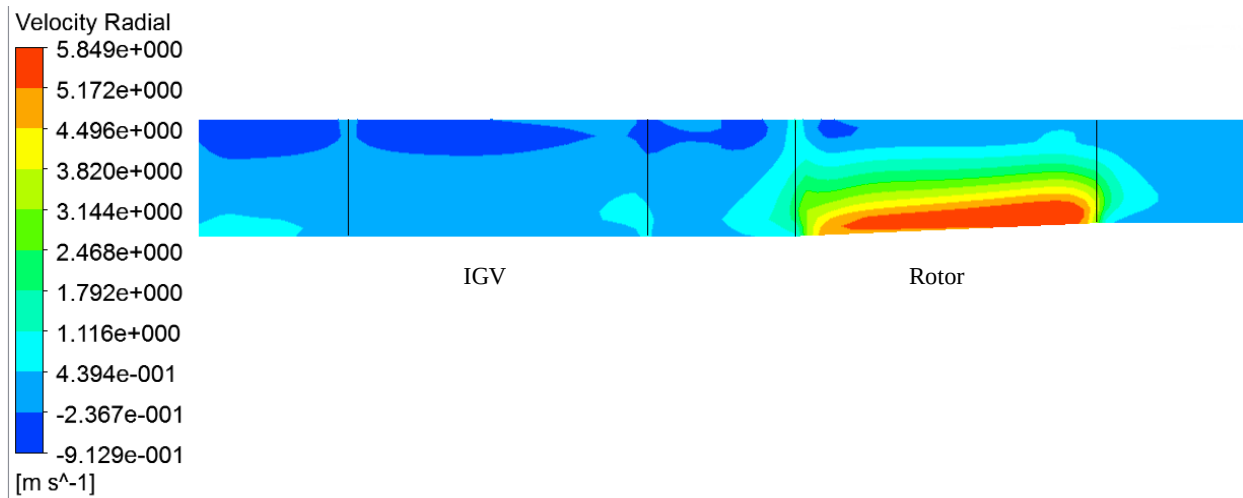


Figure A4.3. Contours of Radial Velocity in Meridional Plane

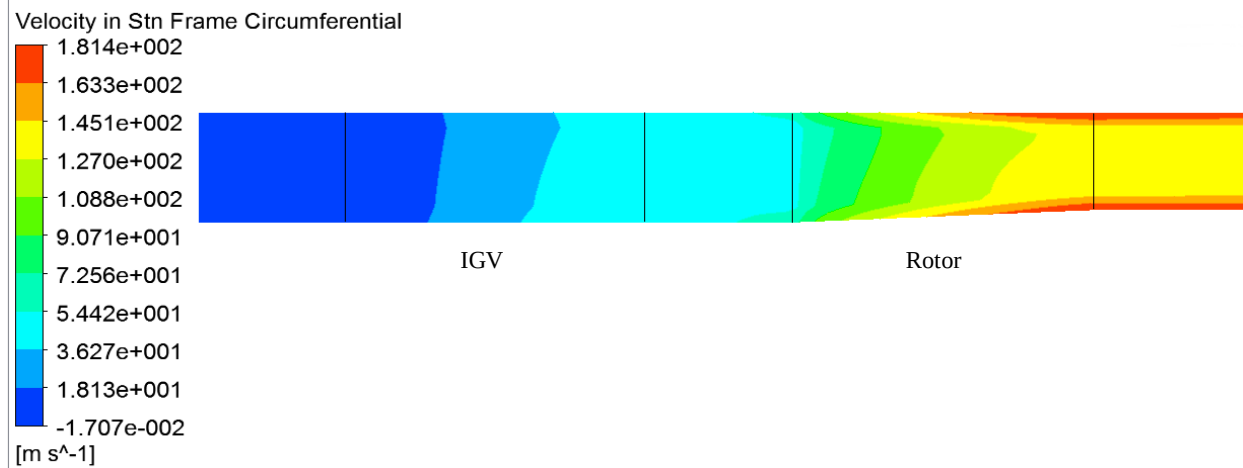


Figure A4.4. Contours of Absolute Tangential Velocity in Meridional Plane

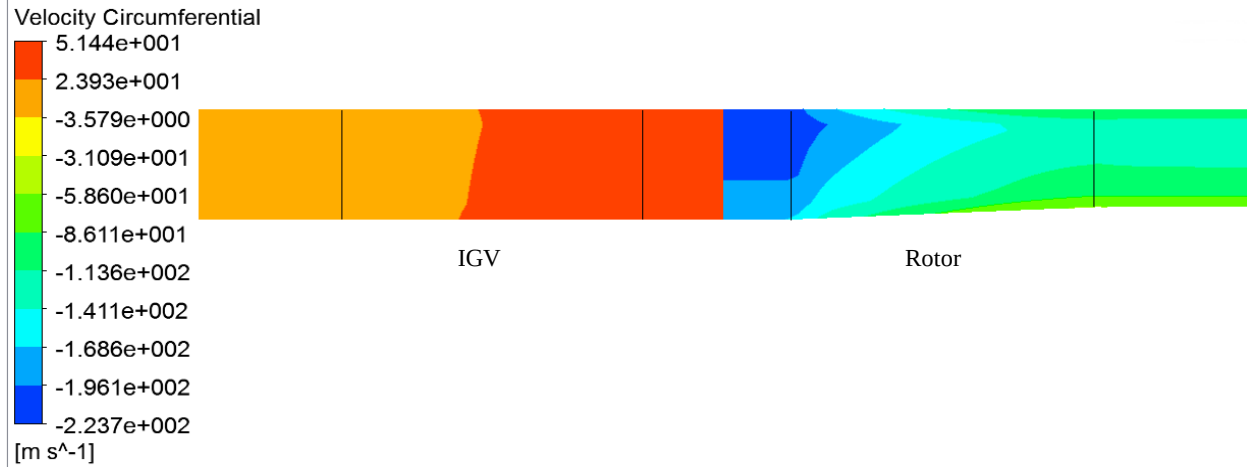


Figure A4.5. Contours of Relative Tangential Velocity in Meridional Plane

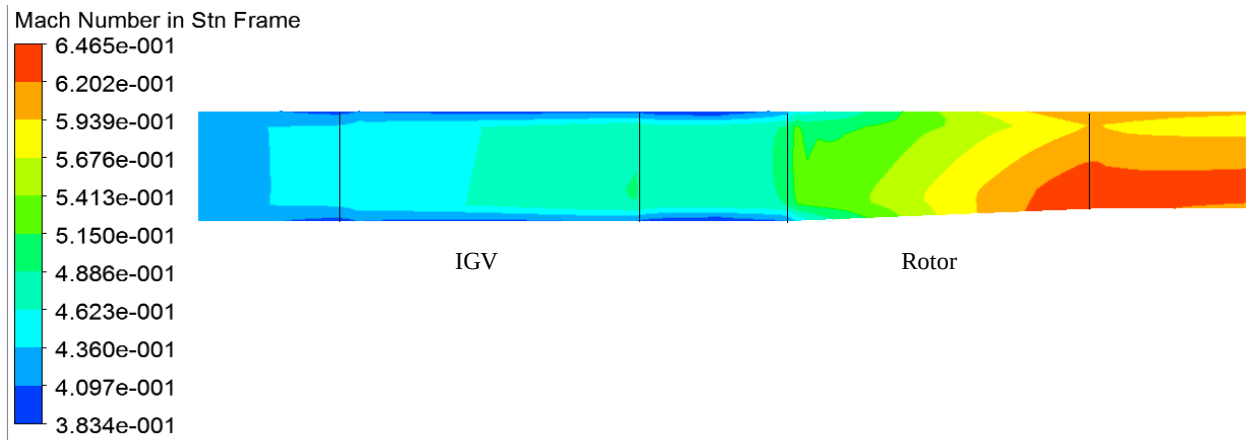


Figure A4.6. Contours of Absolute Mach Number in Meridional Plane

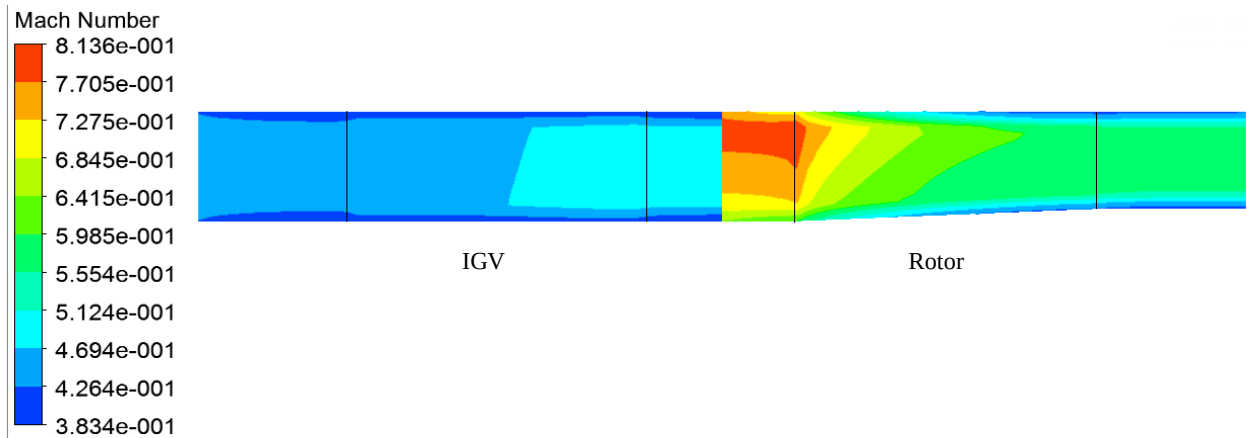


Figure A4.7. Contours of Relative Mach Number in Meridional Plane

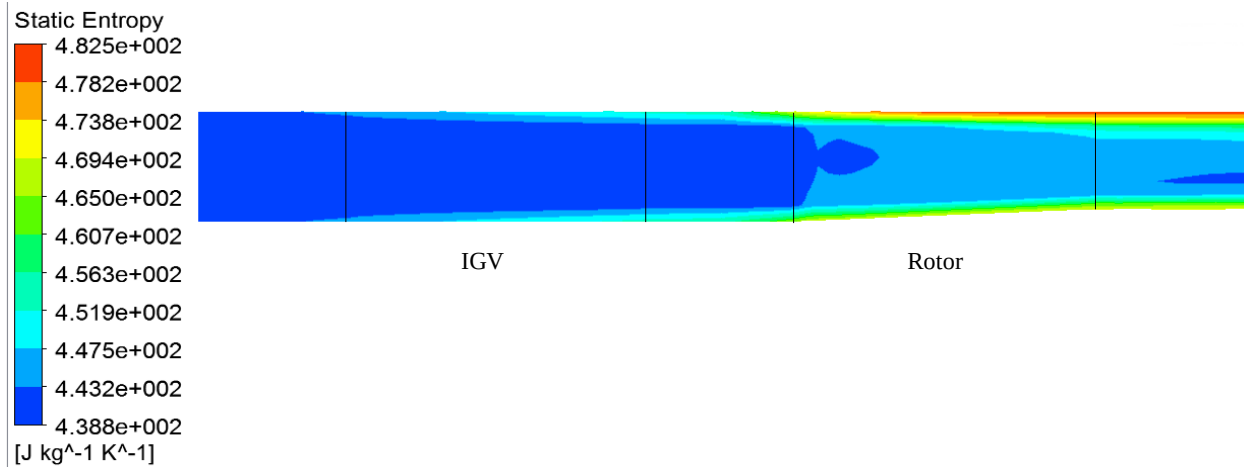


Figure A4.8. Contours of Entropy in Meridional Plane

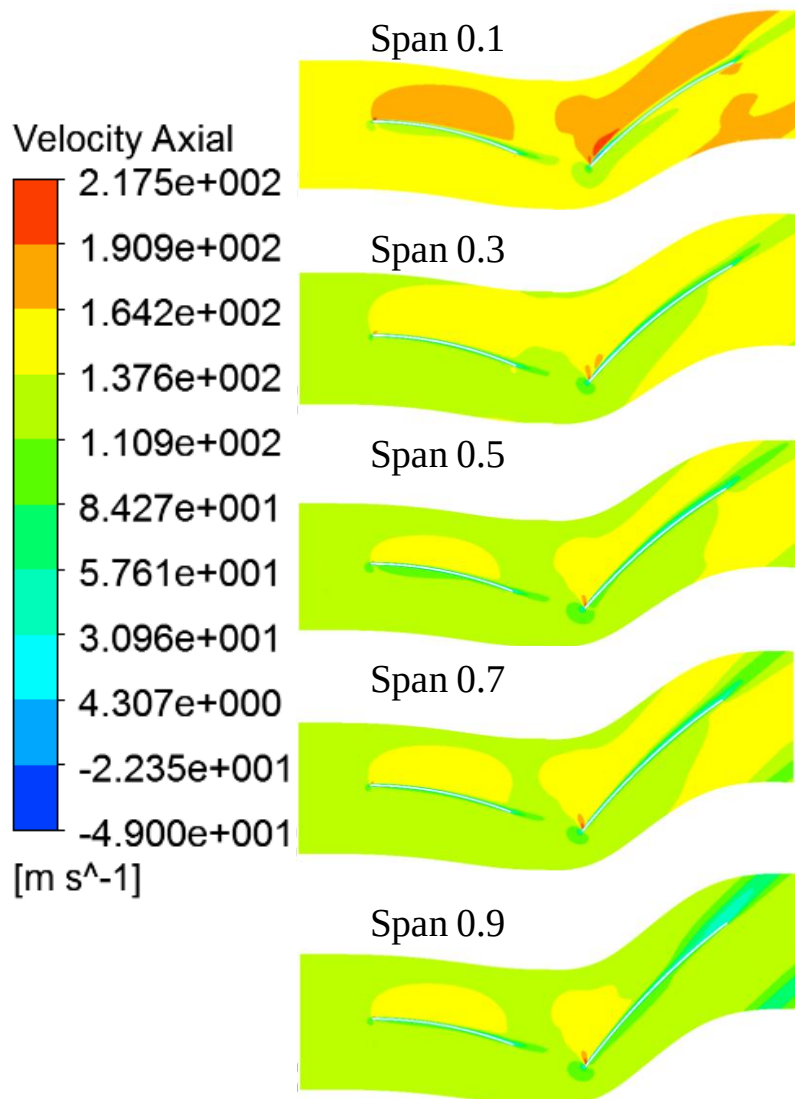


Figure A4.9. Blade-to-Blade View of Axial Velocity Contours

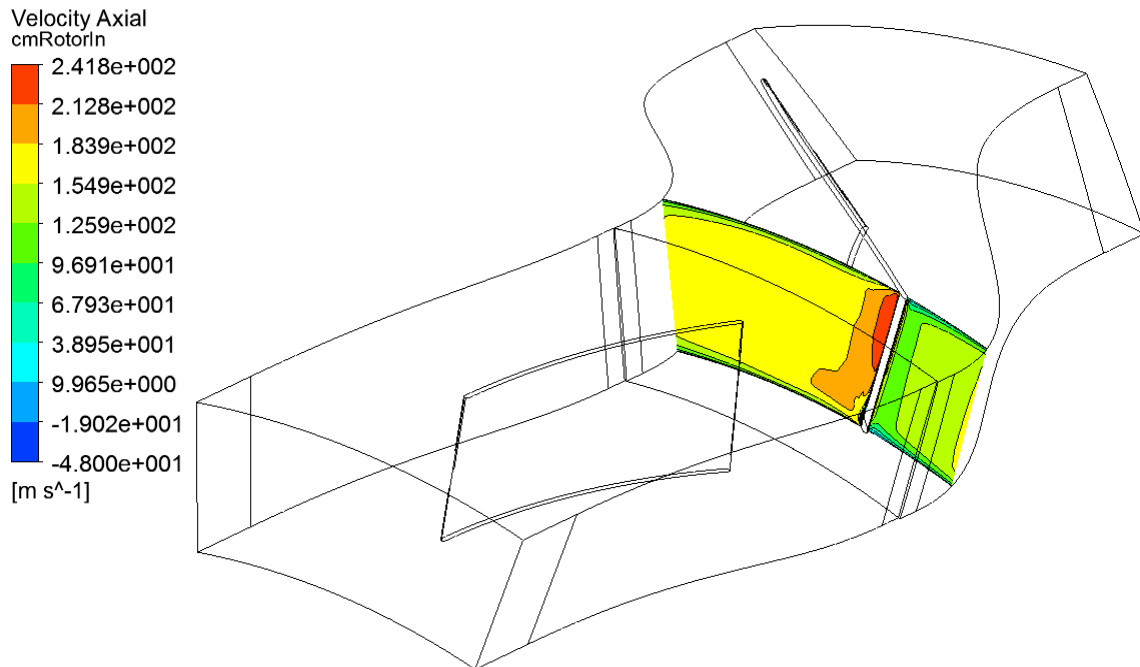


Figure A4.10. Axial Velocity Contours at Rotor Inlet

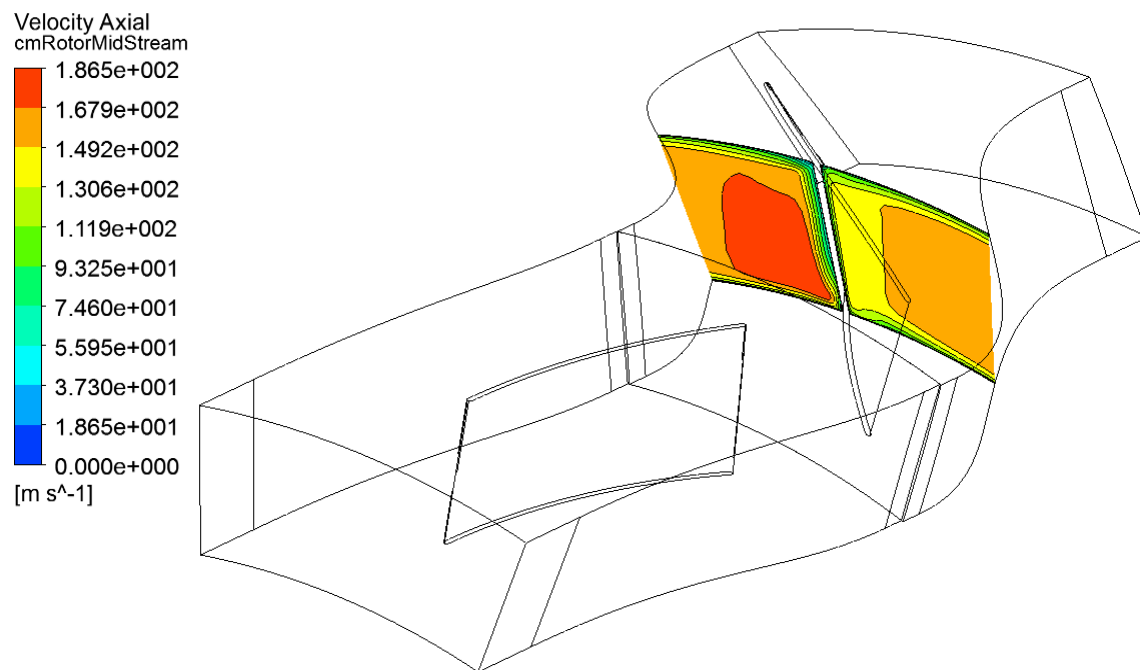


Figure A4.11. Axial Velocity Contours at Rotor Mid-Stream

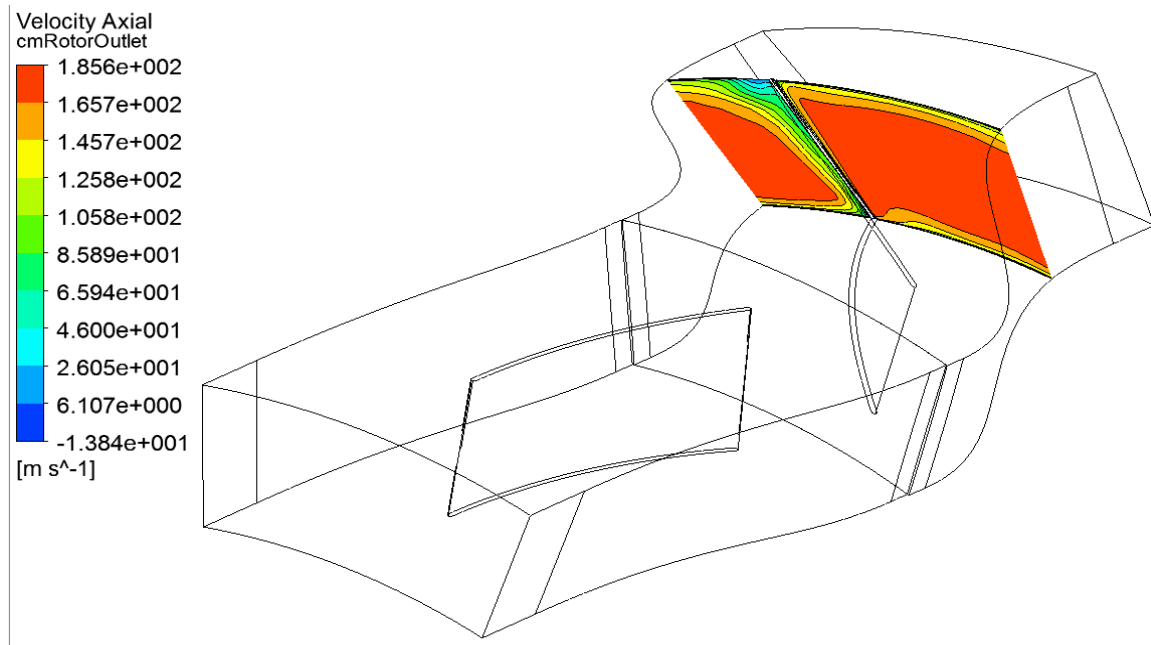


Figure A4.12. Axial Velocity Contours at Rotor Outlet

APPENDIX 5 Multistage Simulation Results

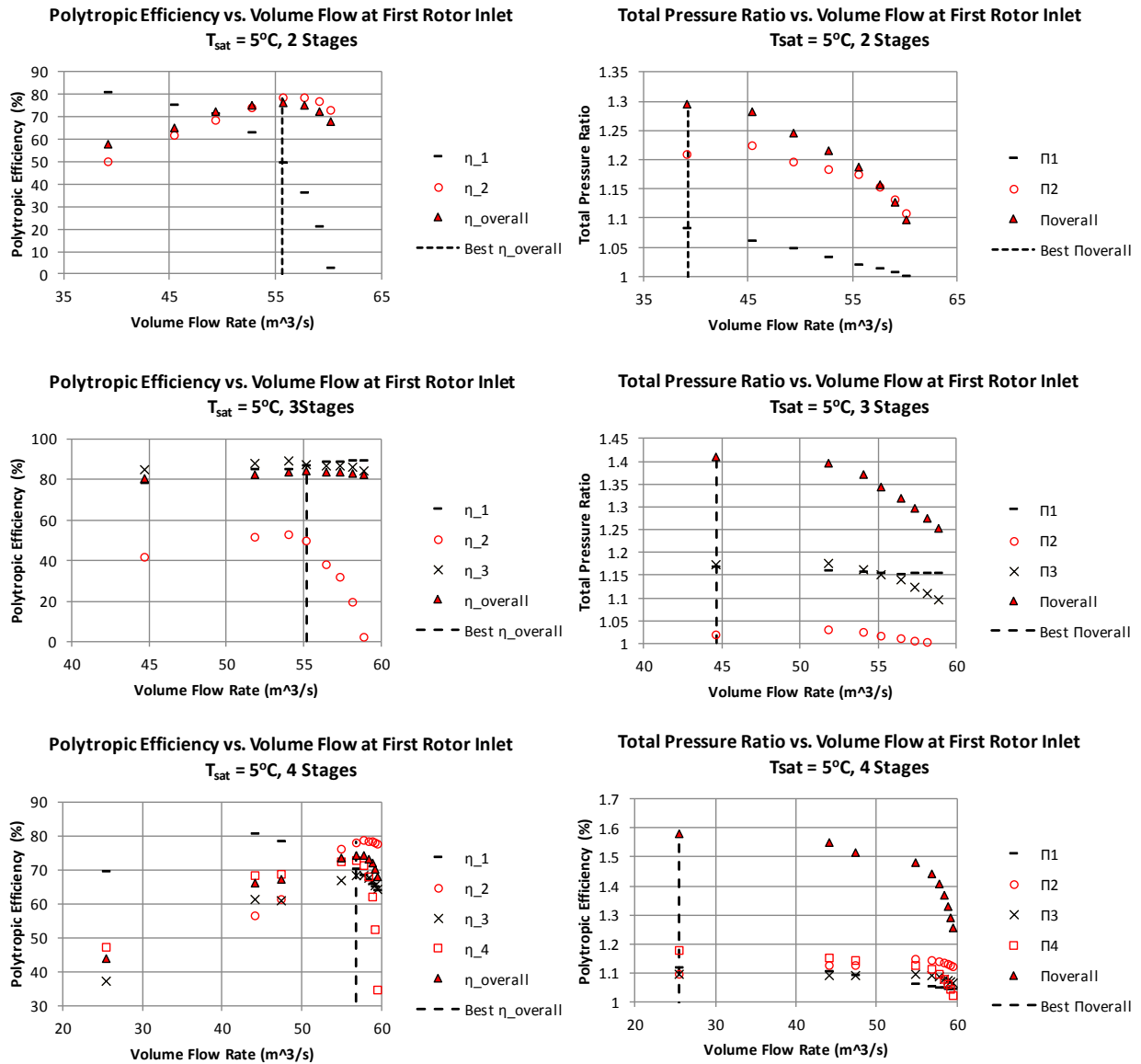


Figure A5.1. 2, 3, and 4 Stage Performance at 5°C

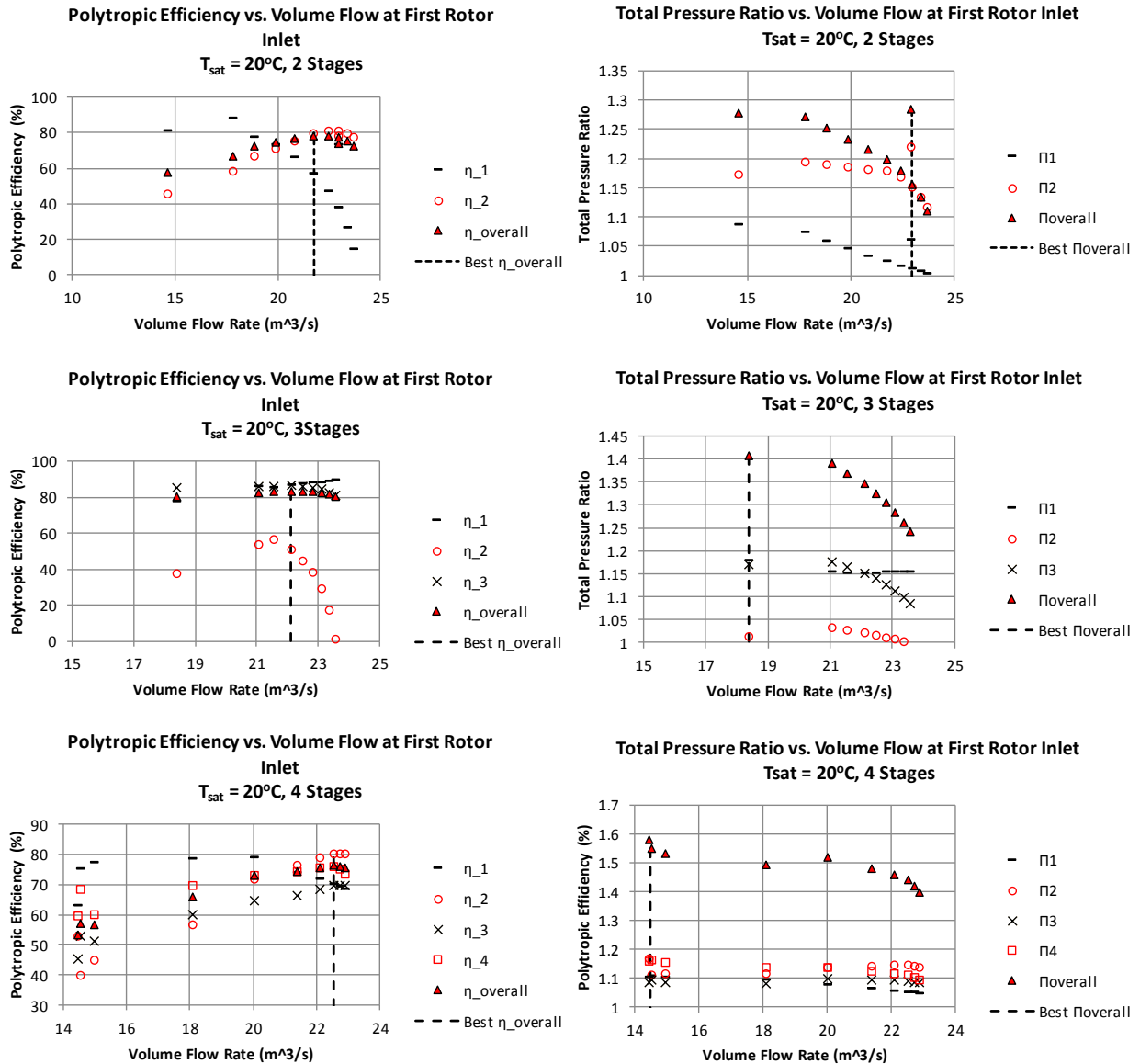


Figure A5.2. 2, 3, and 4 Stage Performance at 20°C

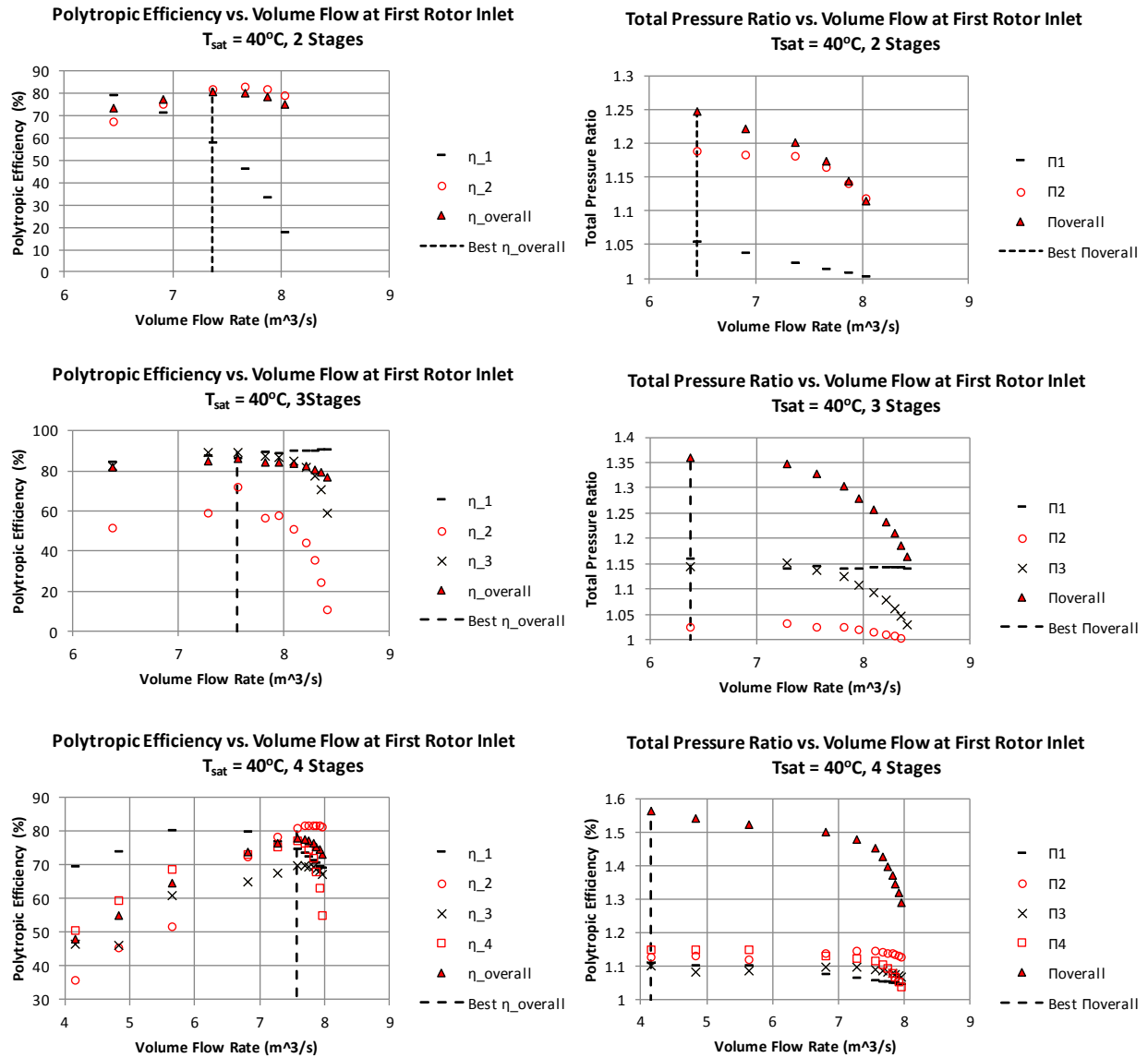


Figure A5.3. 2, 3, and 4 Stage Performance at 40°C

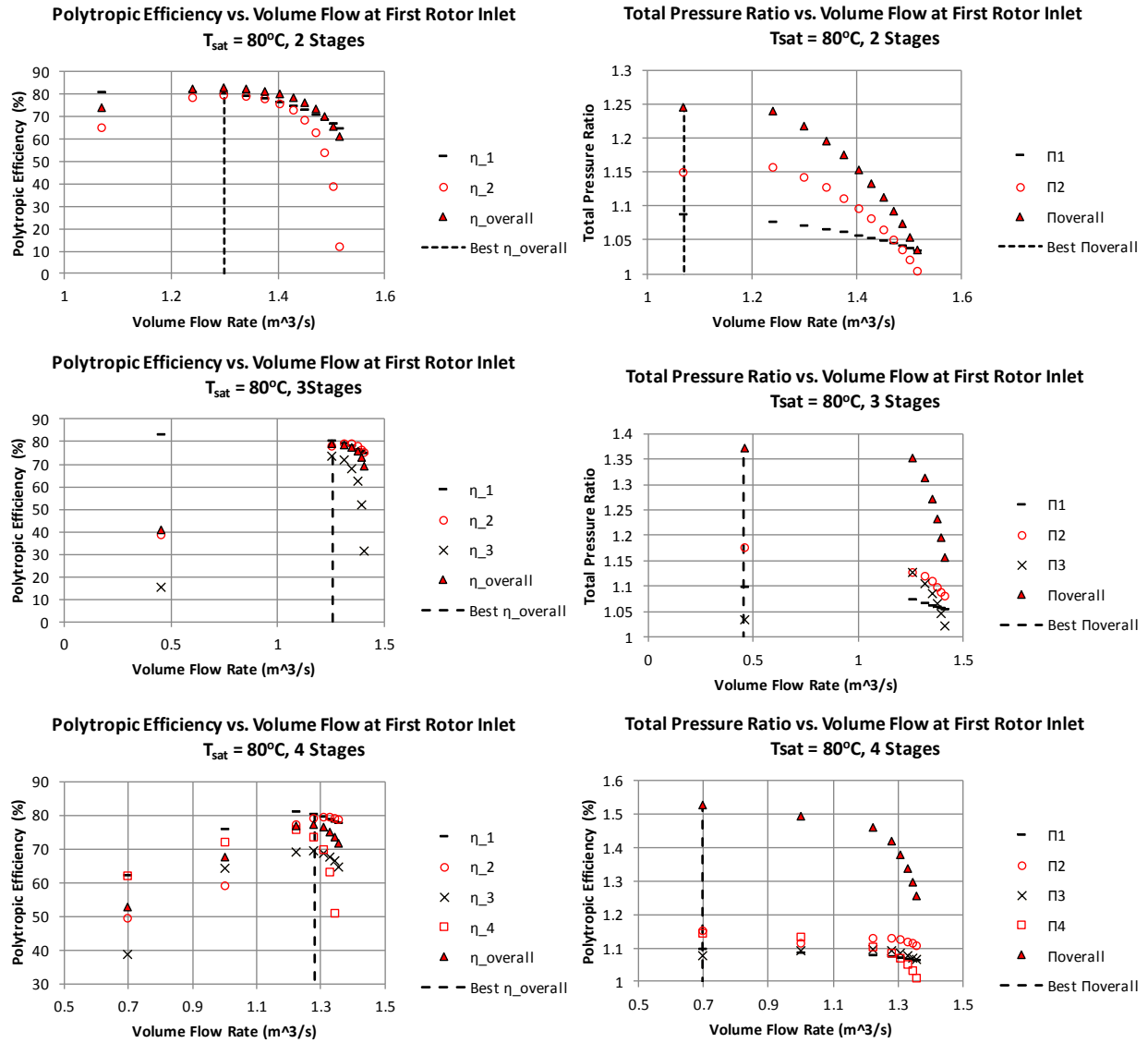


Figure A5.4. 2, 3, and 4 Stage Performance at 80°C

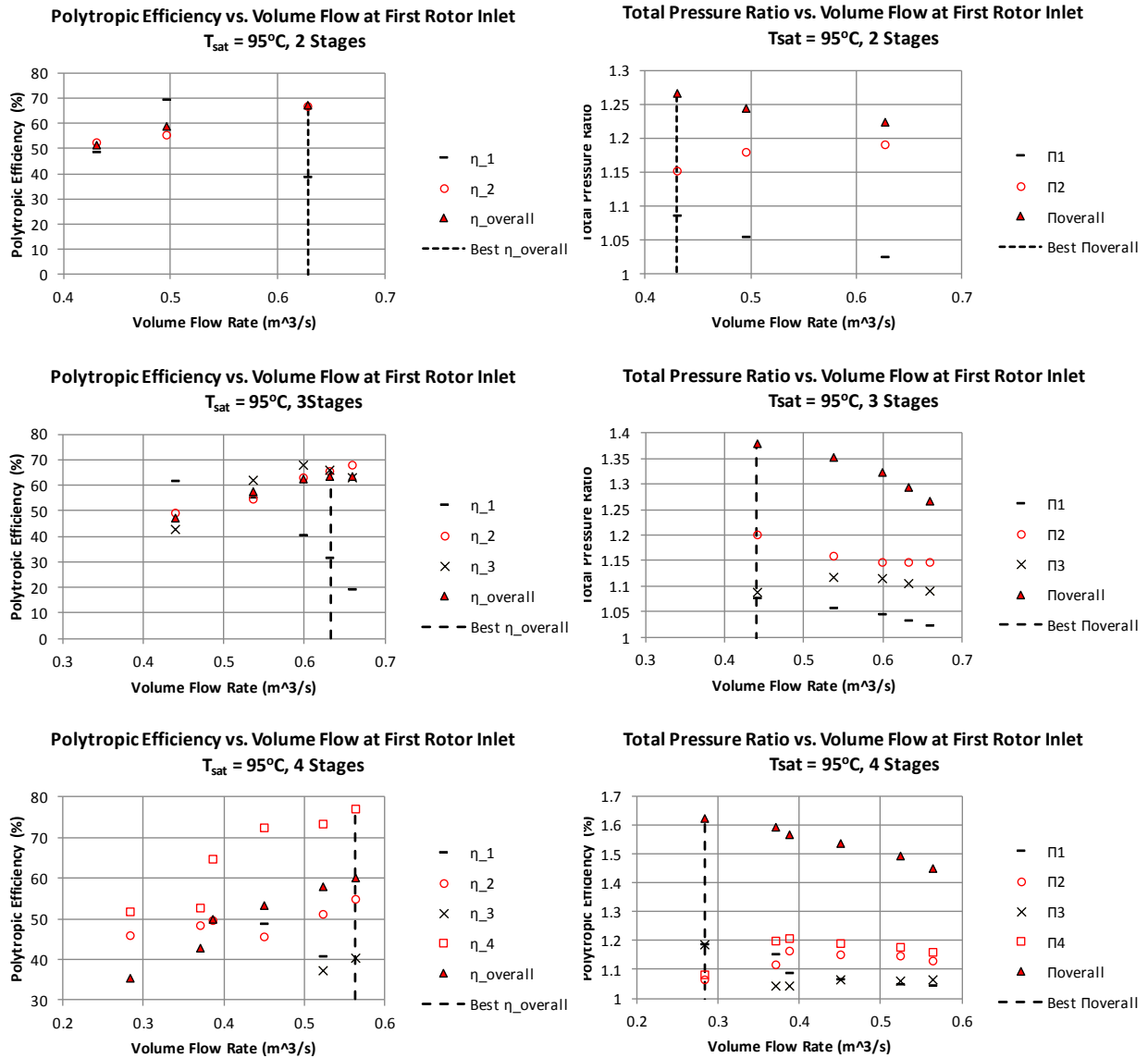


Figure A5.5. 2, 3, and 4 Stage Performance at 95°C

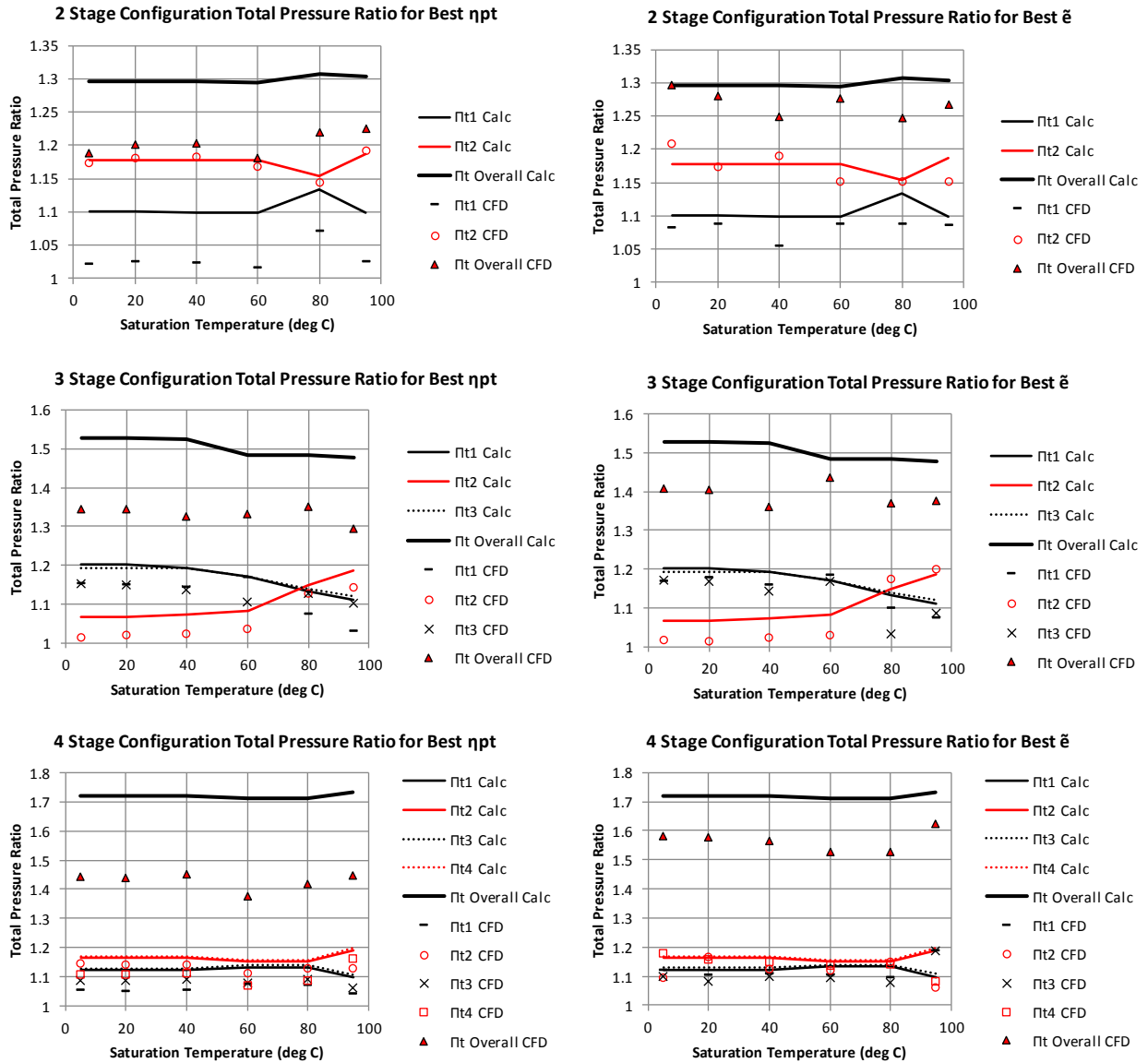


Figure A5.6. Total Pressure Ratio vs. Saturation Temperature at First Rotor Inlet for Cases of Highest Overall Polytropic Efficiency and Largest Combined Work Transfer

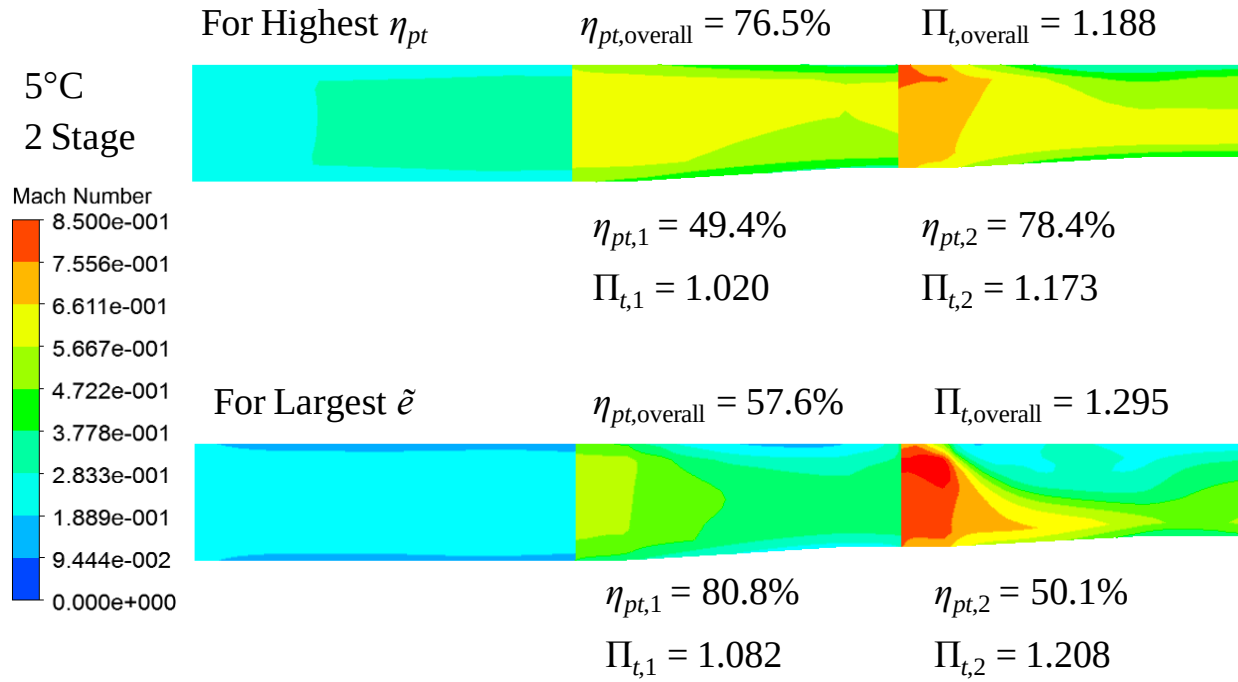


Figure A5.7. Meridional View of Relative Mach Number Contours for 2 Counter-Rotating Stages with Saturated Vapor Inlet at 5°C

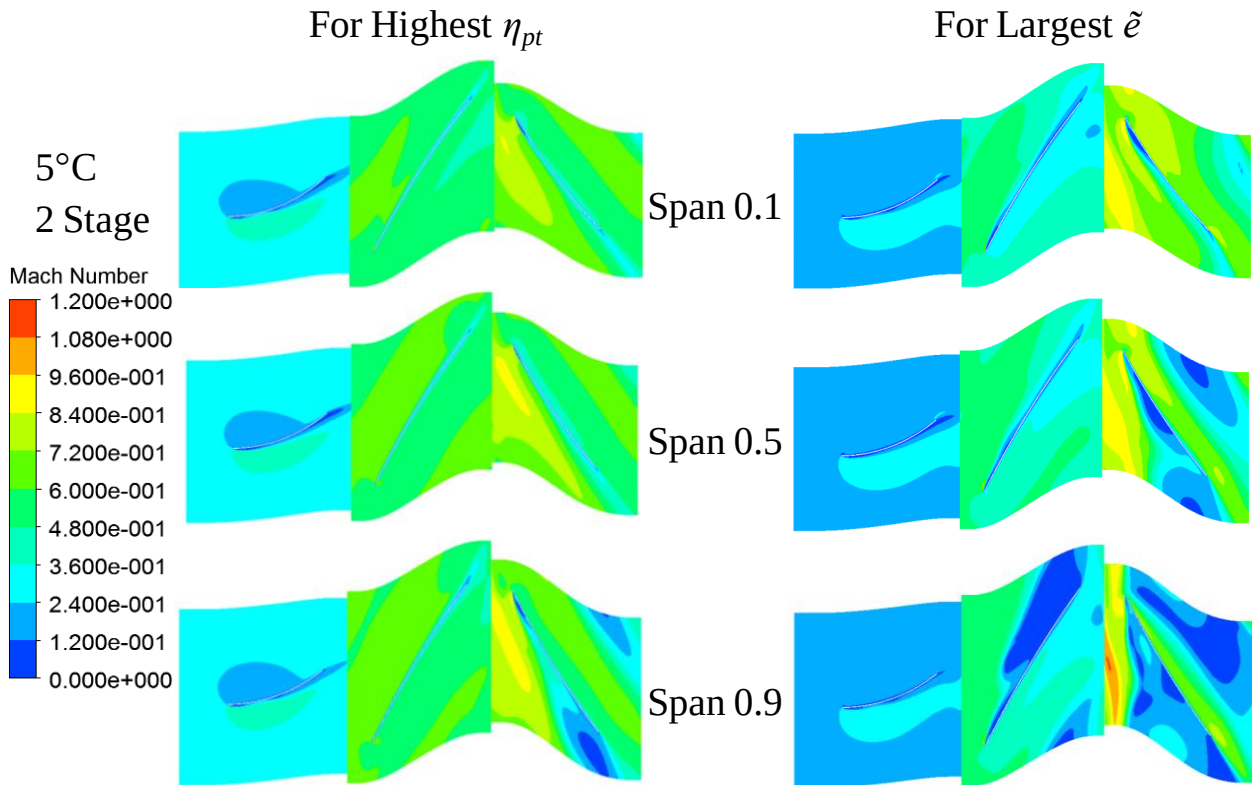


Figure A5.8. Blade-to-Blade View of Relative Mach Number Contours for 2 Counter-Rotating Stages with Saturated Vapor Inlet at 5°C

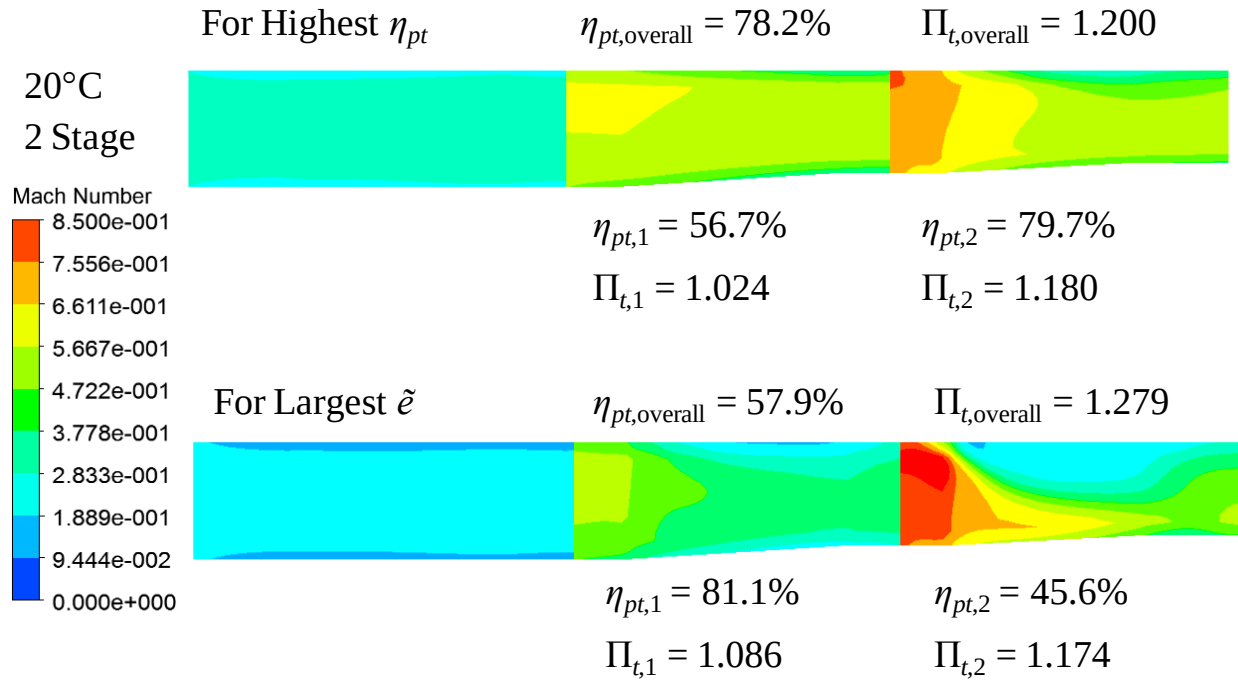


Figure A5.9. Meridional View of Relative Mach Number Contours for 2 Counter-Rotating Stages with Saturated Vapor Inlet at 20°C

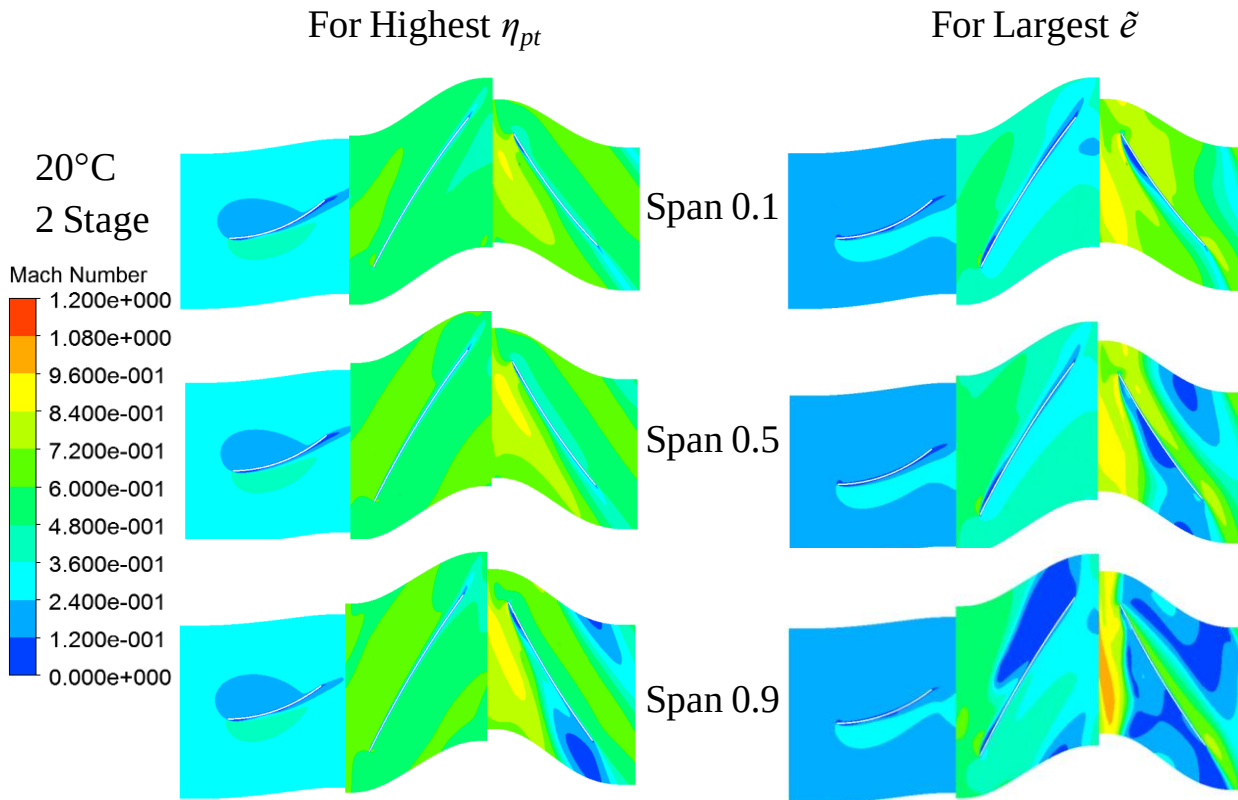


Figure A5.10. Blade-to-Blade View of Relative Mach Number Contours for 2 Counter-Rotating Stages with Saturated Vapor Inlet at 20°C

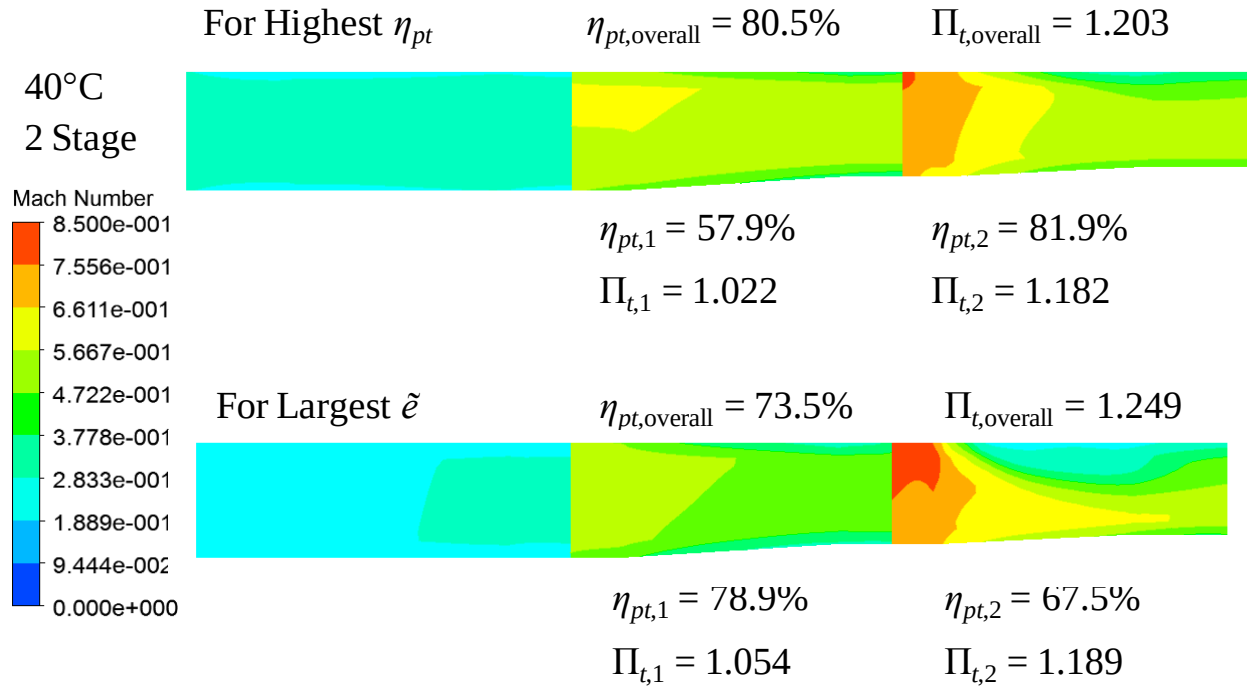


Figure A5.11. Meridional View of Relative Mach Number Contours for 2 Counter-Rotating Stages with Saturated Vapor Inlet at 40°C

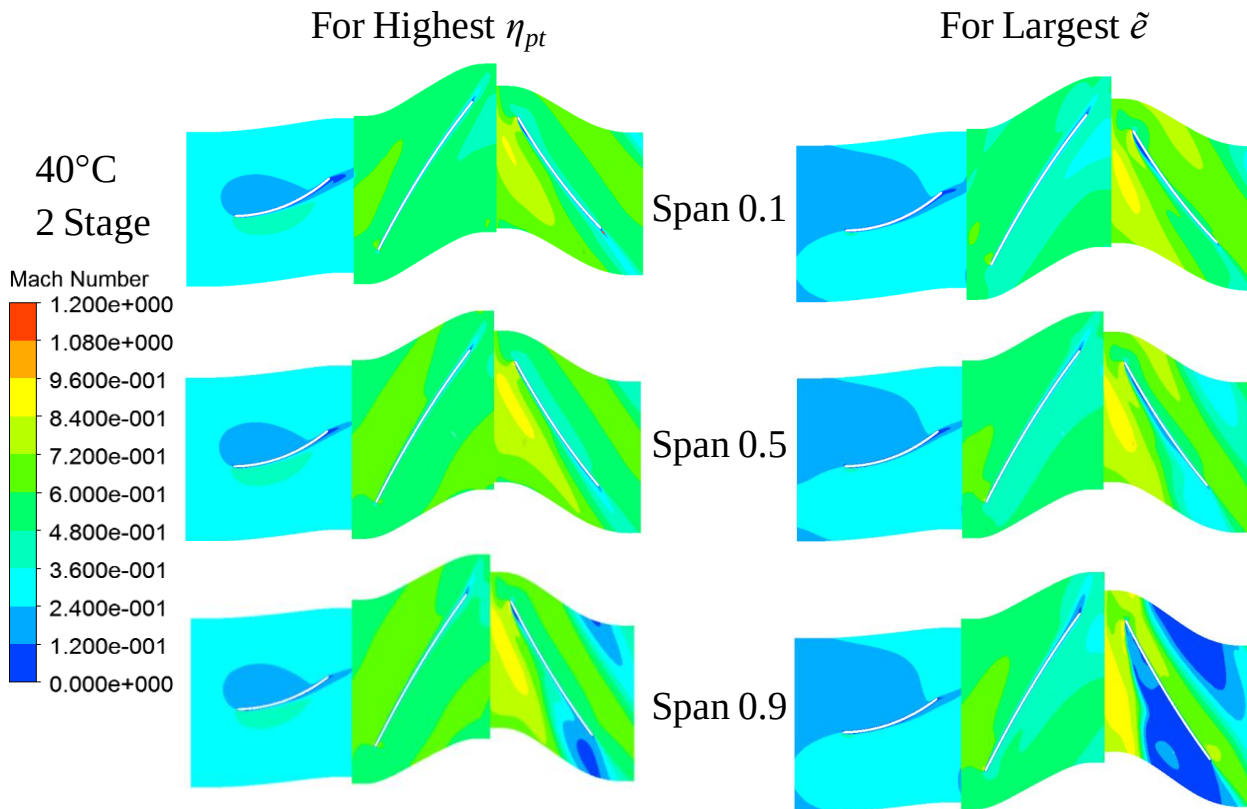


Figure A5.12. Blade-to-Blade View of Relative Mach Number Contours for 2 Counter-Rotating Stages with Saturated Vapor Inlet at 40°C

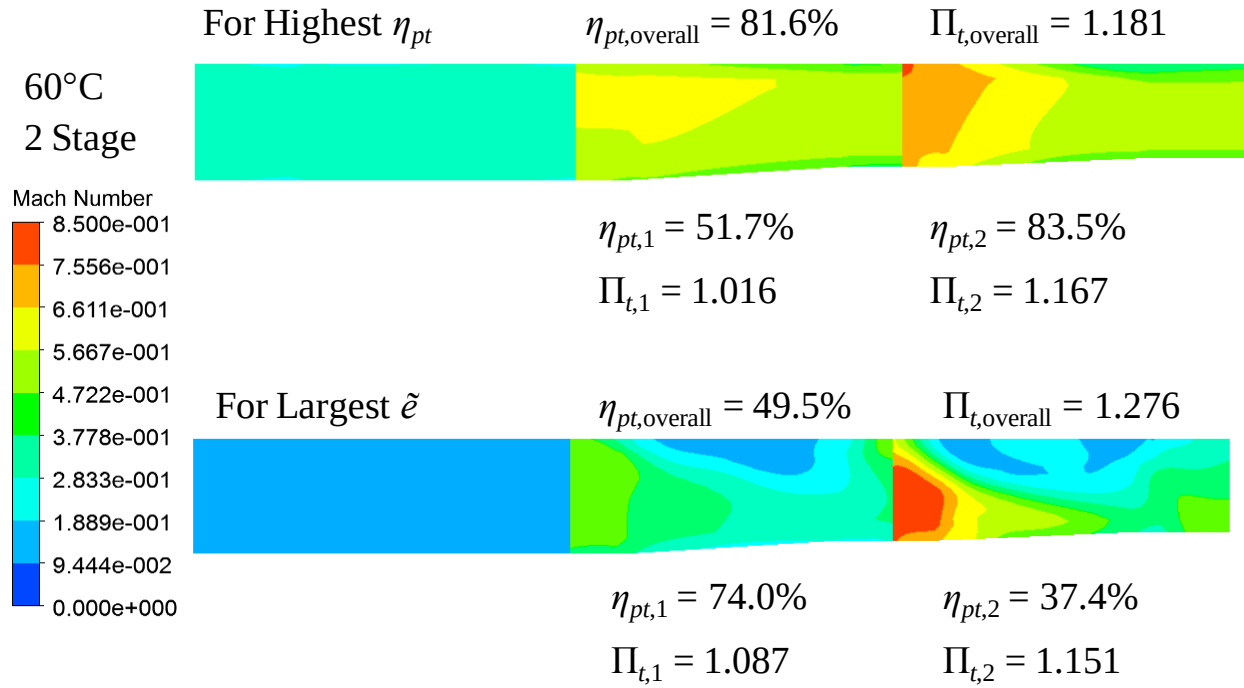


Figure A5.13. Meridional View of Relative Mach Number Contours for 2 Counter-Rotating Stages with Saturated Vapor Inlet at 60°C

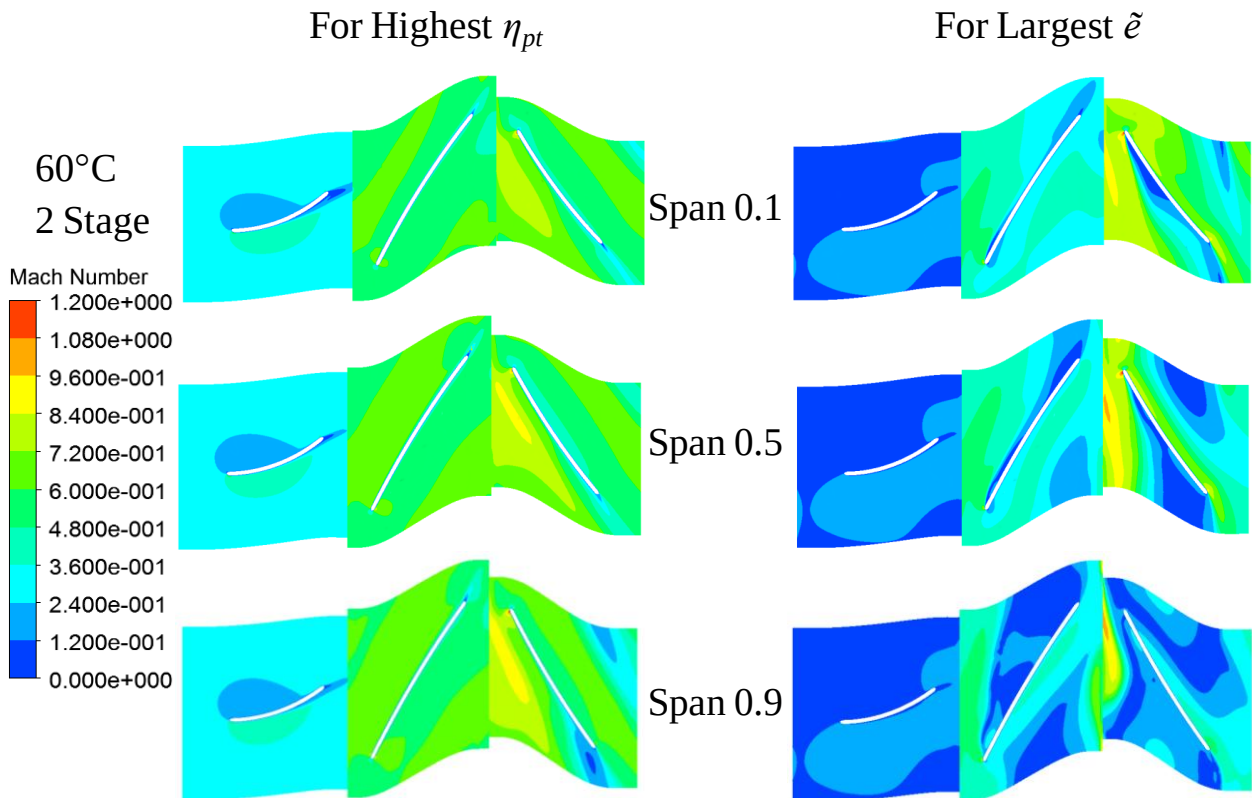


Figure A5.14. Blade-to-Blade View of Relative Mach Number Contours for 2 Counter-Rotating Stages with Saturated Vapor Inlet at 60°C

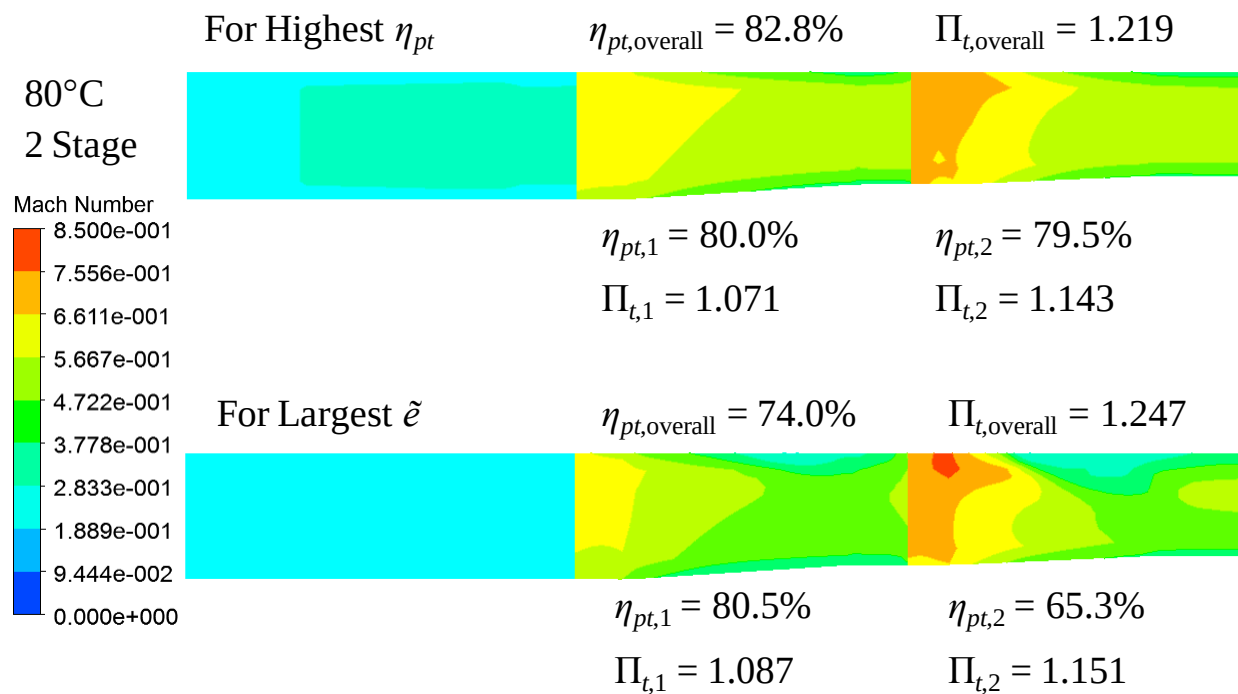


Figure A5.15. Meridional View of Relative Mach Number Contours for 2 Counter-Rotating Stages with Saturated Vapor Inlet at 80°C

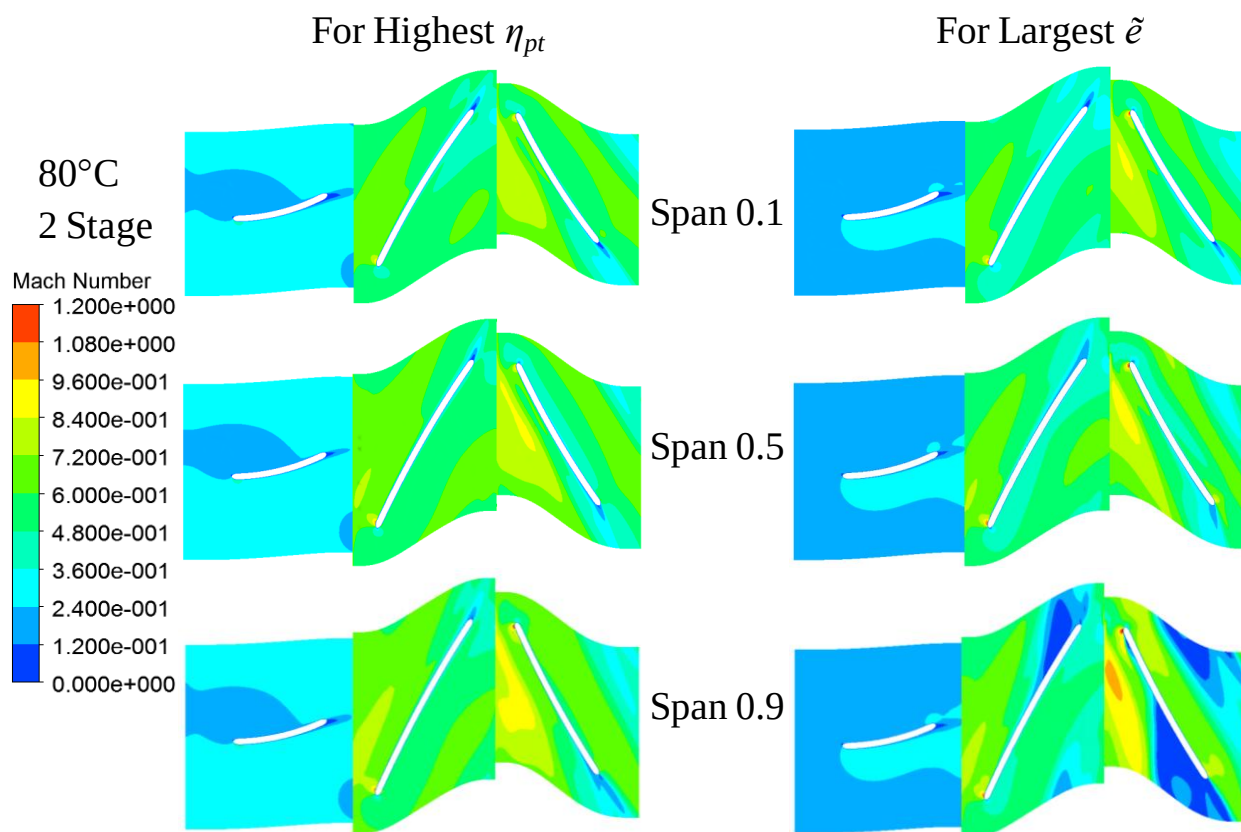


Figure A5.16. Blade-to-Blade View of Relative Mach Number Contours for 2 Counter-Rotating Stages with Saturated Vapor Inlet at 80°C

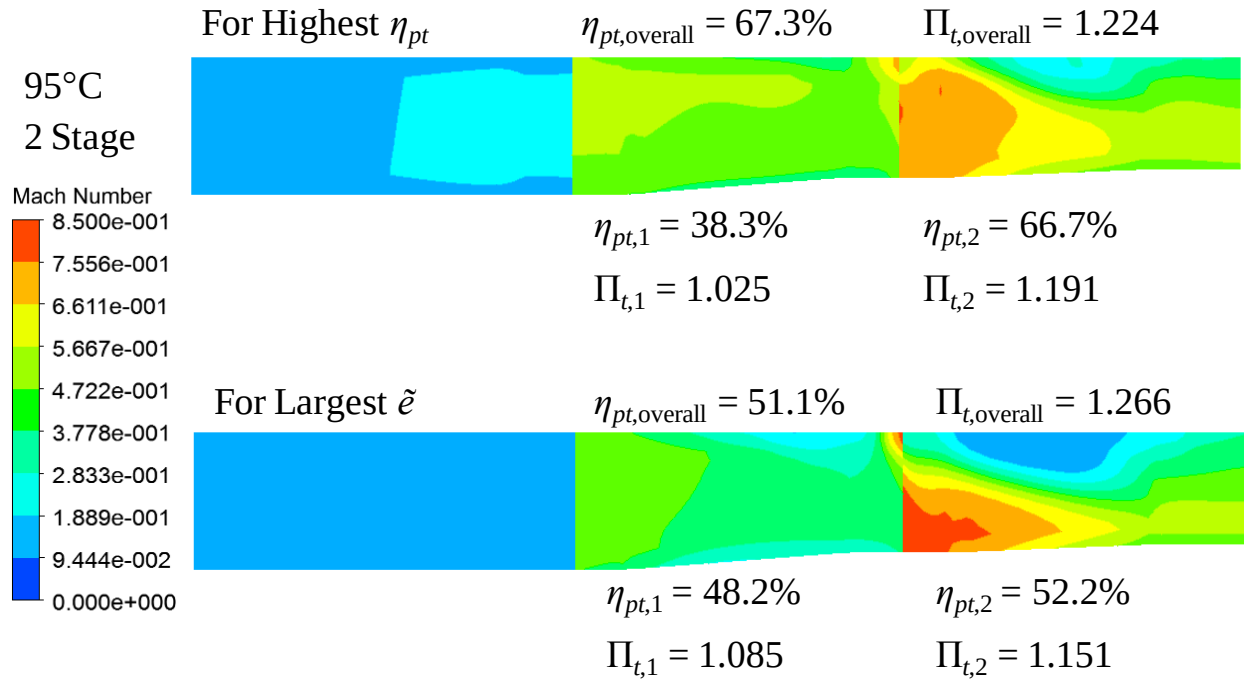


Figure A5.17. Meridional View of Relative Mach Number Contours for 2 Counter-Rotating Stages with Saturated Vapor Inlet at 95°C

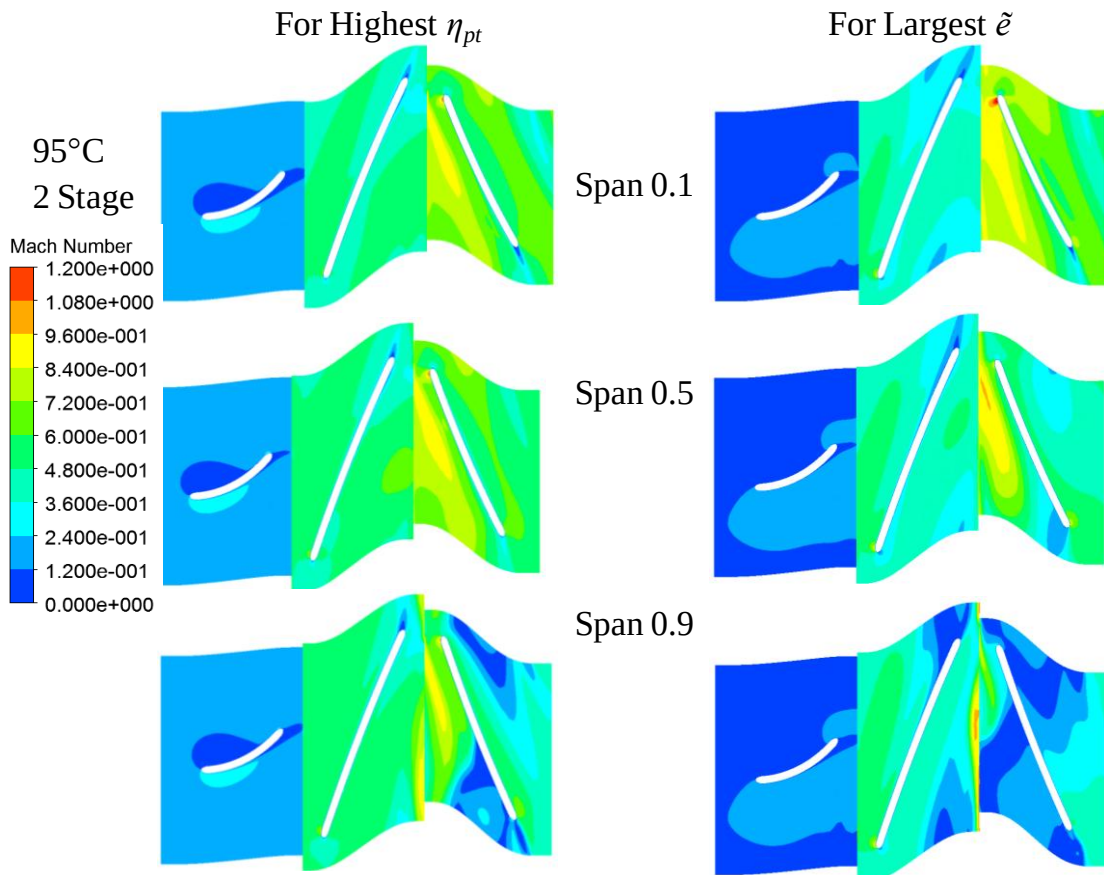


Figure A5.18. Blade-to-Blade View of Relative Mach Number Contours for 2 Counter-Rotating Stages with Saturated Vapor Inlet at 95°C

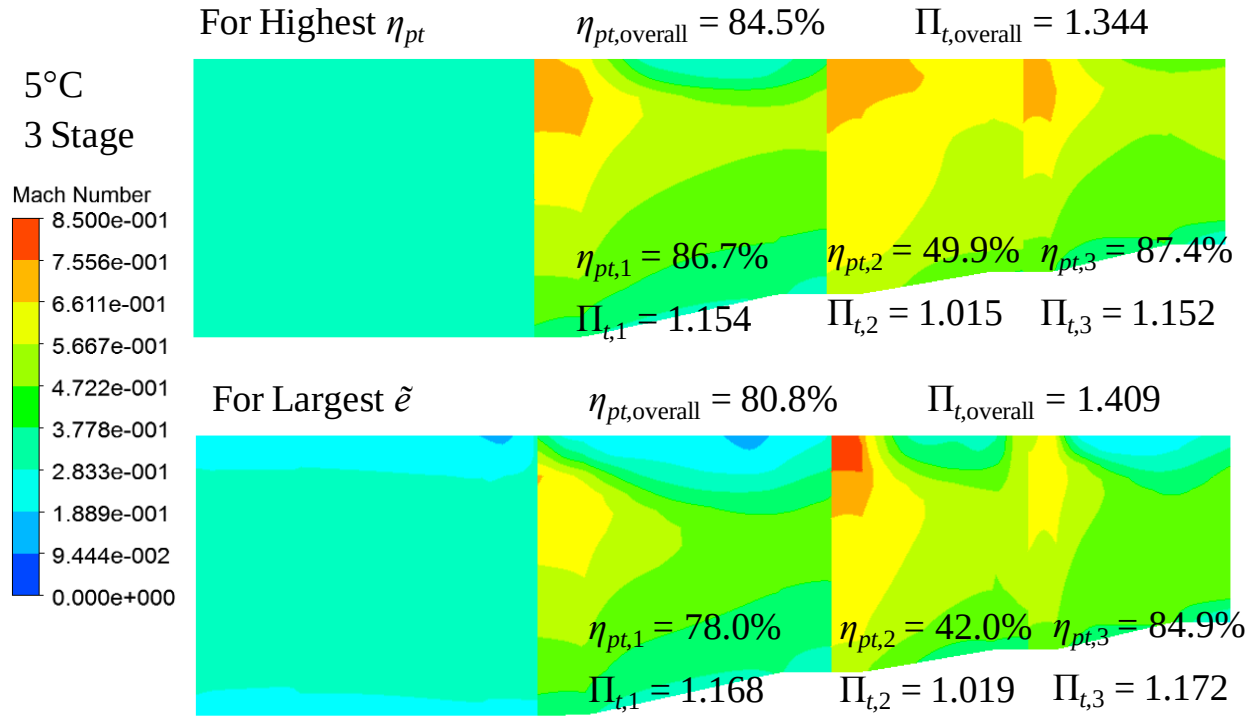


Figure A5.19. Meridional View of Relative Mach Number Contours for 3 Counter-Rotating Stages with Saturated Vapor Inlet at 5°C

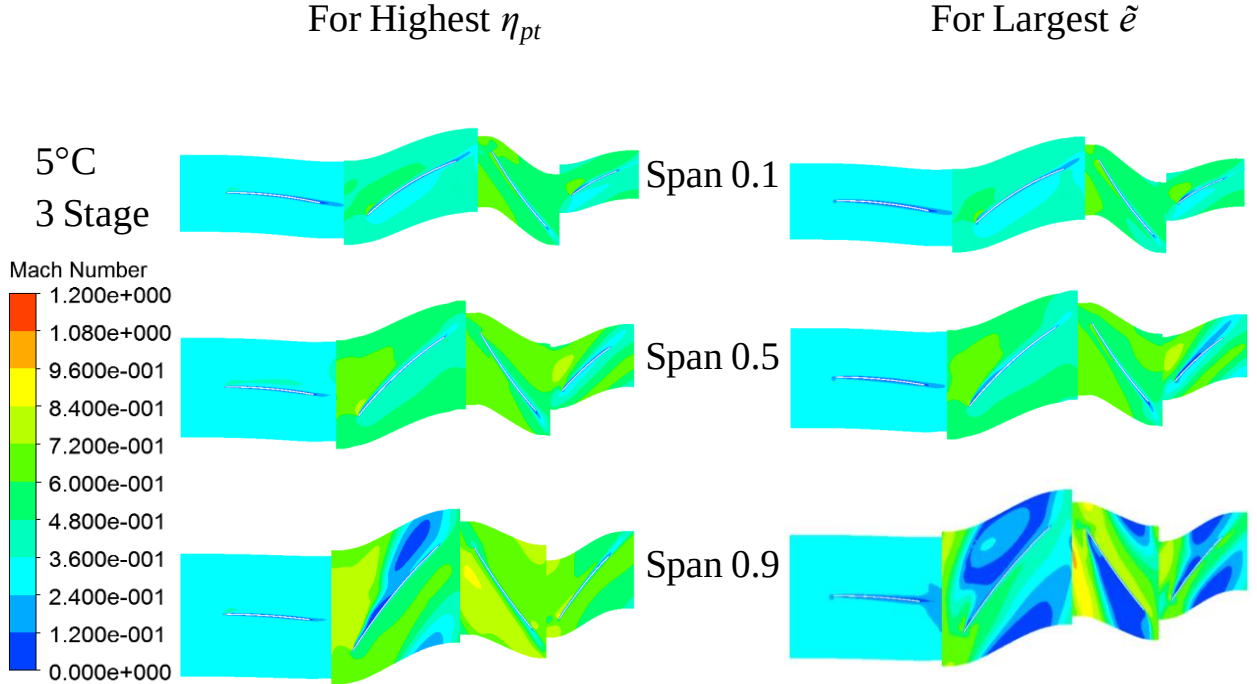


Figure A5.20. Blade-to-Blade View of Relative Mach Number Contours for 3 Counter-Rotating Stages with Saturated Vapor Inlet at 5°C

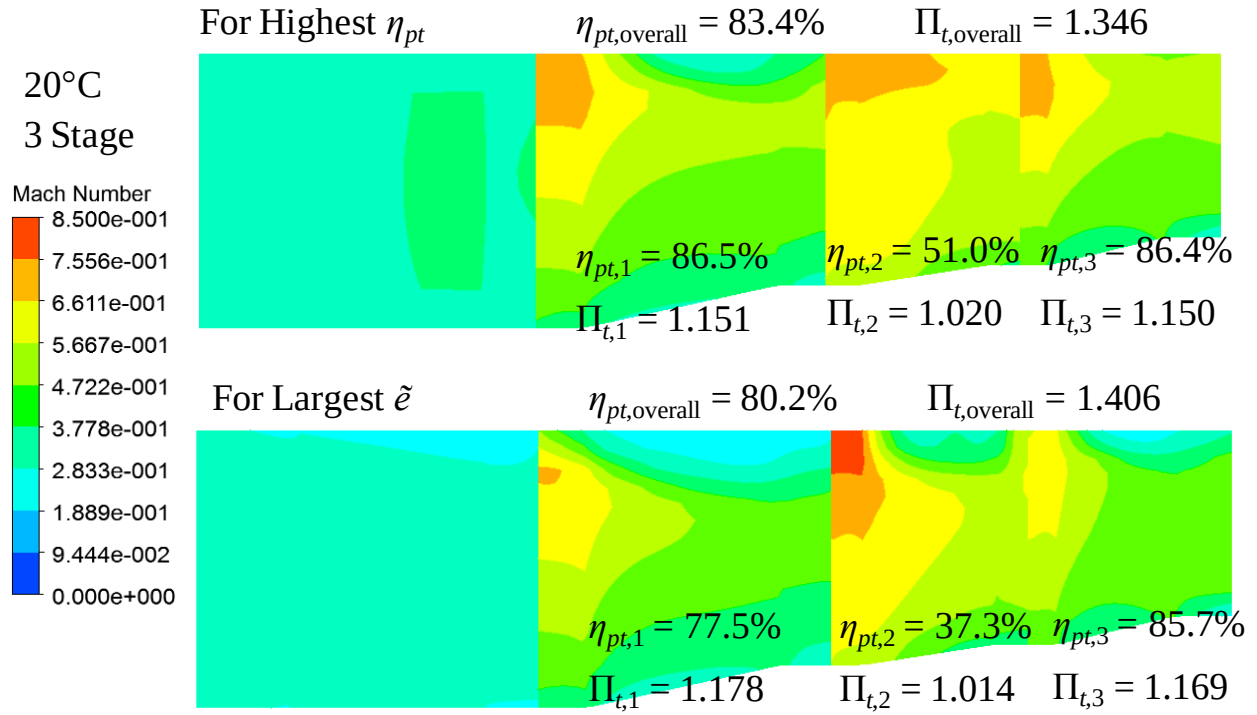


Figure A5.21. Meridional View of Relative Mach Number Contours for 3 Counter-Rotating Stages with Saturated Vapor Inlet at 20°C

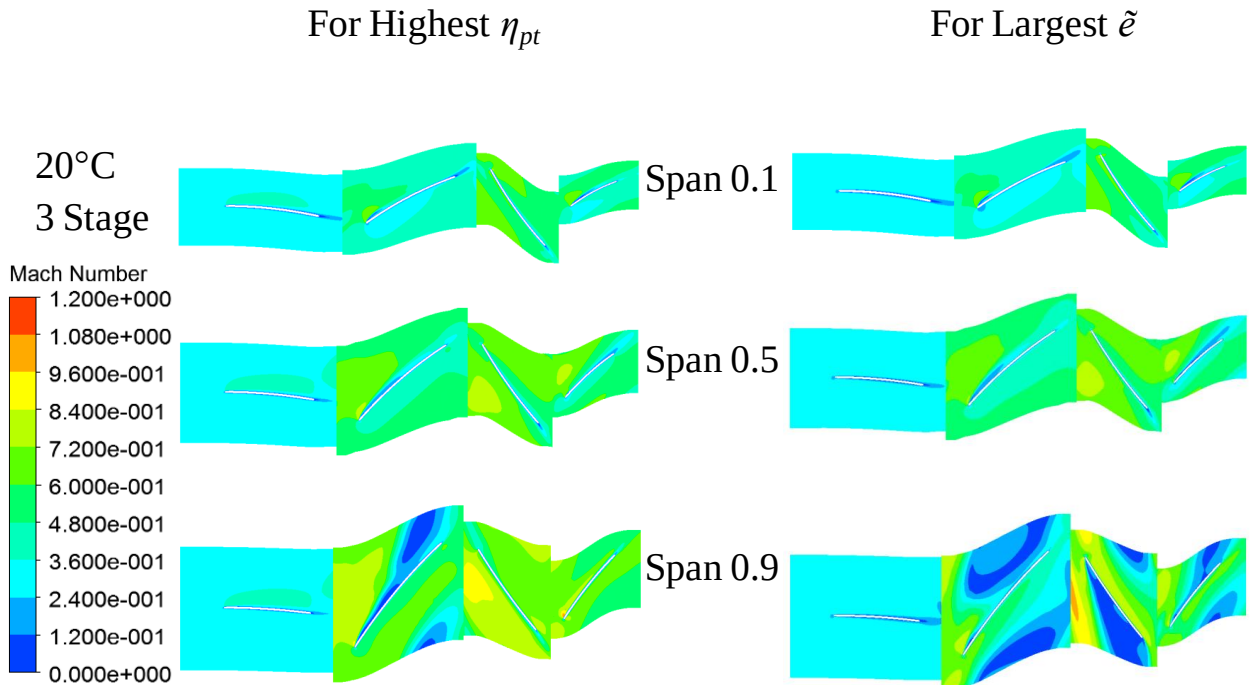


Figure A5.22. Blade-to-Blade View of Relative Mach Number Contours for 3 Counter-Rotating Stages with Saturated Vapor Inlet at 20°C

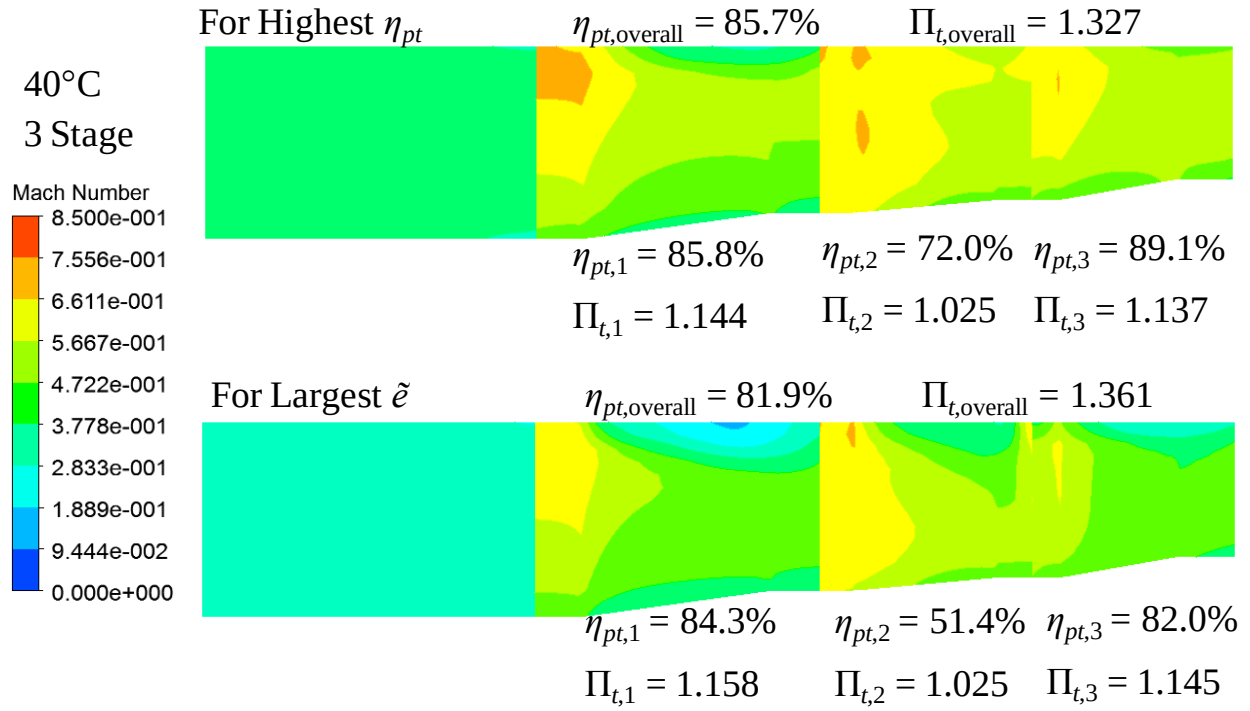


Figure A5.23. Meridional View of Relative Mach Number Contours for 3 Counter-Rotating Stages with Saturated Vapor Inlet at 40°C

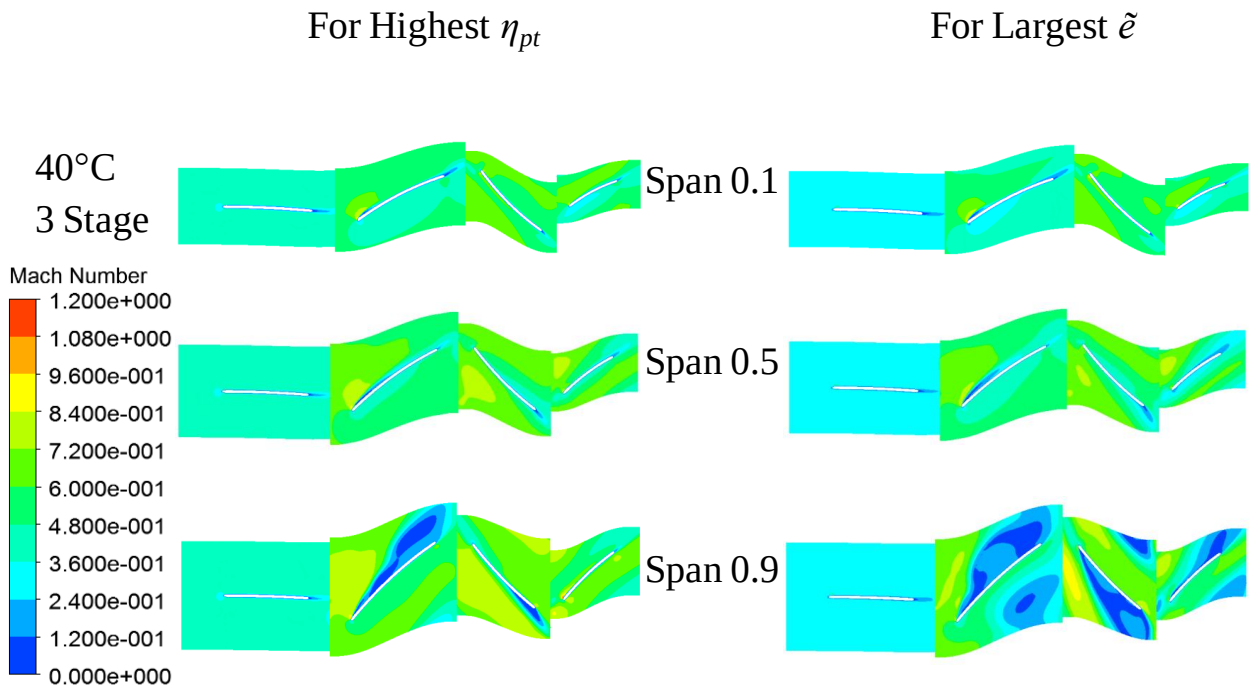


Figure A5.24. Blade-to-Blade View of Relative Mach Number Contours for 3 Counter-Rotating Stages with Saturated Vapor Inlet at 40°C

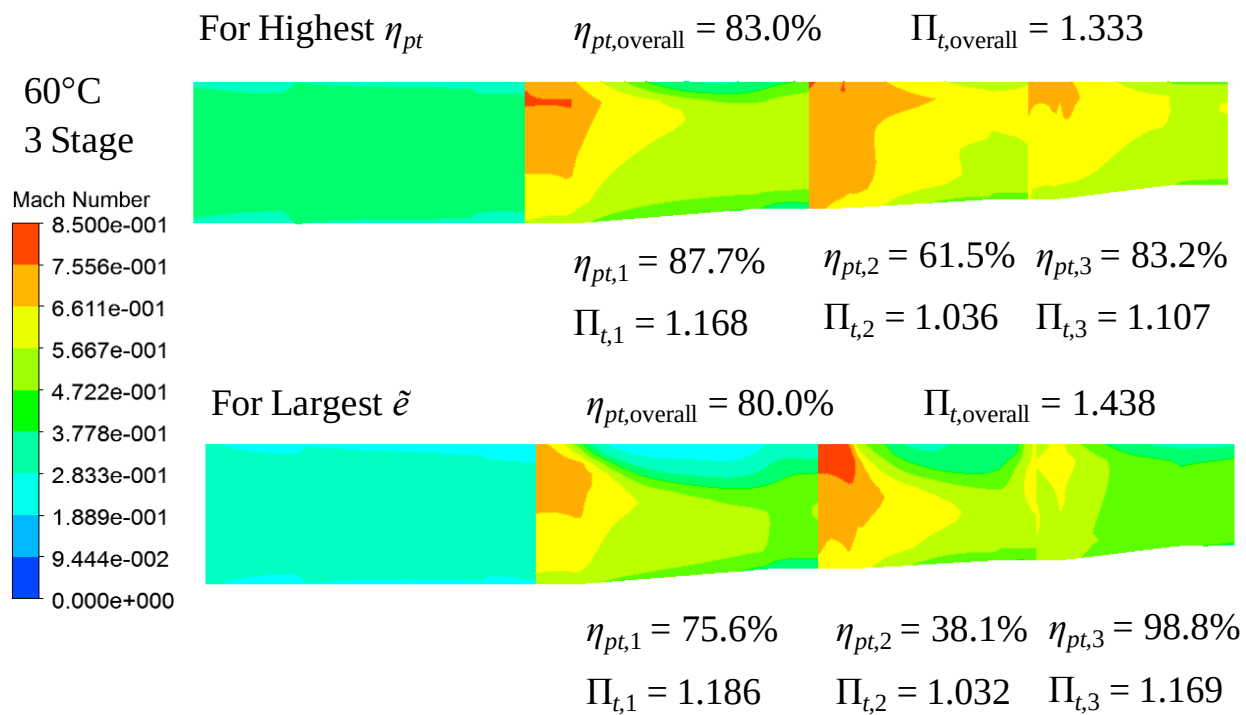


Figure A5.25. Meridional View of Relative Mach Number Contours for 3 Counter-Rotating Stages with Saturated Vapor Inlet at 60°C

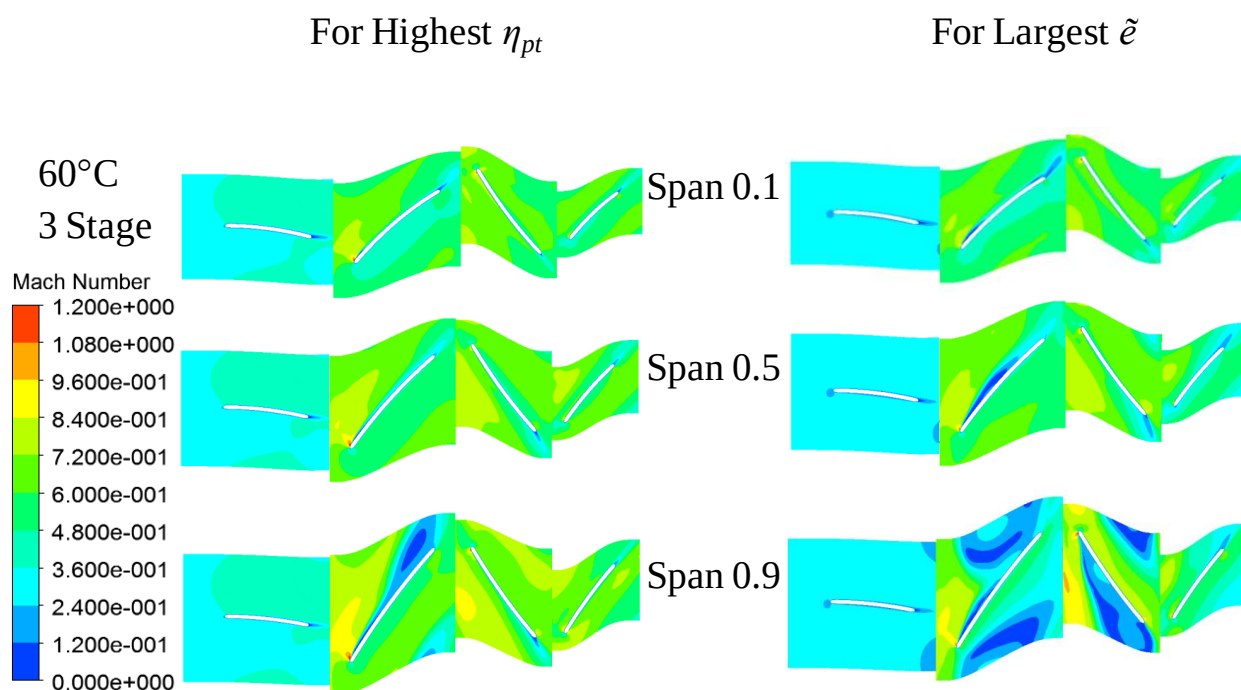


Figure A5.26. Blade-to-Blade View of Relative Mach Number Contours for 3 Counter-Rotating Stages with Saturated Vapor Inlet at 60°C

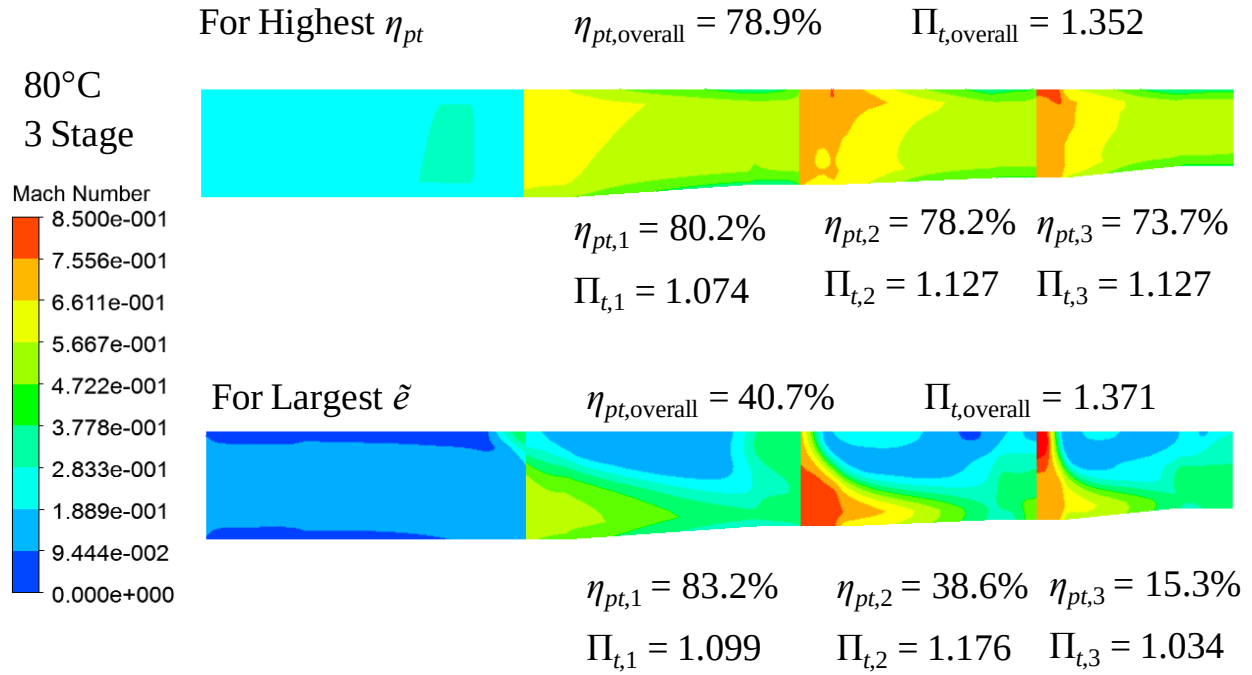


Figure A5.27. Meridional View of Relative Mach Number Contours for 3 Counter-Rotating Stages with Saturated Vapor Inlet at 80°C

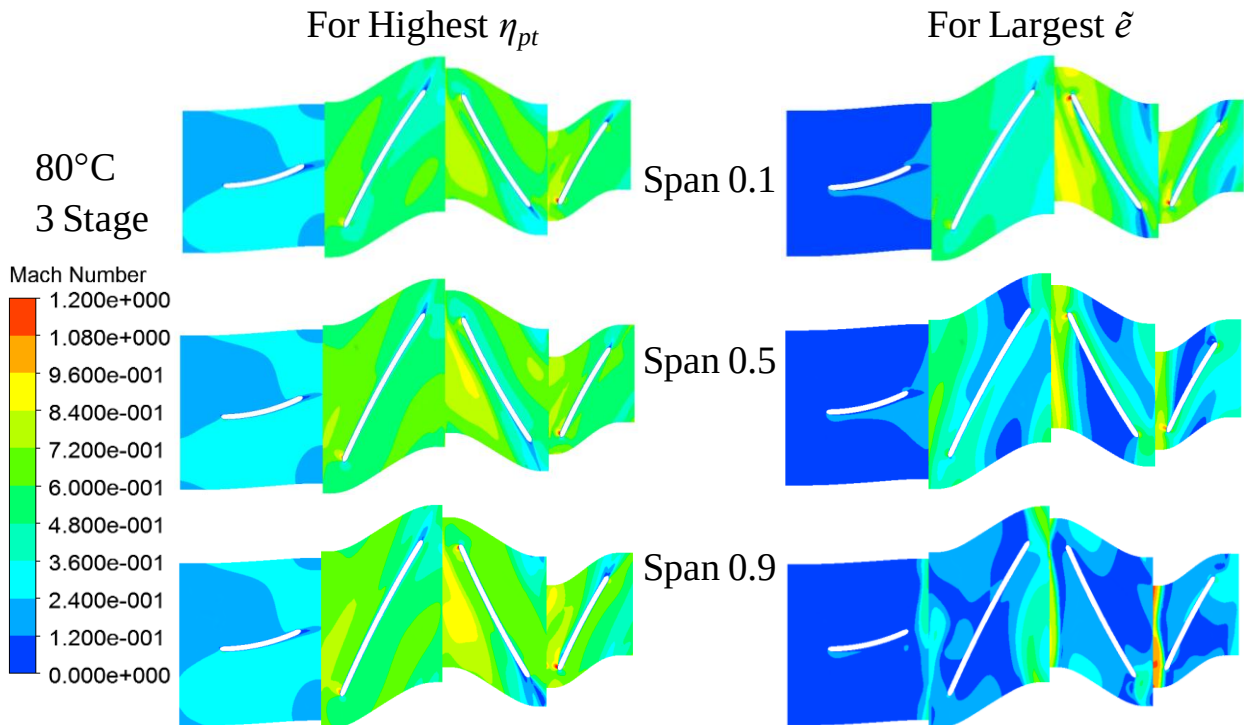


Figure A5.28. Blade-to-Blade View of Relative Mach Number Contours for 3 Counter-Rotating Stages with Saturated Vapor Inlet at 80°C

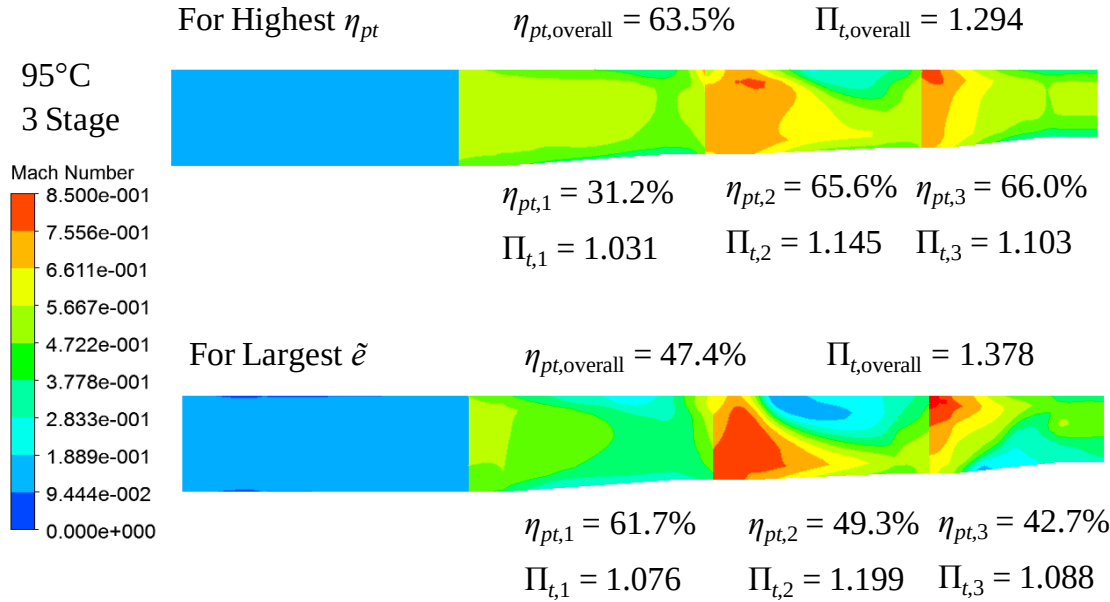


Figure A5.29. Meridional View of Relative Mach Number Contours for 3 Counter-Rotating Stages with Saturated Vapor Inlet at 95°C

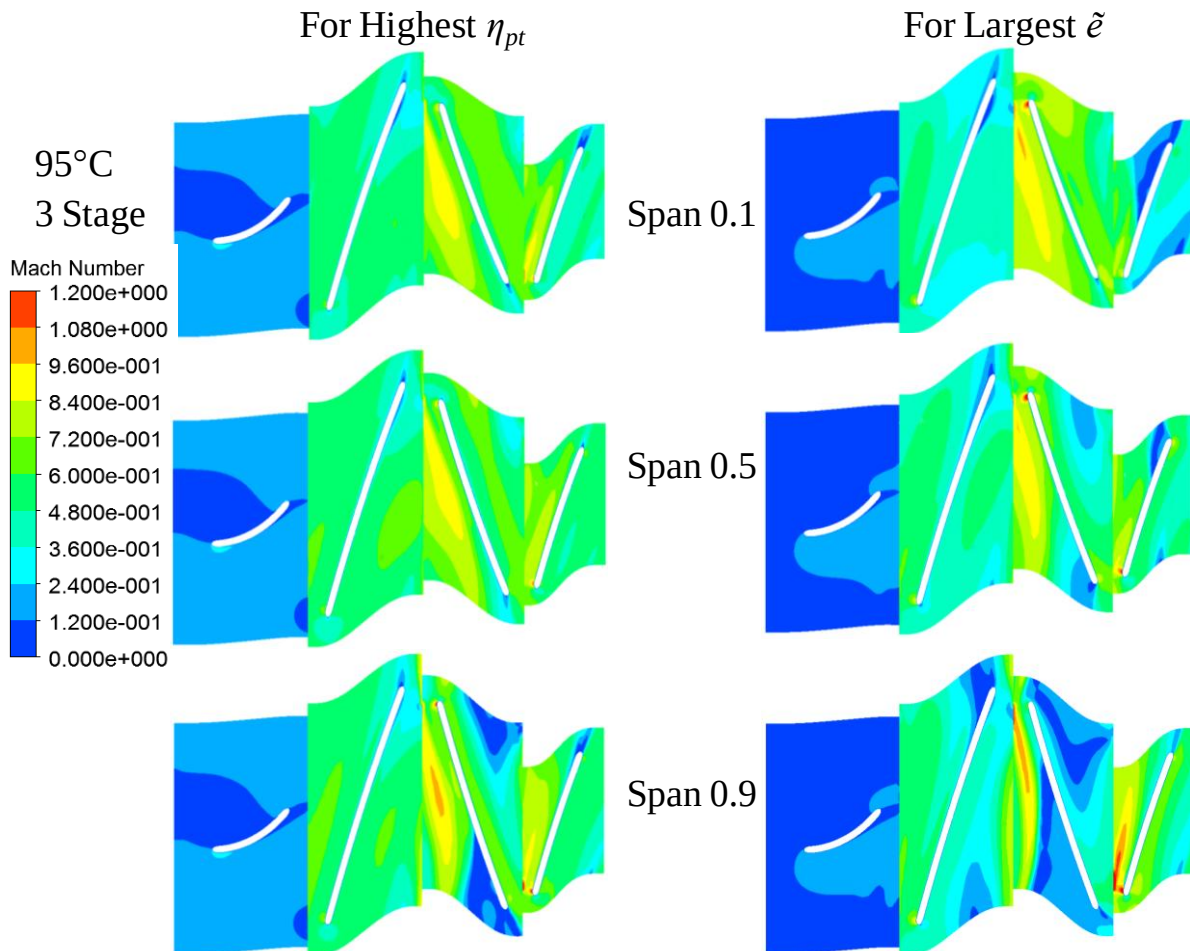


Figure A5.30. Blade-to-Blade View of Relative Mach Number Contours for 3 Counter-Rotating Stages with Saturated Vapor Inlet at 95°C

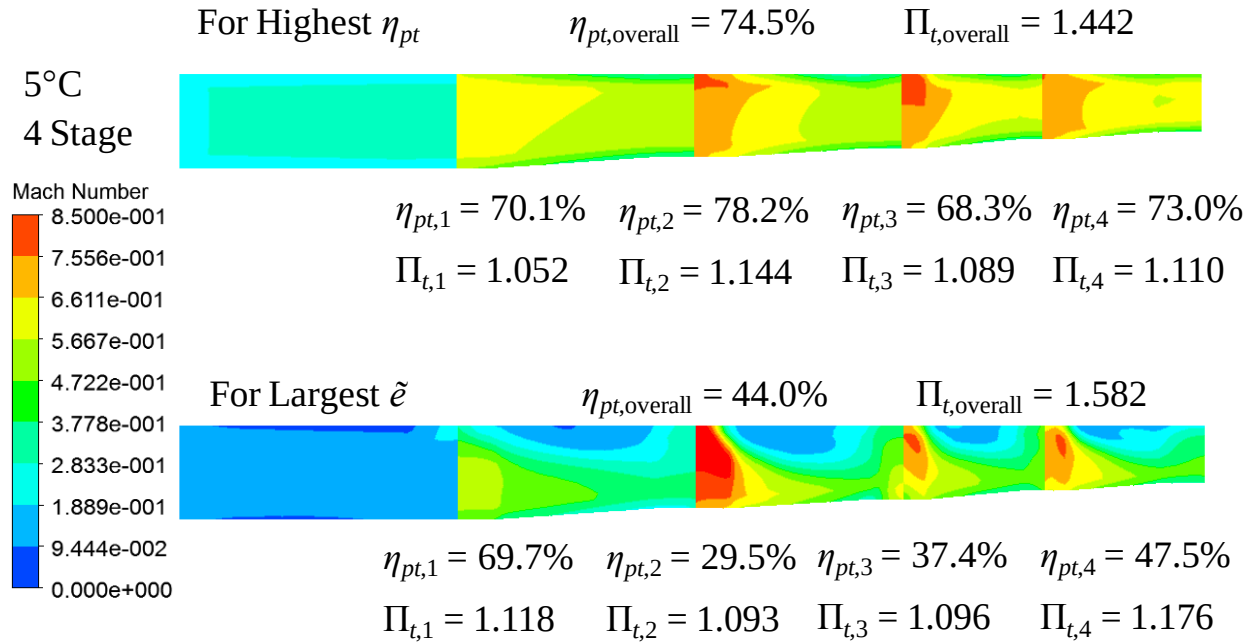


Figure A5.31. Meridional View of Relative Mach Number Contours for 4 Counter-Rotating Stages with Saturated Vapor Inlet at 5°C

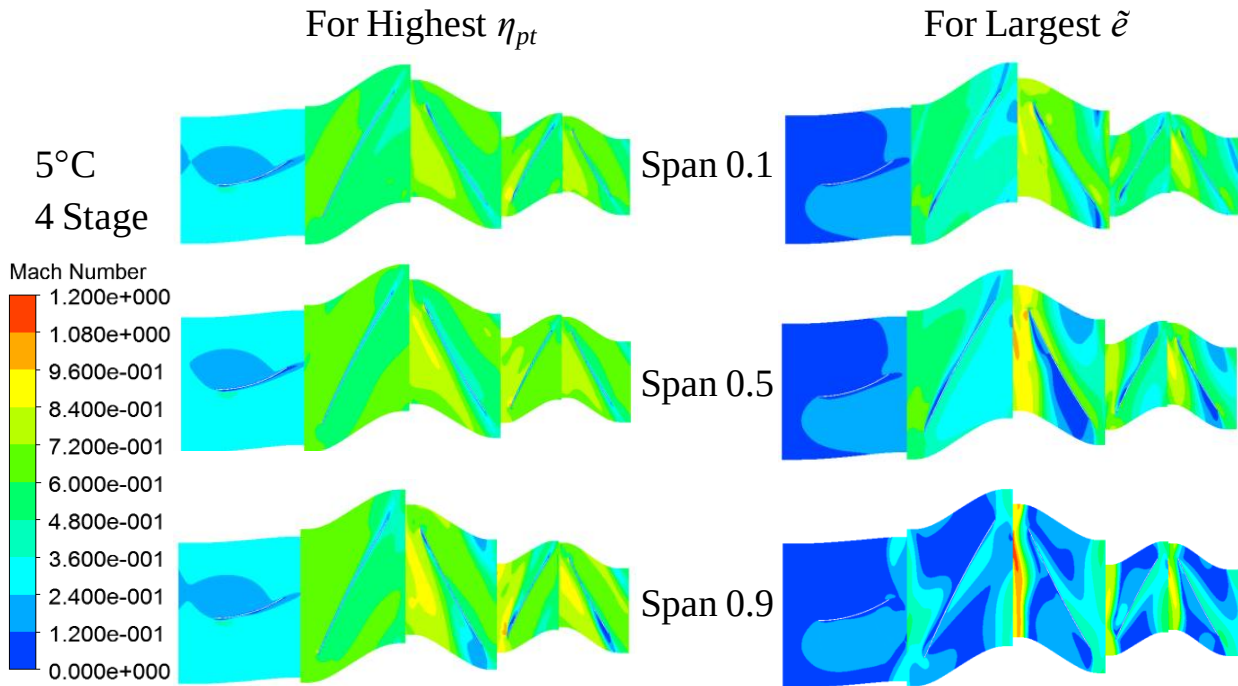


Figure A5.32. Blade-to-Blade View of Relative Mach Number Contours for 4 Counter-Rotating Stages with Saturated Vapor Inlet at 5°C

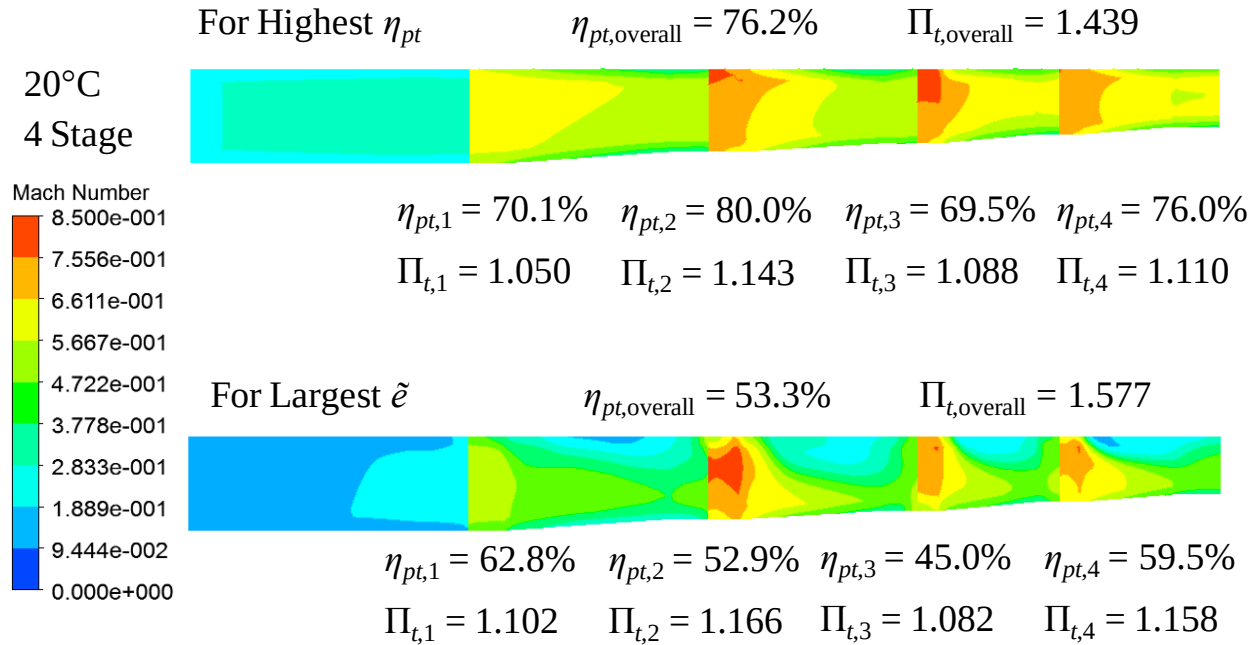


Figure A5.33. Meridional View of Relative Mach Number Contours for 4 Counter-Rotating Stages with Saturated Vapor Inlet at 20°C

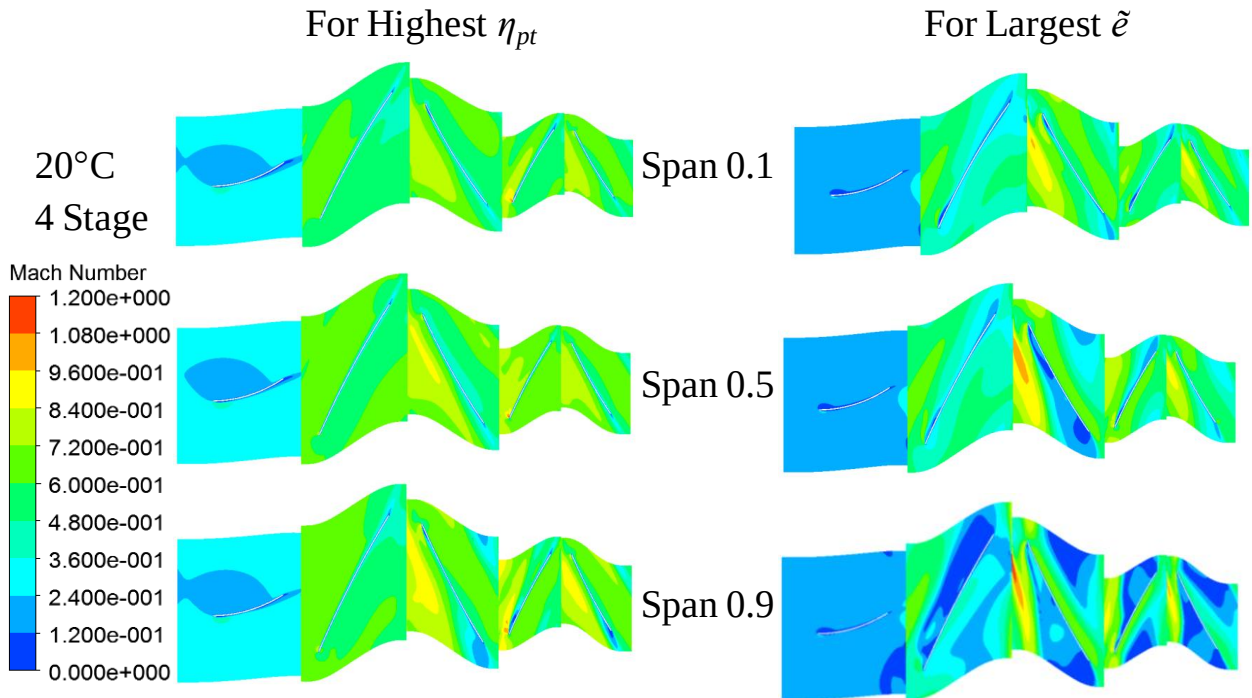


Figure A5.34. Blade-to-Blade View of Relative Mach Number Contours for 4 Counter-Rotating Stages with Saturated Vapor Inlet at 20°C

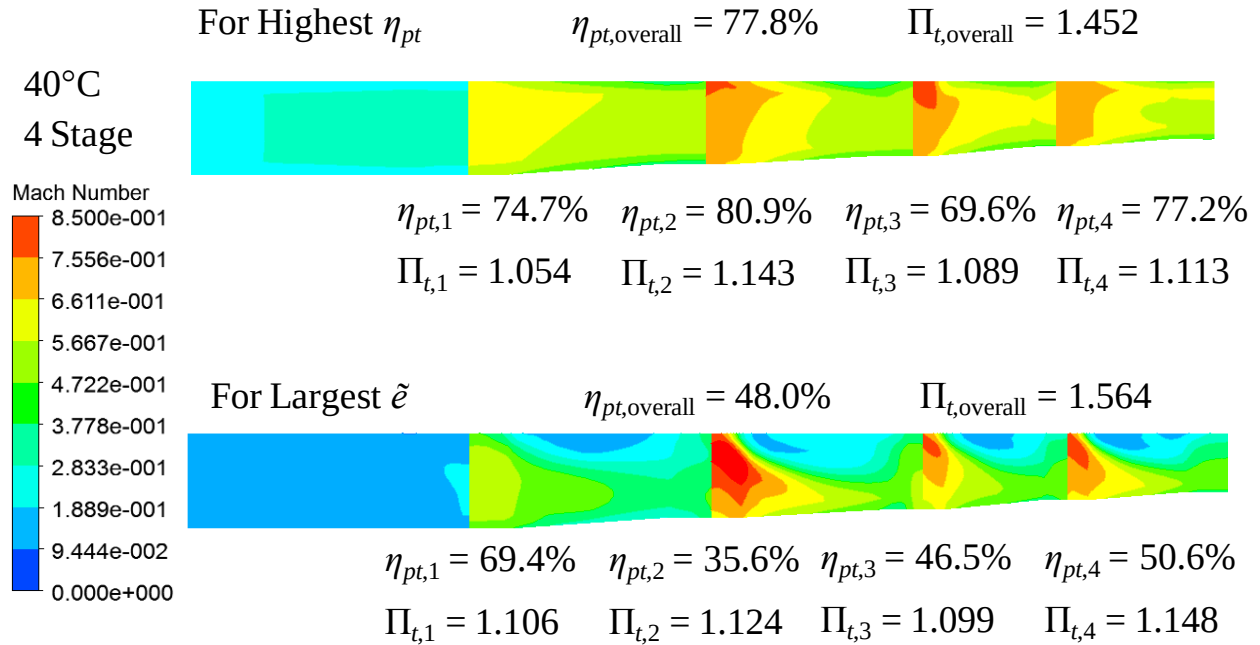


Figure A5.35. Meridional View of Relative Mach Number Contours for 4 Counter-Rotating Stages with Saturated Vapor Inlet at 40°C

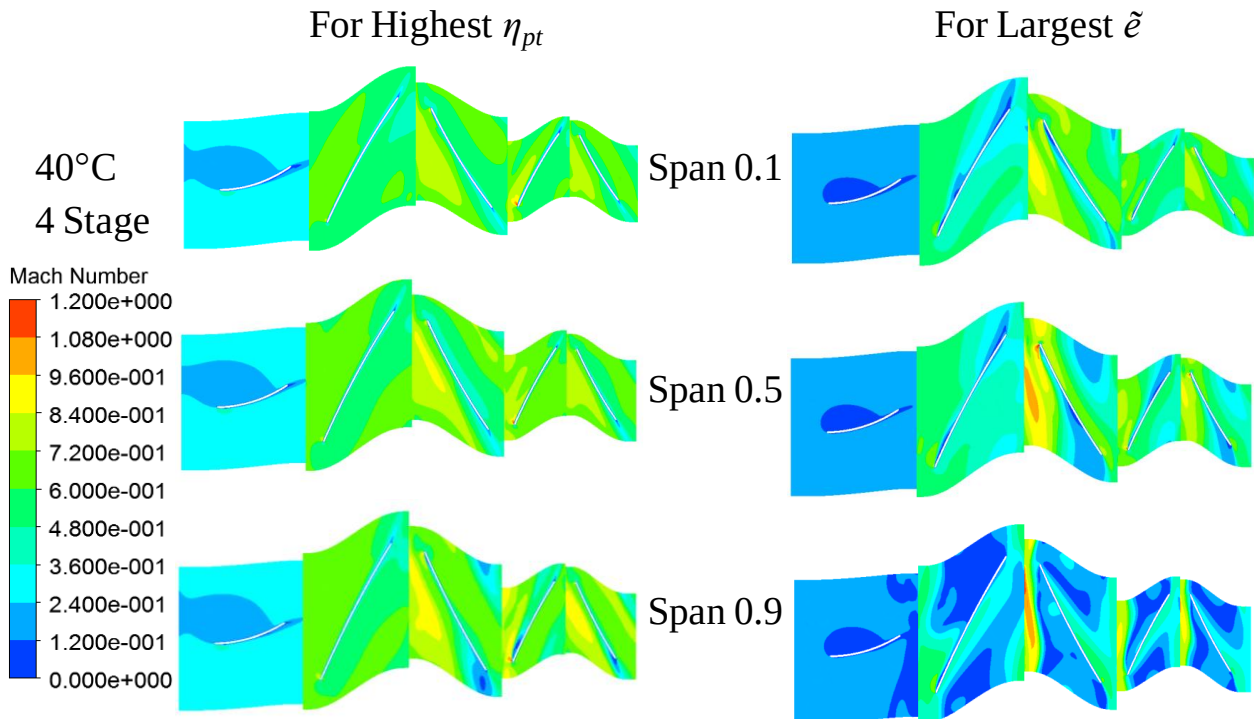


Figure A5.36. Blade-to-Blade View of Relative Mach Number Contours for 4 Counter-Rotating Stages with Saturated Vapor Inlet at 40°C

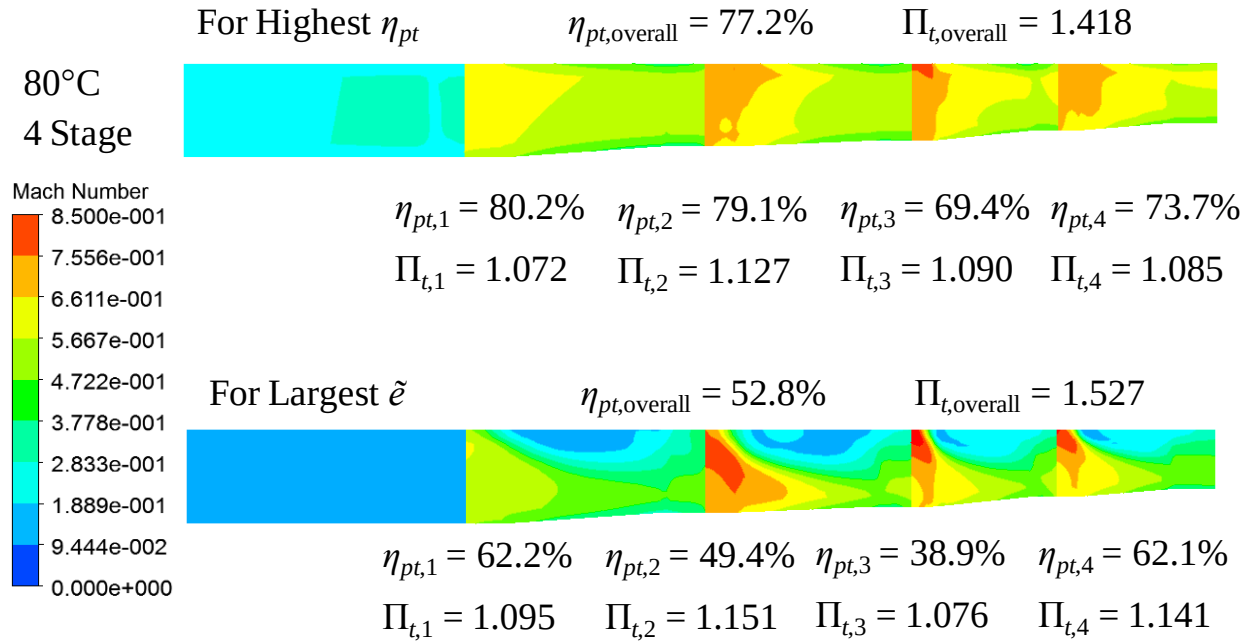


Figure A5.37. Meridional View of Relative Mach Number Contours for 4 Counter-Rotating Stages with Saturated Vapor Inlet at 80°C

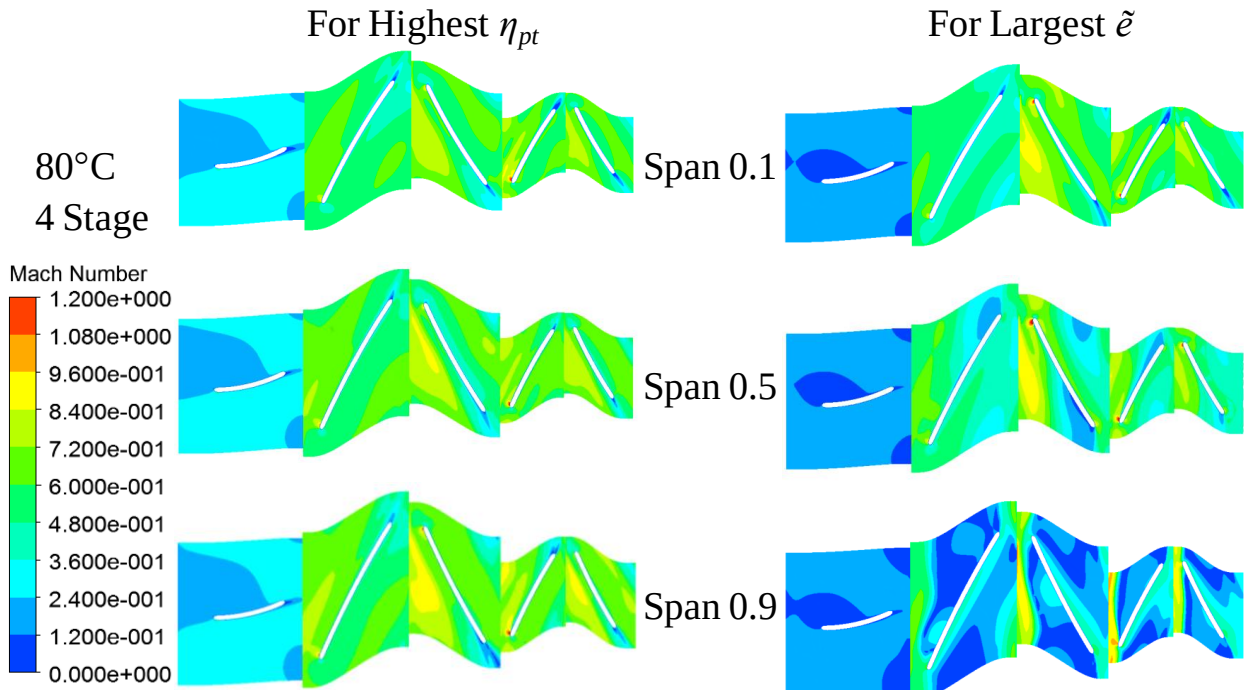


Figure A5.38. Blade-to-Blade View of Relative Mach Number Contours for 4 Counter-Rotating Stages with Saturated Vapor Inlet at 80°C

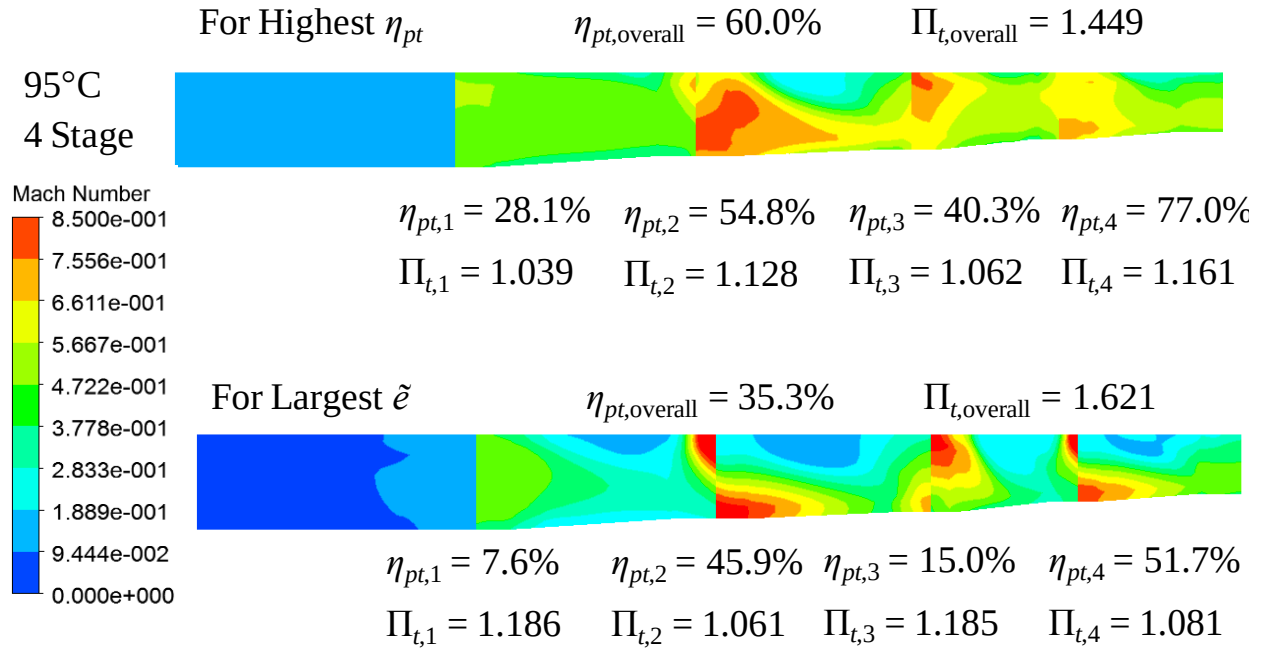


Figure A5.39. Meridional View of Relative Mach Number Contours for 4 Counter-Rotating Stages with Saturated Vapor Inlet at 95°C

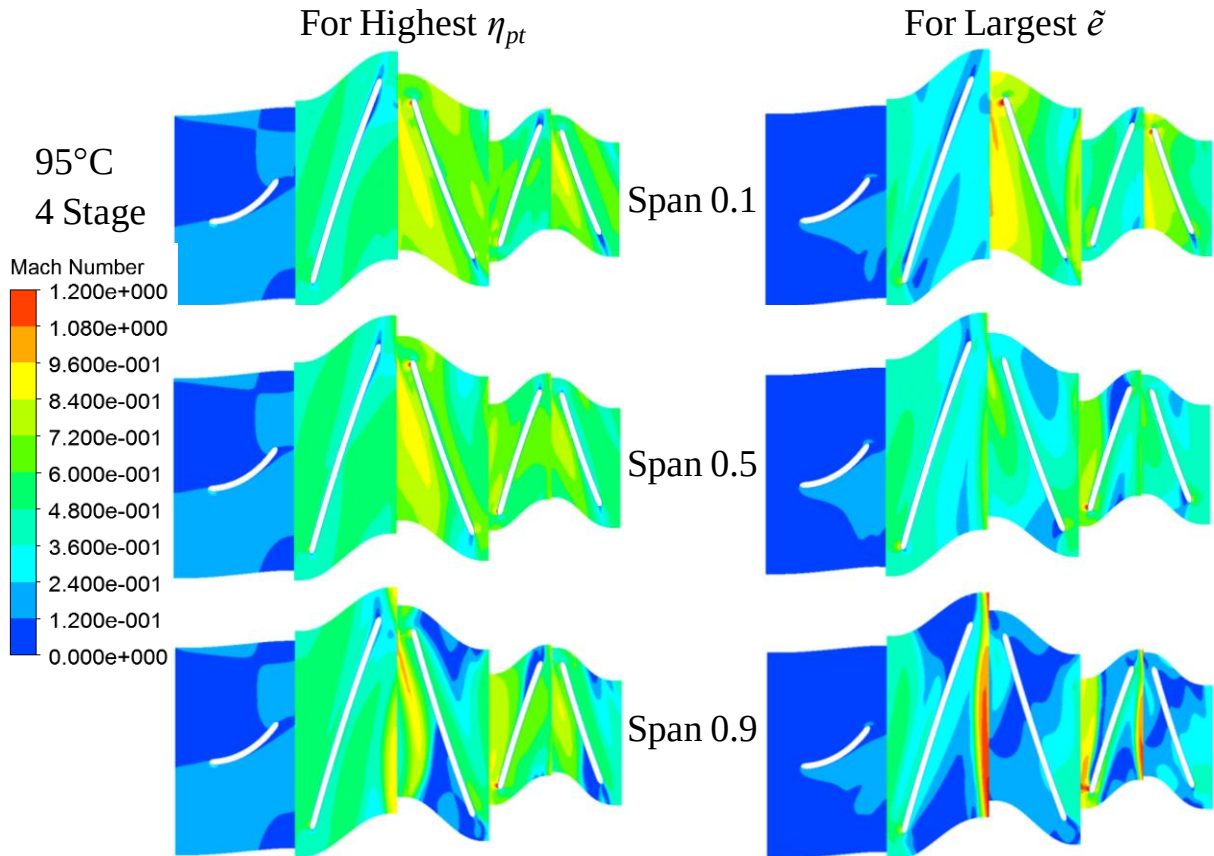


Figure A5.40. Blade-to-Blade View of Relative Mach Number Contours for 4 Counter-Rotating Stages with Saturated Vapor Inlet at 95°C

APPENDIX 6 Best Design Points for Other Gases

A6.1 Air

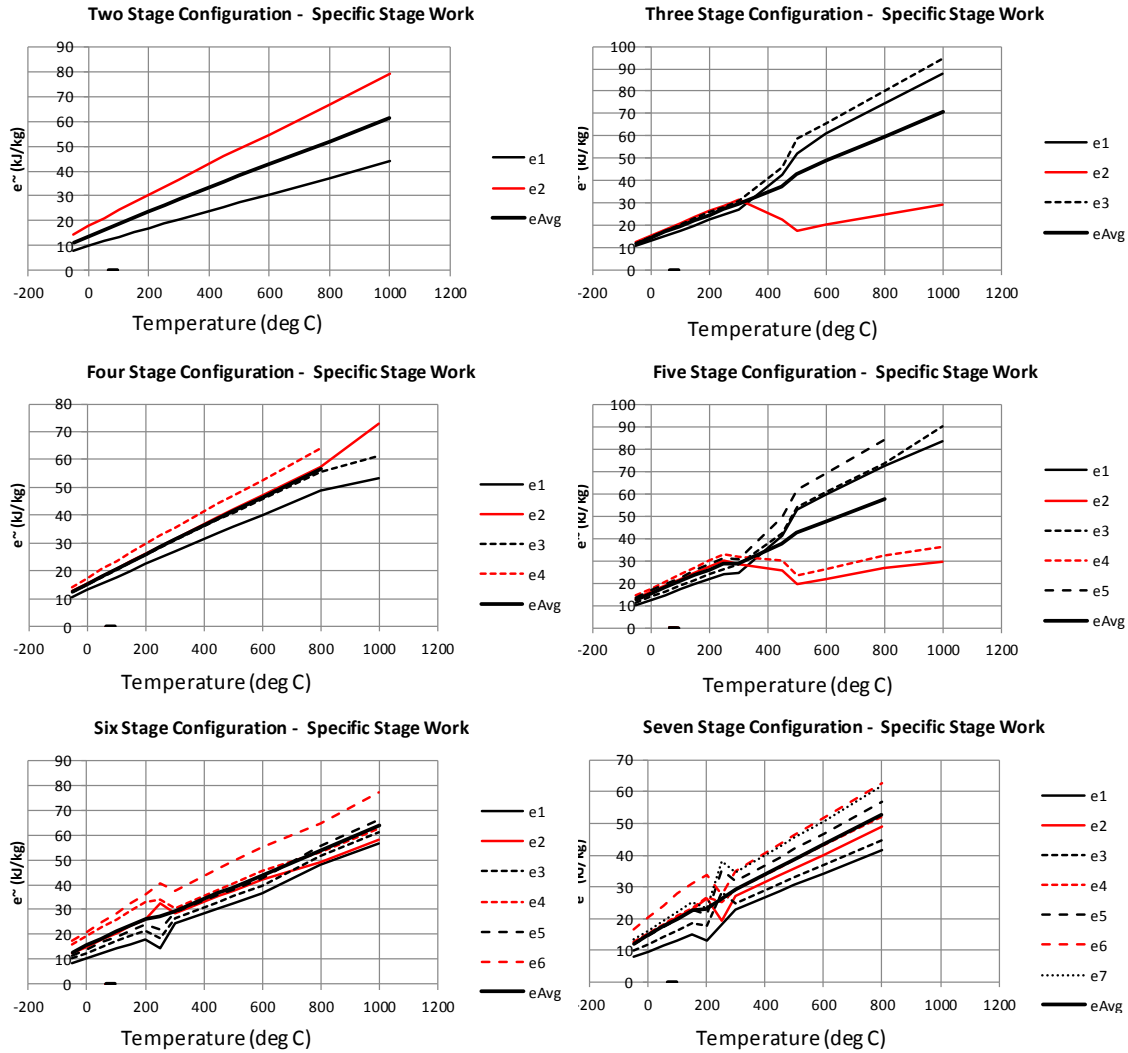
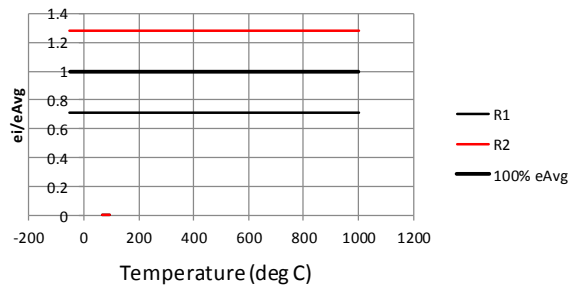
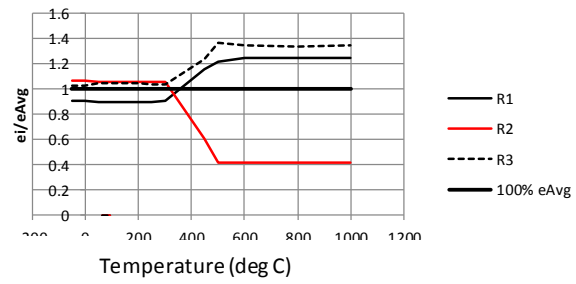


Figure A6.1.1. Specific Work vs. Temperature at First Rotor Inlet (Air)

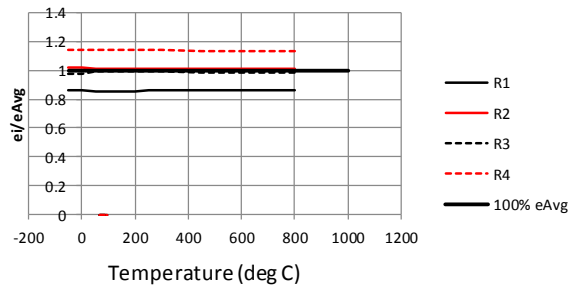
**Two Stage Configuration - Specific Stage Work
Normalized by Average Work Consumed by All Stages**



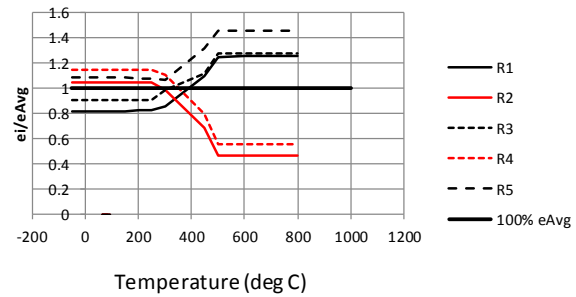
**Three Stage Configuration - Specific Stage Work
Normalized by Average Work Consumed by All Stages**



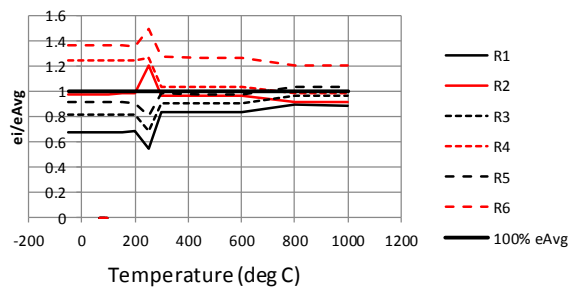
**Four Stage Configuration - Specific Stage Work
Normalized by Average Work Consumed by All Stages**



**Five Stage Configuration - Specific Stage Work
Normalized by Average Work Consumed by All Stages**



**Six Stage Configuration - Specific Stage Work
Normalized by Average Work Consumed by All Stages**



**Seven Stage Configuration - Specific Stage Work
Normalized by Average Work Consumed by All Stages**

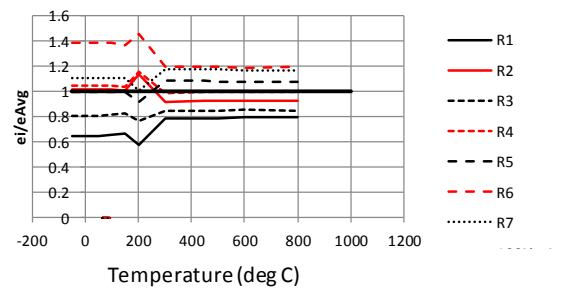


Figure A6.1.2. Normalized Specific Work vs. Temperature at First Rotor Inlet (Air)

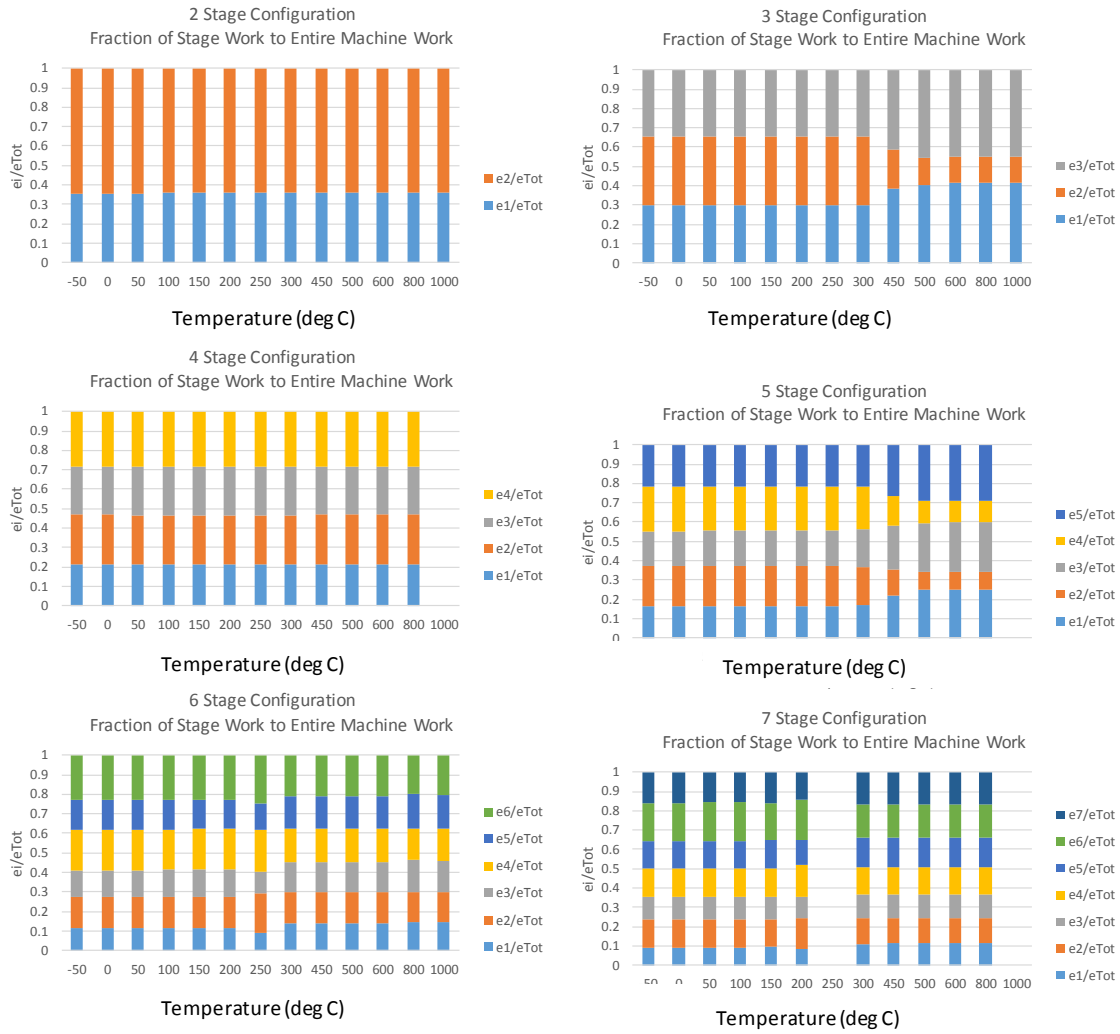


Figure A6.1.3. Rotor Work Fraction of Combined Work Total vs. Temperature at First Rotor Inlet (Air)

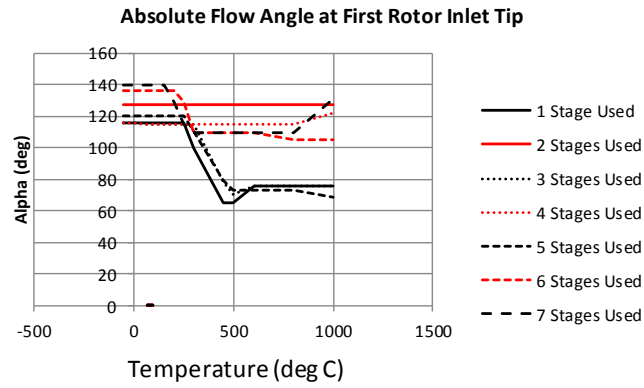


Figure A6.1.4. Absolute Flow Angle at First Rotor Inlet vs. Temperature at First Rotor Inlet (Air)

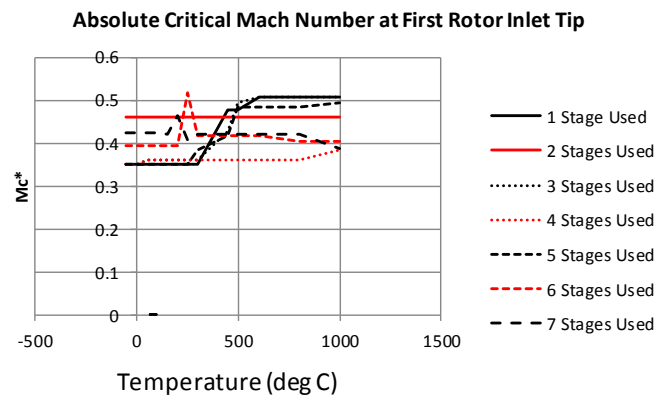


Figure A6.1.5. Absolute Critical Mach Number at First Rotor Inlet vs. Temperature at First Rotor Inlet (Air)

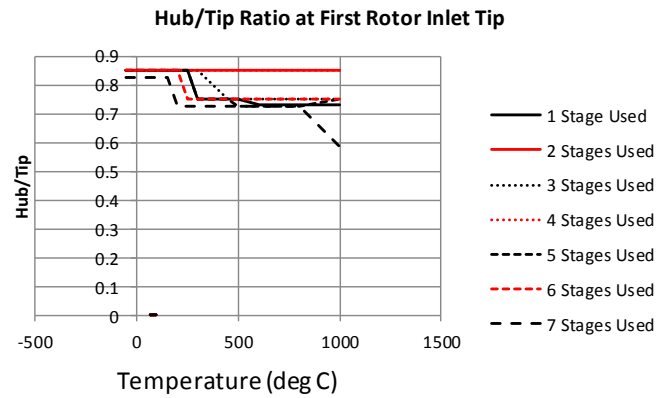


Figure A6.1.6. Hub/Tip Ratio at First Rotor Inlet vs. Temperature at First Rotor Inlet (Air)

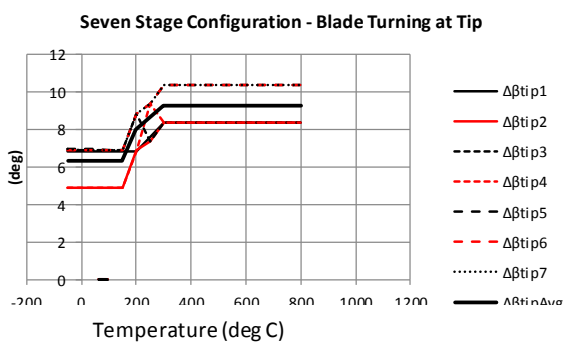
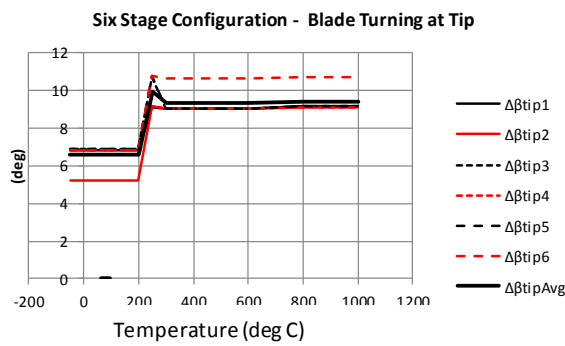
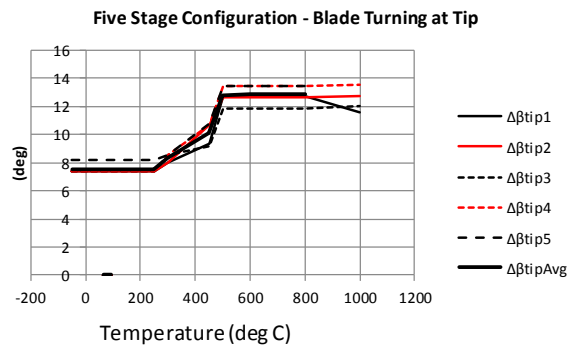
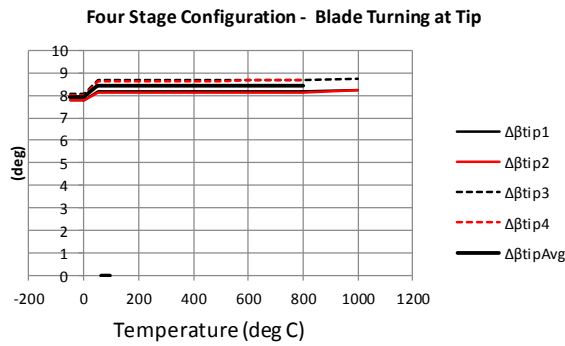
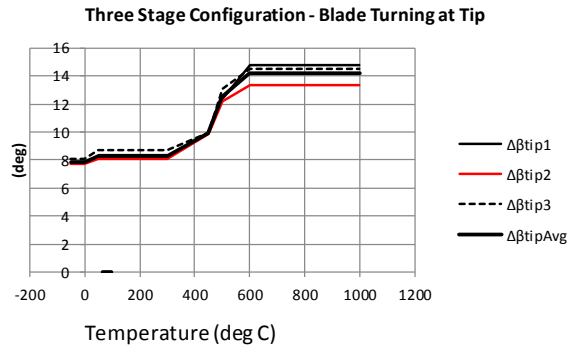
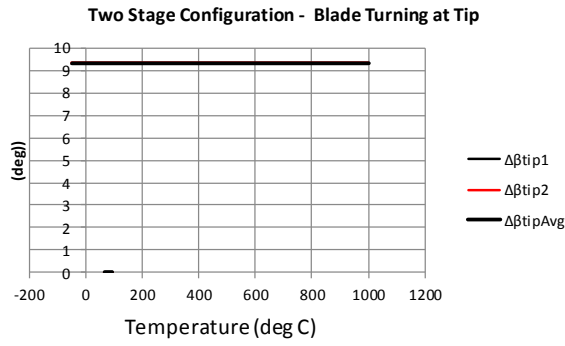


Figure A6.1.7. Blade Turning Angle vs. Temperature at First Rotor Inlet (Air)

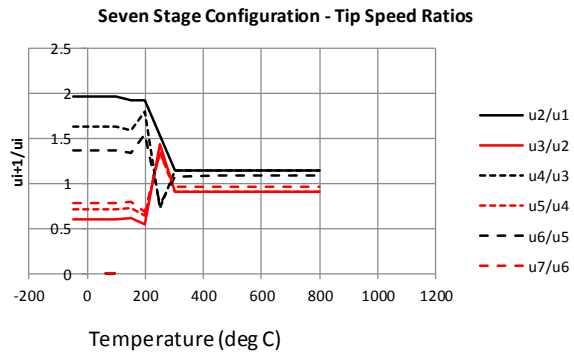
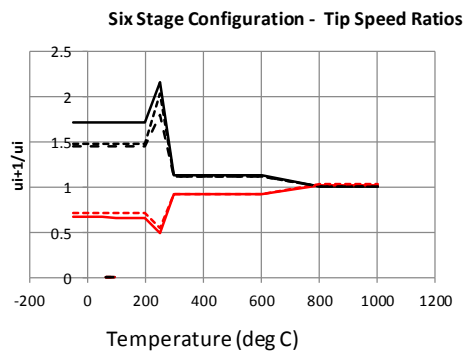
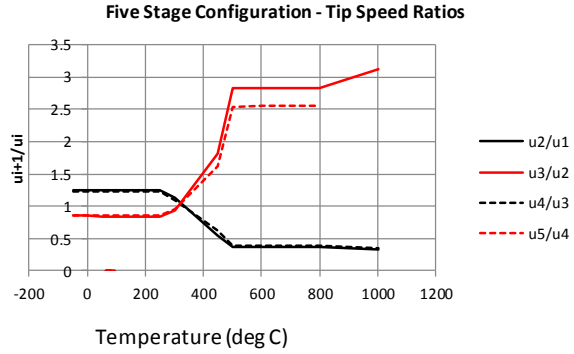
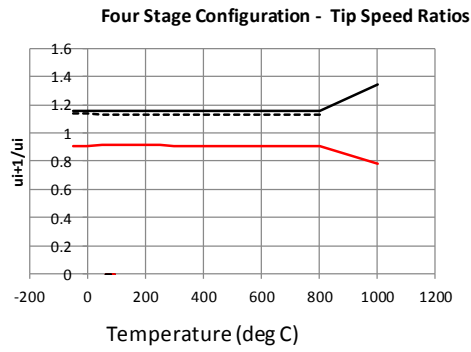
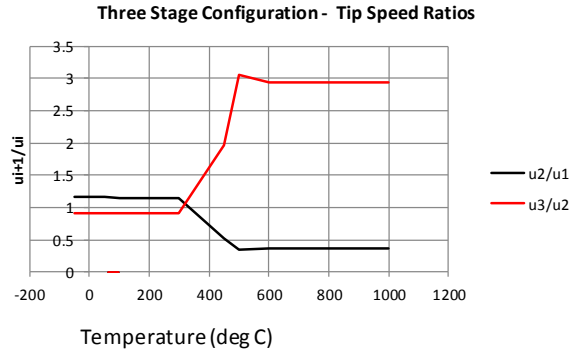
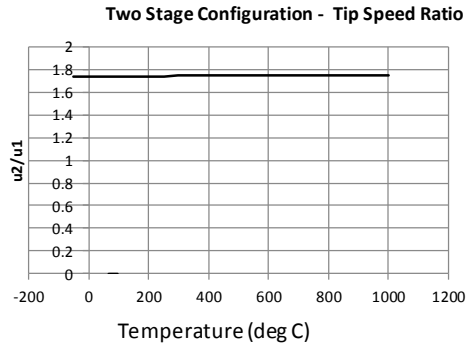


Figure A6.1.8. Tip Speed Ratio vs. Temperature at First Rotor Inlet (Air)

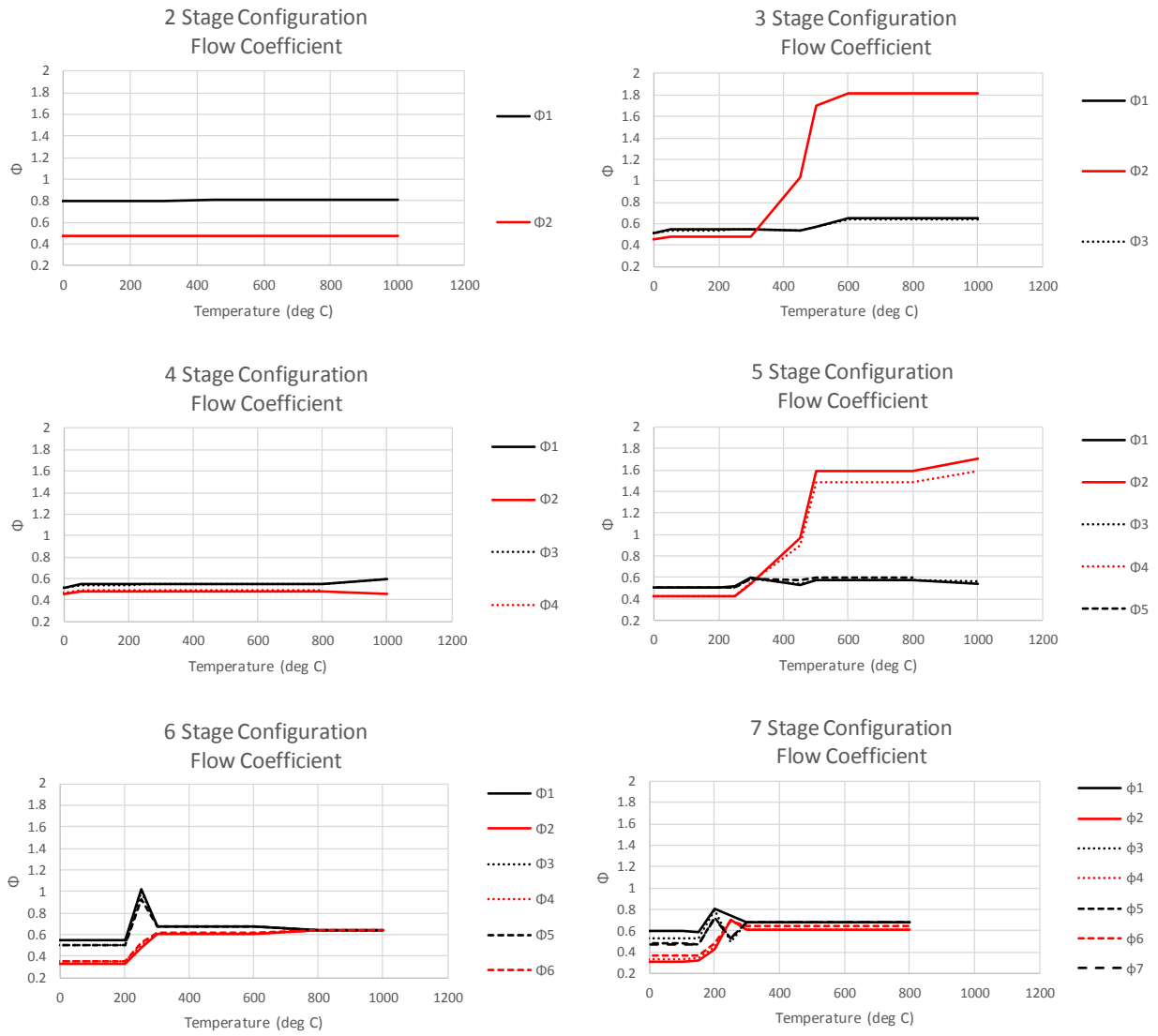


Figure A6.1.9. Flow Coefficient vs. Temperature at First Rotor Inlet (Air)

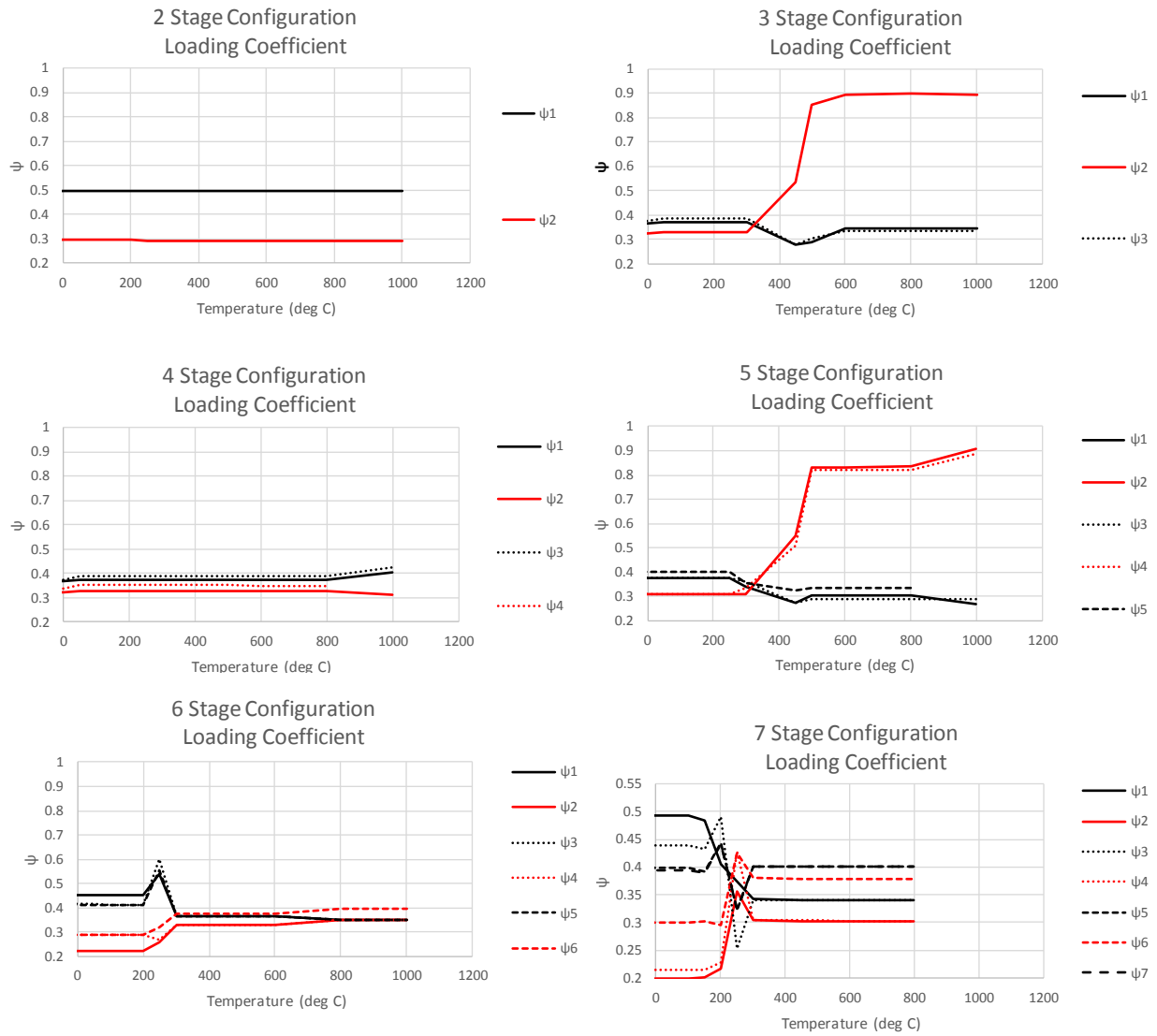


Figure A6.1.10. Blade Loading Coefficient vs. Temperature at First Rotor Inlet (Air)

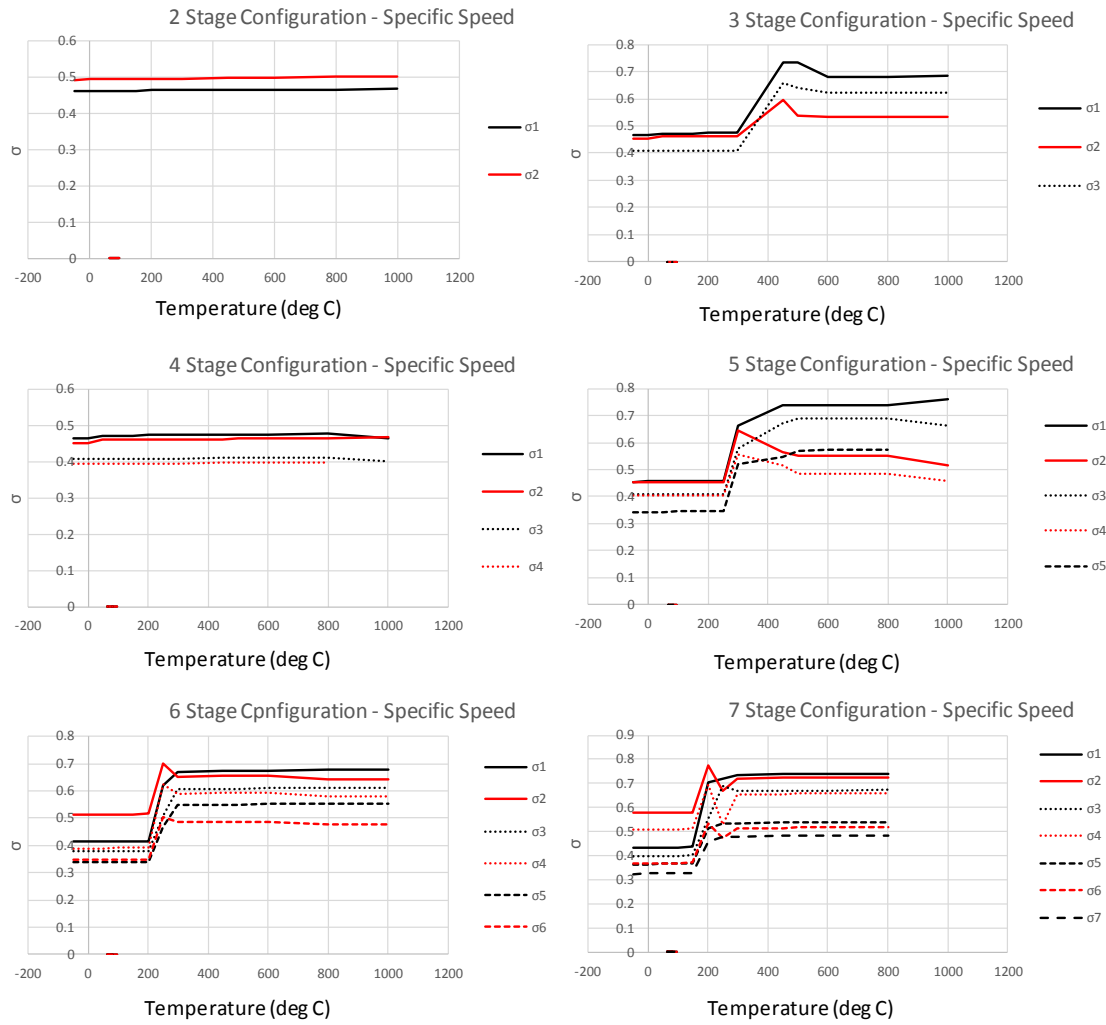


Figure A6.1.11 Specific Speed vs. Temperature at First Rotor Inlet (Air)



Figure A6.1.12 Total Pressure Ratio vs. Temperature at First Rotor Inlet (Air)

A6.2 NCG Mixture

This mixture is representative of the contents of the primary condenser in a geothermal power plant. The NCG is considered as its own mixture. Then a NCG-water vapor-air mixture is considered.

Table A6.2.1. NCG Mixture Components		
Component	Molecular Weight (kg/kmol)	Mass Fraction
CH ₄	16.04	0.0048
CO ₂	44.01	0.9120
H ₂	2.016	0.0011
N ₂	28.02	0.0114
H ₂ S	34.08	0.0680
NH ₃	17.03	0.0023
Ar	39.95	0.0004
NCG	41.47	0.8110
H ₂ O Vapor	18.02	0.1723
Air	28.97	0.0167

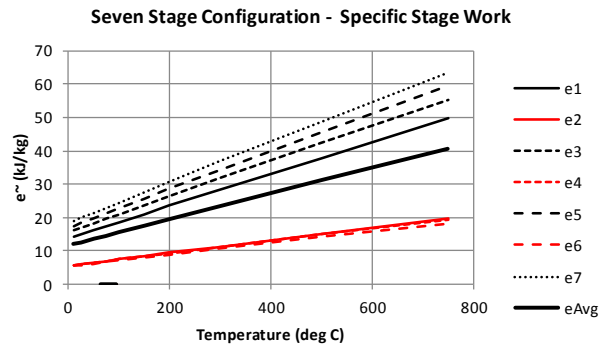
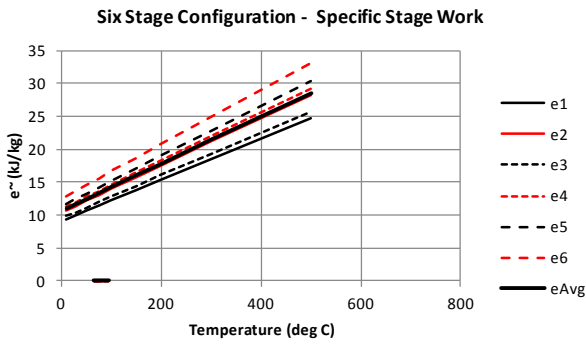
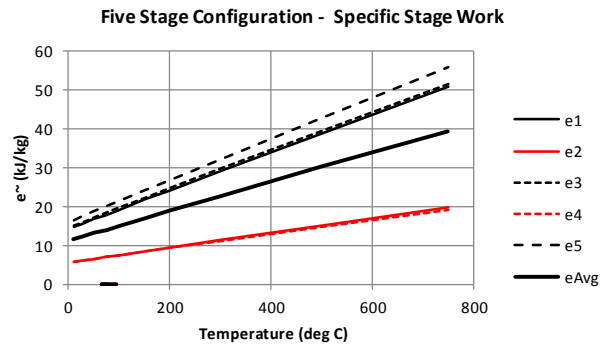
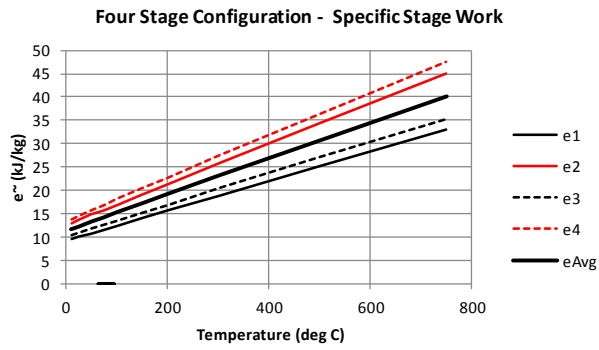
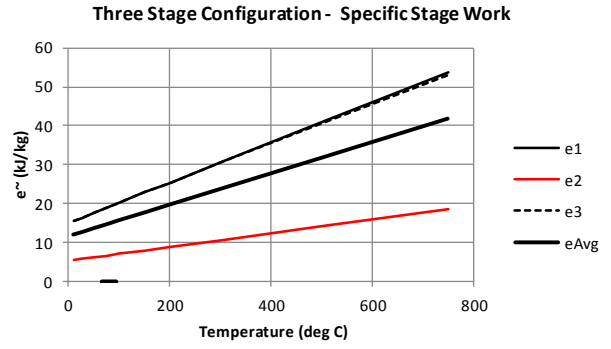
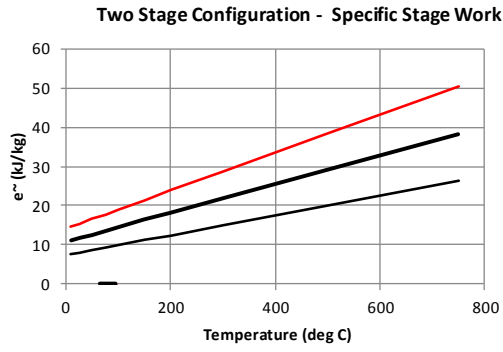
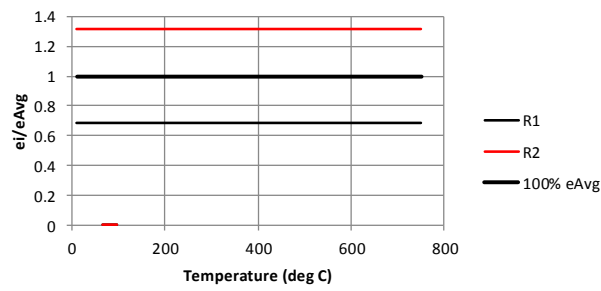
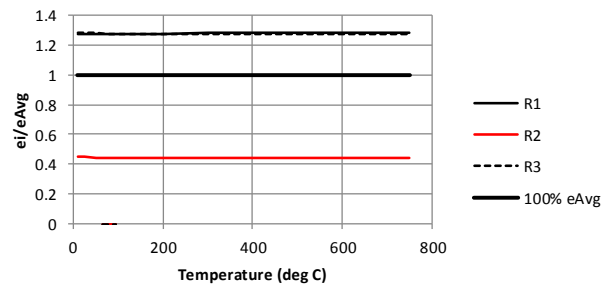


Figure A6.2.1. Specific Work vs. Temperature at First Rotor Inlet (NCG Mix)

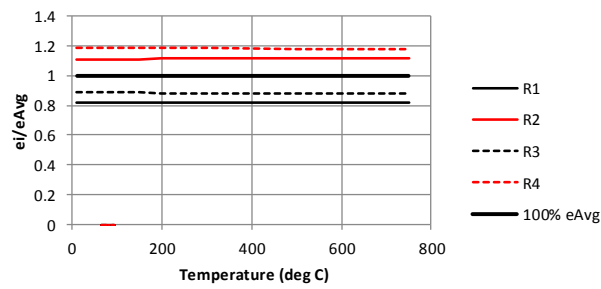
**Two Stage Configuration - Specific Stage Work
Normalized by Average Work Consumed by All Stages**



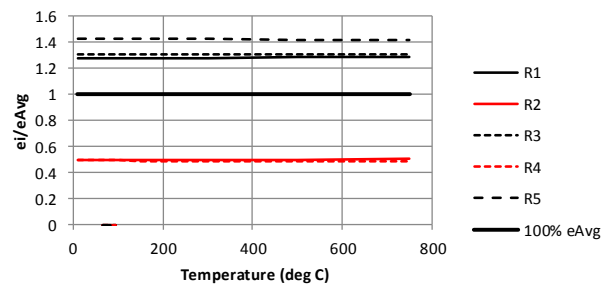
**Three Stage Configuration - Specific Stage Work
Normalized by Average Work Consumed by All Stages**



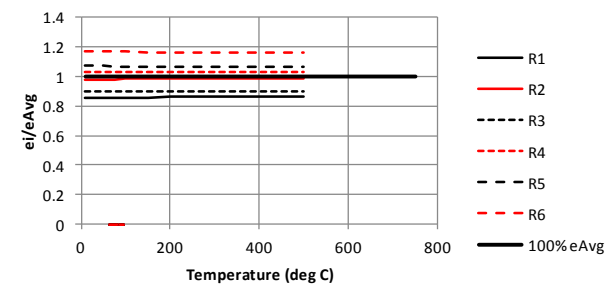
**Four Stage Configuration - Specific Stage Work
Normalized by Average Work Consumed by All Stages**



**Five Stage Configuration - Specific Stage Work
Normalized by Average Work Consumed by All Stages**



**Six Stage Configuration - Specific Stage Work Normalized
by Average Work Consumed by All Stages**



**Seven Stage Configuration - Specific Stage Work
Normalized by Average Work Consumed by All Stages**

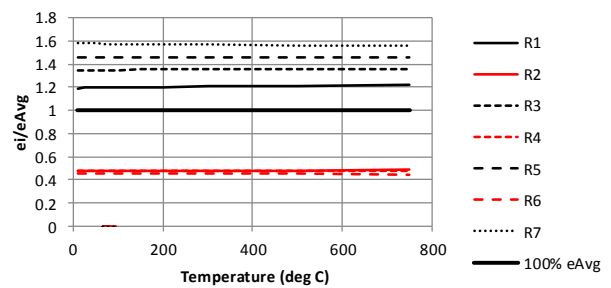


Figure A6.2.2. Normalized Specific Work vs. Temperature at First Rotor Inlet (NCG Mix)

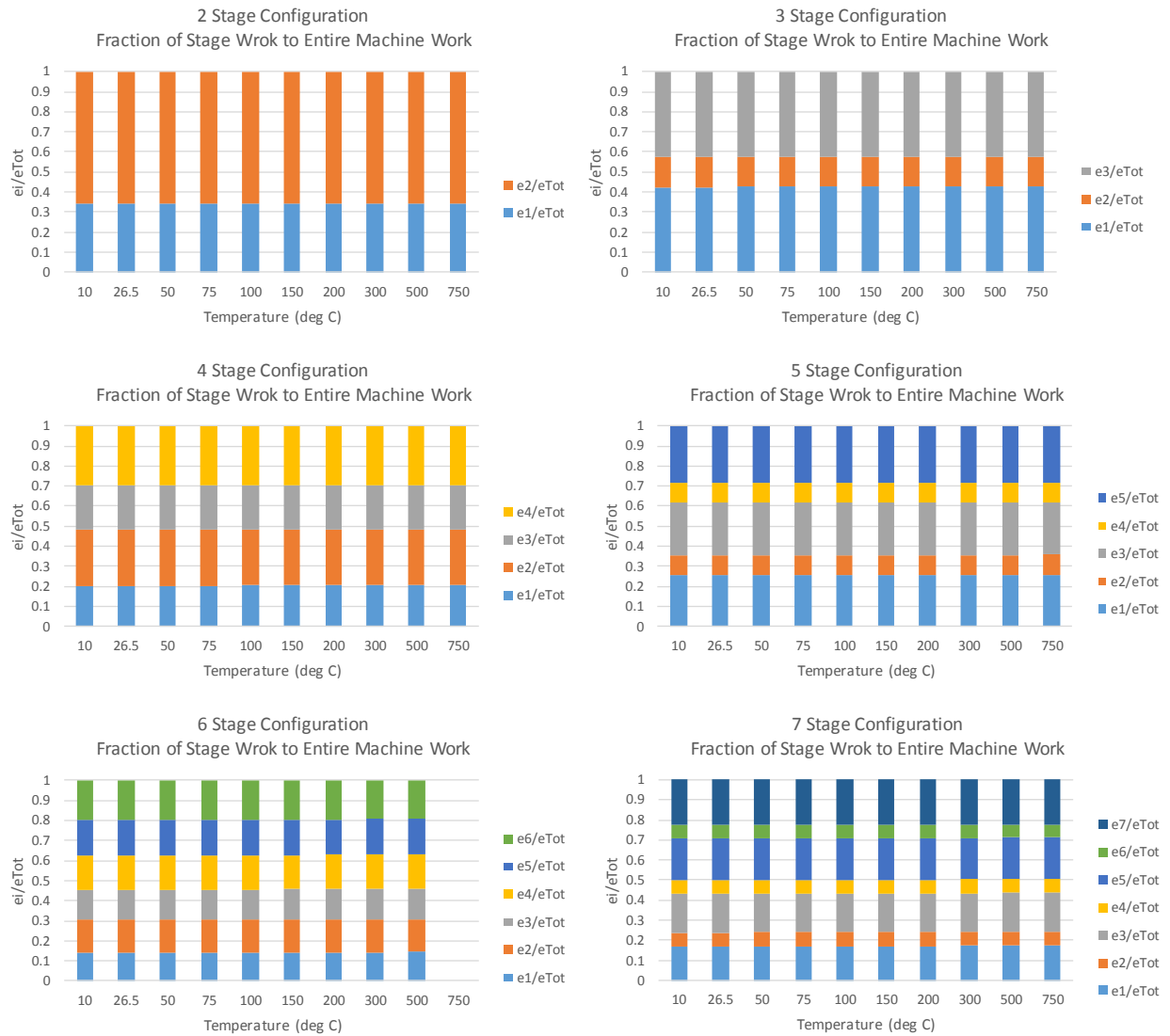


Figure A6.2.3. Rotor Work Fraction of Combined Work Total vs. Temperature at First Rotor Inlet (NCG Mix)

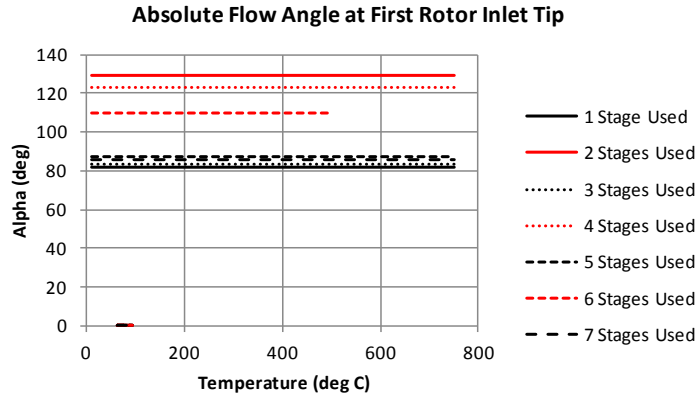


Figure A6.2.4. Absolute Flow Angle at First Rotor Inlet vs. Temperature at First Rotor Inlet (NCG Mix)

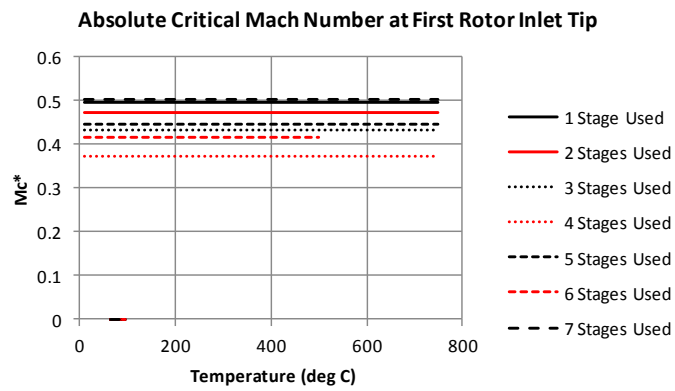


Figure A6.2.5. Absolute Critical Mach Number at First Rotor Inlet vs. Temperature at First Rotor Inlet (NCG Mix)

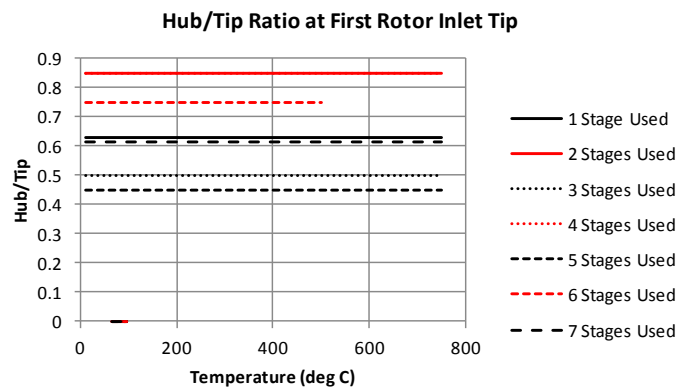


Figure A6.2.6. Hub/Tip Ratio at First Rotor Inlet vs. Temperature at First Rotor Inlet (NCG Mix)

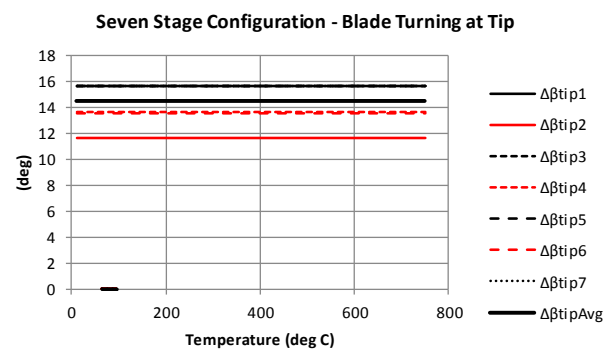
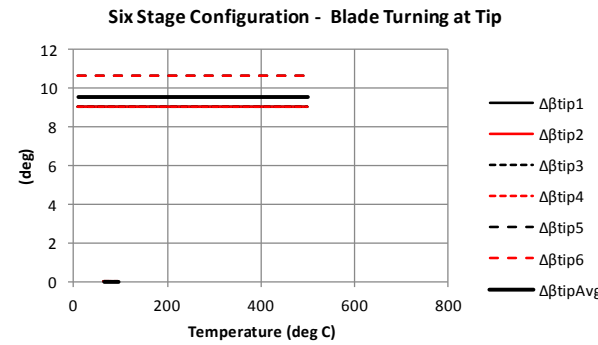
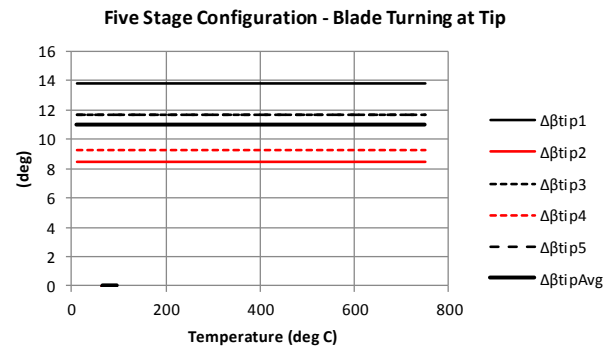
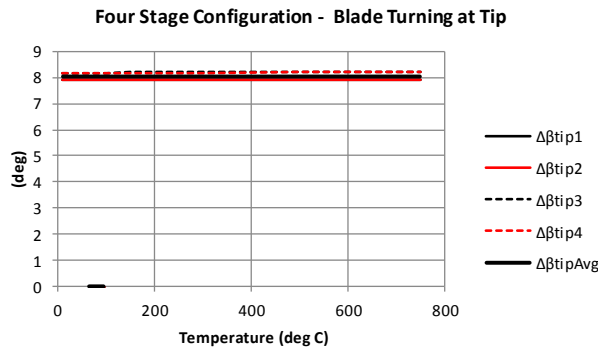
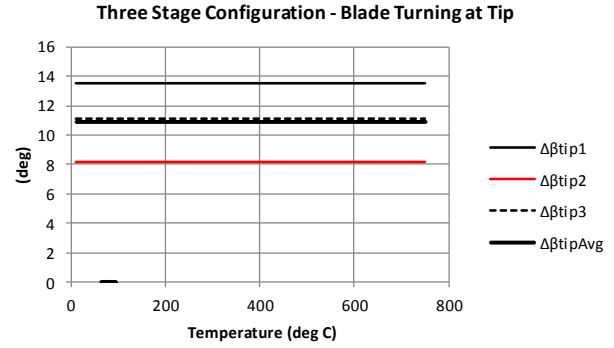
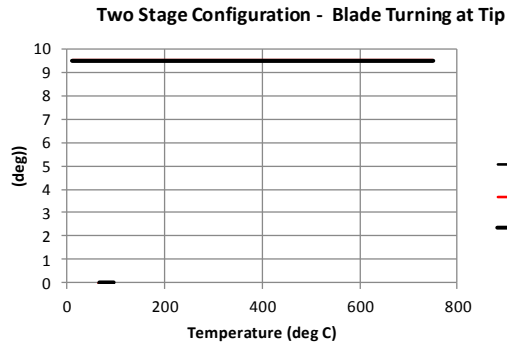


Figure A6.2.7. Blade Turning vs. Temperature at First Rotor Inlet (NCG Mix)

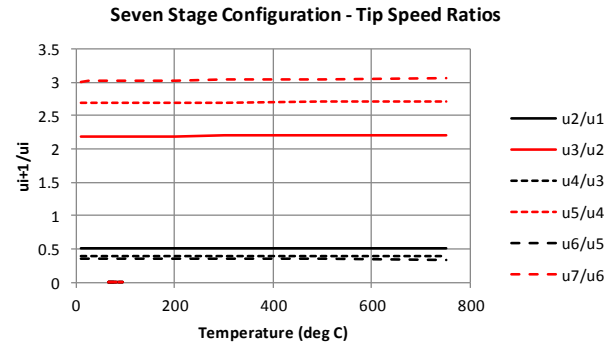
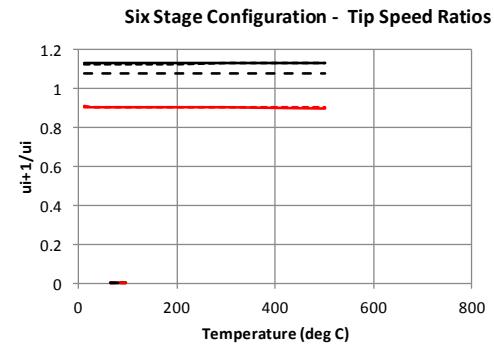
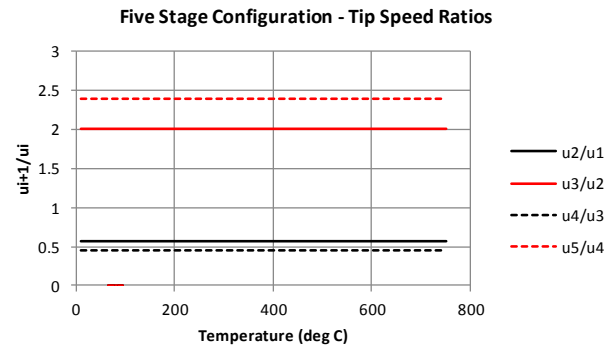
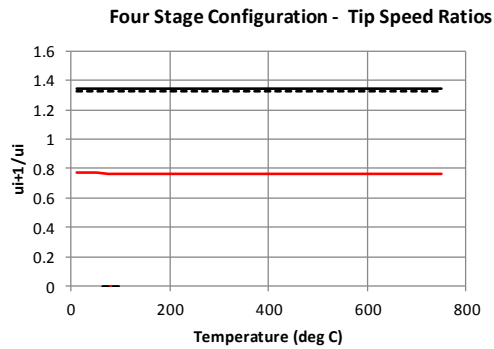
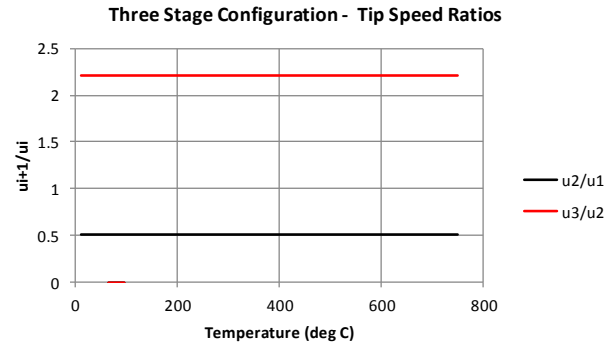
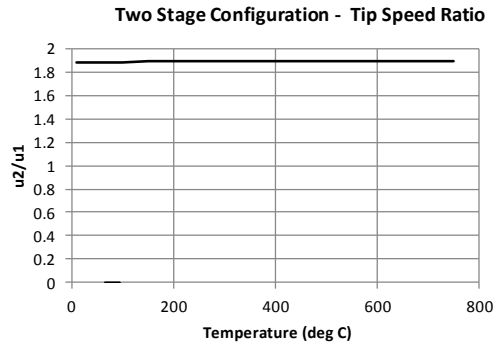


Figure A6.2.8. Tip Speed Ratio vs. Temperature at First Rotor Inlet (NCG Mix)

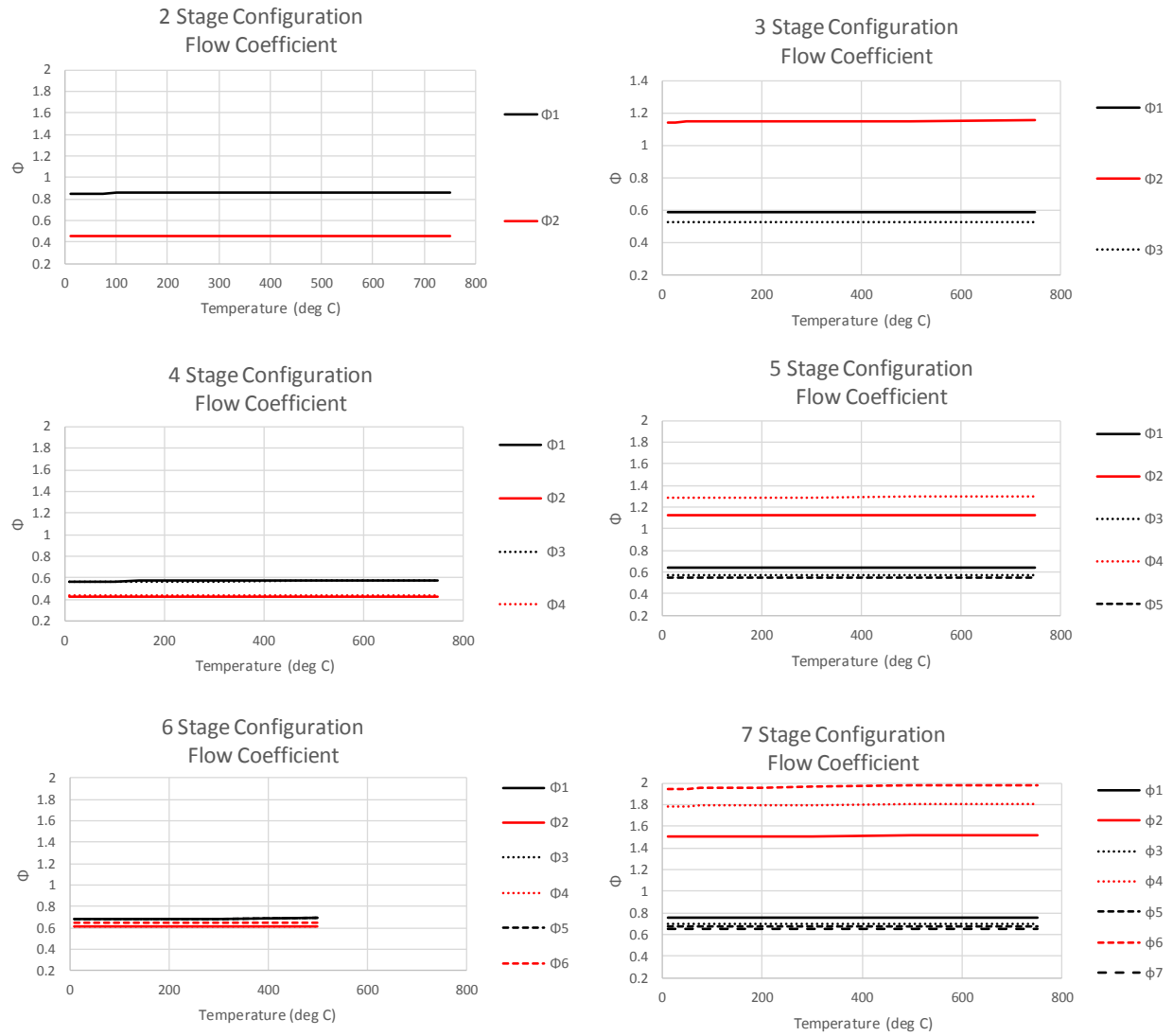


Figure A6.2.9. Flow Coefficient vs. Temperature at First Rotor Inlet (NCG Mix)

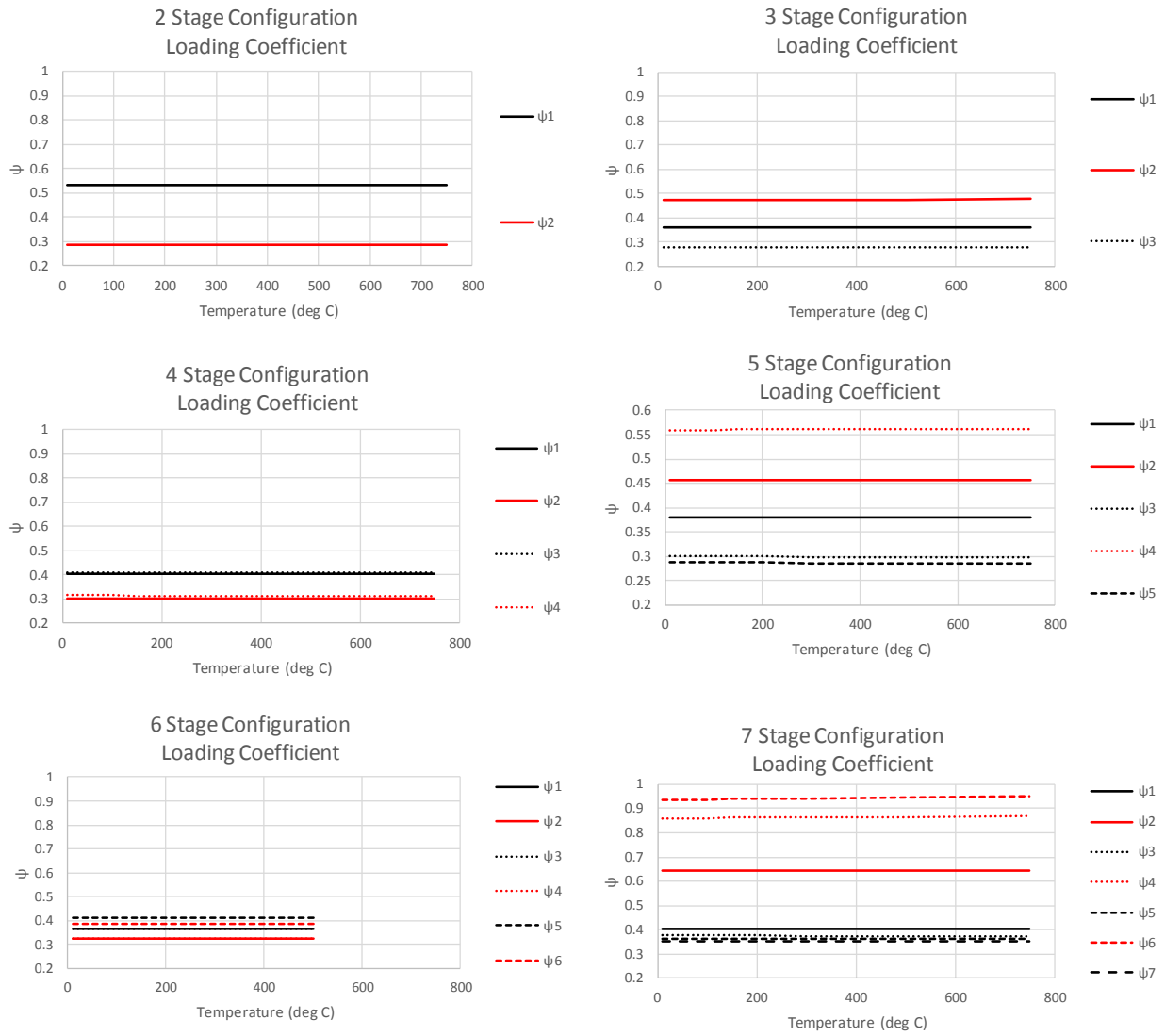


Figure A6.2.10. Blade Loading Coefficient vs. Temperature at First Rotor Inlet (NCG Mix)

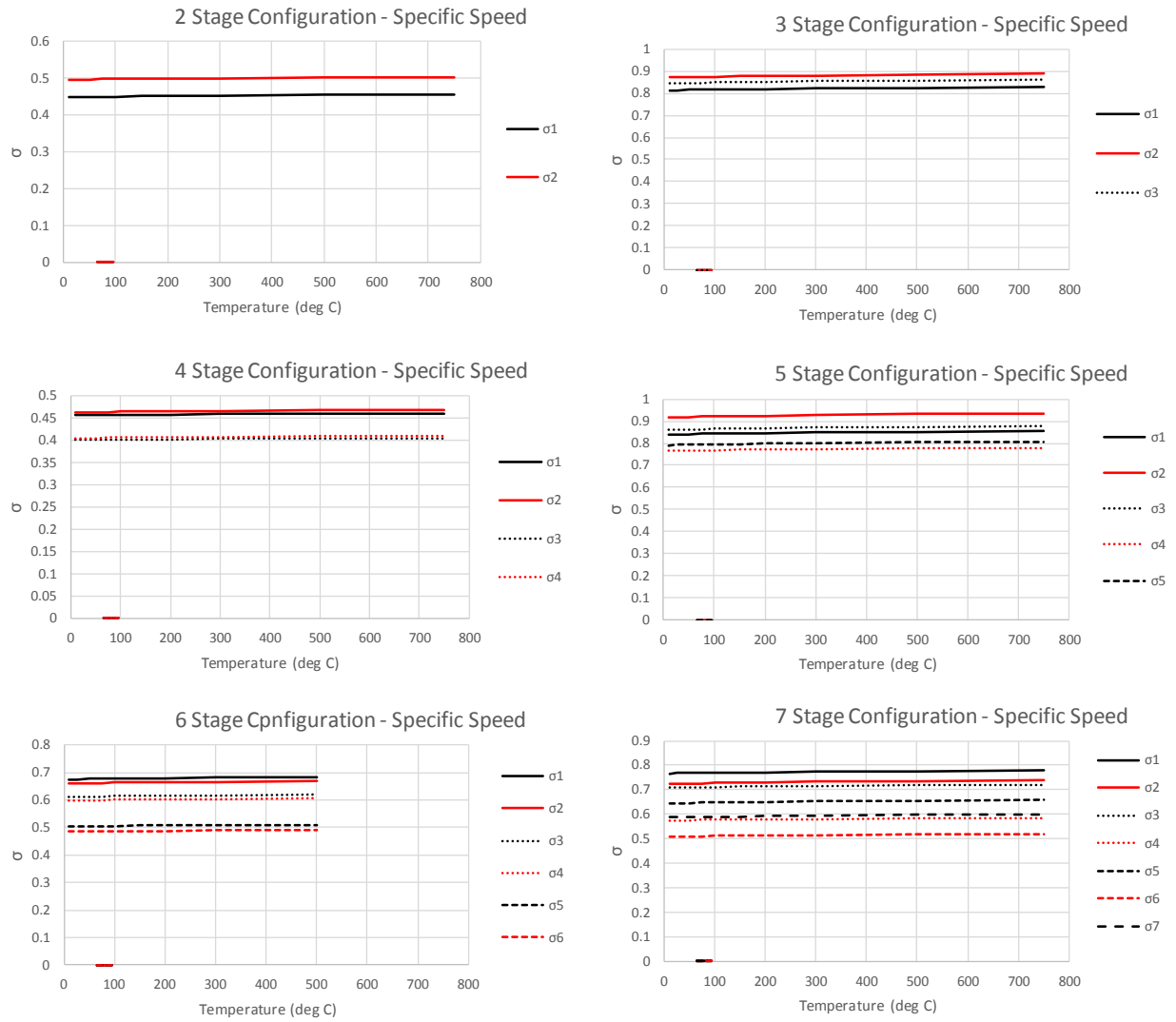


Figure A6.2.11. Specific Speed vs. Temperature at First Rotor Inlet (NCG Mix)

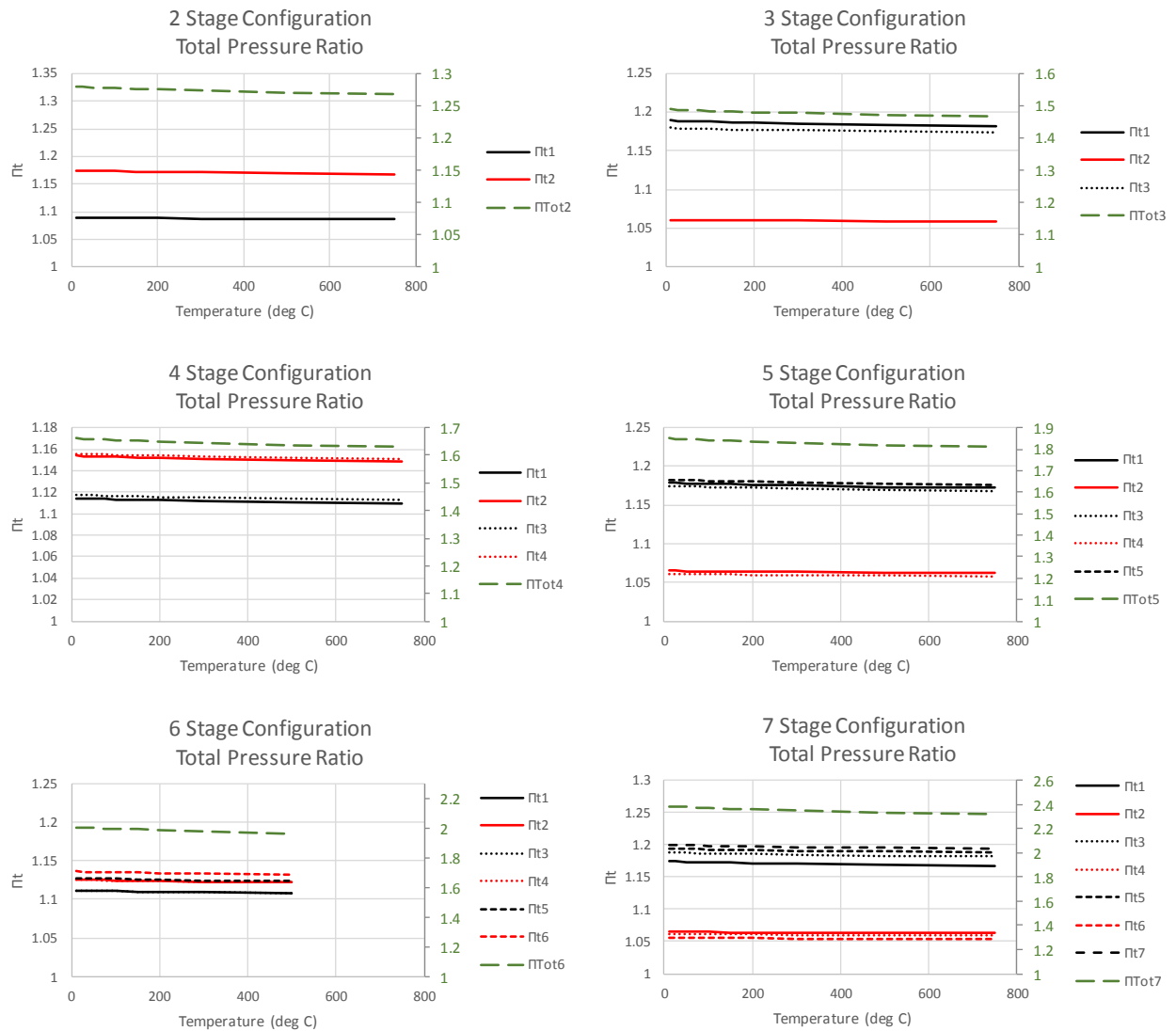


Figure A6.2.12. Total Pressure vs. Temperature at First Rotor Inlet (NCG Mix)

A6.3 Methane

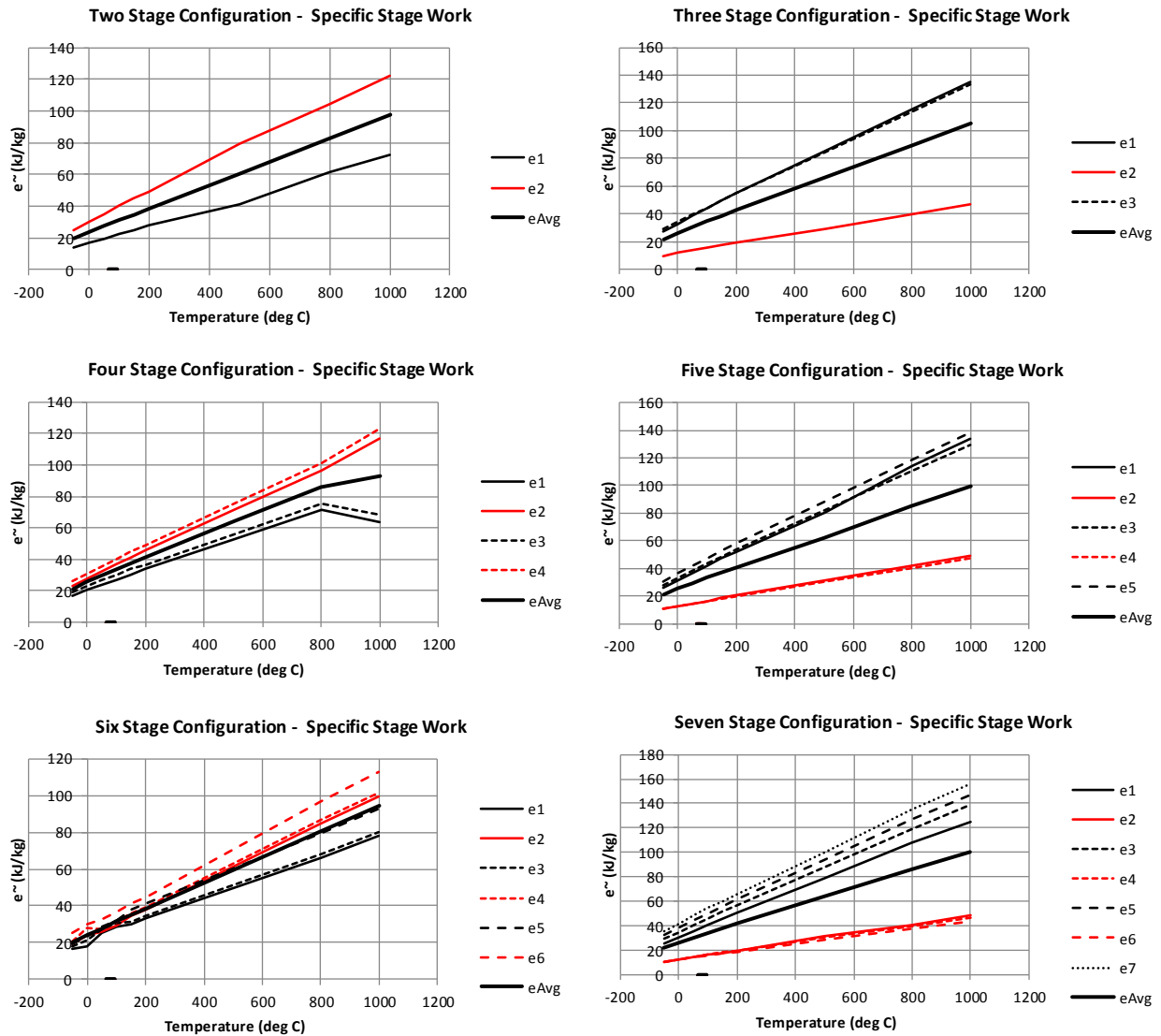
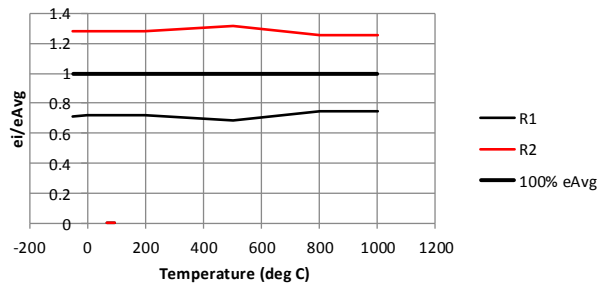
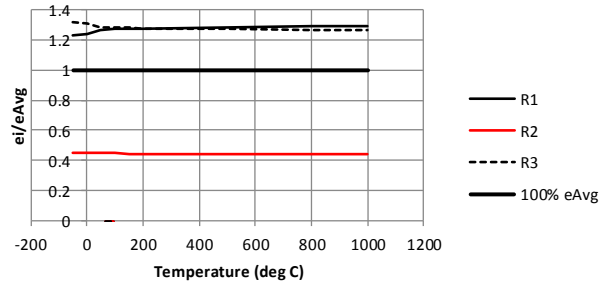


Figure A6.3.1. Specific Work vs. Temperature at First Rotor Inlet (CH_4)

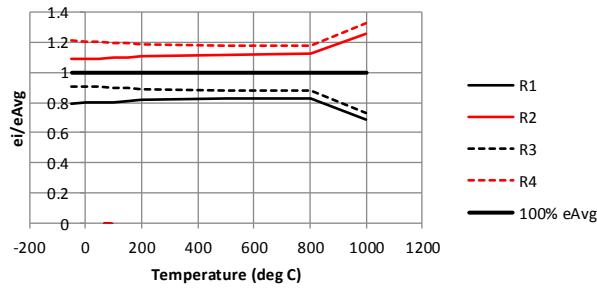
**Two Stage Configuration - Specific Stage Work
Normalized by Average Work Consumed by All Stages**



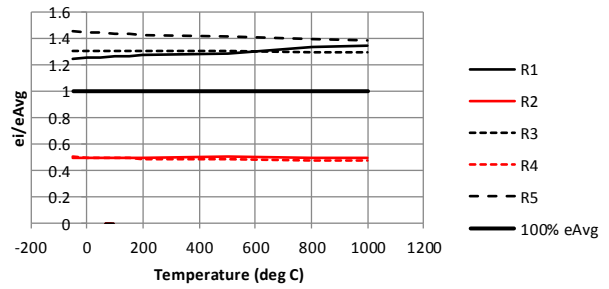
**Three Stage Configuration - Specific Stage Work
Normalized by Average Work Consumed by All Stages**



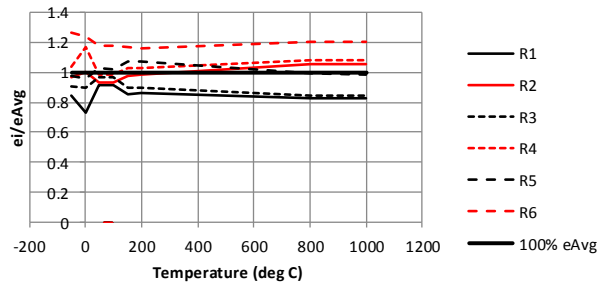
**Four Stage Configuration - Specific Stage Work
Normalized by Average Work Consumed by All Stages**



**Five Stage Configuration - Specific Stage Work
Normalized by Average Work Consumed by All Stages**



**Six Stage Configuration - Specific Stage Work
Normalized by Average Work Consumed by All Stages**



**Seven Stage Configuration - Specific Stage Work
Normalized by Average Work Consumed by All Stages**

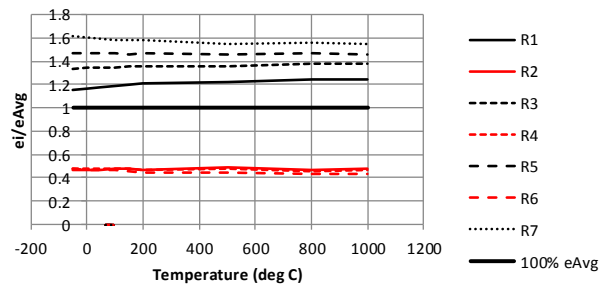


Figure A6.3.2. Normalized Specific Work vs. Temperature at First Rotor Inlet (CH_4)

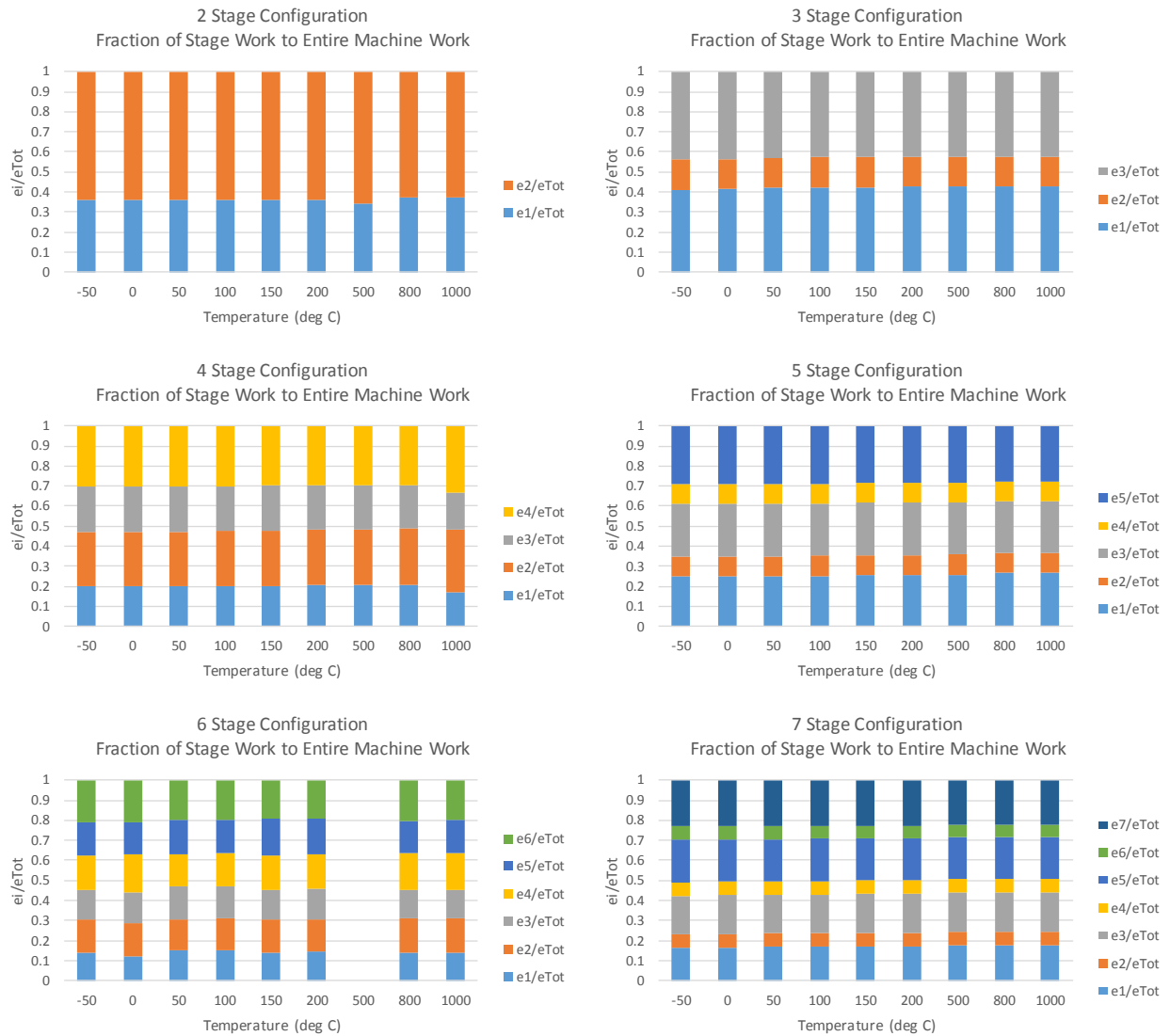


Figure A6.3.3. Rotor Work Fraction of Combined Work Total vs. Temperature at First Rotor Inlet (CH₄)

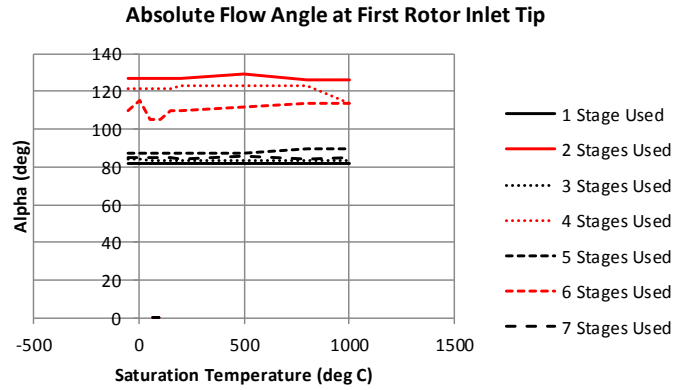


Figure A6.3.4. Absolute Flow Angle at First Rotor Inlet vs. Temperature at First Rotor Inlet (CH₄)

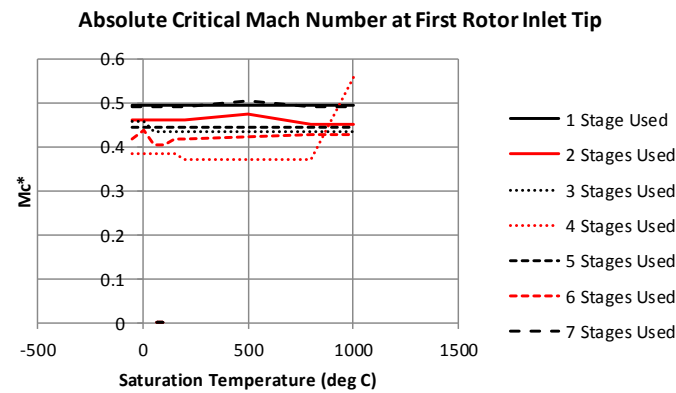


Figure A6.3.5. Absolute Critical Mach Number at First Rotor Inlet vs. Temperature at First Rotor Inlet (CH₄)

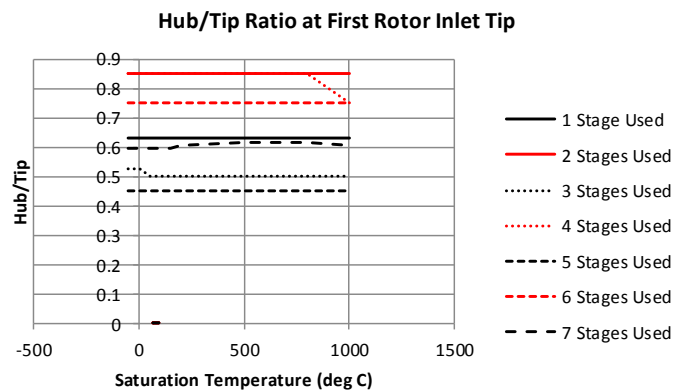
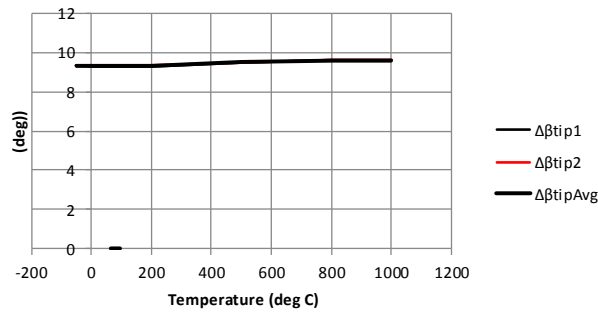
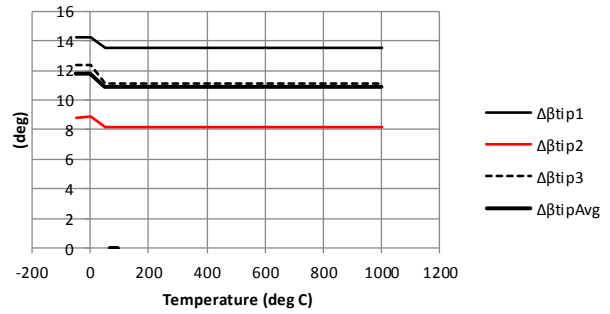


Figure A6.3.6. Hub/Tip Ratio at First Rotor Inlet vs. Temperature at First Rotor Inlet (CH₄)

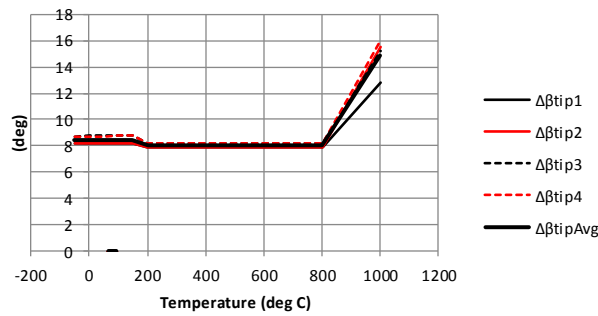
Two Stage Configuration - Blade Turning at Tip



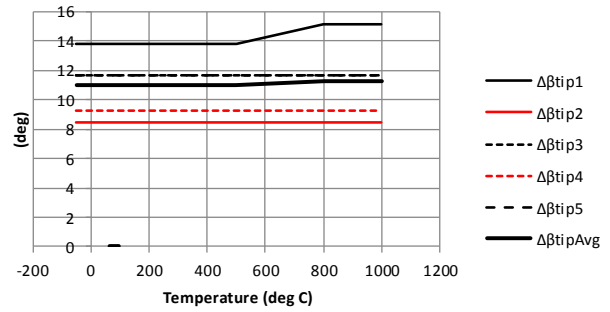
Three Stage Configuration - Blade Turning at Tip



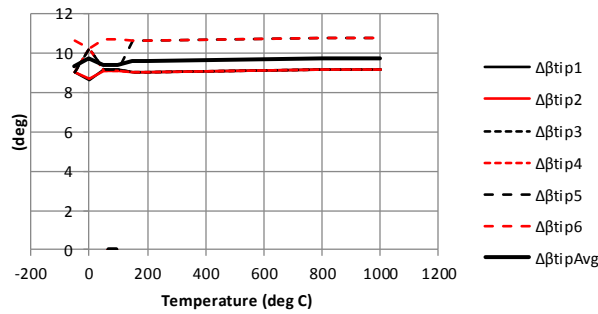
Four Stage Configuration - Blade Turning at Tip



Five Stage Configuration - Blade Turning at Tip



Six Stage Configuration - Blade Turning at Tip



Seven Stage Configuration - Blade Turning at Tip

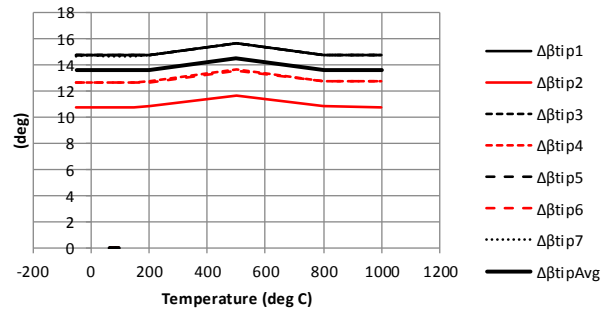


Figure A6.3.7. Blade Turning vs. Temperature at First Rotor Inlet (CH_4)

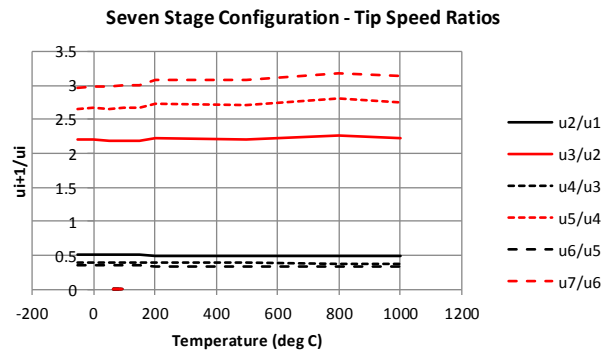
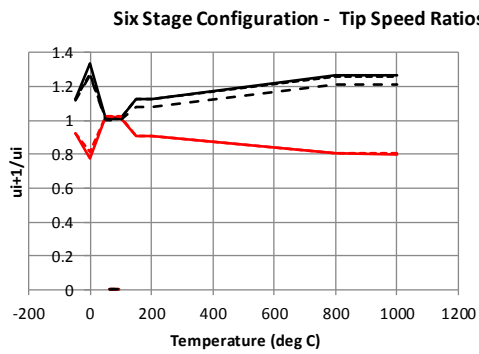
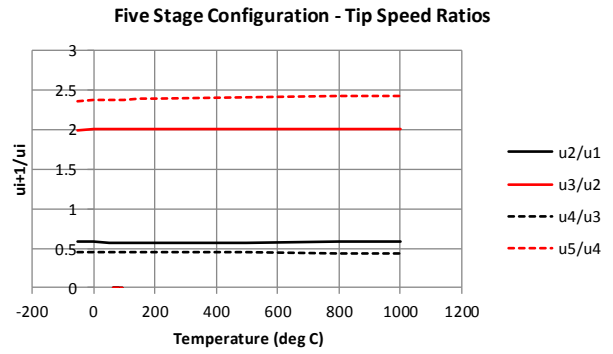
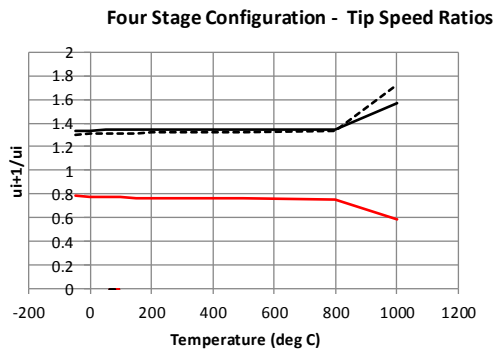
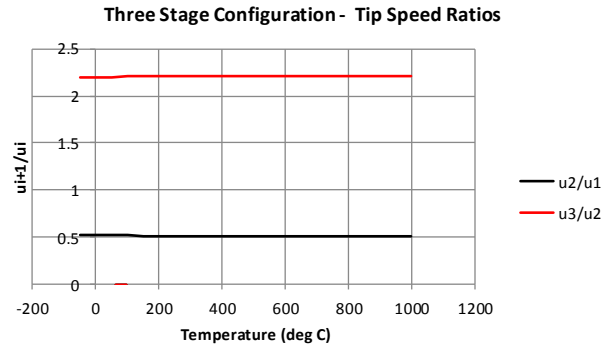
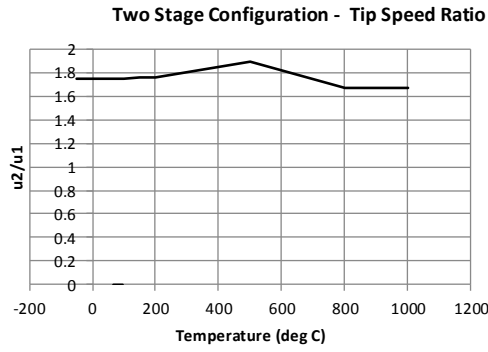


Figure A6.3.8. Tip Speed Ratio vs. Temperature at First Rotor Inlet (CH_4)

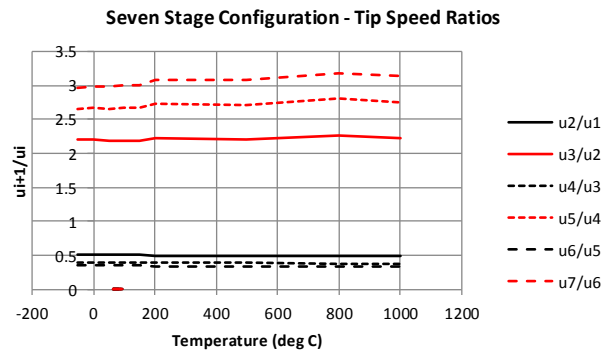
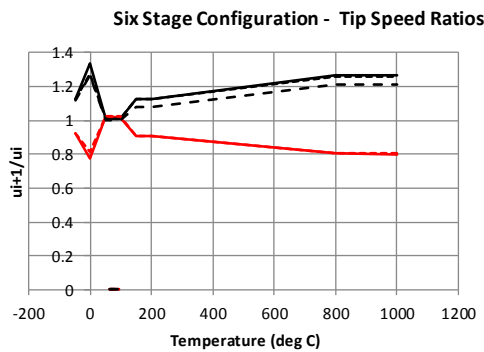
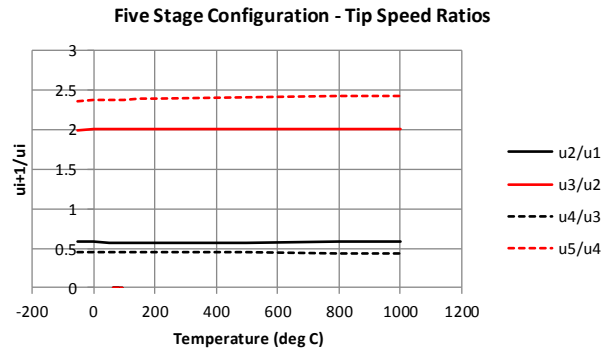
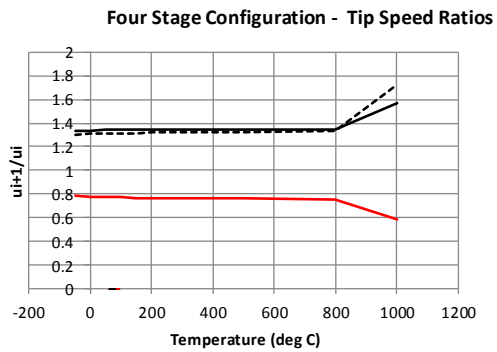
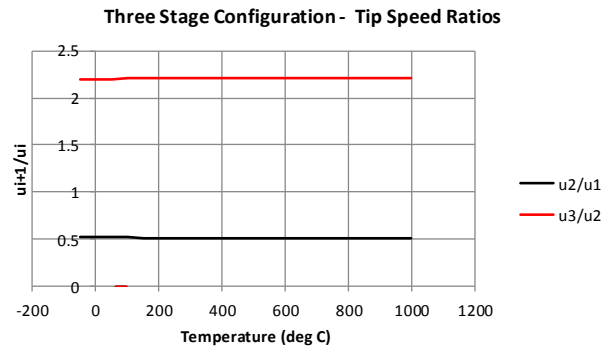
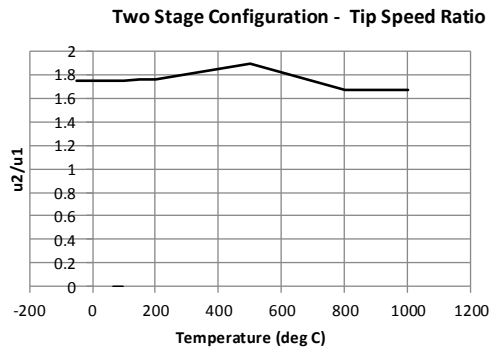


Figure A6.3.9. Flow Coefficient vs. Temperature at First Rotor Inlet (CH₄)

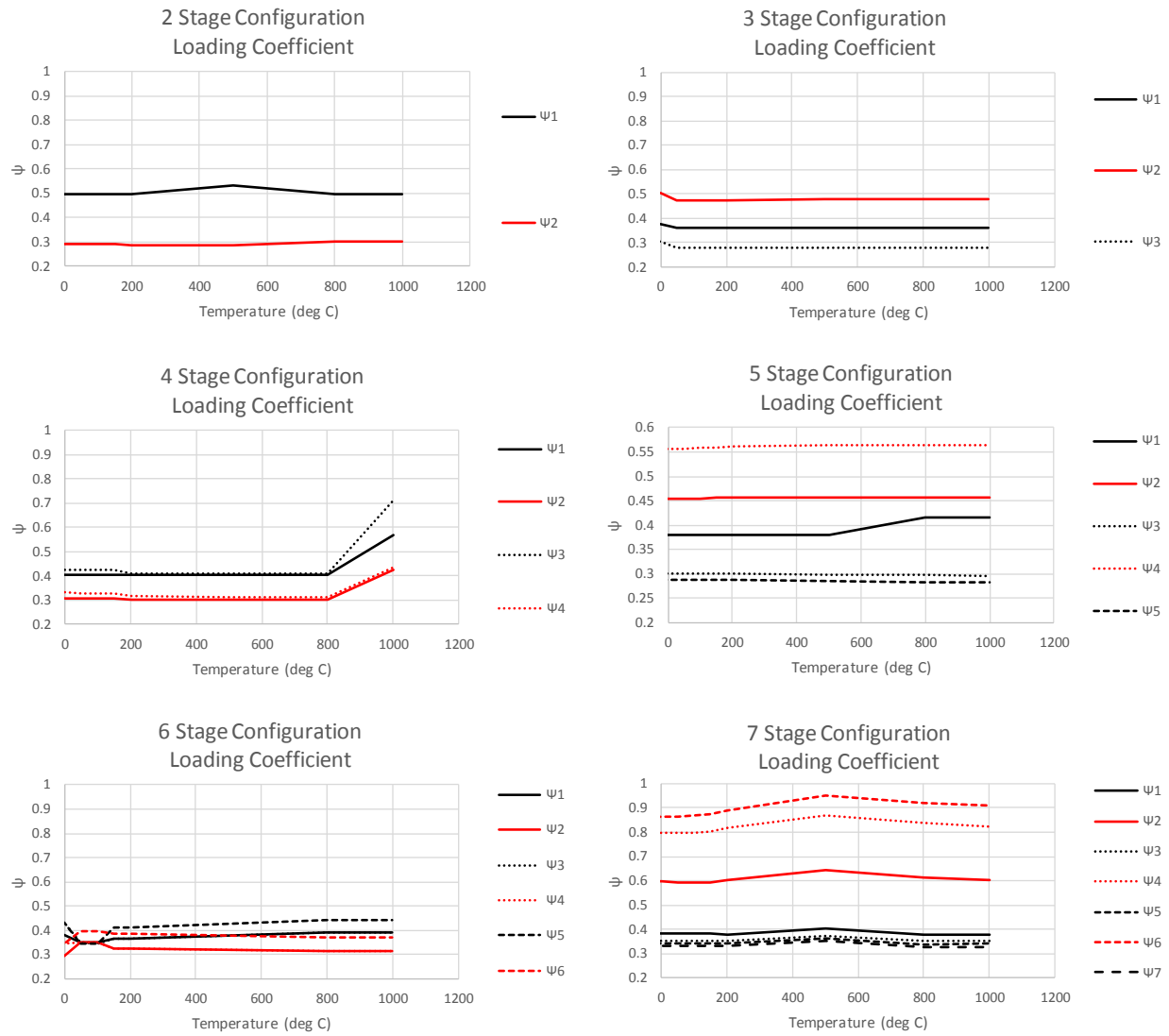


Figure A6.3.10. Blade Loading Coefficient vs. Temperature at First Rotor Inlet (CH₄)

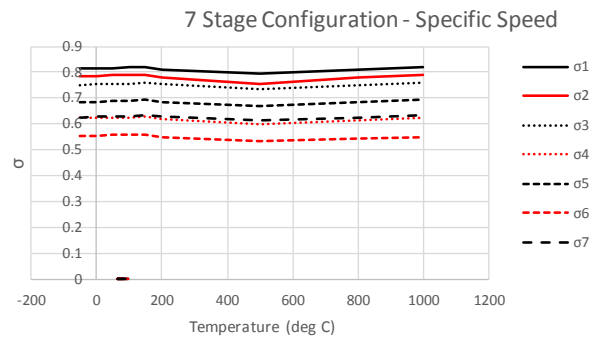
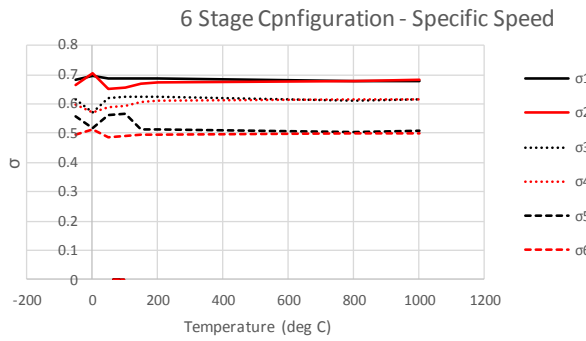
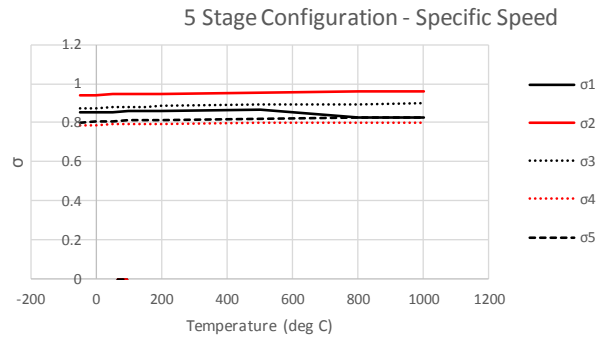
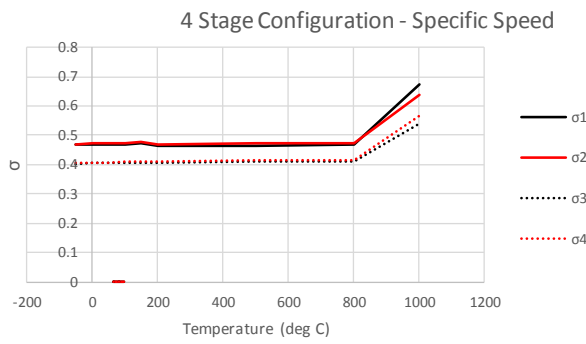
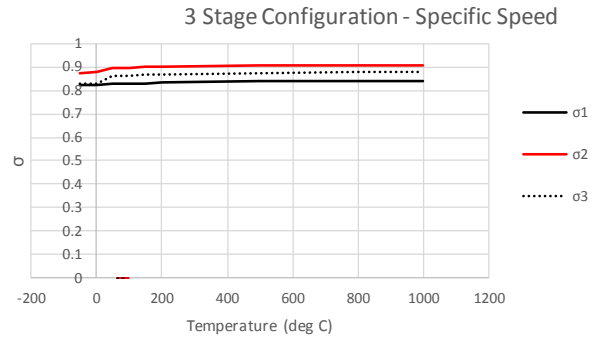
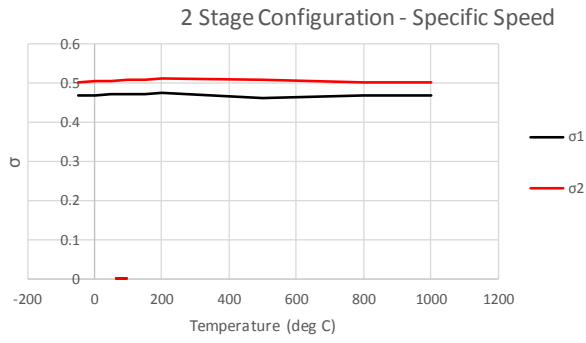


Figure A6.3.11. Specific Speed vs. Temperature at First Rotor Inlet (CH_4)

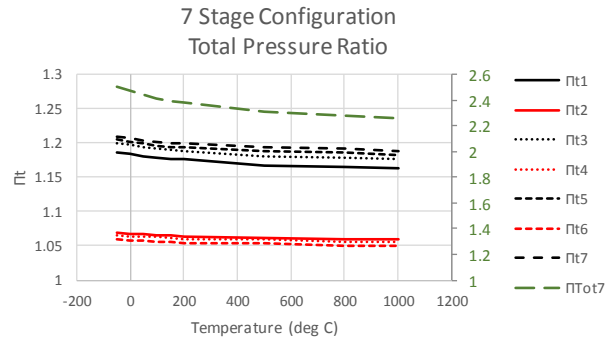
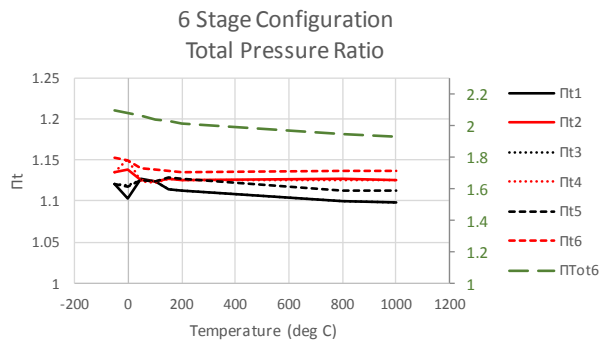
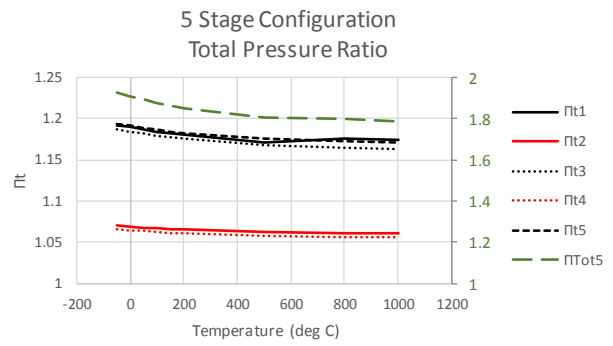
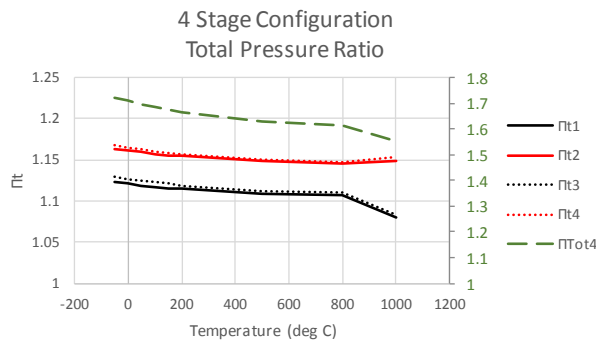
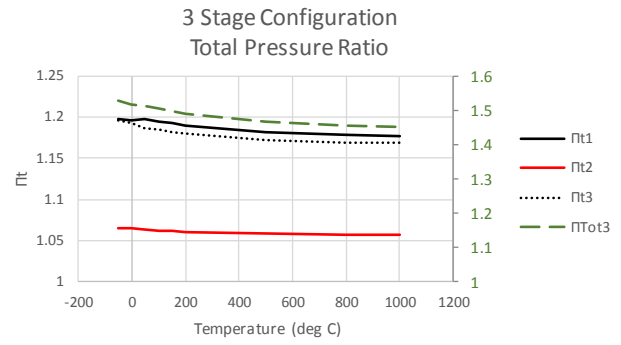
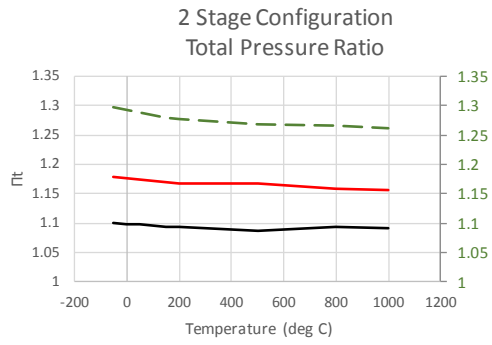


Figure A6.3.12. Total Pressure Ratio vs. Temperature at First Rotor Inlet (CH₄)

A6.4 Carbon Dioxide

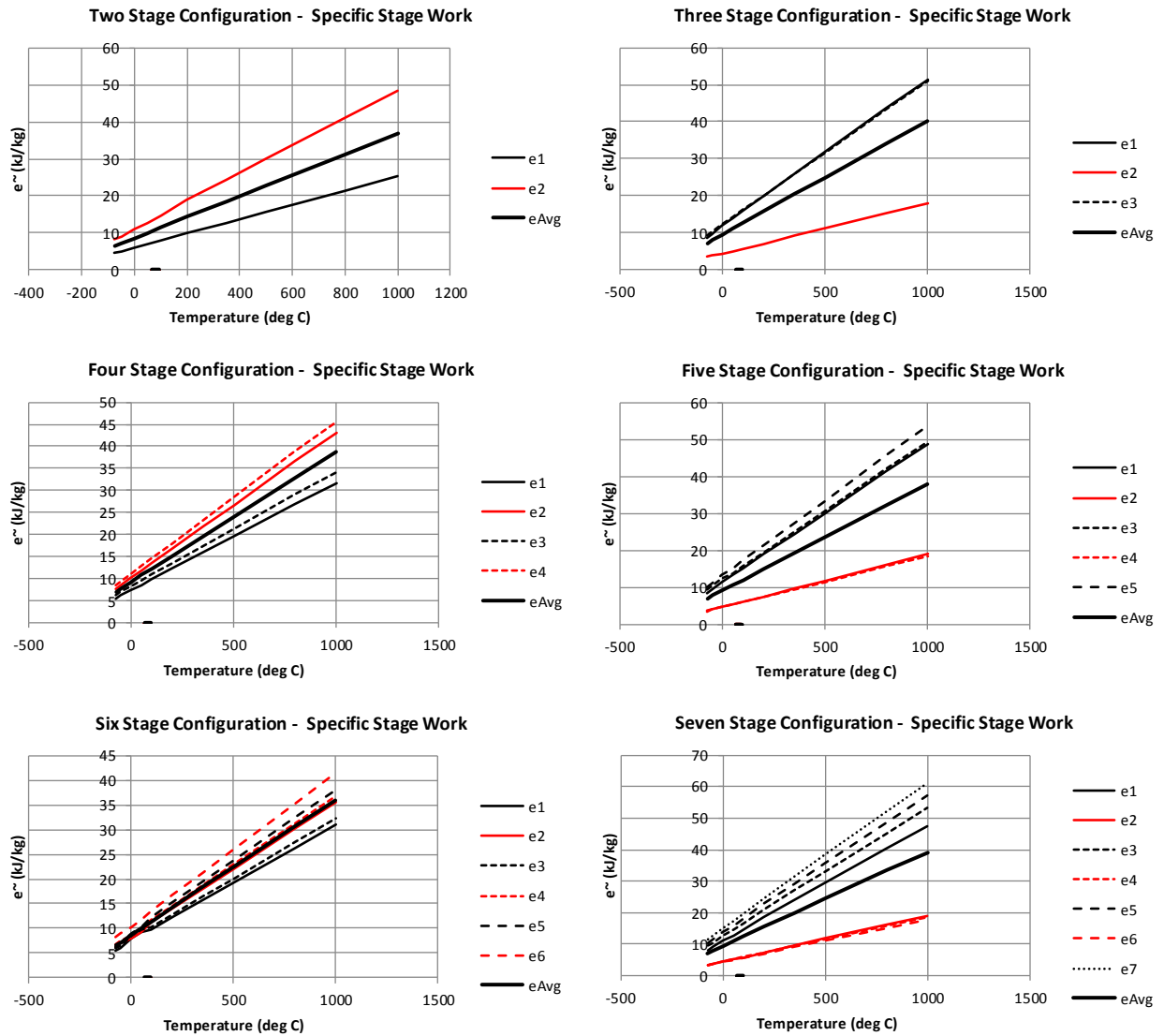


Figure A6.4.1. Specific Work vs. Temperature at First Rotor Inlet (CO₂)

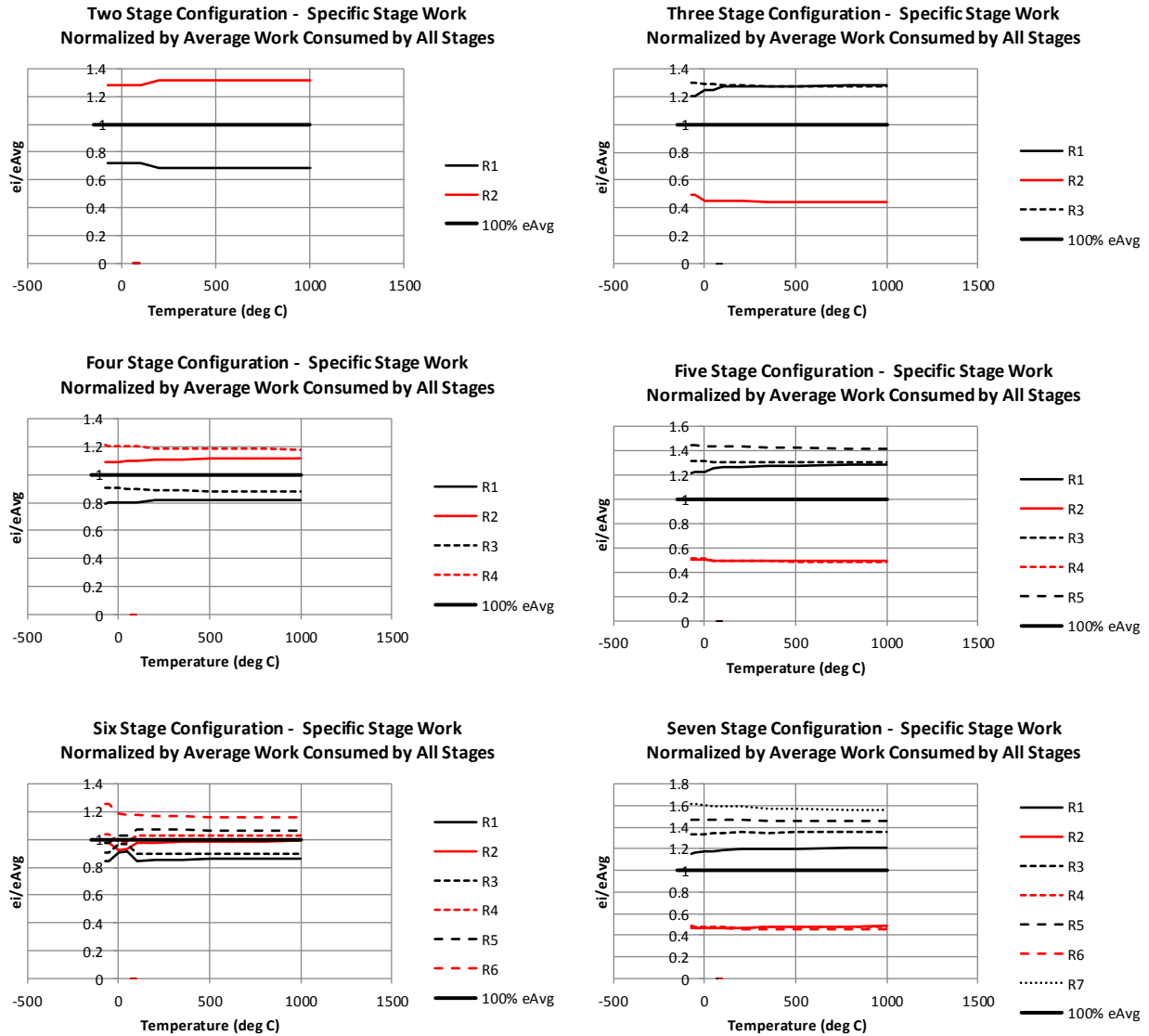


Figure A6.4.2. Normalized Specific Work vs. Temperature at First Rotor Inlet (CO₂)

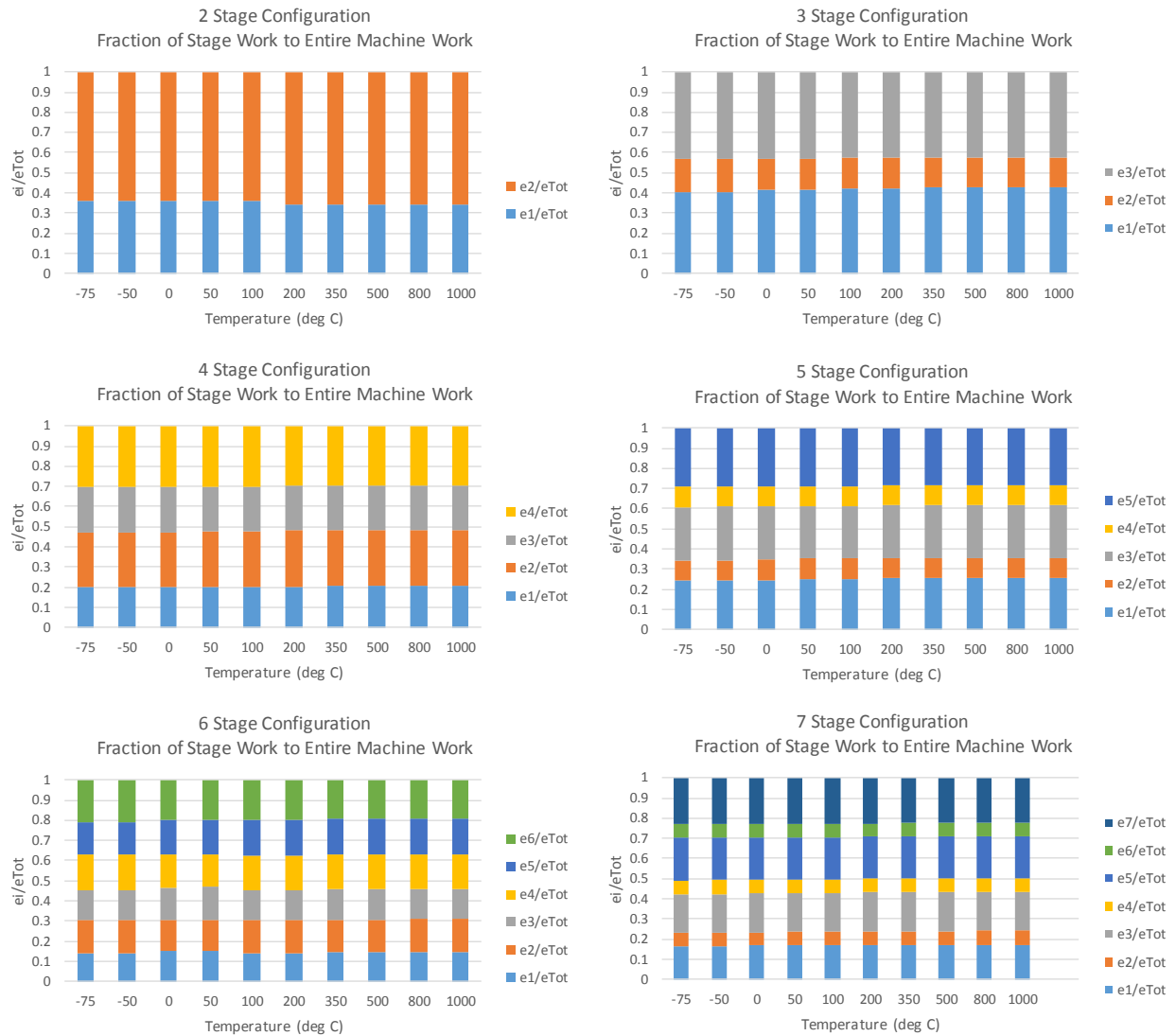


Figure A6.4.3. Rotor Work Fraction of Combined Work Total vs. Temperature at First Rotor Inlet (CO₂)

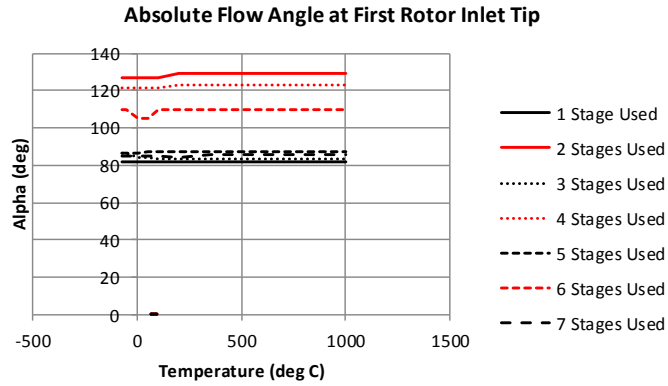


Figure A6.4.4. Absolute Flow Angle at First Rotor Inlet vs. Temperature at First Rotor Inlet (CO₂)

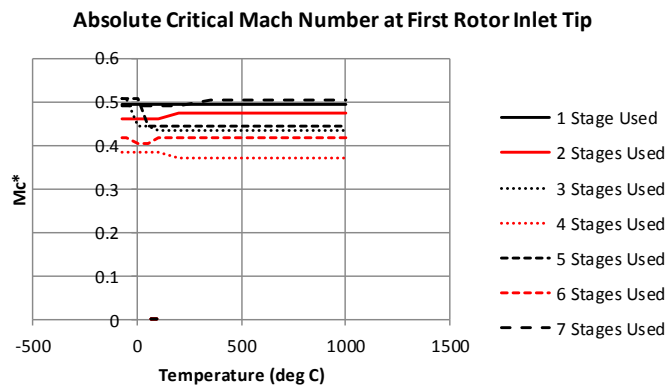


Figure A6.4.5. Absolute Critical Mach Number at First Rotor Inlet vs. Temperature at First Rotor Inlet (CO₂)

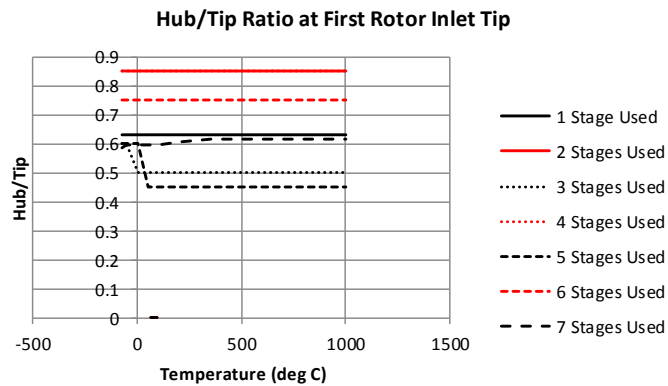


Figure A6.4.6. Hub/Tip Ratio at First Rotor Inlet vs. Temperature at First Rotor Inlet (CO₂)

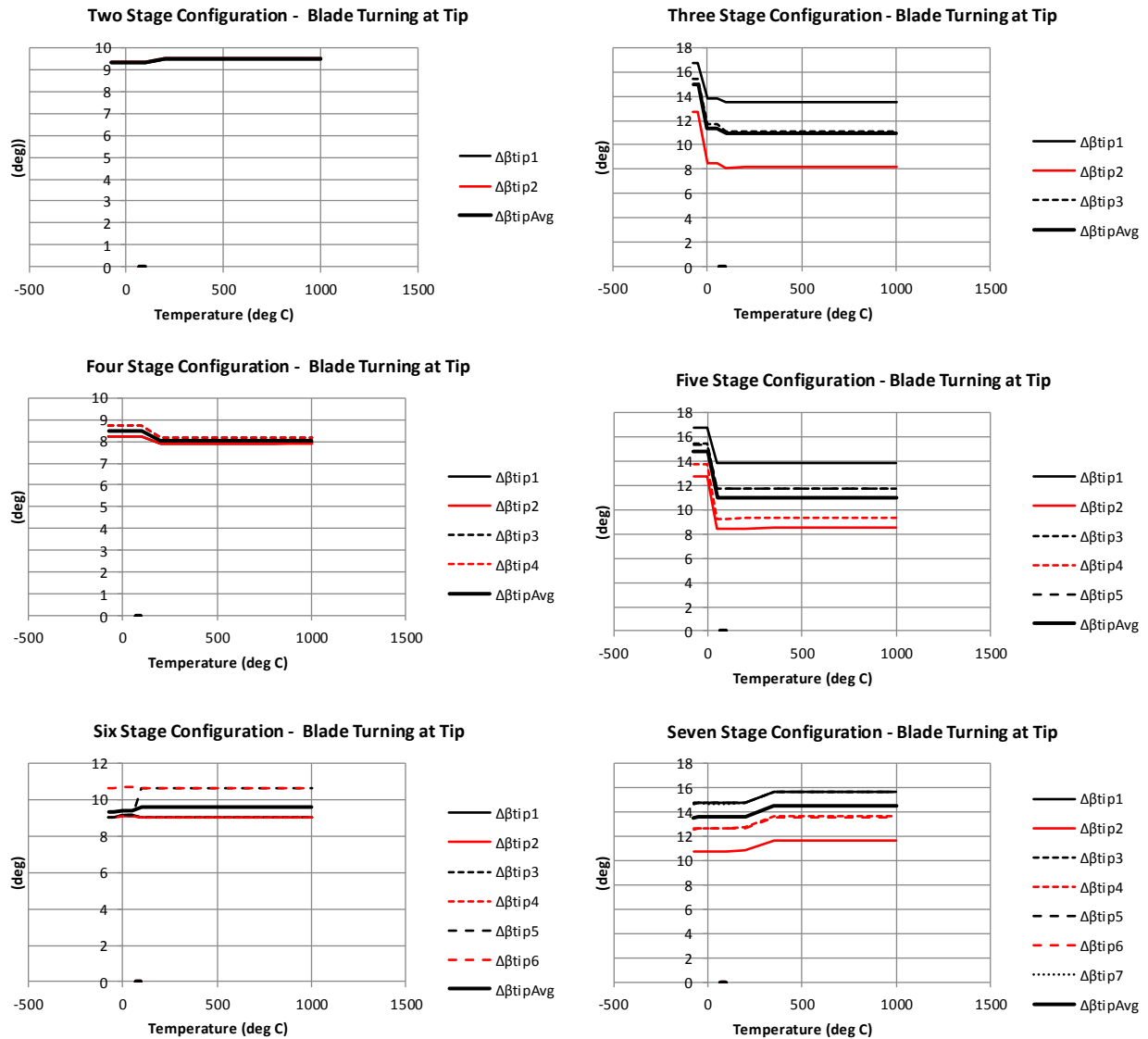


Figure A6.4.7. Blade Turning vs. Temperature at First Rotor Inlet (CO₂)

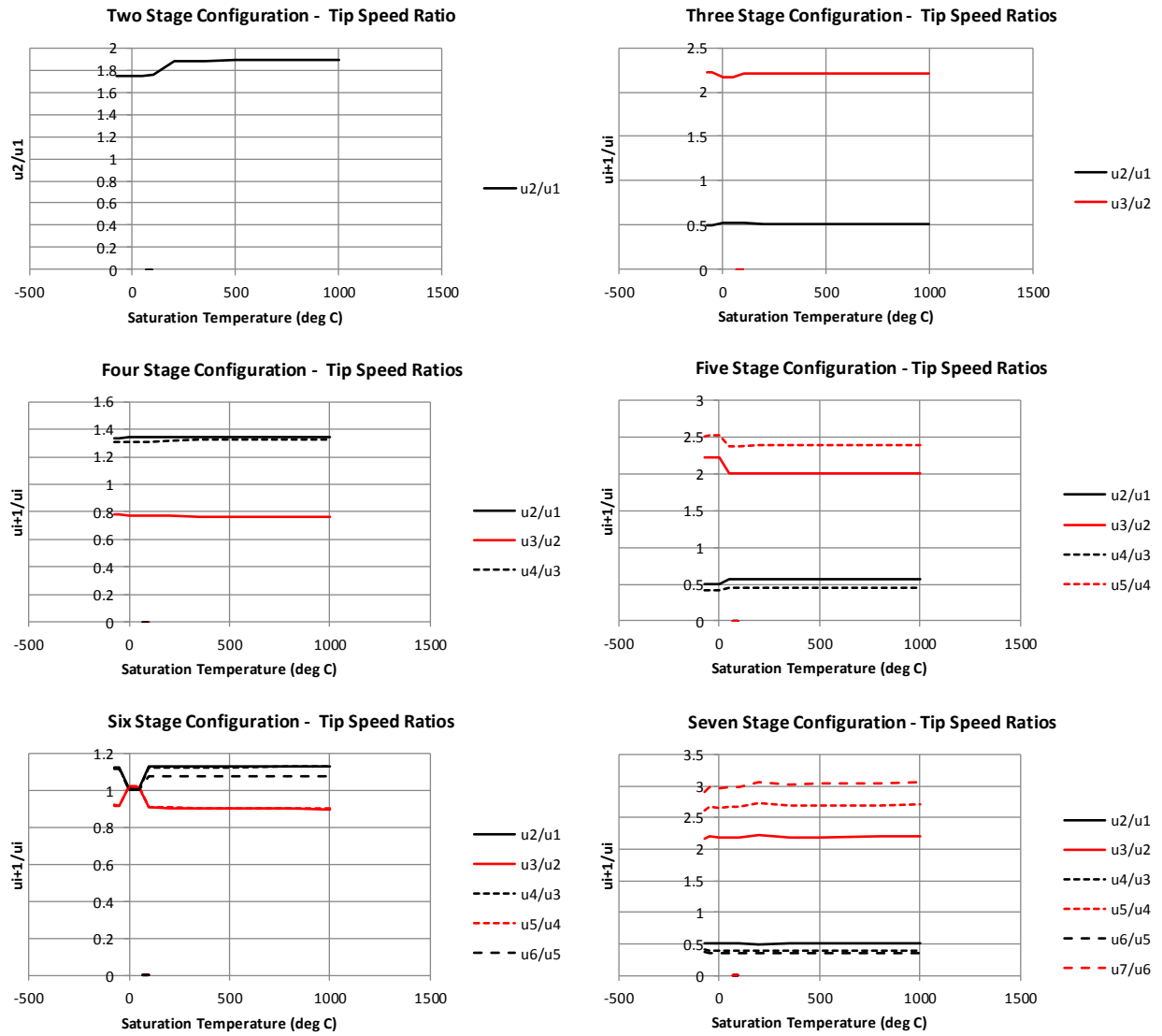


Figure A6.4.8. Tip Speed Ratio vs. Temperature at First Rotor Inlet (CO₂)

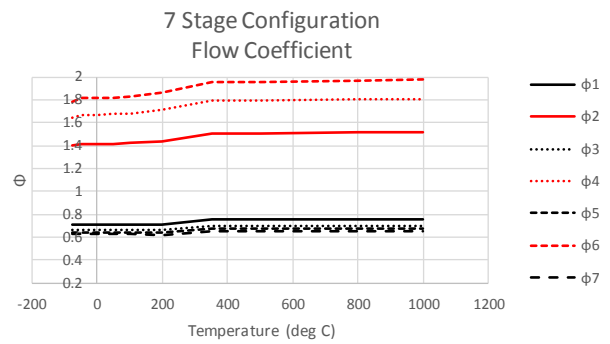
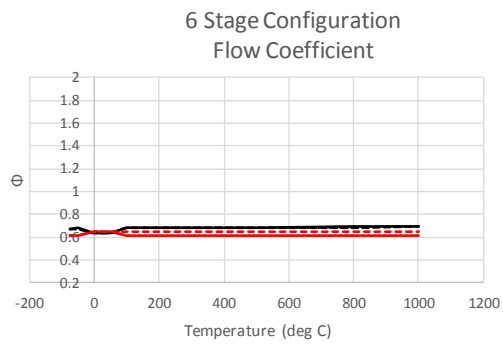
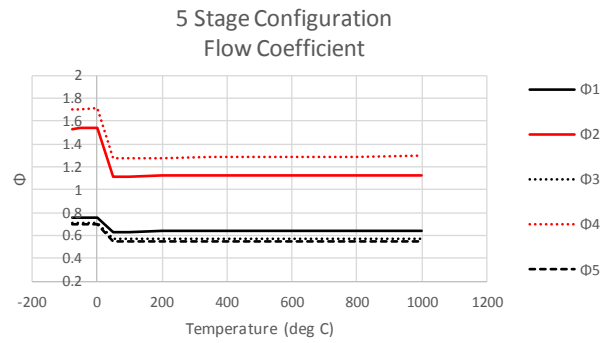
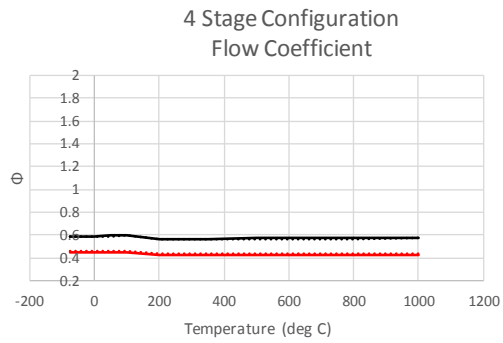
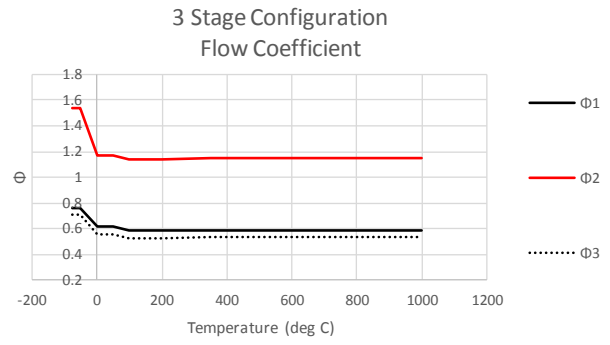
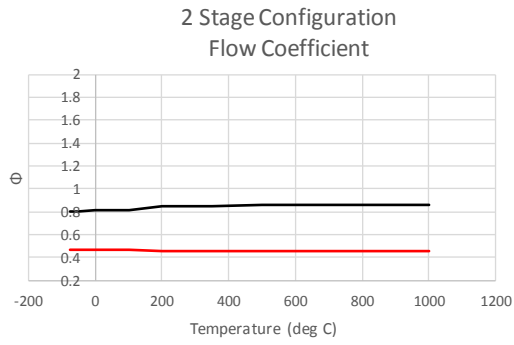


Figure A6.4.9. Flow Coefficient vs. Temperature at First Rotor Inlet (CO₂)

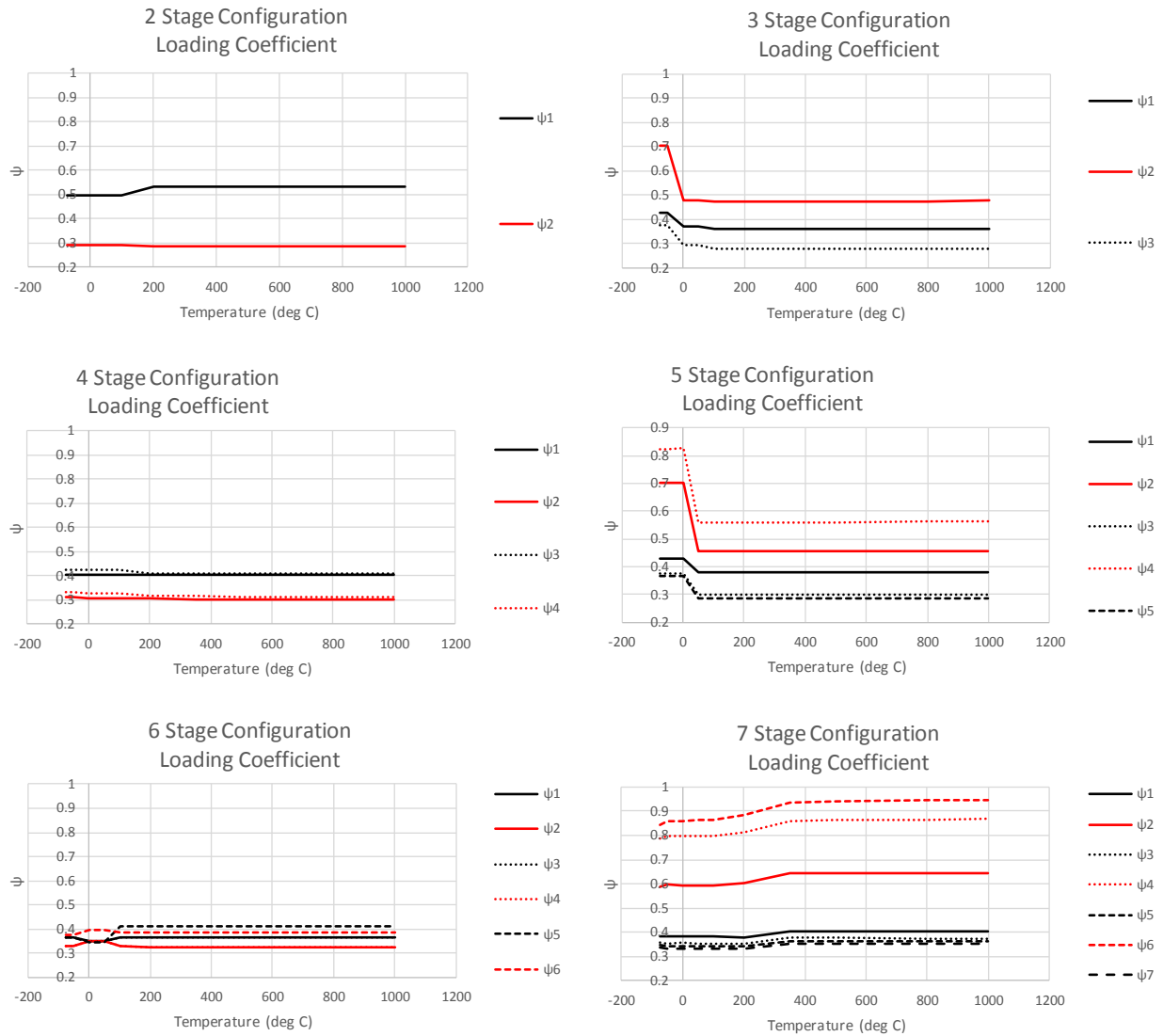


Figure A6.4.10. Blade Loading Coefficient vs. Temperature at First Rotor Inlet (CO₂)

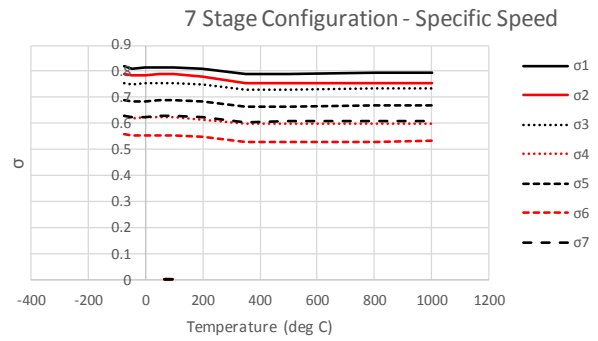
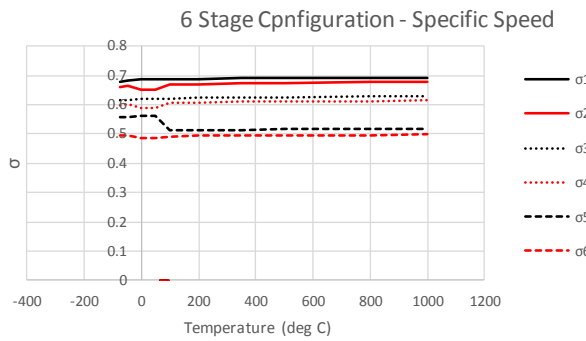
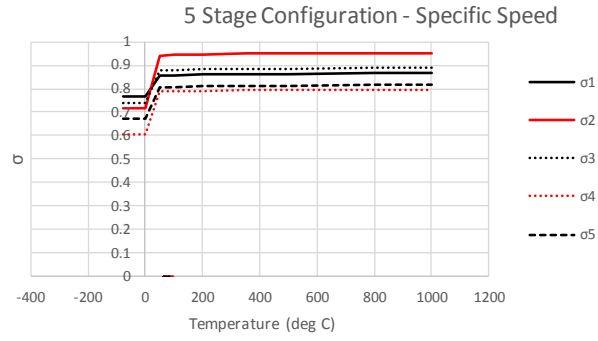
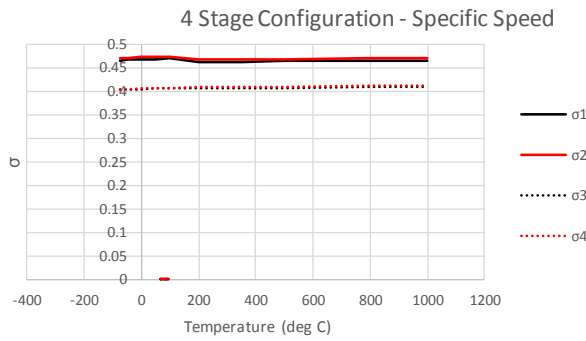
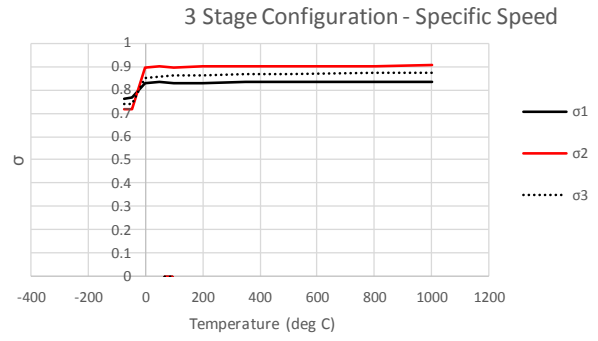
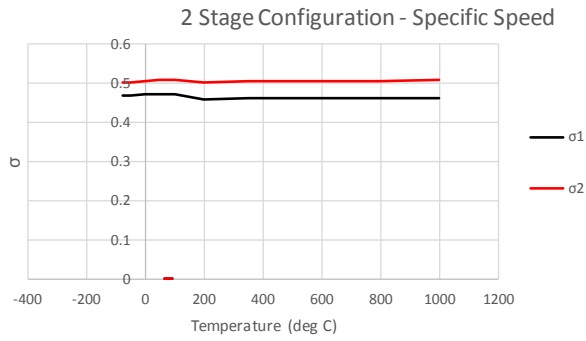


Figure A6.4.11. Specific Speed vs. Temperature at First Rotor Inlet (CO₂)

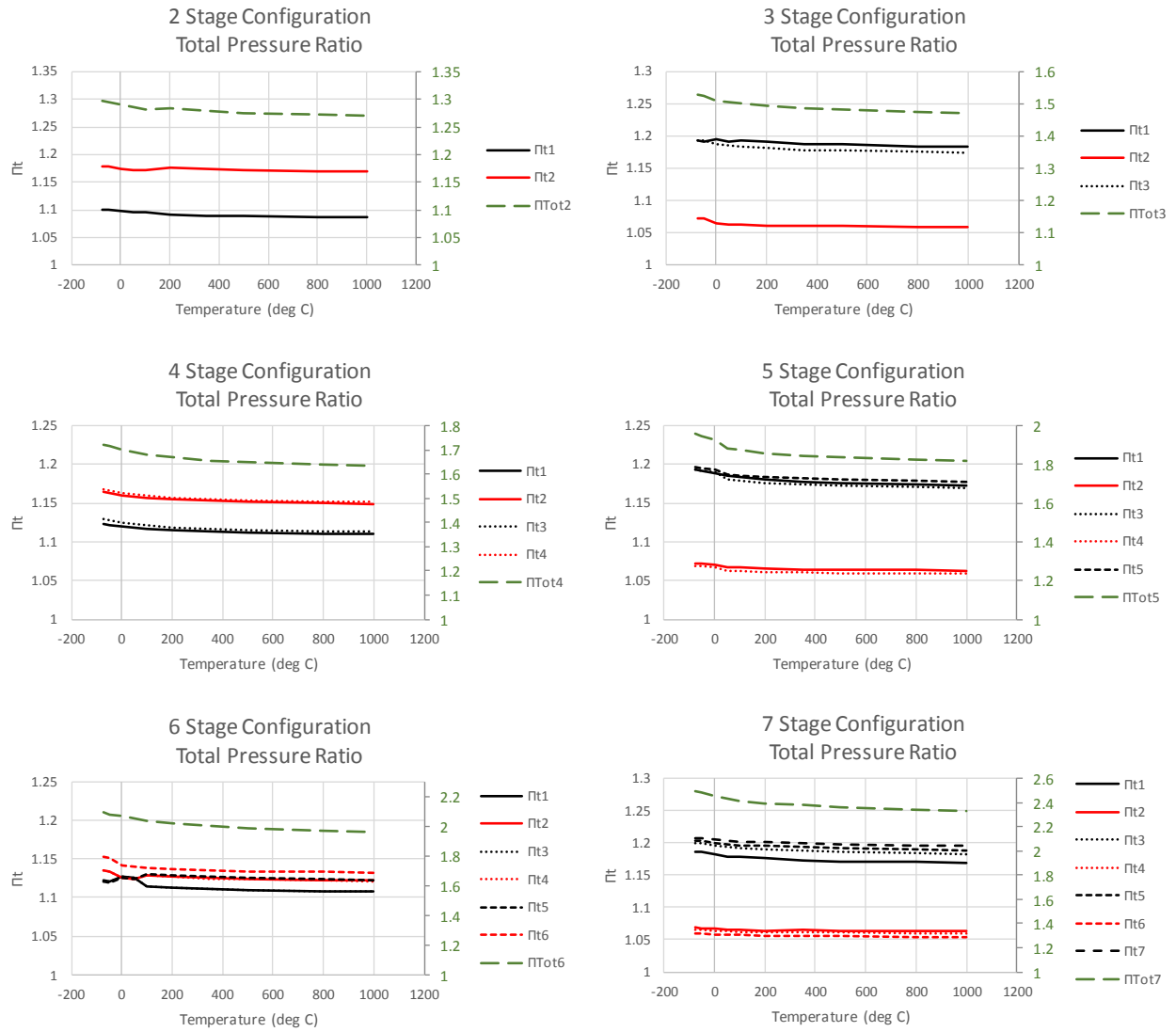


Figure A6.4.12. Total Pressure Ratio vs. Temperature at First Rotor Inlet (CO₂)

A6.5 Hydrogen

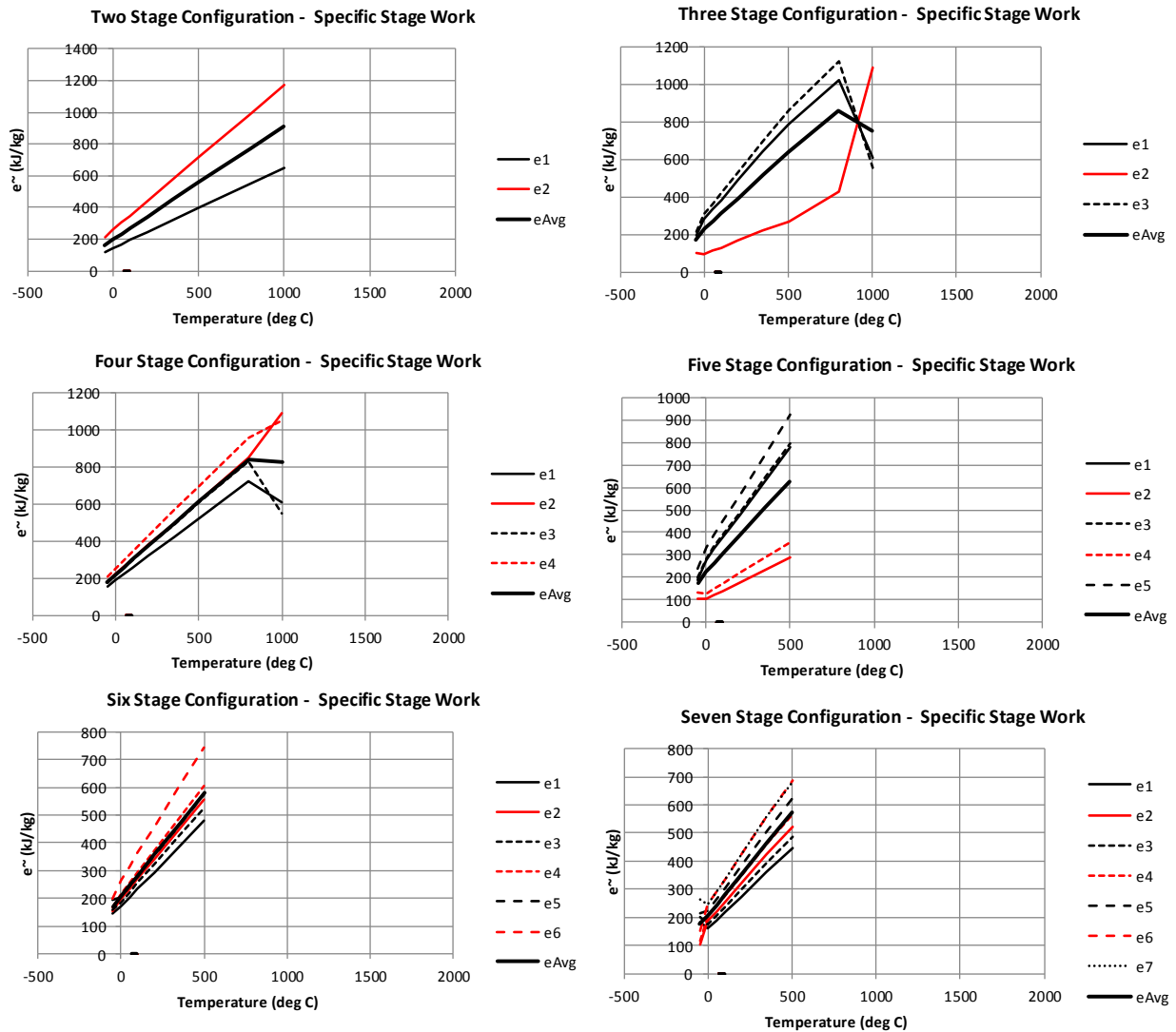


Figure A6.5.1. Specific Work vs. Temperature at First Rotor Inlet (H_2)

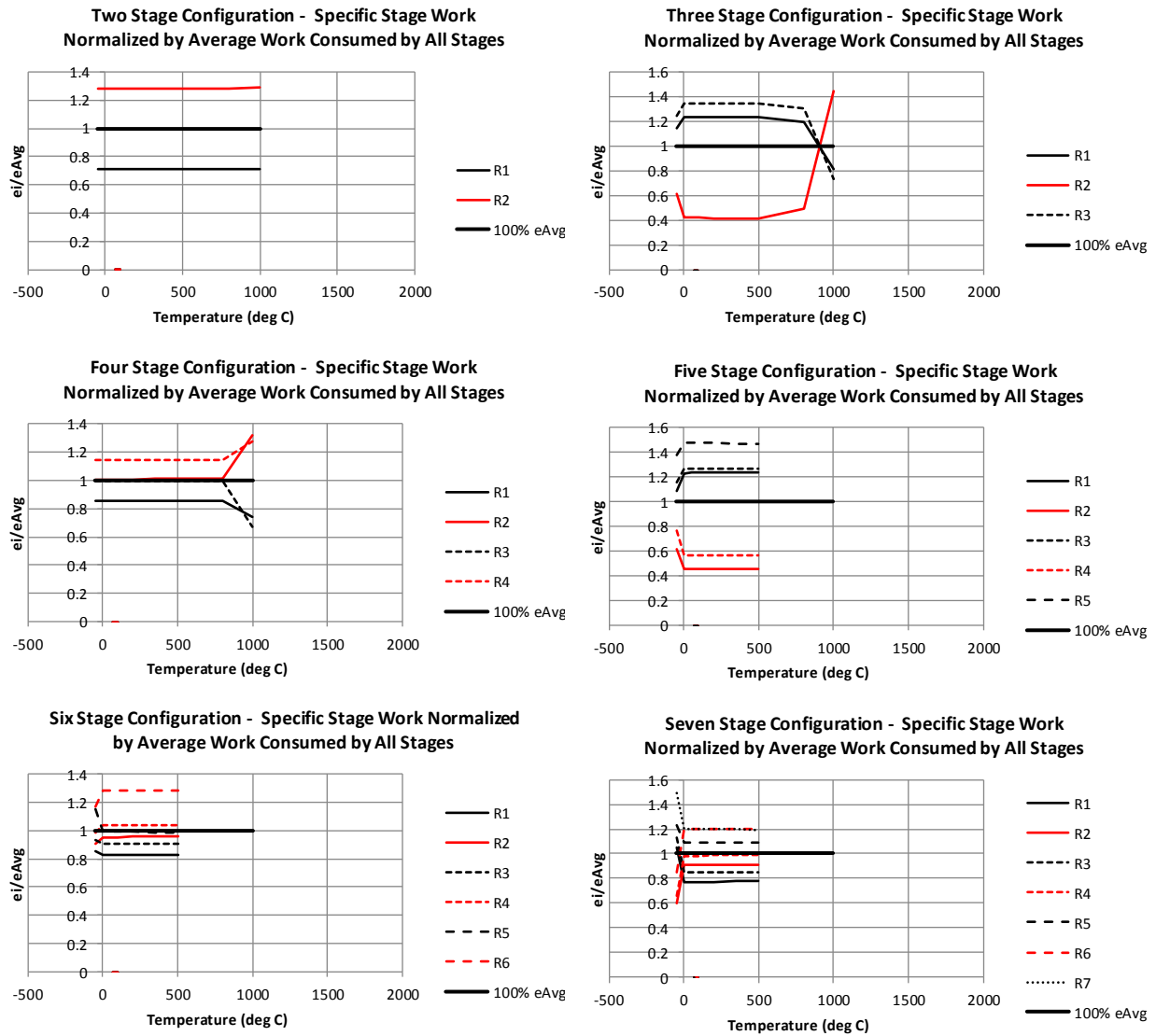


Figure A6.5.2. Normalized Specific Work vs. Temperature at First Rotor Inlet (H_2)

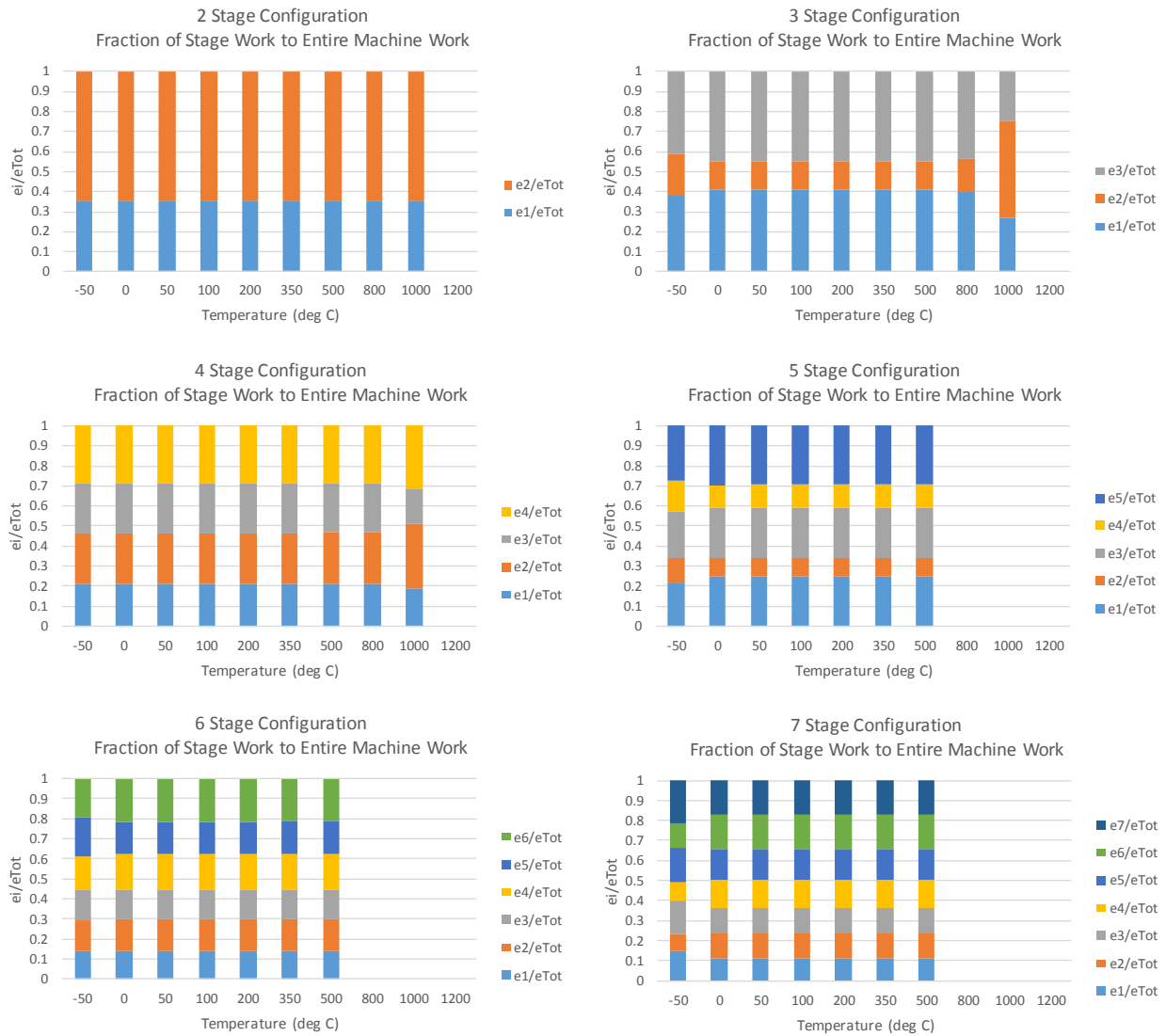


Figure A6.5.3. Rotor Work Fraction of Combined Work Total vs. Temperature at First Rotor Inlet (H_2)

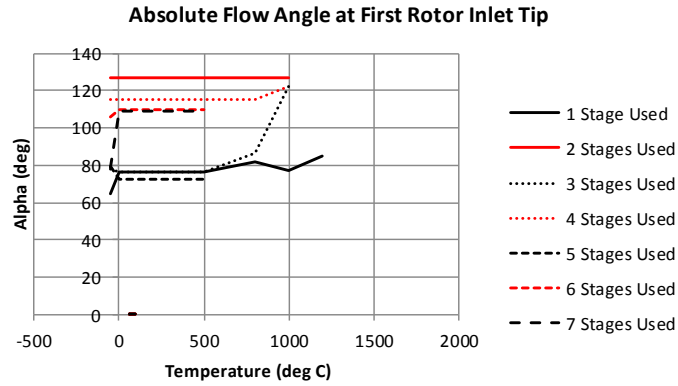


Figure A6.5.4. Absolute Flow Angle at First Rotor Inlet vs. Temperature at First Rotor Inlet (H_2)

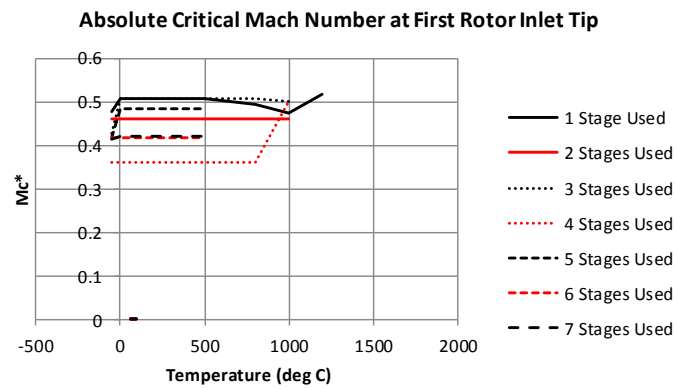


Figure A6.5.5. Absolute Critical Mach Number at First Rotor Inlet vs. Temperature at First Rotor Inlet (H_2)

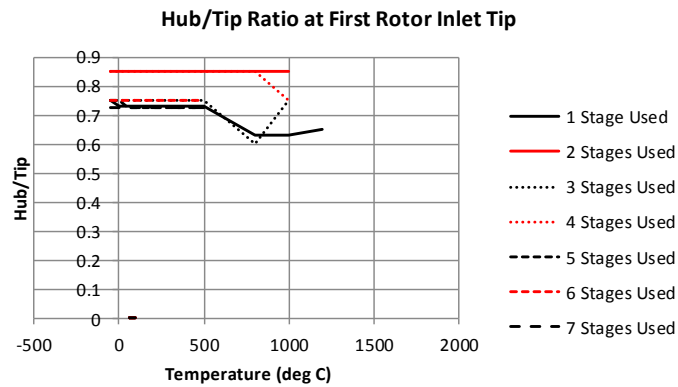
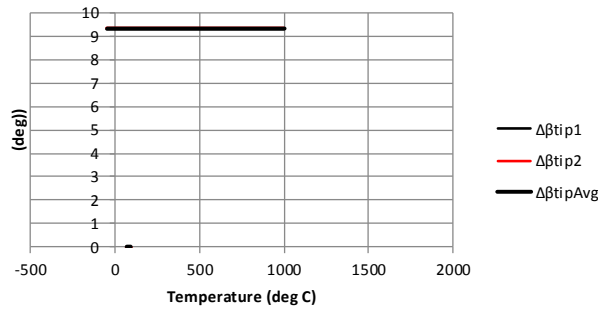
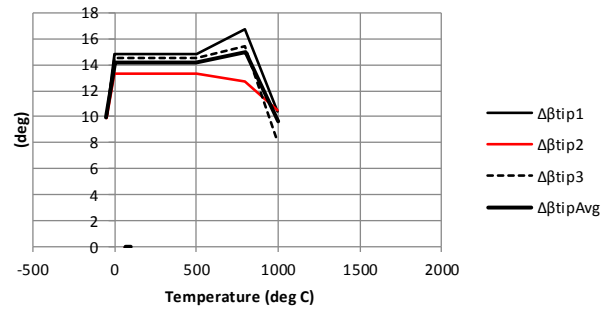


Figure A6.5.6. Hub/Tip Ratio at First Rotor Inlet vs. Temperature at First Rotor Inlet (H_2)

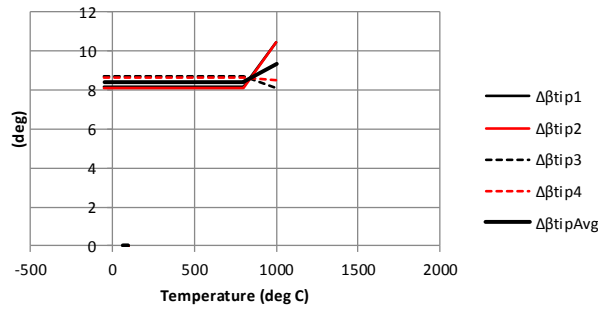
Two Stage Configuration - Blade Turning at Tip



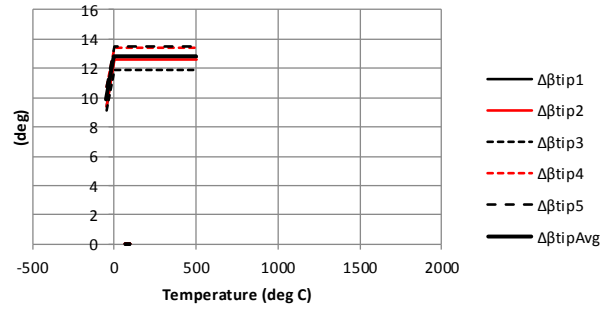
Three Stage Configuration - Blade Turning at Tip



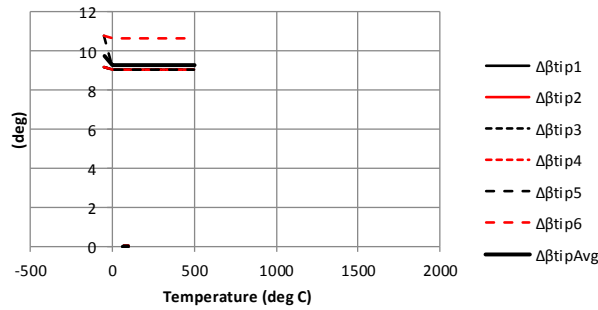
Four Stage Configuration - Blade Turning at Tip



Five Stage Configuration - Blade Turning at Tip



Six Stage Configuration - Blade Turning at Tip



Seven Stage Configuration - Blade Turning at Tip

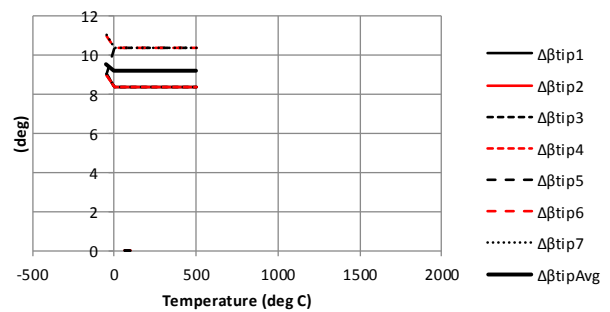


Figure A6.5.7. Blade Turning vs. Temperature at First Rotor Inlet (H_2)

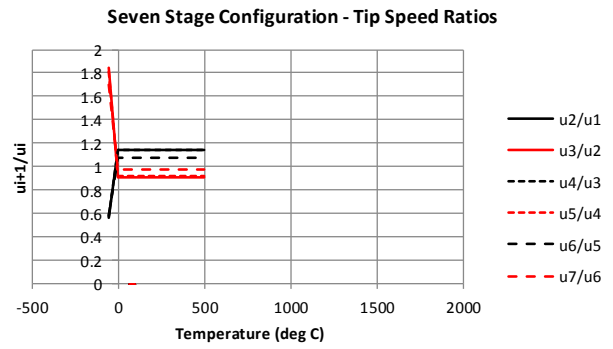
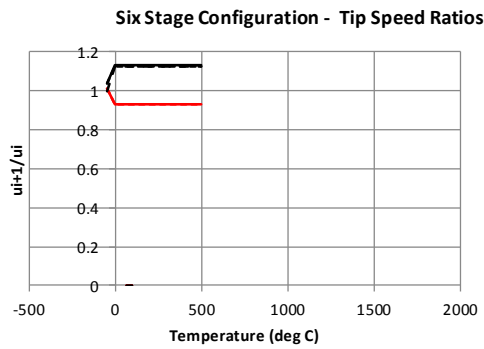
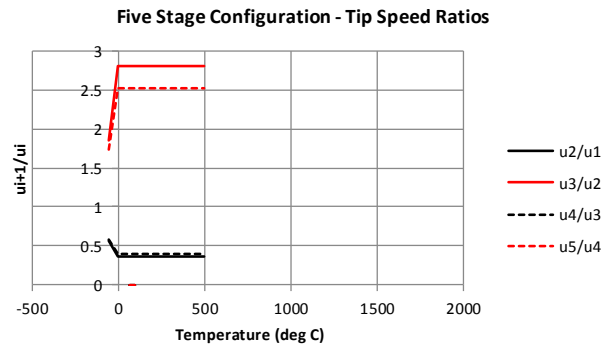
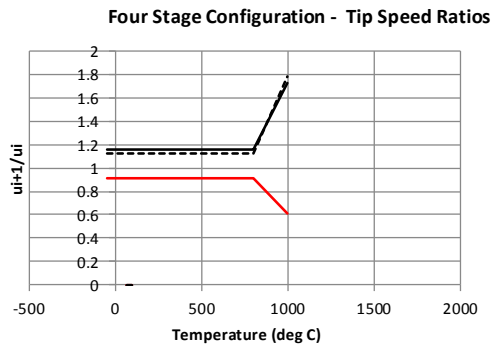
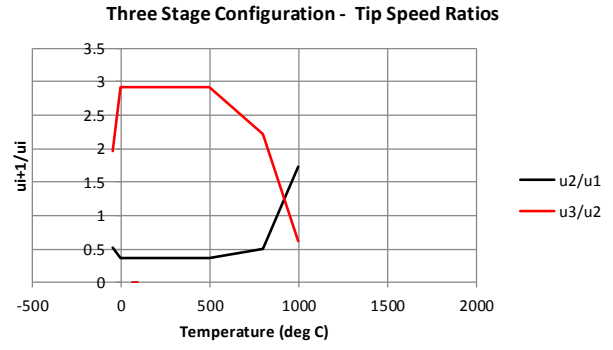
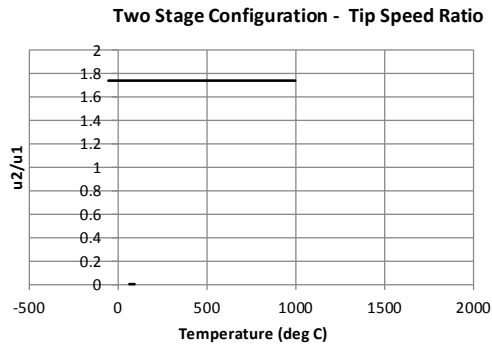


Figure A6.5.8. Tip Speed Ratio vs. Temperature at First Rotor Inlet (H_2)

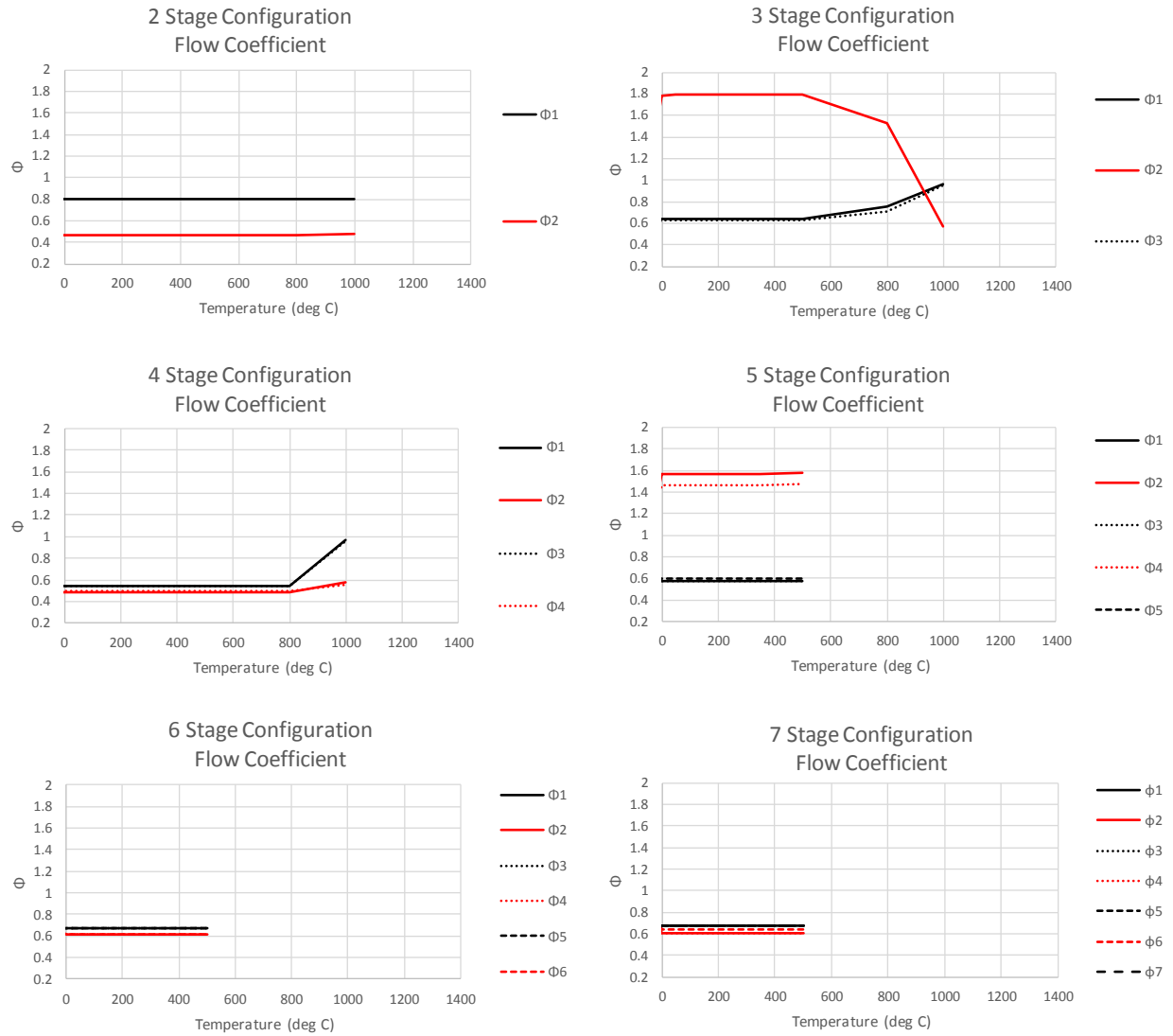


Figure A6.5.9. Flow Coefficient vs. Temperature at First Rotor Inlet (H_2)

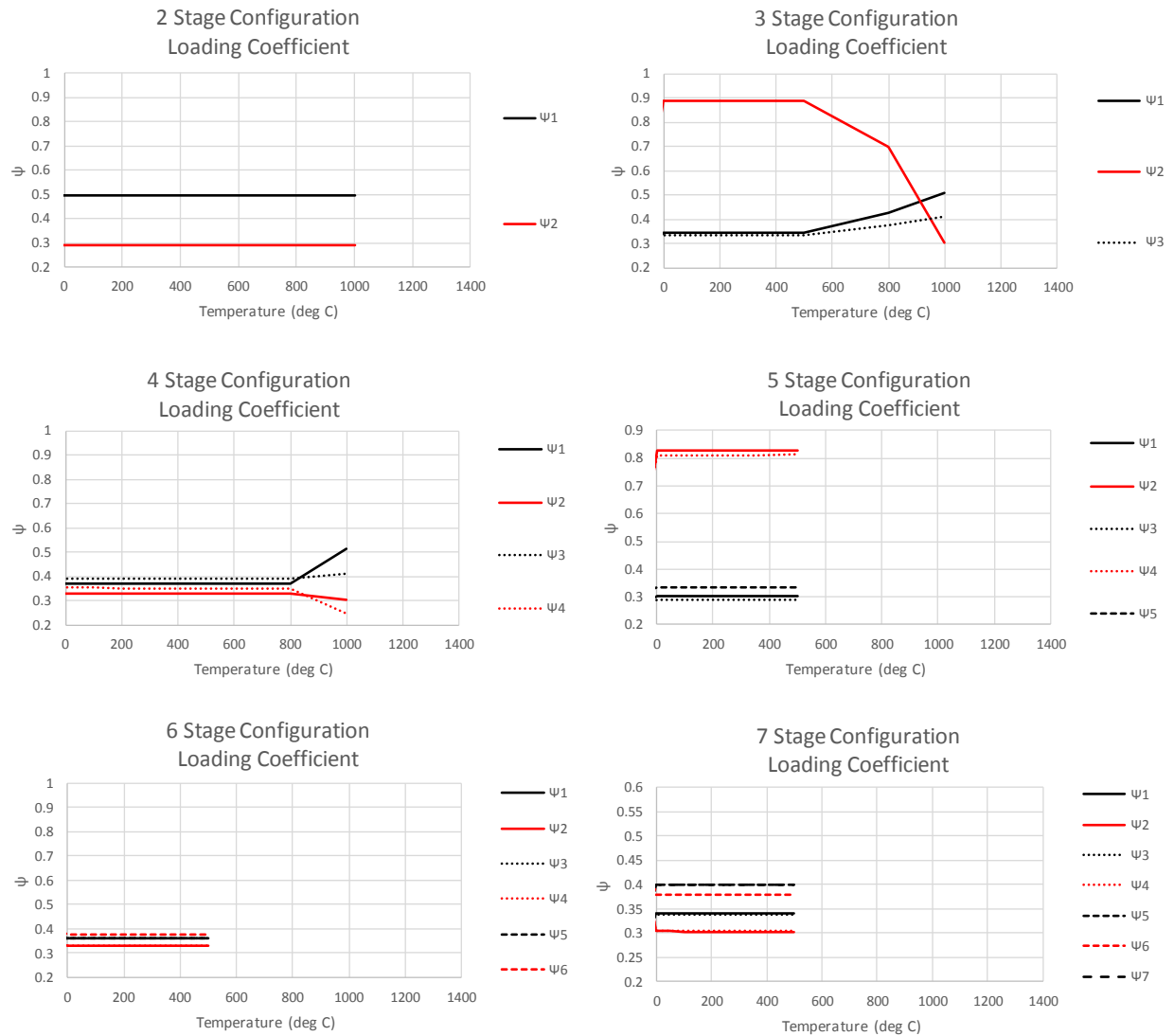


Figure A6.5.10. Blade Loading Coefficient vs. Temperature at First Rotor Inlet (H₂)

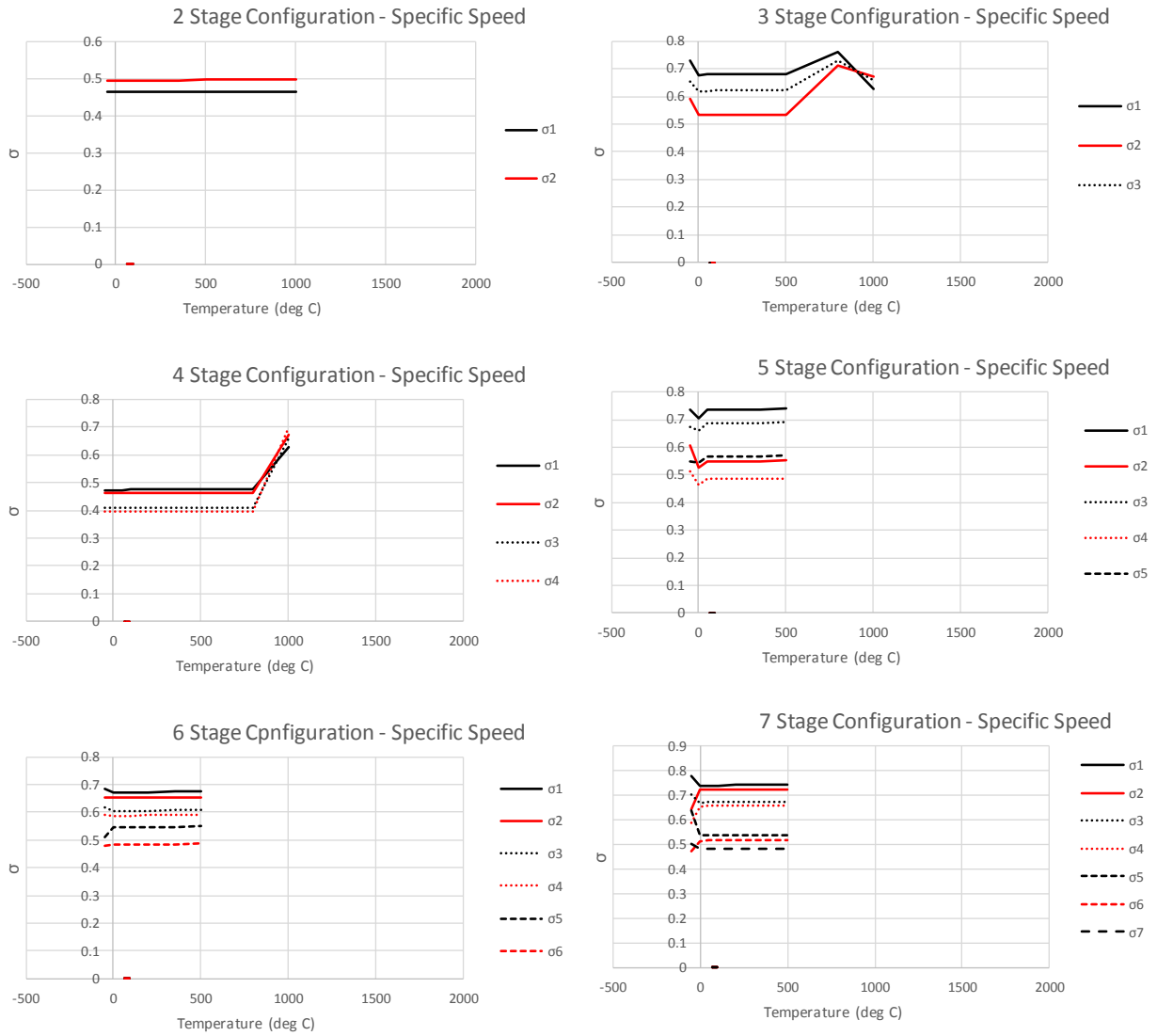


Figure A6.5.11. Specific Speed vs. Temperature at First Rotor Inlet (H₂)

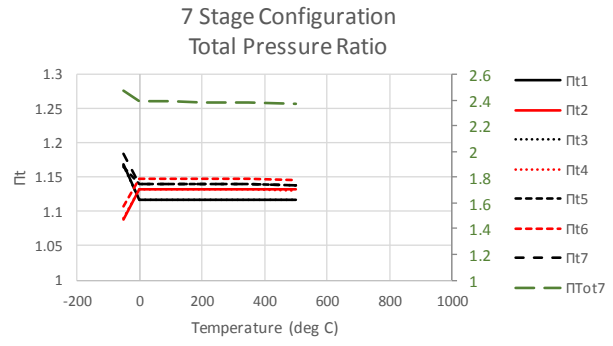
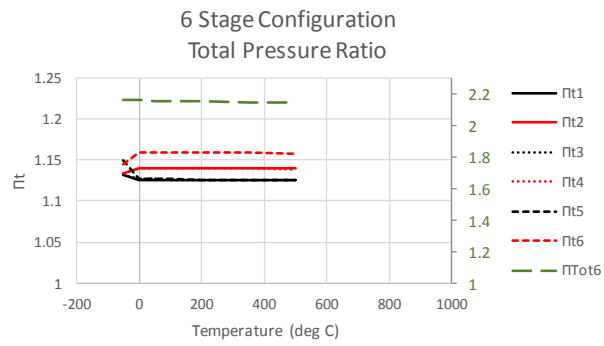
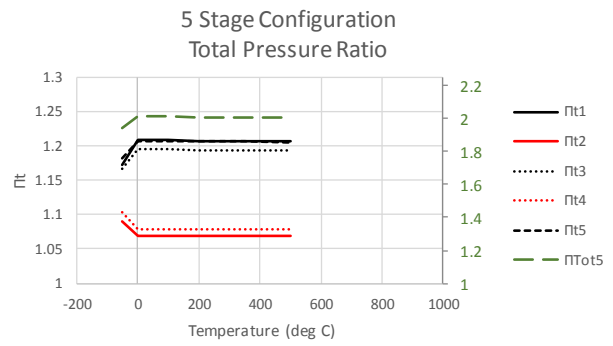
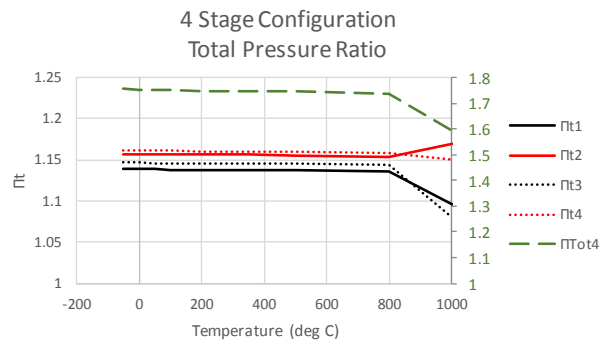
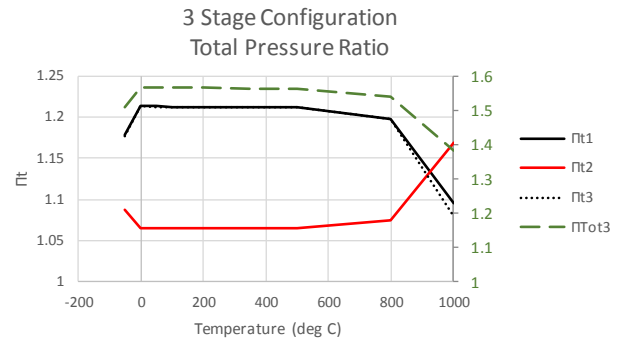
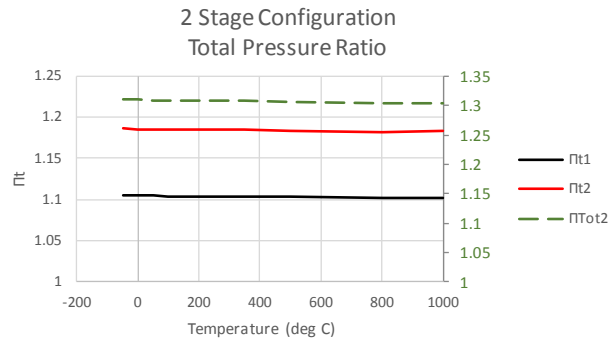


Figure A6.5.12. Total Pressure Ratio vs. Temperature at First Rotor Inlet (H_2)

A6.6 Water Vapor and Carbon Dioxide Mixture

Water vapor is 45% by mass.

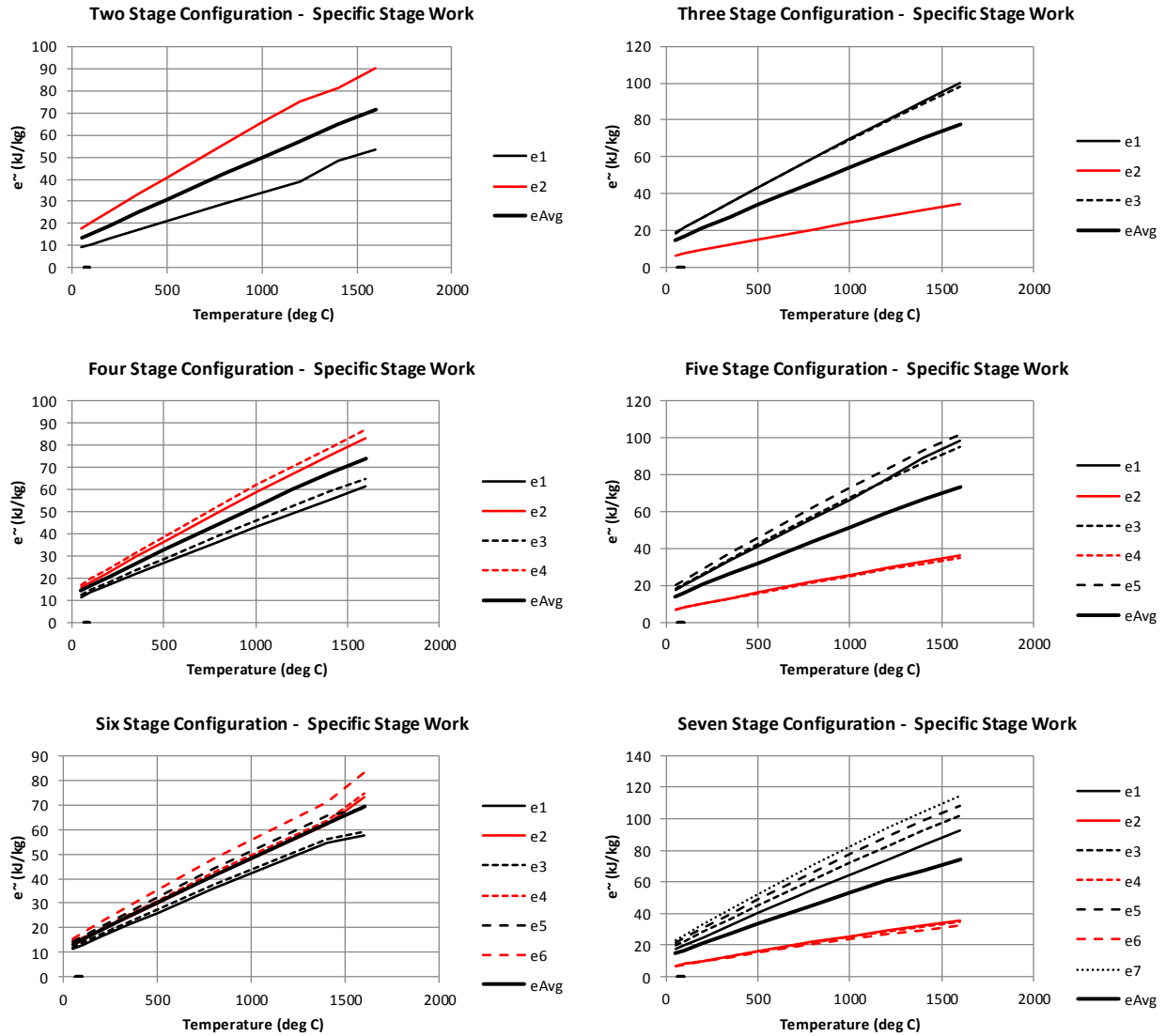


Figure A6.6.1. Specific Work vs. Temperature at First Rotor Inlet (H_2O/CO_2)

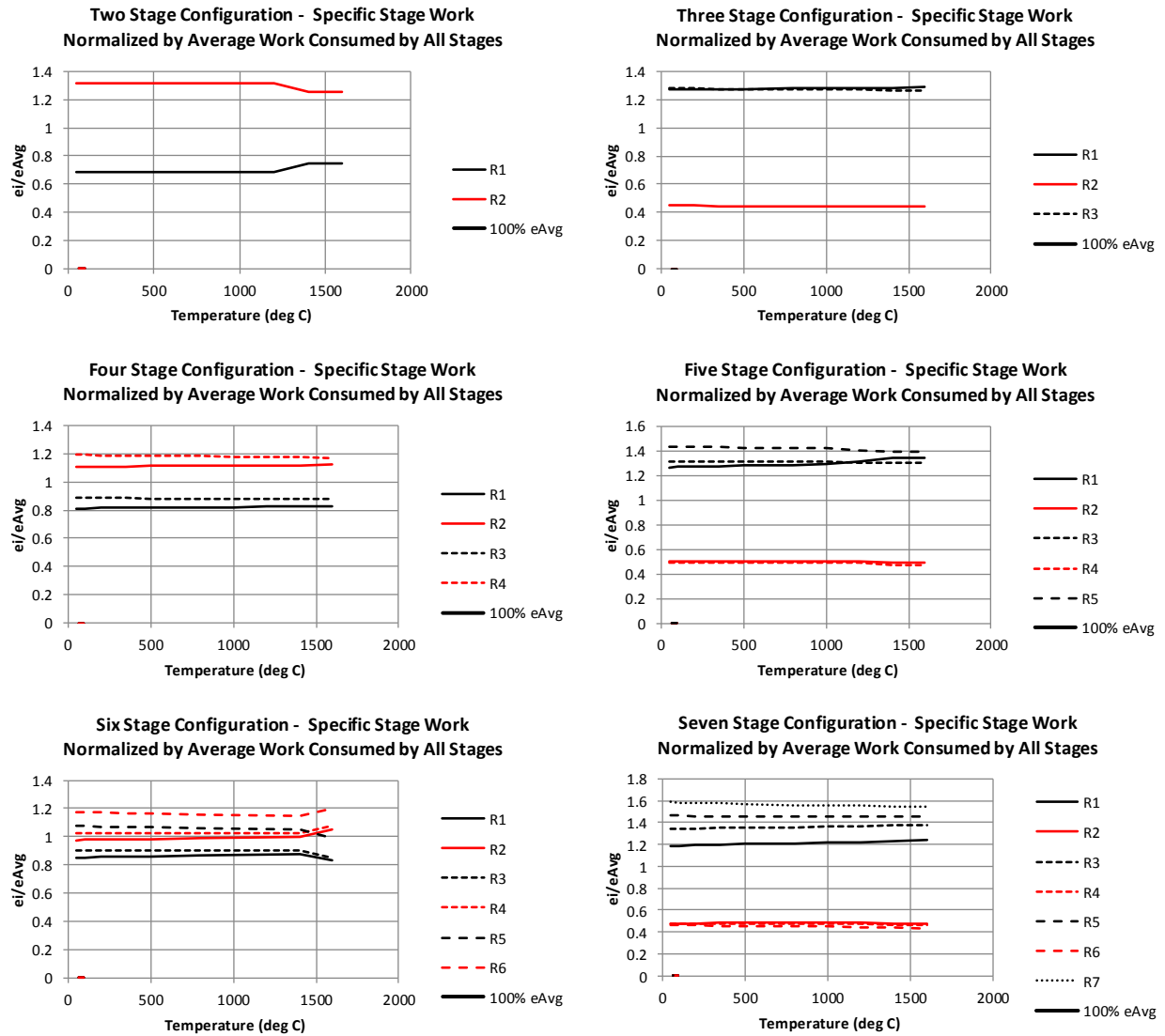


Figure A6.6.2. Normalized Specific Work vs. Temperature at First Rotor Inlet (H₂O/CO₂)

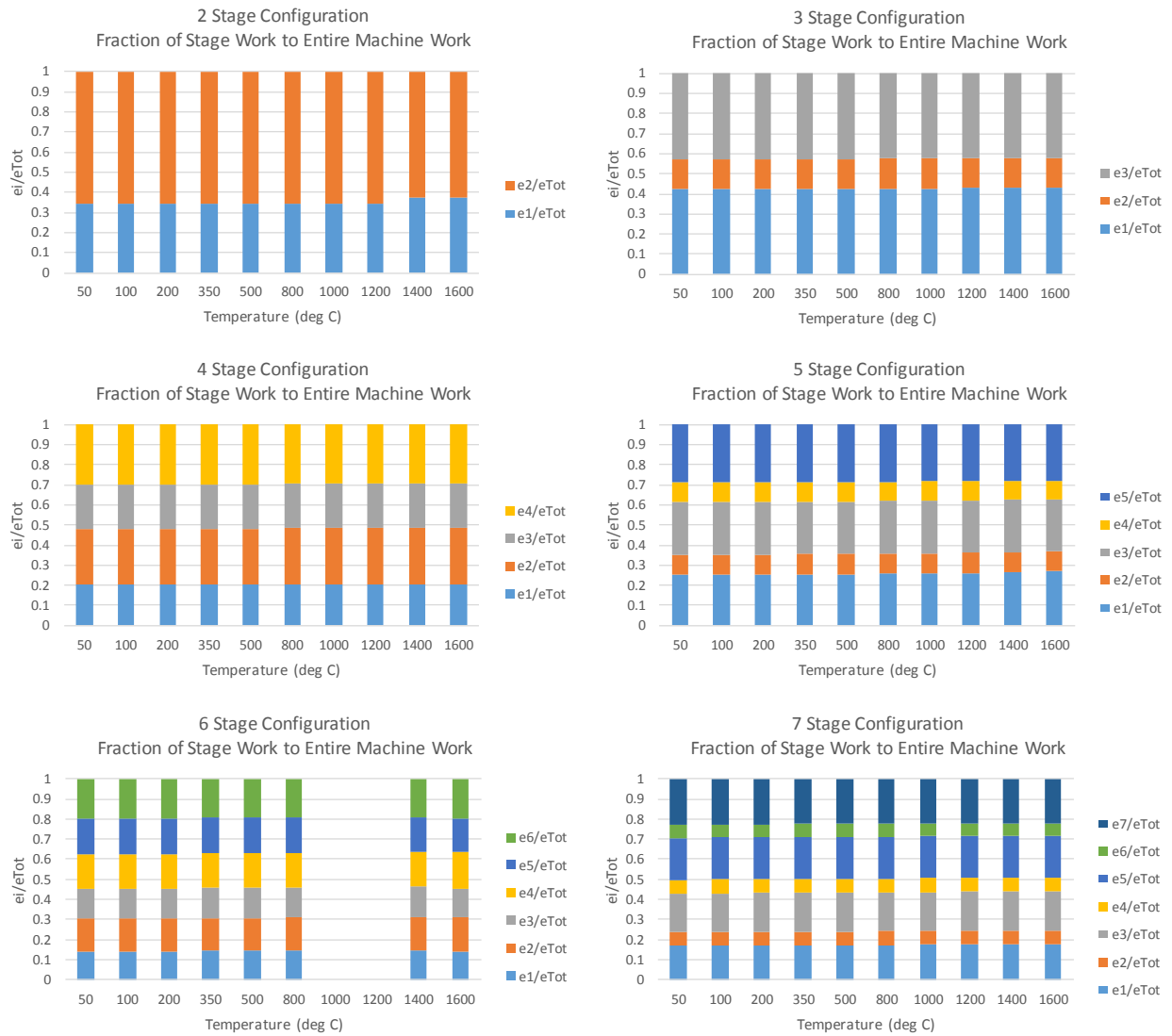


Figure A6.6.3. Rotor Work Fraction of Combined Work Total vs. Temperature at First Rotor Inlet (H₂O/CO₂)

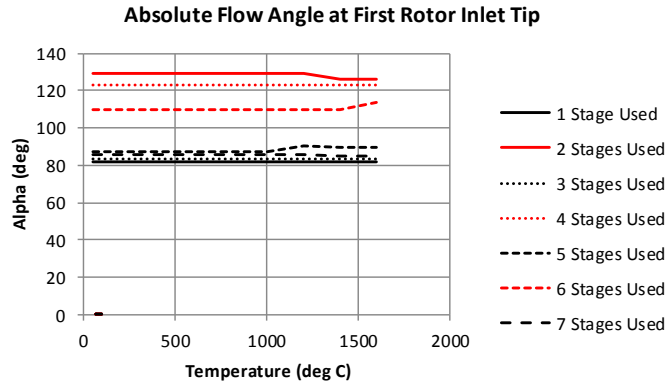


Figure A6.6.4. Absolute Flow Angle at First Rotor Inlet vs. Temperature at First Rotor Inlet (H₂O/CO₂)

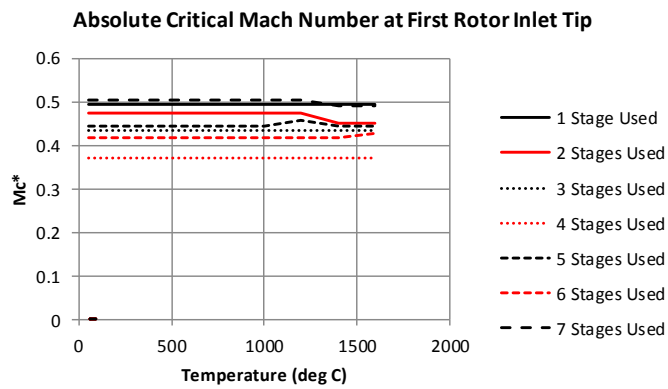


Figure A6.6.5. Absolute Critical Mach Number at First Rotor Inlet vs. Temperature at First Rotor Inlet (H₂O/CO₂)

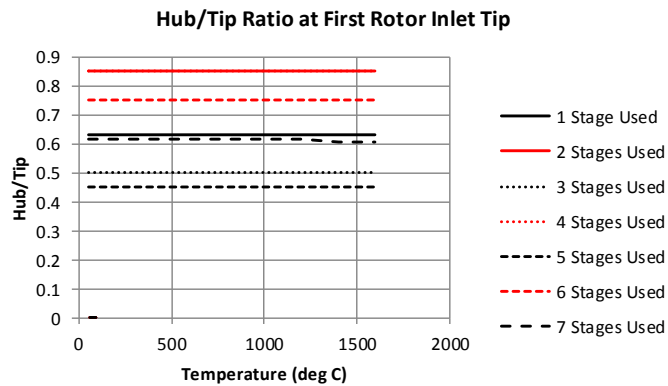
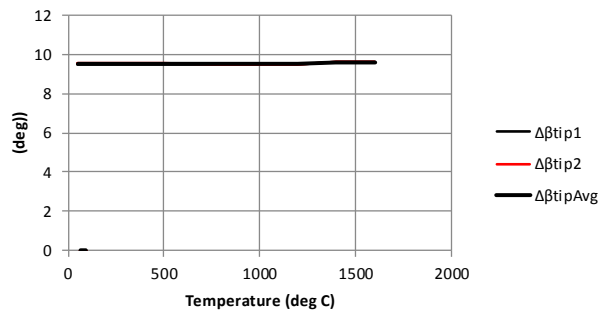
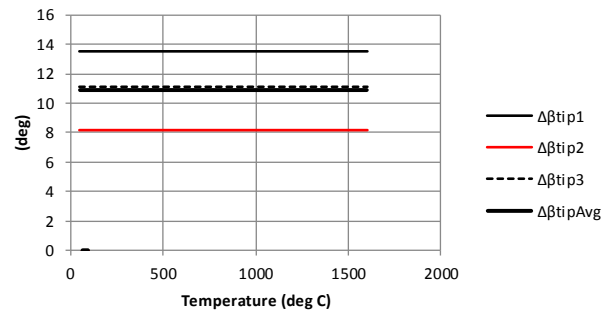


Figure A6.6.6. Hub/Tip Ratio at First Rotor Inlet vs. Temperature at First Rotor Inlet (H₂O/CO₂)

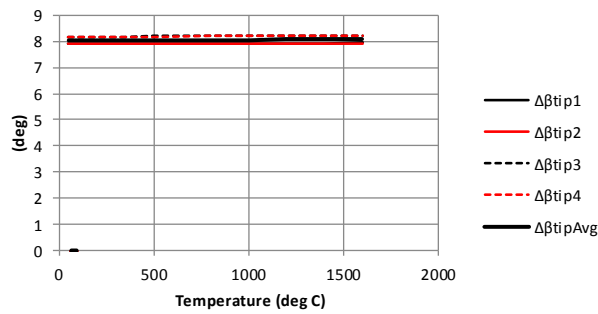
Two Stage Configuration - Blade Turning at Tip



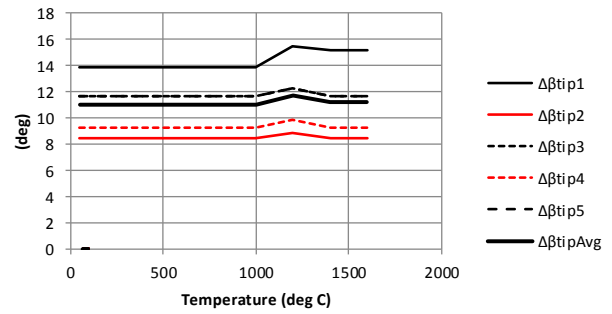
Three Stage Configuration - Blade Turning at Tip



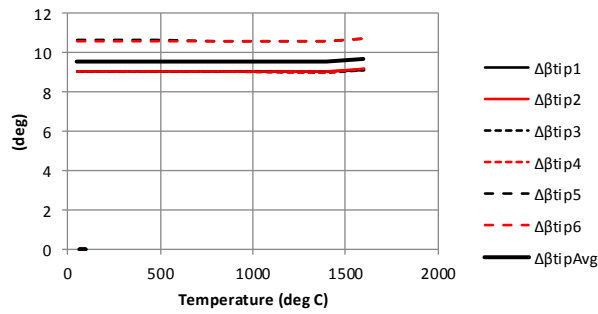
Four Stage Configuration - Blade Turning at Tip



Five Stage Configuration - Blade Turning at Tip



Six Stage Configuration - Blade Turning at Tip



Seven Stage Configuration - Blade Turning at Tip

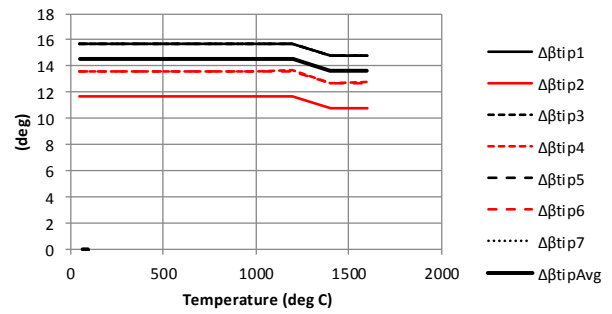


Figure A6.6.7. Blade Turning vs. Temperature at First Rotor Inlet (H_2O/CO_2)

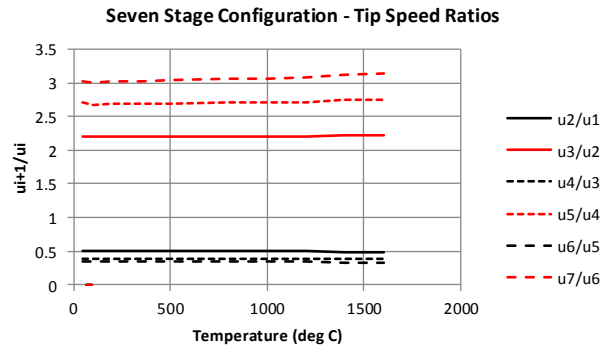
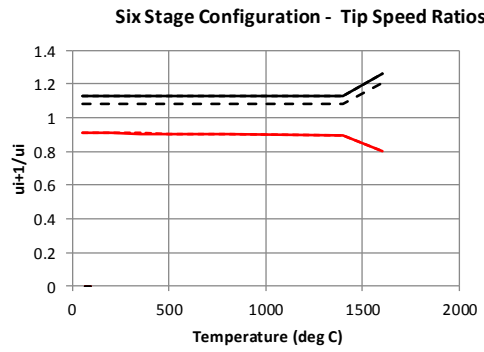
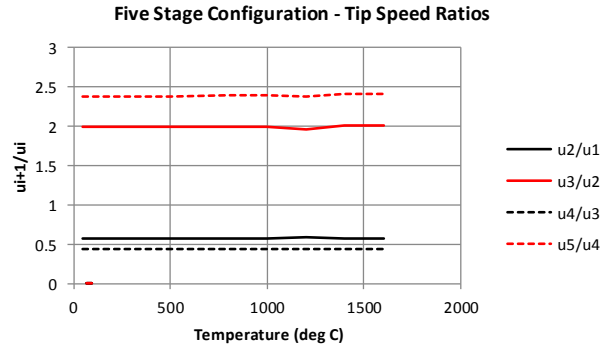
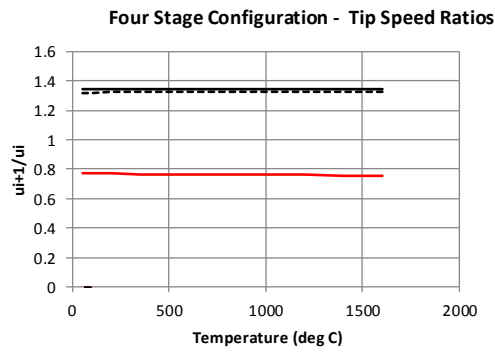
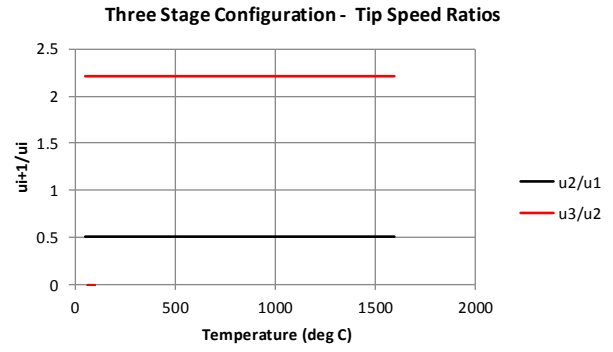
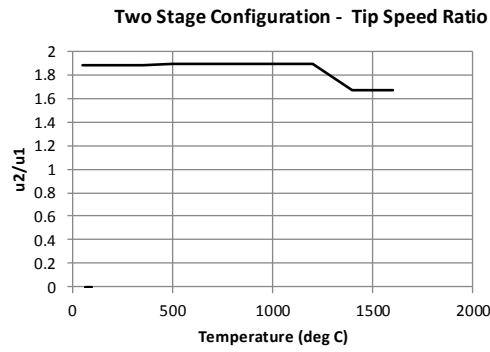


Figure A6.6.8. Tip Speed Ratio vs. Temperature at First Rotor Inlet (H_2O/CO_2)

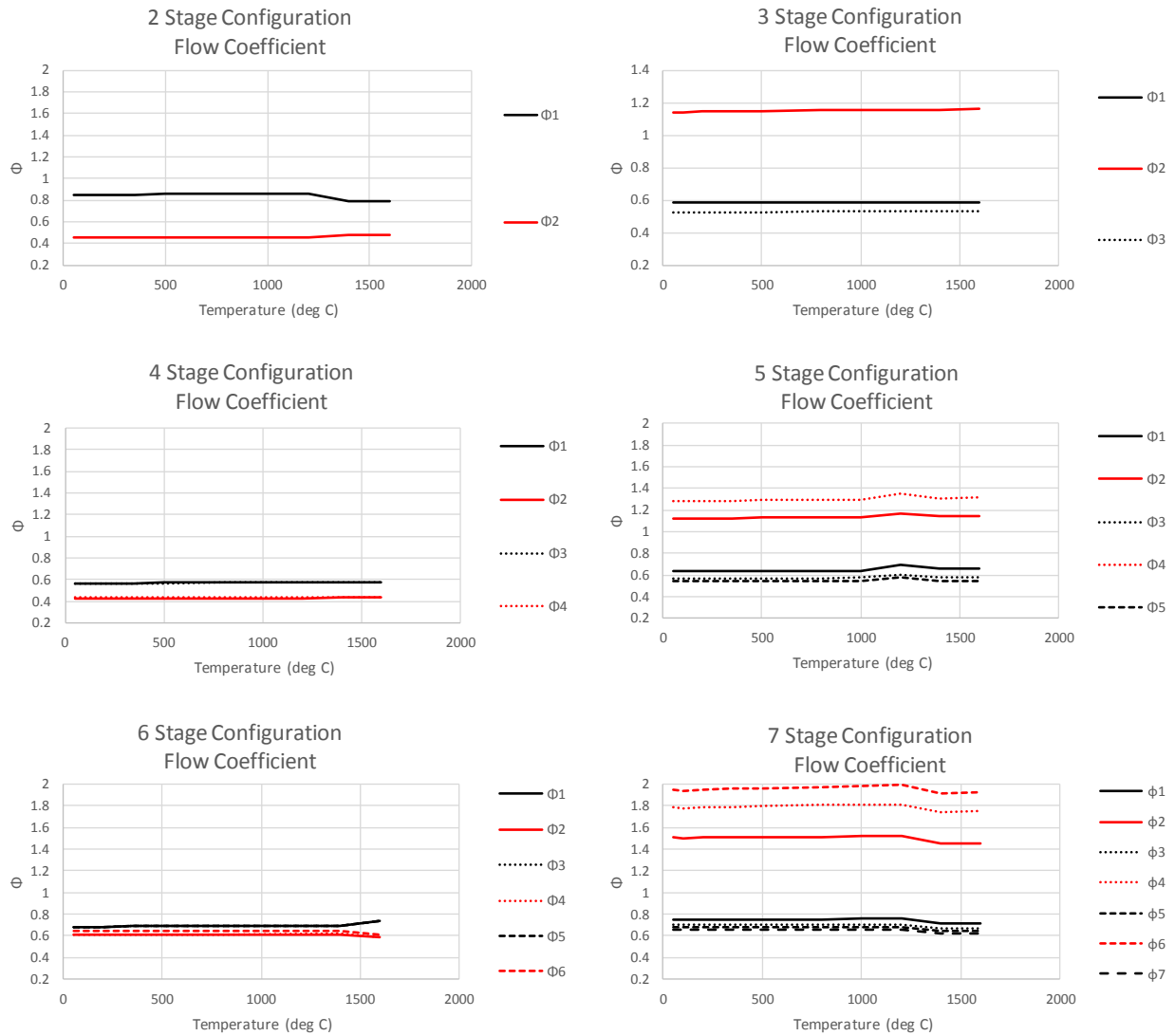


Figure A6.6.9. Flow Coefficient vs. Temperature at First Rotor Inlet (H₂O/CO₂)

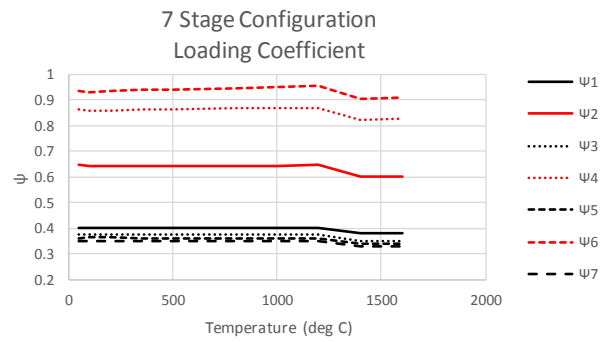
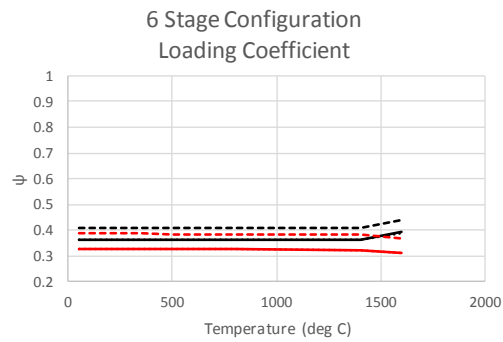
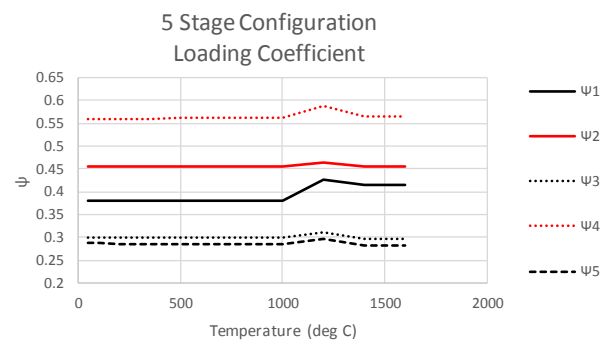
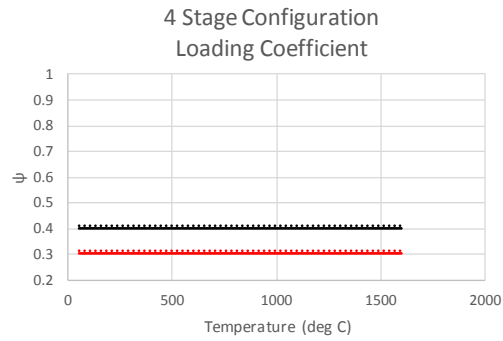
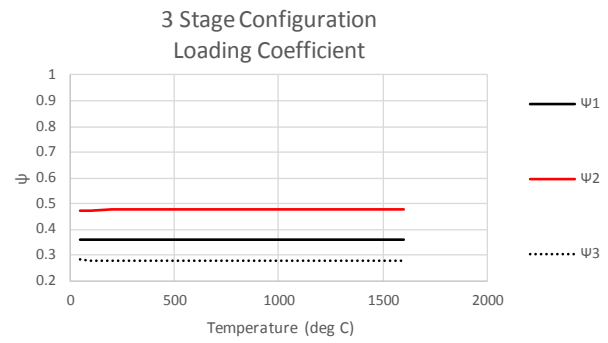
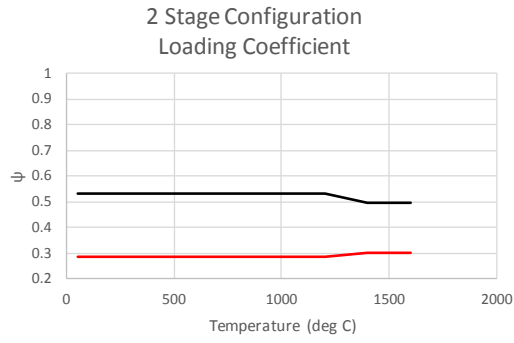


Figure A6.6.10. Blade Loading Coefficient vs. Temperature at First Rotor Inlet (H₂O/CO₂)

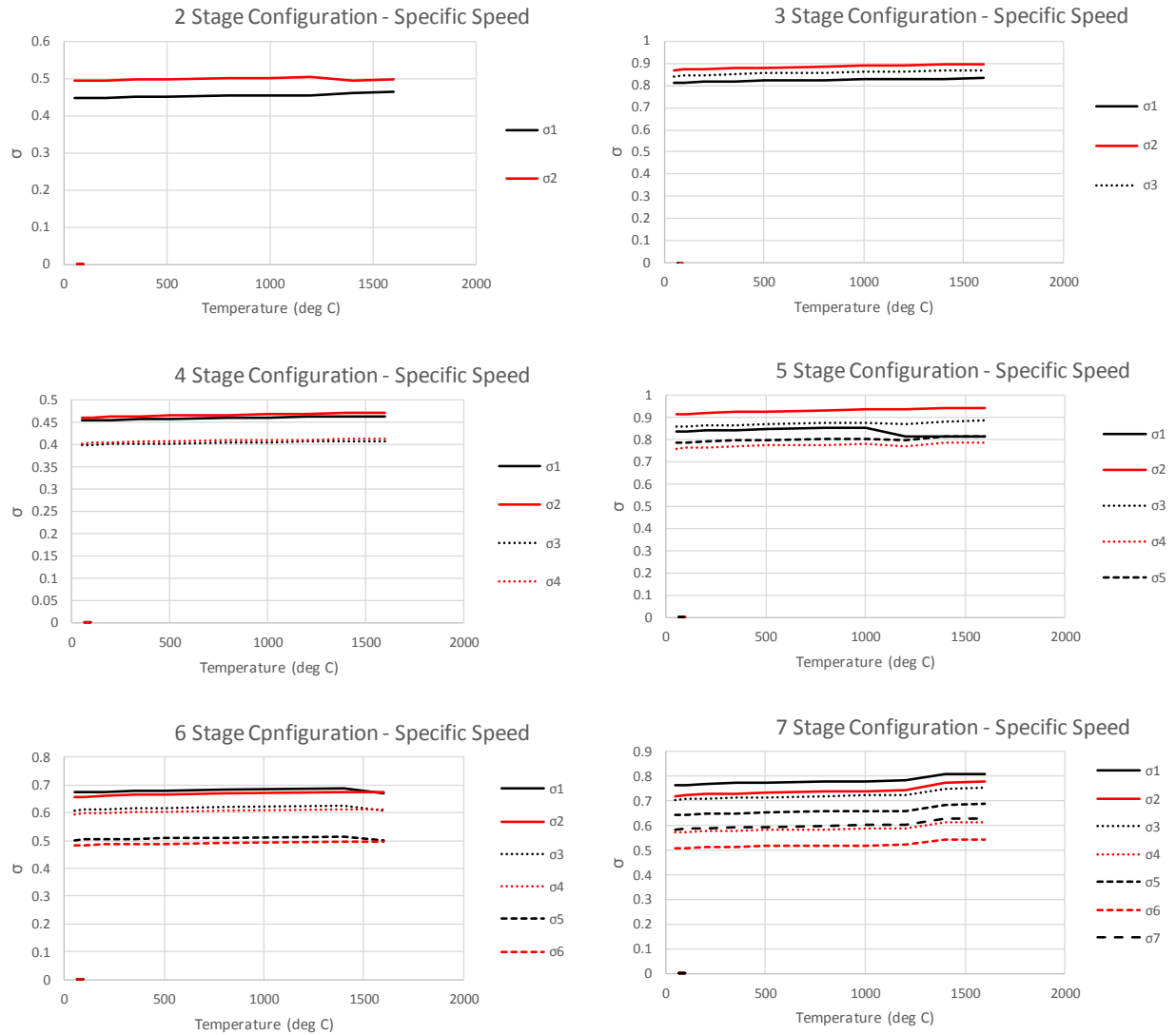


Figure A6.6.11. Specific Speed vs. Temperature at First Rotor Inlet (H₂O/CO₂)

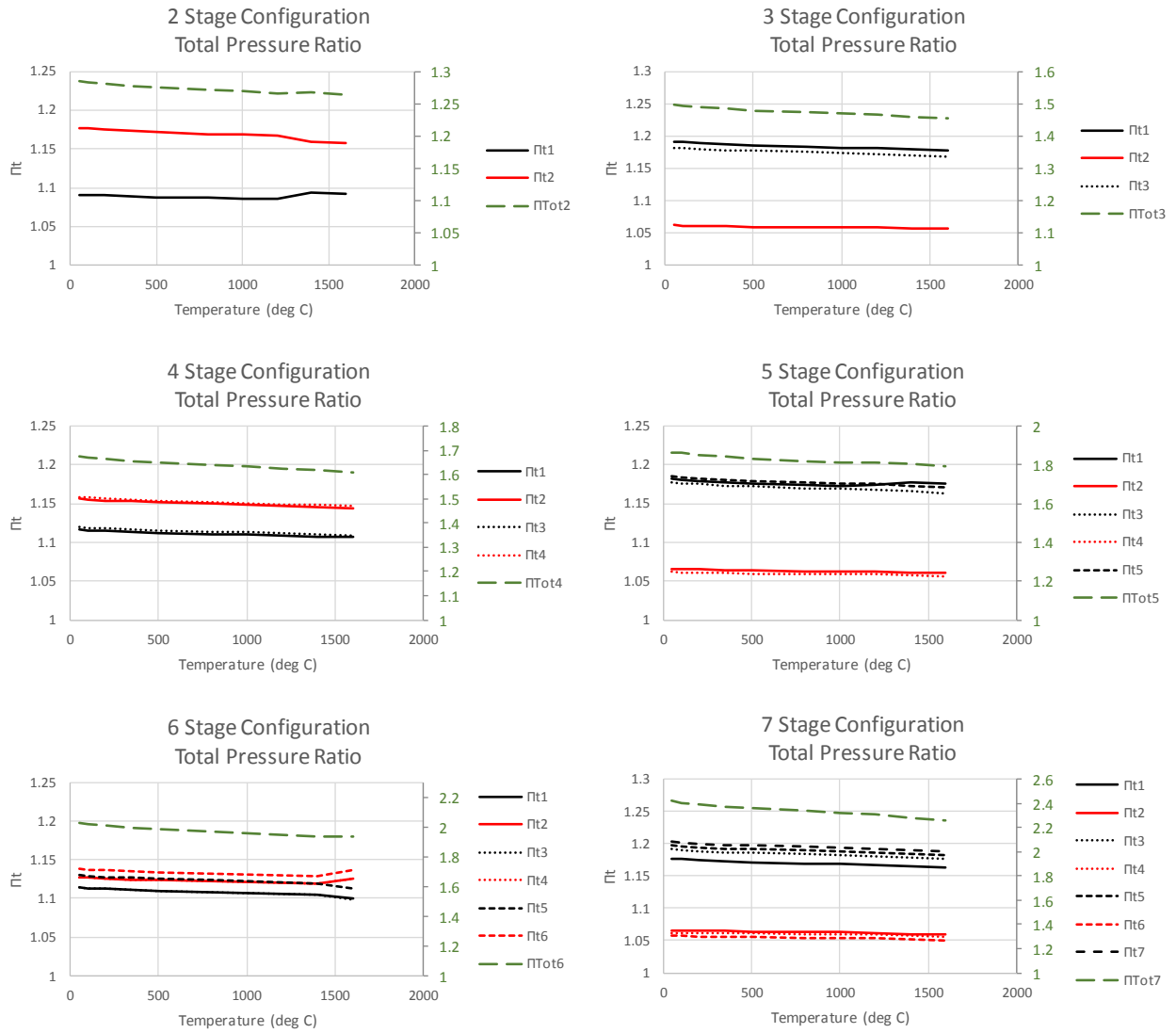


Figure A6.6.12. Total Pressure Ratio vs. Temperature at First Rotor Inlet (H₂O/CO₂)

A6.7 Helium

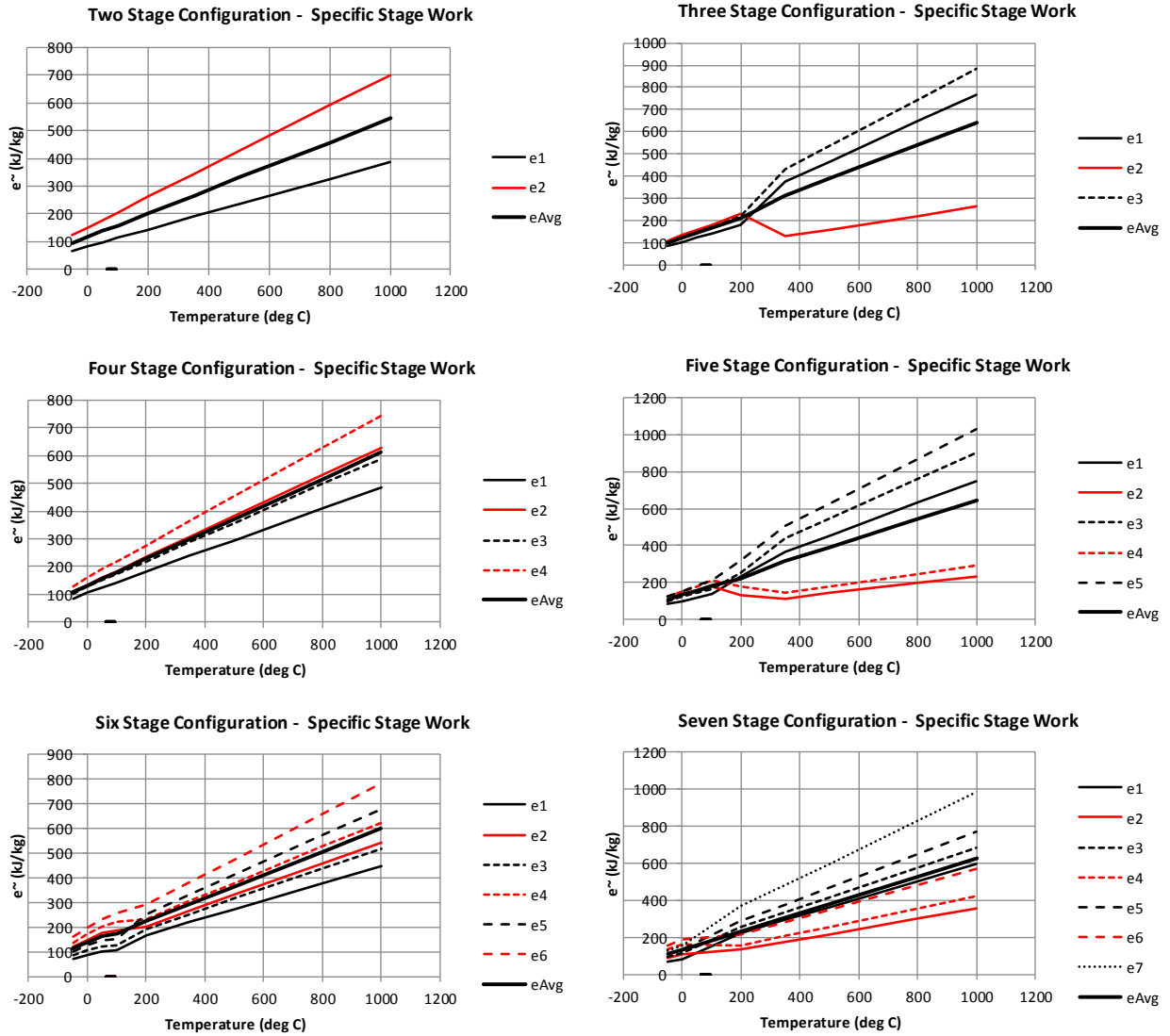
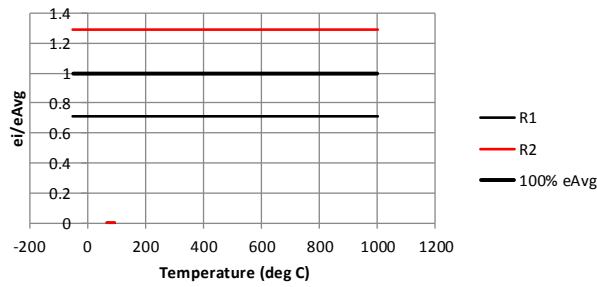
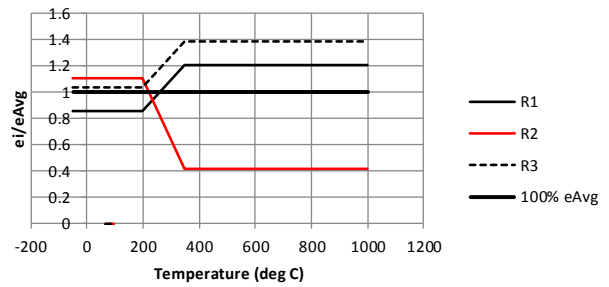


Figure A6.7.1. Specific Work vs. Temperature at First Rotor Inlet (He)

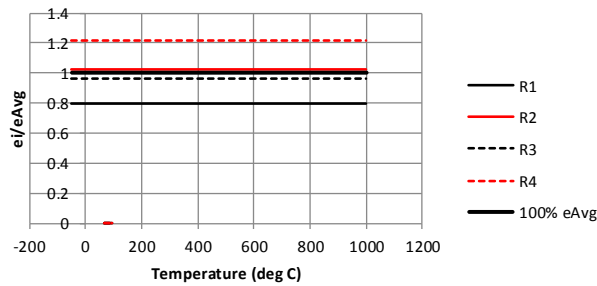
**Two Stage Configuration - Specific Stage Work
Normalized by Average Work Consumed by All Stages**



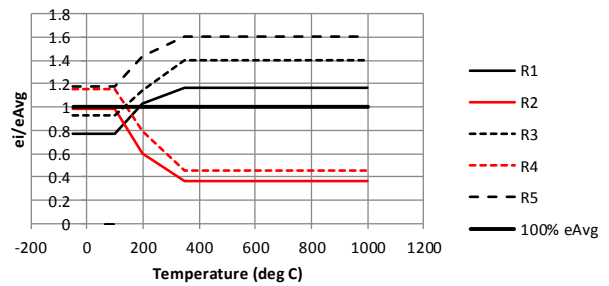
**Three Stage Configuration - Specific Stage Work
Normalized by Average Work Consumed by All Stages**



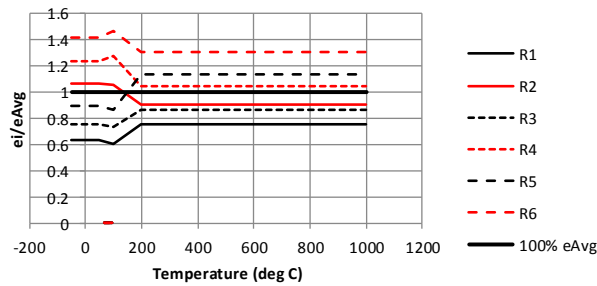
**Four Stage Configuration - Specific Stage Work
Normalized by Average Work Consumed by All Stages**



**Five Stage Configuration - Specific Stage Work
Normalized by Average Work Consumed by All Stages**



**Six Stage Configuration - Specific Stage Work
Normalized by Average Work Consumed by All Stages**



**Seven Stage Configuration - Specific Stage Work
Normalized by Average Work Consumed by All Stages**

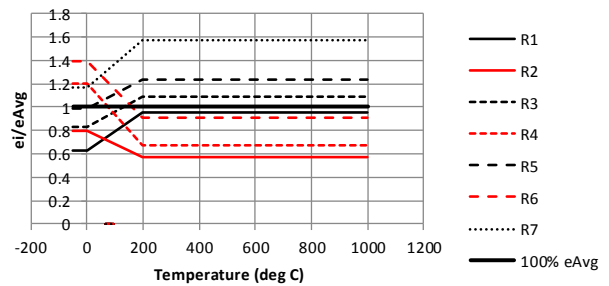


Figure A6.7.2. Normalized Specific Work vs. Temperature at First Rotor Inlet (He)

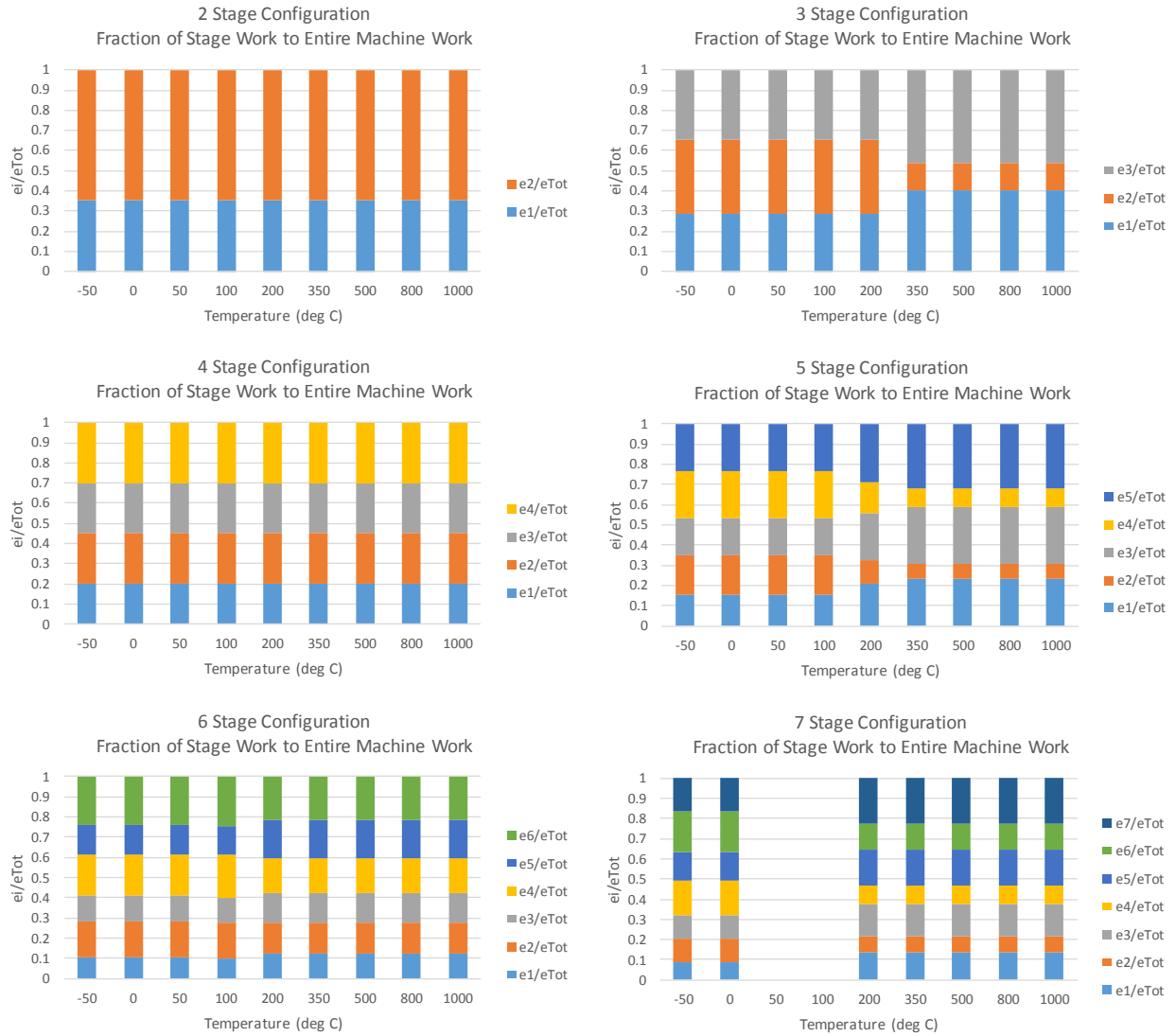


Figure A6.7.3. Rotor Work Fraction of Combined Work Total vs. Temperature at First Rotor Inlet (He)

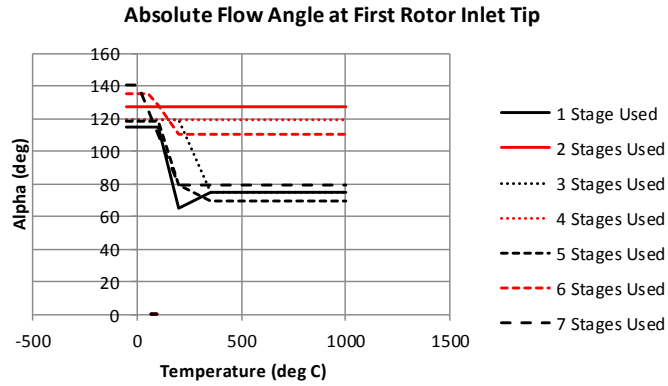


Figure A6.7.4. Absolute Flow Angle at First Rotor Inlet vs. Temperature at First Rotor Inlet (He)

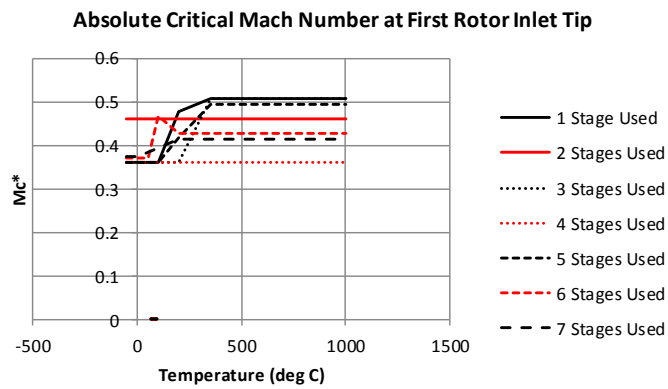


Figure A6.7.5. Absolute Critical Mach Number at First Rotor Inlet vs. Temperature at First Rotor Inlet (He)

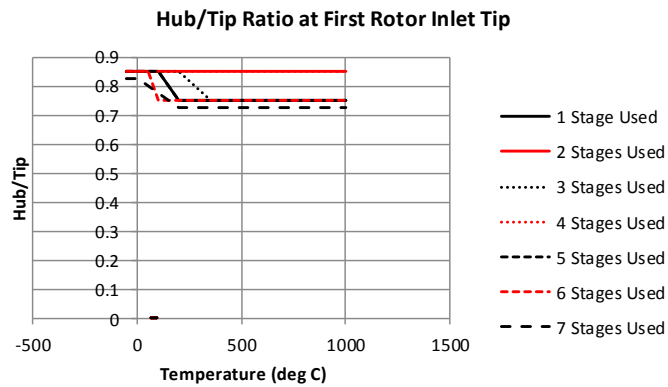


Figure A6.7.6. Hub/Tip Ratio at First Rotor Inlet vs. Temperature at First Rotor Inlet (He)

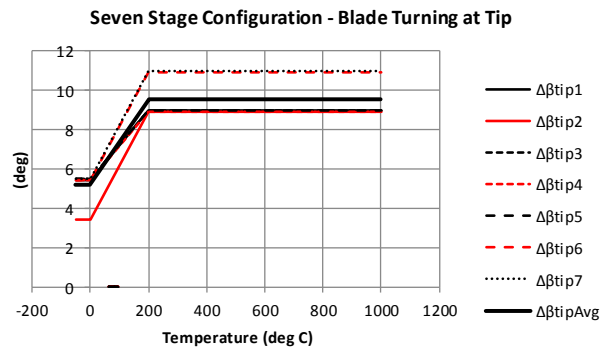
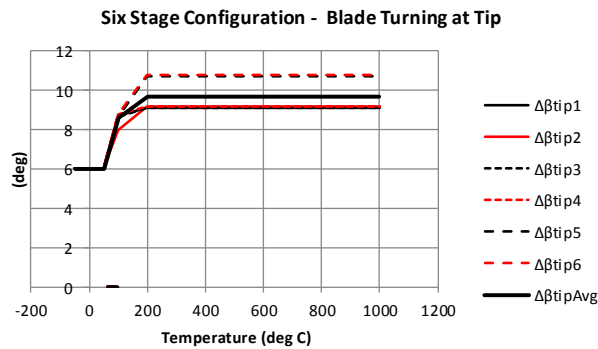
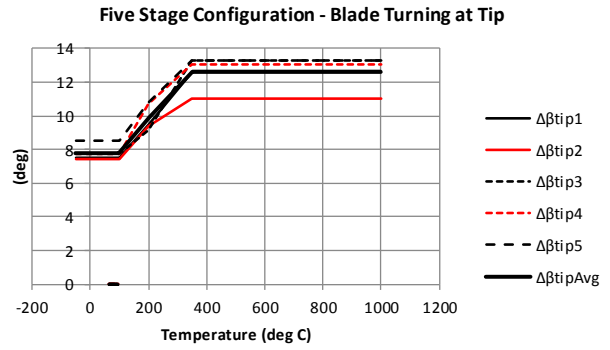
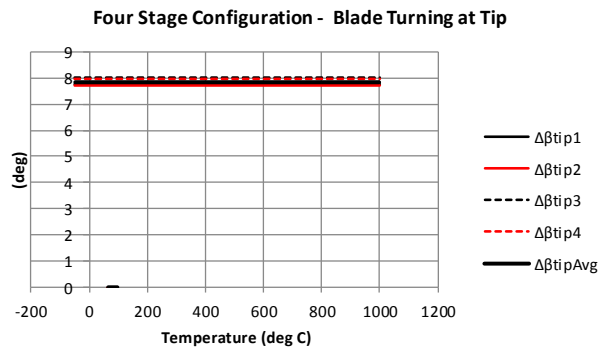
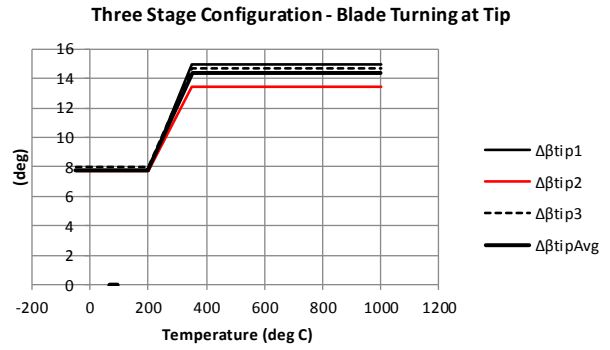
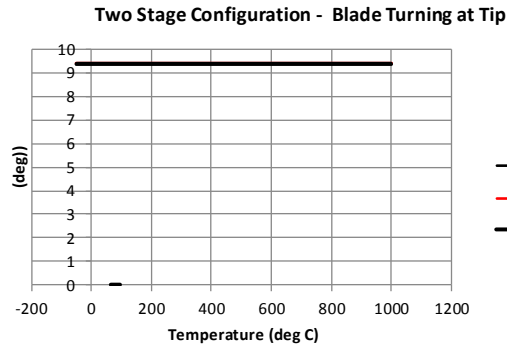


Figure A6.7.7. Blade Turning vs. Temperature at First Rotor Inlet (He)

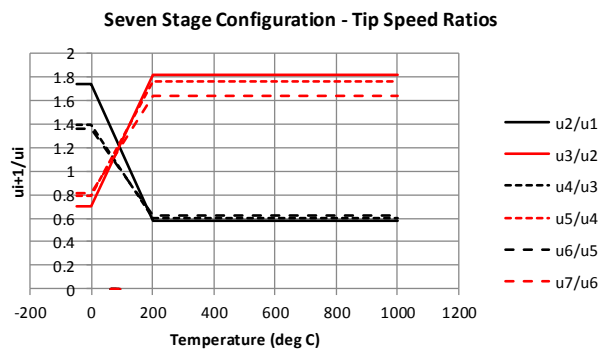
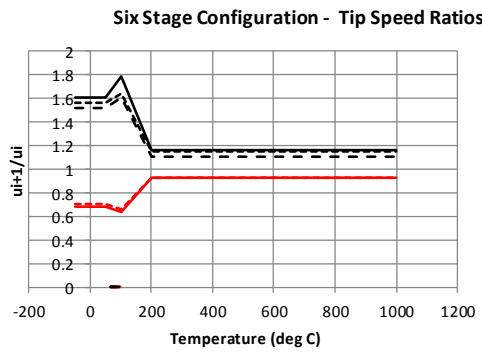
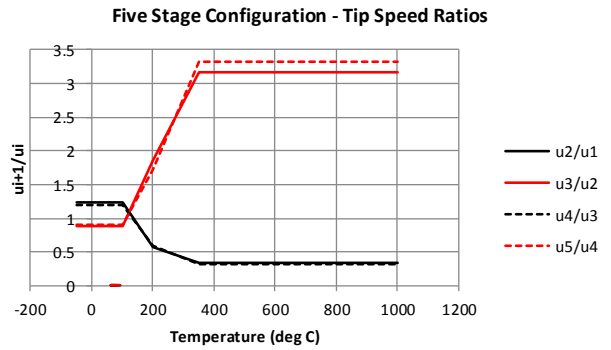
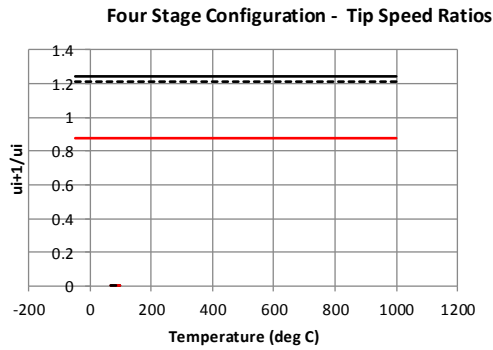
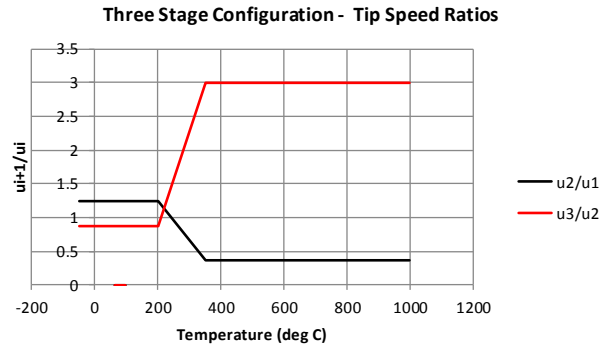
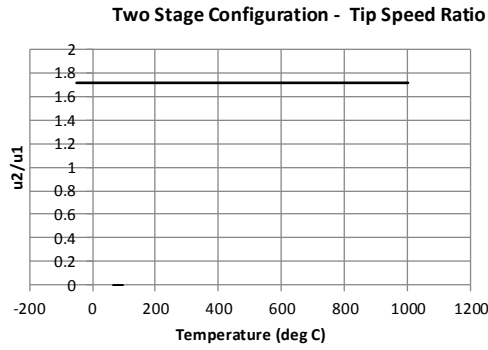


Figure A6.7.8. Tip Speed Ratio vs. Temperature at First Rotor Inlet (He)

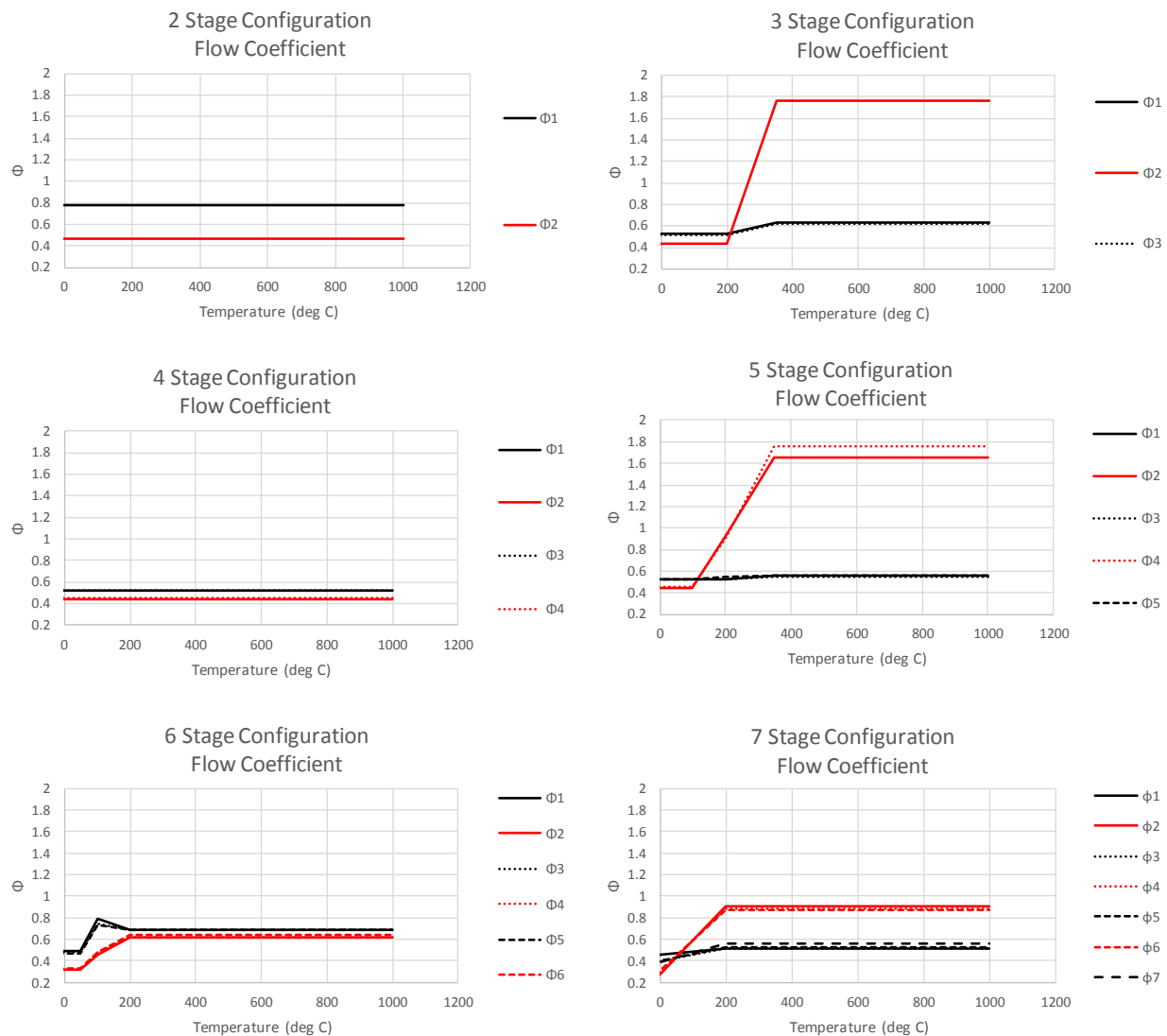


Figure A6.7.9. Flow Coefficient vs. Temperature at First Rotor Inlet (He)

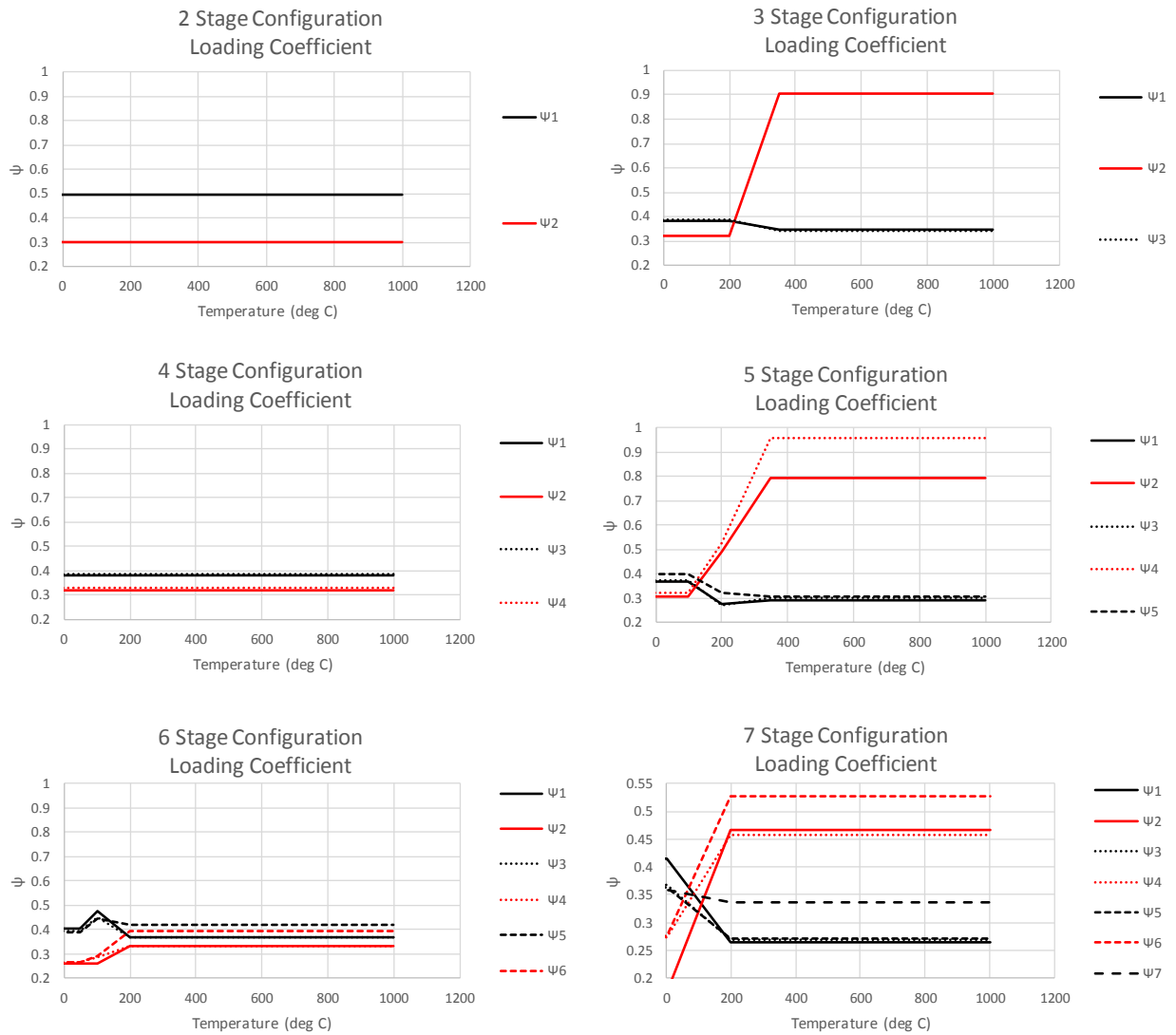


Figure A6.7.10. Blade Loading Coefficient vs. Temperature at First Rotor Inlet (He)

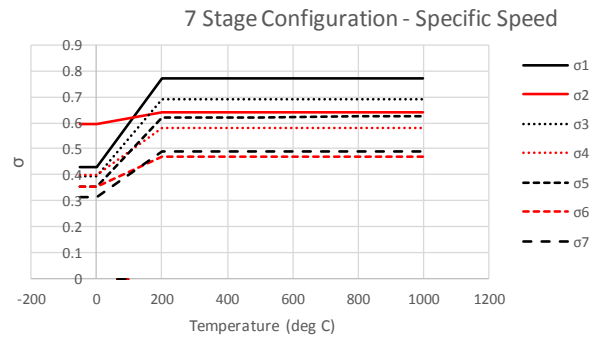
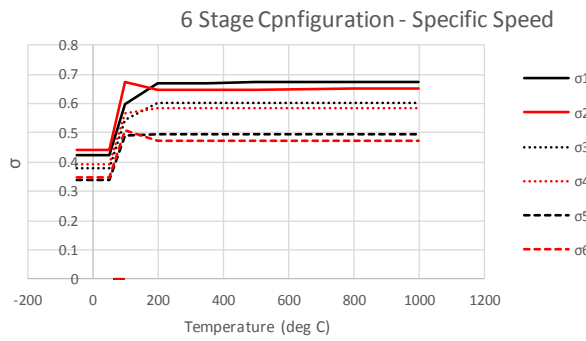
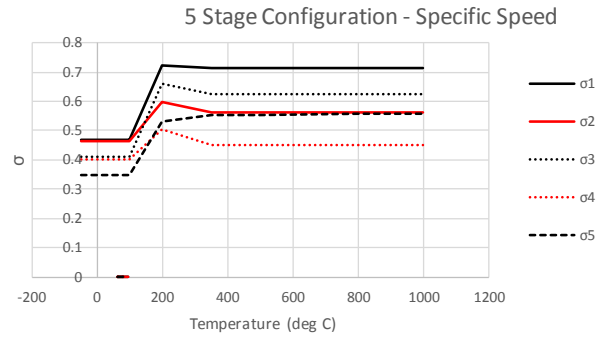
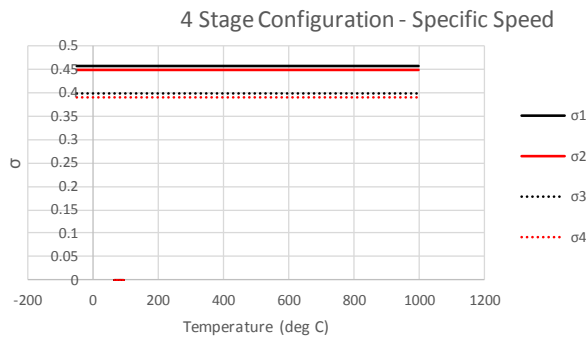
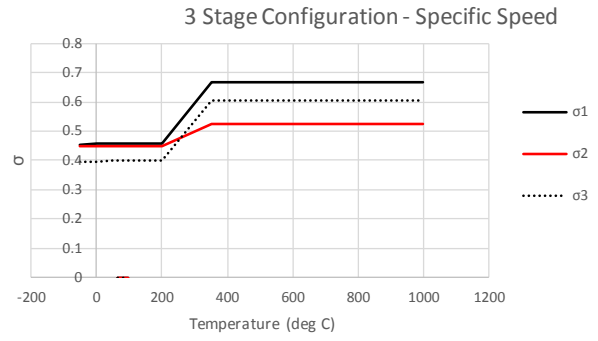
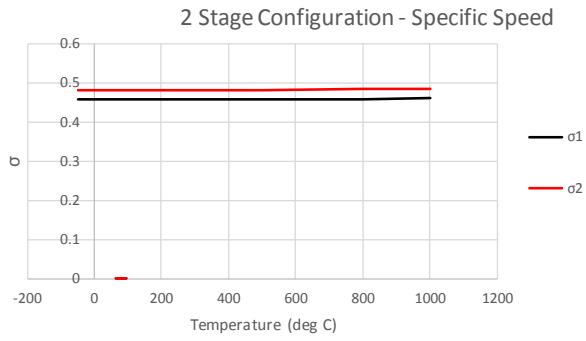


Figure A6.7.11. Specific Speed vs. Temperature at First Rotor Inlet (He)

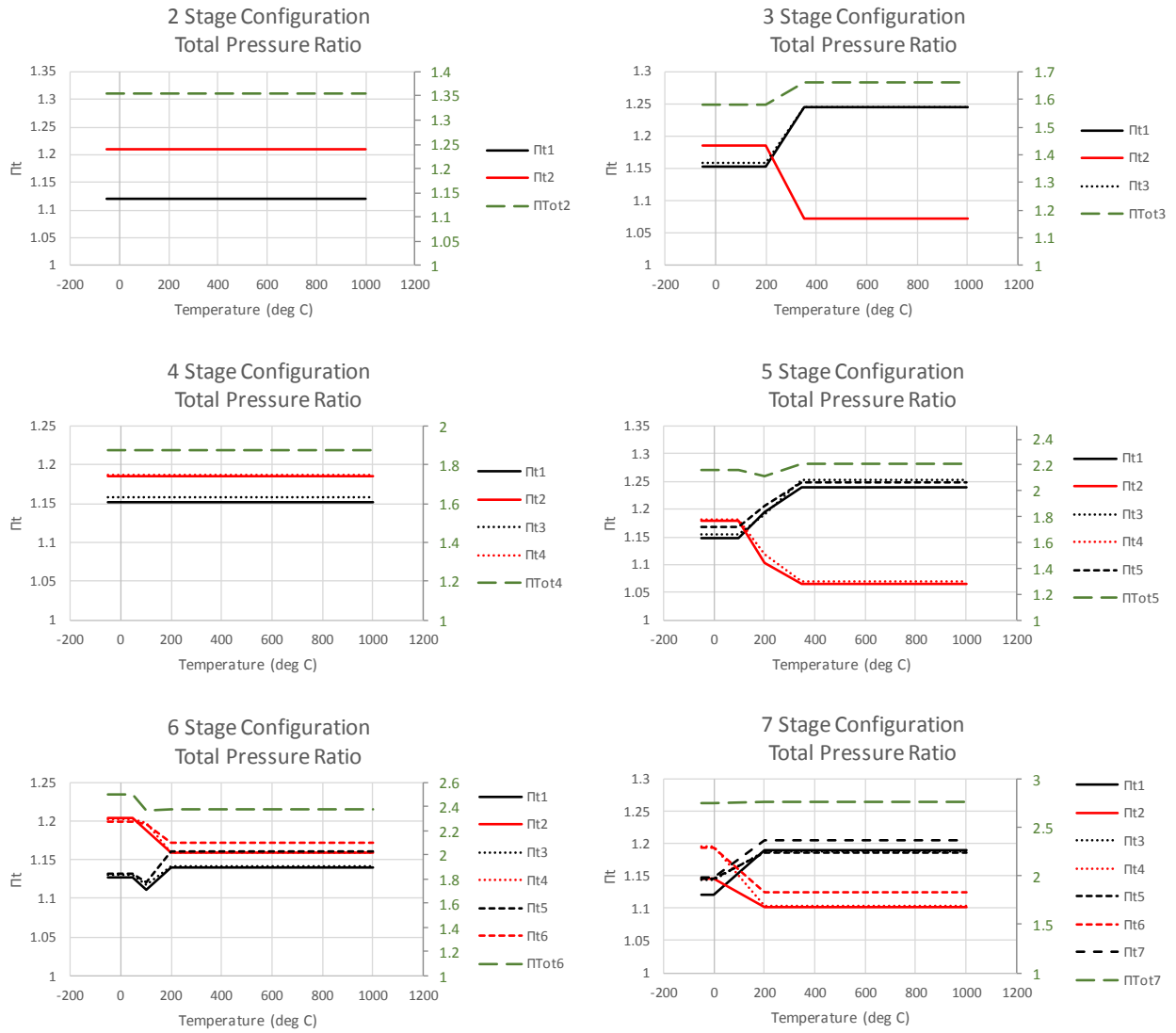


Figure A6.7.12. Total Pressure Ratio vs. Temperature at First Rotor Inlet (He)

REFERENCES

REFERENCES

- [1] N. Müller, “Design of Compressor Impellers for Water as a Refrigerant”. Transaction-American Society of Heating Refrigerating and Air Conditioning Engineers (ASHRAE) Vol. 107 pp. 214-222, 1999.
- [2] M. Moran, H. Shapiro, *Fundamentals of Engineering Thermodynamics*. 6th. John Wiley & Sons, Inc. 2008. Chap 10. Appendix Table A-2, A-10.
- [3] A. Kilicarslan, N. Müller, “A Comparative Study of Water as a Refrigerant with Some Current Refrigerants”, *Int. J. Energ. Res.*, 29 (2005), pp. 947–959.
- [4] Q. Li, J. Piechna, N. Müller, “Design of a Novel Axial Impeller as a part of Counter-Rotating Axial Compressor to Compress Water Vapor as Refrigerant”, *Appl Energy*, 88 (9) (2011), pp. 3156–3168.
- [5] <http://www.nrcan.gc.ca/energy/publications/efficiency/ee-products/roomaircond/6667>
- [6] J. Paul. 1998. “District Cooling with Water as Refrigerant Combined with Energy Storage Systems,” IIF-IIR, Commission B2, with B1, E1&E2, Oslo, Norway, p.82-92.
- [7] B. Lachner, G. Nellis, D. Reindl. “The Commercial Feasibility of the use of Water Vapor as a Refrigerant”. In: *International Journal of Refrigeration* 30 (4 June 2006), pp. 699-708.
- [8] S.E. Wight, Y. Tsukasa, B.A. Le Drew, N.C. D’Orsi, “The Efficiency Limits of Water Vapor Compressors”, Technical Report ARTI-21CR/605-10010-01, Air-Conditioning and Refrigeration Technology Institute, 2000.
- [9] R. Aungier, "Axial-Flow Compressors: A Strategy for Aerodynamic Design and Analysis," New York, ASME Press, 2003.
- [10] Q. Li, J. Wang, N. Müller, “On the Realization of Filament Wound Composite Impeller for Water Refrigerant”, ASME paper IMECE2009-12786.
- [11] N. Müller, Board of Trustees of Michigan State University, “Woven Turbomachine Impeller”, U.S. Patent: 7938627 B2, issued date May 10, 2011.
- [12] Q. Li. “Design and Analysis of a Novel Composite Axial Impeller for Compressing Water Vapor as Refrigerant”. Dissertation. Michigan State University, 2010.
- [13] M. Patil. “Composite Wound Axial Turbomachinery Impeller for Green-Renewable Energy: Applications and Numerical Structural Analysis”. Dissertation. Michigan State University, 2014.

- [14] Crystic Composites Handbook, Scott Bader Company Limited, Wollaston, Wllingborough, Northamptonshire, Issued December 2005.
- [15] P. Valdimarsson. "Geothermal Power Plant Cycles and Main Components", in: *Short Course on Geothermal Drilling, Resource Development and Power Plants*, Santa Tecla, El Salvador, January 16-22, 2011.
- [16] R. Bertani, "Geothermal Power Generation in the World 2005-2010 Update Report," in *World Geothermal Congress*, Bali, Indonesia, 2010.
- [17] R. Bertani, "World Geothermal Power Generation in the Period 2001-2005," *Geothermics*, pp. 651-669, 2005.
- [18] H. Gupta, S. Roy, Worldwide Status of Geothermal Resource Utilization in Geothermal Energy, and Alternative Resource for the 21st Century, 1st, Oxford: Elsevier, 2007.
- [19] G. Coury, H. Guillen and D. Cruz, "Geothermal Noncondensable Gas Removal from Turbine Inlet Steam," in: *Energy Conversion Engineering Conference*, 1996.
- [20] M. Vorum and E. A. Fritzler, "Comparative Analysis of Alternative Means for Removing Noncondensable Gases from Flash-Steam Geothermal Power Plants," NREL (National Renewable Energy Laboratory), 2000.
- [21] <http://www.usvacuumpumps.mx/upload/image/LiquidRingDiagram.gif>
- [22] A. Khawaji, I. Kutubkhanah, J-M. Wie, "Advances in Seawater Desalination Technologies," *Desalination*, 221 (2008) 47-69.
- [23] *Water: The Power, Promise, and Turmoil of North America's Fresh Water*, National Geographic Special Edition, November 1993.
- [24] IDA Desalting Inventory 2004: Desalination Business Stabilized on a High Level, *Int. Desal. Water Reuse*, 14 (2) (2004) 14–17.
- [25] S. Bou-Hamad, M. Abdel-Jawad, M. Al-Tabtabaei and S. Al-Shammari, "Comparative Performance Analysis of Two Seawater Reverse Osmosis Plants: Twin Hollow Fine Fiber and Spiral Wound Membranes", *Desalination*, 120 (1–2) (1998) 95–106.
- [26] H.I. Shaban, "Reverse Osmosis Membranes for Seawater Desalination State-of-the-Art", *Separ. Puri. Methods*, 19 (2) (1990) 121–131.
- [27] I. Moch, Jr., "Hollow Fiber Permeators & Reverse Osmosis", *Desal. Water Reuse*, 3 (2) (1993) 30–37.
- [28] S.S. Whipple and C.R. Granlund, "Spiral Wound Reverse Osmosis", Part I: Technology, *Desal. Water Reuse*, 3 (2) (1993) 44–46.

- [29] A.M. Hassan, A.M. Al-Abanmy, M. Al-Thobiety, T. Al-Lohabi, I. Al-Masudi, A.A. Al-Grier, L.M. Bakheet, M.M.I. Amri, K. Aryah and M. Al-Hydaibi, "Performance Evaluation of SWCC SWRO Plants", in: Proceedings of the IDA World Conference on Desalination and Water Reuse, Washington, DC, August 25–29, 1991.
- [30] A.D. Khawaji, T. Khan and J.M. Wie, "Gas Turbine Operating Experience in a Power/Seawater Desalination Cogeneration Mode", ASME ASIA 1997 Congress & Exhibition, Paper Reprint No 97-AA-120, Singapore, September 30–October 2, 1997.
- [31] B.A. Kamaluddin, S. Khan and B.M. Ahmed, "Selection of Optimally Matched Cogeneration Plants", Desalination, 93 (1993) 311–321.
- [32] A.M. El-Nashar, "Cogeneration for Power and Desalination-State of the Art Review", Desalination, 134 (2001) 7–28.
- [33] U.S. Environmental Protection Agency (EPA). 2010. Evaluation of Energy Conservation Measures for Wastewater Treatment Facilities. EPA 832-R-10-005.
- [34] Water Environment Federation (WEF). 2009. Manual of Practice (MOP) No. 32: Energy Conservation in Water and Wastewater Facilities. Prepared by the Energy Conservation in Water and Wastewater Treatment Facilities Task Force of the Water Environment Federation. McGraw Hill, New York.
- [35] Evaporator Handbook, AVP, SPX Corporation, Charlotte NC, Issued 02/2009 10003-01-08-2008.
- [36] J. Anderson. *Modern Compressible Flow with Historical Perspective*. 3rd. McGraw-Hill, 2004.
- [37] Gordon, McBride. *Computer Program for Calculation of Complex Chemical Equilibrium Compositions, Rocket Performance, Incident and Reflected Shocks, and Chapman-Jouget Detonations*. NASA SP-273, March 1976.
- [38] Greenwood, Donald T. *Principles of Dynamics*. 2nd. Prentice-Hall, 1988.
- [39] M. Schobeiri. *Turbomachinery Flow Physics and Dynamic Performance*. 2nd. Springer, 2012.
- [40] W. Briggs. *Effect of Mach Number on the Flow and Application of Compressibility Corrections in a Two-Dimensional Subsonic-Transonic Compressor Cascade Having Varied Porous-Wall Suction at the Blade Tips*. Technical Note 2649. Cleveland, Ohio: NACA, March 1952.
- [41] S. Pope. *Turbulent Flows*. Cambridge University Press, 2000.
- [42] P. de Haller. "Das Verhalten von Tragflugelgittern in Axialverdichtern und in Windkanal", *Bernstoff-Warmer-Kraft* Vol. 5, Heft 10, 1953.

- [43] S. Lieblein, F. Schwenk, R. Broderick. *Diffusion Factor for Estimating Losses and Limiting Blade Loadings in Axial-Flow-Compressor Blade Elements*. Research Memorandum RM E53D01. Cleveland, Ohio: NACA, Lewis Flight Propulsion Laboratory, June 1953.
- [44] S. Korpela. *Principles of Turbomachinery*. John Wiley & Sons, Inc. 2011.
- [45] J. Anderson, *Computational Fluid Dynamics: The Basics with Applications*. New York: McGraw-Hill, 1994.
- [46] *ANSYS CFX-Solver Theory Guide*. Version Release 14.5. ANSYS, Inc., 2012.
- [47] M. Potter, D. Wiggert. *Mechanics of Fluids*. 3rd. Prentice Hall, Inc., 2002.
- [48] D. Wilcox, “Multiscale Model for Turbulent Flows”, *AIAA J.* 26, 1311-1320, 1988.
- [49] D. Wilcox, *Turbulence Modeling for CFD*. La Cañada, CA: DCW Industries, 1993.
- [50] F. Menter. “Zonal Two Equation k- ω Turbulence Models for Aerodynamic Flows”. In: 24th Fluid Dynamics Conference. (July 6, 1993). 93-2906. AIAA. Orlando, FL.
- [51] D. Zill, M. Cullen. *Advanced Engineering Mathematics*. 3rd. Jones and Bartlett Publishers, LLC, 2006.
- [52] *ANSYS Meshing User’s Guide*. Version Release 14.5. ANSYS, Inc. 2012.
- [53] J. Pohl. “Development and Setup of a Closed Loop Test Stand for a Woven Wheel Axial Compressor”. MS. RWTH Aachen University, 2016.
- [54] S. Dixon. *Fluid Mechanics, Thermodynamics of Turbomachinery*. 5th. Butterworth-Heinemann, 2005.
- [55] R. L. Simpson. “Turbulent Boundary Layer Separation”, *Ann. Rev. Fluid Mech.* 21, 205-34, 1989.
- [56] P. K. Kundu, I. M. Cohen. *Fluid Mechanics*. 4th, Elsevier Inc., 2008.
- [57] W. M. Benson. “Tests of a Contra-Propeller for Aircraft”. NACA Tech Note 677, Nov, 1938
- [58] F. W. Lancaster. “Contra-Props”, *Flight*, pp. 418-419, December 11, 1941.
- [59] C. W. McCutchen. “A Theorem on Swirl Loss in Propeller Wakes”, *J. of Aircraft*, Vol. 22, No. 4, pp. 609-617, 1985.
- [60] W. C. Strack, G. Knip, A. L. Weisbrich, J. Godston, E. Bradley. “Technology and Benefits of Aircraft Counter-Rotation Propellers”. NASA Technical Memo 82983, 1990.

- [61] T. J. Sullivan, "Aerodynamic Performance of a Scale-Model Counter Rotating Unducted Fan". *ASME J. of Turbomachinery*, 112 (4), pp. 579-586, 1990.
- [62] J. S. Vanderover, K. D. Visser. "Analysis of a Contra-Rotating Propeller Driven Transport Aircraft", AIAA 2006.
- [63] C. S. Mistry, A. M. Pradeep, "Effect of Speed Ratio and Axial Spacing Variations on the Performance of a High Aspect Ratio, Low Speed Contra-Rotating Fan", GT2012 Proceedings of ASME Turbo Expo, June 2012.
- [64] X. Mao, B. Liu, "A Numerical Study on the Unsteady Effect of Axial Spacing on the Performance in a Contra-Rotating Axial Compressor". *J. of Mechanical Engineering Science*. Proc. IMechE 1-12, 2016.
- [65] J. Wang, F. Ravelet, F. Bakir, "Performance Gains Provided by the Use of a Counter-Rotating Axial-Flow Fan with Respect to a Conventional Rotor-Stator Stage". *J. Energy Power Sources*, Vol. 2, No. 4, pp.164-170, 2015.
- [66] P. B. Sharma, K. Chaudhry, "A Study of Aeroacoustic Performance of a Contra-Rotating Axial Flow Compressor Stage". *Def Sci J*. Vol. 41, No. 2, pp.165-180, 1991.
- [67] A. A. Merchant, "Design and Analysis of Axial Aspirated Compressor Stages", Doctoral Thesis, Massachusetts Institute of Technology, 1999.
- [68] J. H. Freedman, "Design of a Multi-Spool, High-Speed, Counter-Rotating, Aspirated Compressor". M.S. Thesis, Massachusetts Institute of Technology, 2000.
- [69] B. Roy, L. Agrawal, "Casing Boundary Layer Control by Recess Vaned Casing for a Twin-Rotor Contra-Rotating Axial Flow Fan Unit". 94-GT-478, ASME International Gas Turbine and Aeroengine Congress and Exposition, The Hague, Netherlands, June 13-16, 1994.
- [70] A. Alexiou, I. Roumeliotis, N. Aretakis, A. Tsalavoutas, K. Mathioudakis, "Modeling Contra-Rotating Turbomachinery Components for Engine Performance Simulations: The Geared Turbofan with Contra-Rotating Core Case". *J. Engineering for Gas Turbines and Power*. Vol. 134/111701, 2012.
- [71] R. Rzadkowski, M. Drewczynski, "Coupling of Vibration of Several Bladed Discs on a Shaft Part II. Forced Vibration Analysis". *Advances in Vibration Engineering*, Vol. 9, No. 4, October 2010.
- [72] N. A. Cumpsty. *Compressor Aerodynamics*. Krieger Publishing Company, 2004.
- [73] J. H. Horlock. *Axial Flow Compressors: Fluid Mechanics and Thermodynamics*. Robert E. Krieger Publishing Company, 1973