

Development of a Computational Fluid Dynamics Model for a High-Speed Centrifugal
Compressor

Thesis

Presented in Partial Fulfillment of the Requirements for the Degree Bachelor of Science in
Aerospace Engineering with Honors Research Distinction at The Ohio State University

By

Tammy Nguyen-Huynh

Undergraduate Program in Aerospace Engineering

The Ohio State University

2023

Thesis Committee

Randall Mathison, Advisor,
The Ohio State University

Kiran D'Souza,
The Ohio State University

Herbert Harrison,
NASA Glenn Research Center

Abstract

Computational fluid dynamics (CFD) has become widely used in the design and analysis of turbomachinery components such as centrifugal compressors. However, CFD is only a limited representation of experimental cases and struggles to model complex flows or can lack small model details to increase computation speed. To make advancements in compressor technology, designers need tools like CFD that can help them predict flow behavior in new designs. The High Efficiency Centrifugal Compressor (HECC) was designed by United Technologies Research Center (UTRC) for NASA to investigate the difficulties behind improving centrifugal compressor technology and to provide an open case to the turbomachinery research community. CFD analysis was completed by UTRC to aid in the design process. The pre-test study significantly overpredicted the compressor's performance, and while post-test analysis was more representative, its results have yet to be publicly available. CFD technology has also since improved in recent years. In this study, a high-fidelity computational model for the vaneless configuration of HECC operating at its design point was developed and compared to an equivalent experimental case. This model incorporated GPU versions of mesh generation and solver codes from AeroDynamic Solutions, Inc. The development of such a model is significant since there has yet to be a published numerical simulation of HECC's vaneless configuration. Additionally, the use of GPU codes decreases computation times for the model, which has allowed for the inclusion of impeller blade fillets. Initial results showed a general overprediction

of compressor performance by the model. Predictions were more accurate for the impeller compared to predictions for the full compressor stage. Spanwise analyses showed that the model tended to follow flow behavior patterns exhibited in the experiment. Beyond this study, this model will be iterated upon and experimentally validated to further examine HECC performance.

Dedication

*To John, Bryan, Louis, and Brenton,
and to all my other friends in Houston –
thank you for the encouragement to explore,
for this project wouldn't be possible without you.*

Acknowledgments

This project has been one of the most adventurous opportunities in my undergraduate career that I was so fortunate to have, and I wouldn't have been able to get this far without the support of so many. I am very lucky to have you all.

My advisor, Randall Mathison, has spent so many hours guiding me throughout this past year. He agreed to be my research advisor 20 minutes after I sent him an email asking about the possibility. Sometimes, I fear that I've been more trouble than he signed up for, but he's always shown me his dedication to my success. Not only has he provided me with insight for this work, but he has also given me a ton of advice as I navigated graduate school applications. He has opened his office to listen to me whenever I needed to rant about my struggles as a student or my insecurities about being an effective engineer. I've stayed afloat all throughout my senior year thanks to him.

I also would like to thank my thesis committee member, Kiran D'Souza, for the time he has spent reviewing my work and for his patience and flexibility as my schedule became unexpectedly chaotic as I wrapped up this thesis.

This thesis is based upon my work in NASA's Pathways Intern Employment Program at NASA Glenn Research Center. Much direction was provided by my mentor, Trey Harrison. I began working with Trey as a committed spacecraft GNC nerd, having never touched CFD or turbomachinery in my life (and honestly, being quite afraid of it). I've never seen him exhausted by my many questions as I've made progress with learning about everything turbomachinery related. He's provided me with much needed encouragement and advice on graduate school and my career, often lending from his own personal experiences. I couldn't have asked for a better mentor – you truly are a Guiding Star.

I wanted to give a big shout out to the rest of the Turbomachinery and Turboelectric Systems Branch as well for welcoming me to be part of your community. I especially am appreciative of my supervisor, Mark Celestina, who has been a huge advocate of me bringing this project to OSU. Special thanks to Tim Beach who assisted me as I stumbled through Paraview while post-processing.

This research is funded by the Transformational Tools and Technologies Project from the Transformative Aeronautics Concepts Program within NASA's Aeronautics Research Mission Directorate. Their support is greatly appreciated. Thank you to project manager Nick Georgiadis, who has not only been supportive of my work but has also shared with me some of his expansive CFD knowledge.

Another thanks to Christina Krupa, Pathways Internship coordinator at NASA Glenn, who sorted out many details for me regarding the continuation of my Pathways internship while I was away at school.

Additionally, I am grateful for my colleagues at AeroDynamic Solutions, Inc., especially Gregorio Robles Vega and Michael Ni. I am very appreciative of their patience as I learned how to work with CFD by using their codes. They have spent countless hours guiding me through the software installation process and teaching me how to use their tools. Their troubleshooting efforts have been key behind my progress. They also assisted me by providing several of the meshes used in the grid convergence study.

I view this project as the culmination of my time as an undergraduate student at OSU. I am particularly thankful for the many friends I've had in the past five years who have provided me with company and plenty of encouragement. Thank you for the car rides, for laughing at my ridiculous memes, and for putting up with my busy schedule. A special shout out to Adithya Ramaswami, who went on the research distinction journey alongside me with his own amazing project, as well as Sam Yoder, who provided me with inspiration as I went about my own project.

I'm especially grateful for my parents and my family, who have given me everything I need to be successful. Thank you for letting me drag you to three different aerospace museums multiple times each. I promise we can visit the American History museum instead of the Air and Space one next time.

Finally, I'd like to acknowledge everyone, many of whom are listed above, who have supported me while I fought to finish this thesis during the most unfortunately eventful month in my undergraduate career. Turns out, being young does not mean that I am invincible to stress. There were some days where I was so sick that I spent many hours in bed, and others where I had to rush home to support my grieving family. It became frustrating when it felt like life continued without me, but I've been comforted by many of your personal anecdotes and stories. Thank you for showing me what it means to be human.

Table of Contents

Abstract	ii
Dedication	iv
Acknowledgments.....	v
List of Tables	x
List of Figures	xi
1. Introduction.....	1
1.1 Motivation.....	1
1.2 The High Efficiency Centrifugal Compressor	4
1.3 Research Objectives.....	6
2. Experimental Facility and Numerical Model Development	9
2.1 Experimental Facility.....	9
2.2 Mesh Development	14
2.2.1 Impeller Mesh Development.....	14
2.2.2 Duct Mesh Development	19
2.3 Model Initialization.....	21
2.4 Grid Convergence Study	22
3. Numerical Model Comparison.....	26
3.1 Post-Processing Method for Overall Performance Assessment.....	26
3.2 Stage Exit Comparisons.....	27
3.2.1 Overall Performance	27
3.2.2 Spanwise Analysis: Stage Total Pressure Ratio.....	28
3.2.3 Spanwise Analysis: Stage Total Temperature Rise Ratio.....	29
3.2.4 Spanwise Analysis: Stage Adiabatic Efficiency	31
3.3 Impeller Exit Comparisons	32
3.3.1 Overall Performance	32

3.3.2 Spanwise Analysis: Impeller Total Pressure Ratio	32
3.3.3 Spanwise Analysis: Impeller Total Temperature Rise Ratio	34
3.3.4 Spanwise Analysis: Impeller Adiabatic Efficiency	36
4. Conclusions and Future Work	37
4.1 Conclusions	37
4.2 Future Work	38
Bibliography	40
Appendix A. Spanwise Analysis Plots.....	42
Appendix B. Full Passage Contours	46

List of Tables

Table 1: Circumferential locations of impeller exit probes	12
Table 2: Circumferential locations of stage exit probes	12
Table 3: Grid counts for each grid resolution	22
Table 4: Comparisons between model and experiment for performance prediction at stage exit	28
Table 5: Comparisons between model and experiment for performance prediction at impeller exit	32

List of Figures

Figure 1: Centrifugal Compressor with vaned diffuser [3].....	2
Figure 2: HECC vaned configuration	4
Figure 3: Pre-test CFD results compared to experimental data for 100% design speed [3]	5
Figure 4: Post-test CFD results compared to experimental data for various design speeds [3]	5
Figure 5: HECC vaneless configuration	7
Figure 6: Small Engine Components Compressor Test Facility at NASA Glenn, where HECC is tested [8].....	9
Figure 7: Meridional view of HECC flow path with station numbers [8]	10
Figure 8: Forward-looking-Aft view of HECC impeller with cylindrical coordinate frame overlaid	11
Figure 9: Meridional view of HECC with impeller and stage exit probes denoted.....	13
Figure 10: Meridional view of HECC zoomed into diffuser with location of impeller and stage exit probes denoted	14
Figure 11: HECC impeller geometry with main blade colored pink (left), compared to extracted main blade profiles (right)	15
Figure 12: Meridional view of HECC hub at impeller comparing hot and cold coordinates	16
Figure 13: Computational domain for impeller	17
Figure 14: Comparison between surface mesh excluding (left) and including (right) fillets	18
Figure 15: Mesh distortion at main blade hub trailing edge	19
Figure 16: Full stage computational domain	20
Figure 17: Inputs and outputs of CFD model	21
Figure 18: Grid convergence study results for stage total pressure ratio.....	23
Figure 19: Grid convergence study results of stage total temperature rise ratio	24
Figure 20: Grid convergence study results for adiabatic efficiency	25
Figure 21: Stage TPR comparison and total pressure contour.....	28
Figure 22: Stage TTR comparison and total temperature contour.....	29
Figure 23: Full stage exit total temperature distribution.....	31
Figure 24: Stage exit adiabatic efficiency comparison	31
Figure 25: Impeller TPR comparison and full passage total pressure contour	33
Figure 26: Impeller TPR comparison with survey data	34
Figure 27: Impeller exit TTR comparison and full passage total temperature contour	35
Figure 28: Impeller exit adiabatic efficiency comparison	36
Figure 29: Per rake spanwise comparison for stage TPR	42
Figure 30: Per rake spanwise comparison for stage TTR.....	43
Figure 31: Per rake spanwise comparison for impeller TPR	44
Figure 32: Per rake spanwise comparison for impeller TTR.....	45

Figure 33: Full passage contour for stage exit total pressure.....	46
Figure 34: Full passage contour for stage exit total temperature.....	47
Figure 35: Full passage contour for impeller exit total pressure	48
Figure 36: Full passage contour for impeller exit total temperature.....	49

1. Introduction

1.1 Motivation

Aeroengine centrifugal compressors are usually found in rotorcraft and small-sized aircraft. Compared to axial compressors, they are known to have higher stage total pressure ratios due to the centrifugal force imparted onto the flow as it is being pressurized [1]. However, axial compressors usually have higher stage efficiencies than centrifugal compressors due to effects from intense secondary flows and flow paths of longer lengths [2]. For this reason, axial compressors are often preferred over centrifugal compressors in aircraft engines to increase the fuel efficiency of aircraft. The disadvantage to using axial compressors is that they require multiple stages to compress air by the same magnitude as a single-stage centrifugal compressor, consuming more physical space in an aircraft configuration. This creates a strong case for further developing centrifugal compressor technology so that these compressors are more efficient.

Centrifugal compressors are characterized by the shape of the rotating component, which is known as the impeller. The blades on the impeller have a variable radius along the axial direction. A complete centrifugal compressor stage includes the stationary component, which is called a diffuser. Diffusers can either be vaned, meaning the diffuser contains airfoils, or vaneless [1]. A configuration of a centrifugal compressor with a vaned diffuser, NASA's CC3 compressor, can be seen in Figure 1.

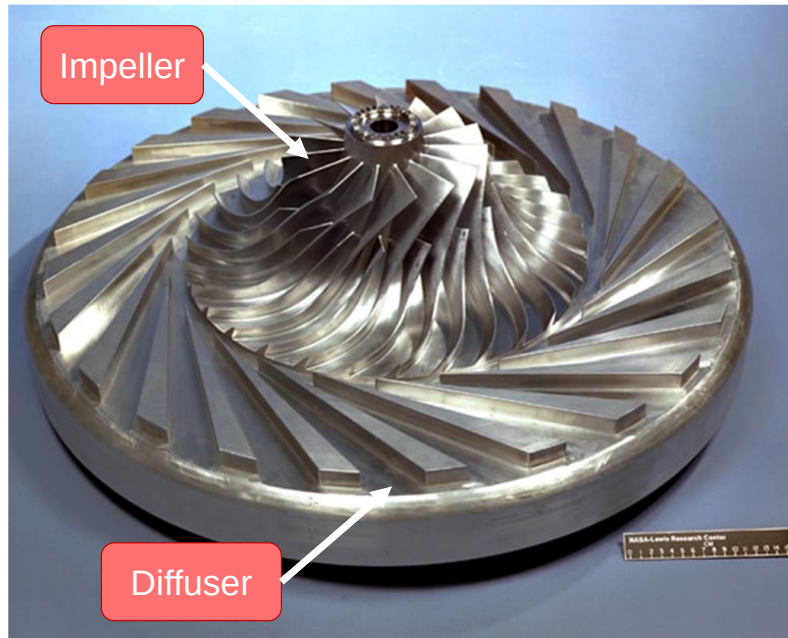


Figure 1: Centrifugal Compressor with vaned diffuser [3]

The use of Computational Fluid Dynamics (CFD) as a design and analysis tool in turbomachinery has become more common over recent decades. CFD is often utilized to simulate fluid flows for cases where fluid behavior is difficult to predict; in the context of turbomachinery, these would include the behavior of components such as centrifugal compressors. A struggle that stems from designing compressors is the difficulty behind predicting the behavior of new designs. Designers tend to use geometries and configurations that are proven to be reliable because of this. In order to develop new designs, there is a need for design tools such as CFD that can predict flow behavior. When the time for testing comes, another issue that arises is the limitations of collecting data. Sensors and probes can only collect data on flow conditions at their location in the flow passage, so the use of CFD allows for insight into the flow behavior at points in the flow passage between sensors and probes.

However, CFD is only a limited representation of real-world compressor behavior. One of these limits is the methodology used to model complex flows. Existing turbulence models have trouble modeling flows with curved streamlines, rotation, and adverse pressure gradients, which are all prevalent in compressor aerodynamics [3]. Another limitation of CFD is its computational speed. Cases may take from several hours to several weeks to converge, which can be highly inconvenient for designers who iterate through compressor designs at a rapid pace. To make a case converge faster, physical features such as surface roughness or heat transfer are sometimes neglected to simplify the model [4]. These simplifications, however, make the model a less accurate representation of the real-world case.

Previous research on centrifugal compressor numerical simulations have assessed prediction abilities for general performance, flow behavior, and specific flow phenomena, such as compressor surge and stall. These predictions are often experimentally validated as part of this research. These CFD models vary in the turbulence models used and the level of physical details included in them [4,5]. Such studies show that a ‘correct’ CFD model highly depends on a compressor’s geometry and highlight the need for continually studying the capabilities of CFD as its technologies improve.

There has been a shift towards development of smaller-sized centrifugal compressors in recent years with their potential for reducing the size of aeroengines [3]. The challenge with reducing compressor size is developing a design that performs the same as a larger compressor. For this effort to be successful, consideration needs to be made for the changes in flow behavior that come with decreasing the compressor’s size. CFD can be used as a design tool to help predict flow behavior, but limited representations of compressor geometry and complex flows

adds uncertainty to these predictions [3]. Greater understanding of the limitations of CFD will allow the turbomachinery community to determine which level of uncertainty in CFD modeling is acceptable. Insight to these limitations will additionally lead to advances in CFD that will help create more accurate representations of flow behavior, making the design and analysis processes of centrifugal compressors easier and more reliable.

1.2 The High Efficiency Centrifugal Compressor

The High Efficiency Centrifugal Compressor (HECC) was developed by United Technologies Research Center (UTRC) in response to NASA's need for a smaller, more efficient successor to their previous centrifugal compressor. The HECC compressor stage consists of an impeller with 15 main-splitter blade pairs, a vaned diffuser with 20 main-splitter blade pairs, and 60 exit guide vanes [6]. HECC has vaned and vaneless configurations. Figure 2 shows the vaned configuration.

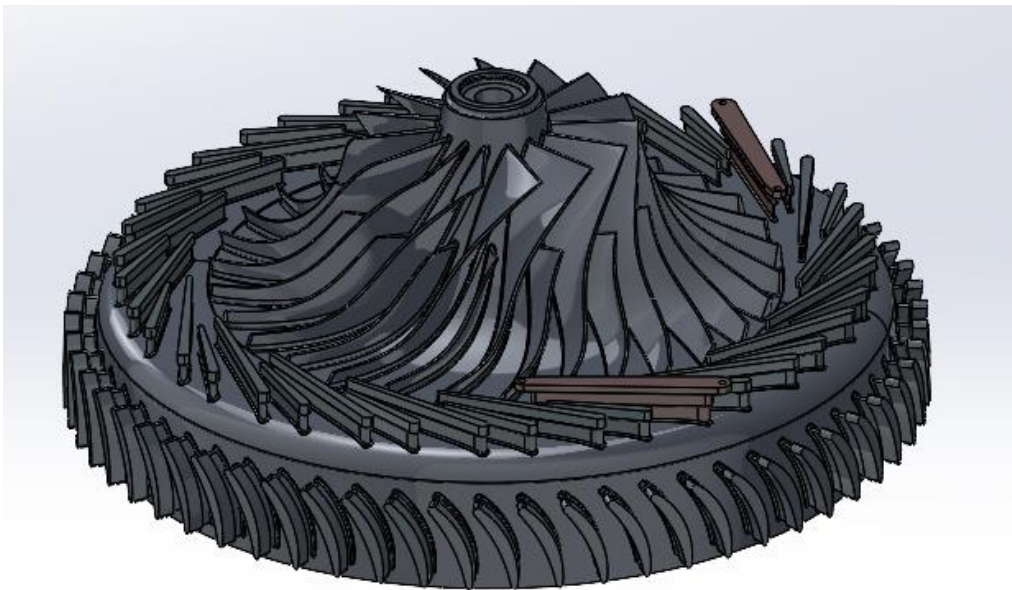


Figure 2: HECC vaned configuration

During the development of HECC, UTRC predicted the compressor's performance with CFD prior to the collection of experimental data. The pre-test CFD model significantly overpredicted the performance of HECC, an example of which can be seen in Figure 3. A post-test CFD analysis was conducted by UTRC to determine the faults of the pre-test model. Concerns with the pre-test computational model included the use of an inaccurate turbulence model and insufficient studies using unsteady CFD [3]. The results of the post-test CFD were a better reflection of HECC performance, an example of which is pictured in Figure 4.

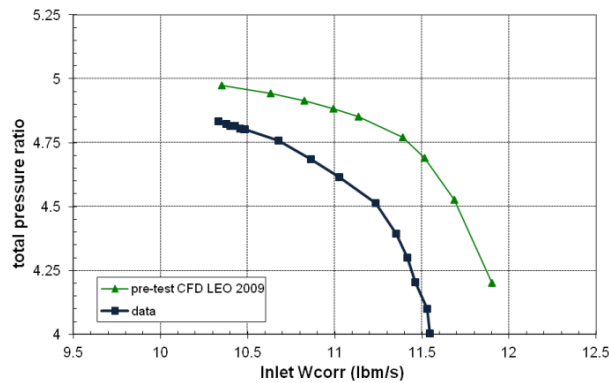


Figure 3: Pre-test CFD results compared to experimental data for 100% design speed [3]

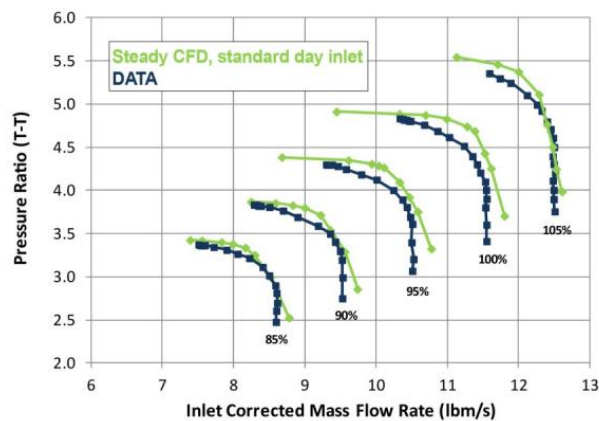


Figure 4: Post-test CFD results compared to experimental data for various design speeds [3]

The overarching goals of the research performed on HECC are to investigate the obstacles to accurate prediction of centrifugal compressor performance and to provide an accessible open data set of the compressor's geometry and performance to the research community. This data can be used to test and validate computational tools, including new or improved CFD models.

Results from UTRC's post-test study currently are not publicly available and computing capabilities have continued to improve since publication of that work. Additionally, there is potential for improving the project's documentation.

An unsteady computational model for this compressor stage was previously developed at NASA Glenn Research Center with TURBO as the solver. The vaned configuration was modeled and included fillets on the vanes of the diffuser. This study found that the presence of fillets reduced the amount of flow separation occurring at the hub and shroud on the main vanes in the simulation. Additionally, performance predictions were better matched to the unsteady model from UTRC's post-test study compared to the TURBO model excluding fillets [6]. Other numerical models utilizing the HECC stage include a study performed by the developers of the TCAE environment. This case study, which was a steady-state model, also used the vaned configuration of HECC but neglected smaller geometric features present on the physical compressor [7].

1.3 Research Objectives

A steady CFD model of the vaneless configuration of HECC was developed using the GPU version of LEO from AeroDynamic Solutions, Inc. (ADS) as the solver. The vaneless configuration is depicted in Figure 5. Codes from ADS were used for this study over the codes in

the model from [6] based on the availability of GPU codes and ease of use. The scope of this study is for cases running at the design speed because that is the most common operating condition for the compressor (usually during cruising flight). Simulation results were then compared to experimental data collected at NASA Glenn Research Center's Small Engine Components Compressor Test Facility.

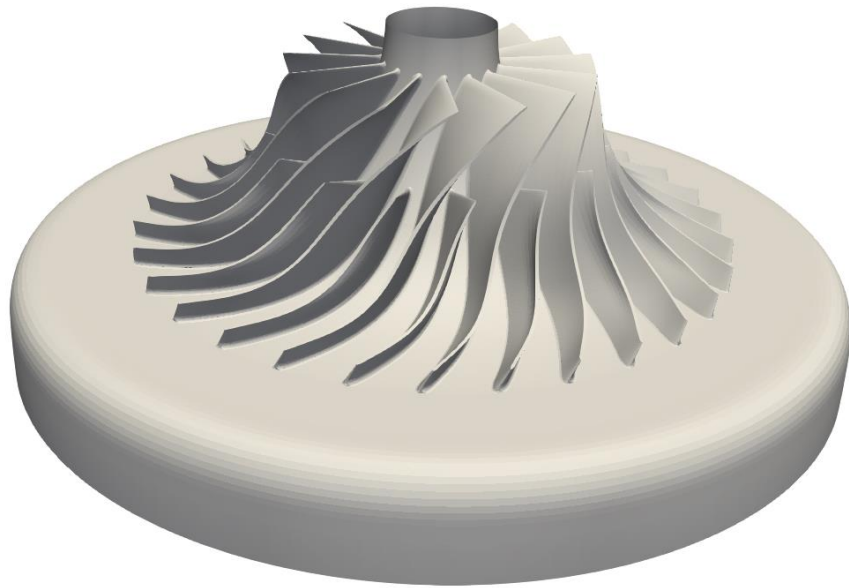


Figure 5: HECC vaneless configuration

A simulation of HECC's vaneless configuration is significant because, at the time of writing, there has yet to be a published study involving a vaneless diffuser. Experimentation with a vaneless diffuser was recommended by UTRC after experimental and post-test CFD studies showed that losses in efficiency were potentially caused by impeller-diffuser interactions potentially prompted by the diffuser vanes. A numerical simulation of the vaneless configuration will contribute to this type of research. Additionally, the use of GPU codes results in

significantly faster computations. This allows for the addition of previously neglected details, such as the fillets present on the impeller blades, to create a higher fidelity model than previously.

This study not only provides a more detailed analysis of HECC's performance but will also give insight to CFD technology available today in the context of turbomachinery applications. In order to use a high-fidelity CFD model for analysis, it needs to be developed and experimentally validated.

2. Experimental Facility and Numerical Model Development

2.1 Experimental Facility

Experiments on HECC are conducted at NASA Glenn Research Center in the Small Engine Components Compressor Test Facility. A diagram of the facility can be seen in Figure 6.

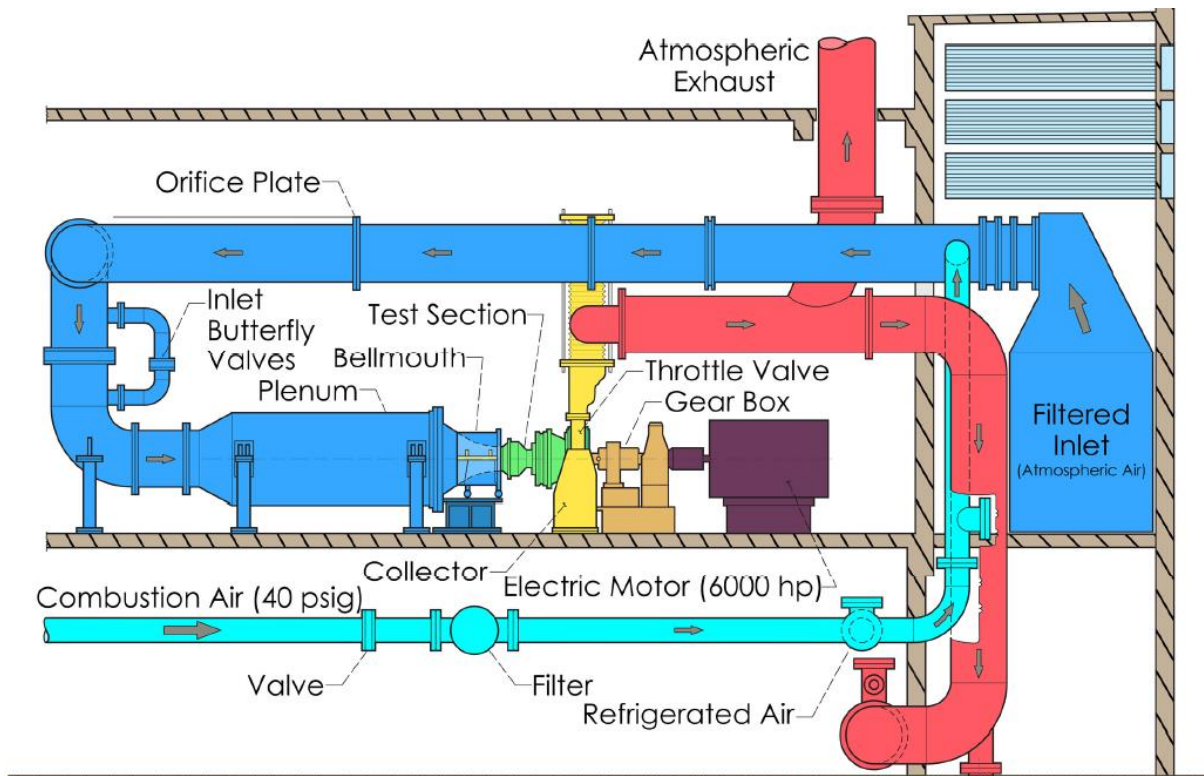


Figure 6: Small Engine Components Compressor Test Facility at NASA Glenn, where HECC is tested [8]

The path to the inlet measuring plane is in blue. Inlet air is drawn from an atmospheric intake. Downstream of the inlet is an orifice plate that measures mass flow rate. Two butterfly valves are used to manage inlet pressure. As flow passes through the plenum, two screens decrease flow velocity and ensure that the flow is uniform once it reaches the inlet.

The compressor is in the test section, colored green. The flow through the test section and compressor can be assumed axisymmetric, as any disturbances that cause asymmetry are located a significant distance downstream. Past the compressor stage exit measuring plane is the collector, in yellow. The collector contains an annulus-shaped throttle valve with discharge ports. The flow is finally exhausted to the atmosphere through the red pipe [8].

Interfaces between compressor components are assigned station numbers, shown in the compressor's meridional view in Figure 7. Stations of interest for this study are the stage inlet, diffuser leading edge (LE)/impeller exit, and the stage exit.

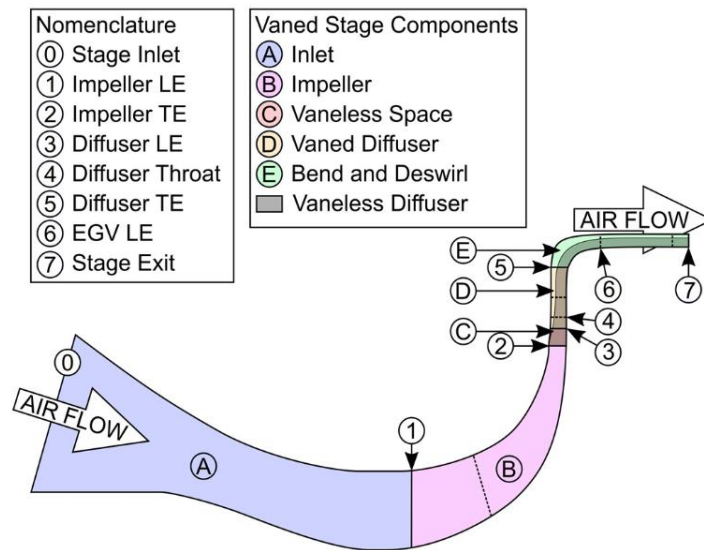


Figure 7: Meridional view of HECC flow path with station numbers [8]

Within the compressor stage, total pressure and temperature are measured using rakes. The rakes of most significance to this study are those located at the stage inlet, impeller exit, and stage exit planes. Inlet total pressure and temperature are recorded in the plenum [8]. At the impeller exit, there are three total temperature rakes and four total pressure rakes, with each rake having five probes. The probes are located at 10%, 30%, 50%, 70%, and 90% span. Table 1 shows the circumferential location of each total pressure and total temperature rake in the impeller exit. It is important to note that while a cylindrical coordinate system is used to explain rake locations, the $\theta = 0^\circ$ axis is at a 12 o'clock position in relation to an analog clock and the $+\theta$ direction is counterclockwise forward-facing-aft, as seen in Figure 8.

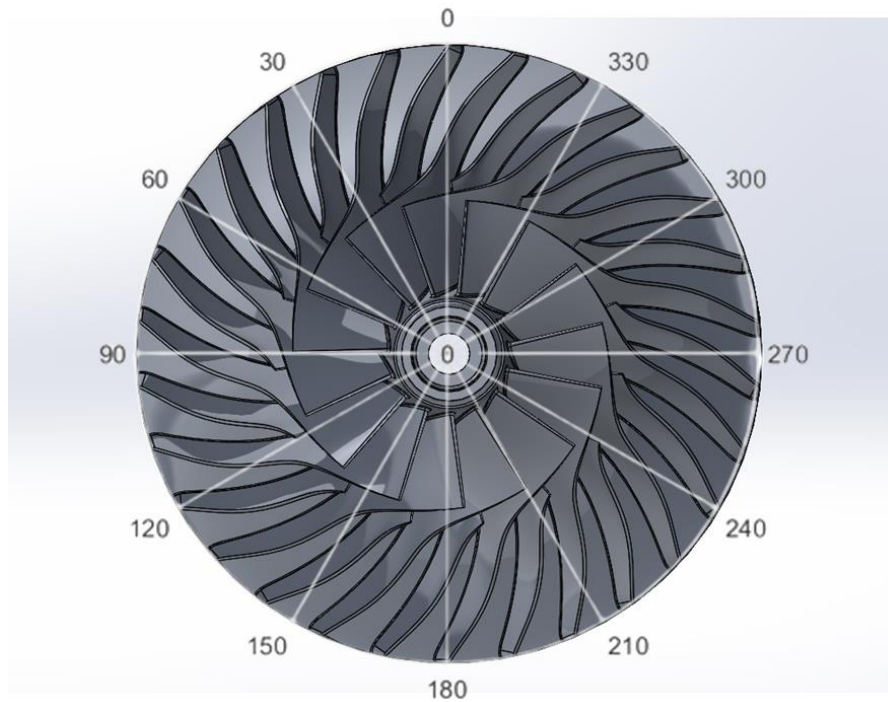


Figure 8: Forward-looking-Aft view of HECC impeller with cylindrical coordinate frame overlaid

Table 1: Circumferential locations of impeller exit probes

Rake	Circumferential Location [degrees]
Impeller Exit – Total Temperature (1)	0
Impeller Exit – Total Temperature (2)	76
Impeller Exit – Total Temperature (3)	226
Impeller Exit – Total Pressure (1)	181
Impeller Exit – Total Pressure (2)	256
Impeller Exit – Modular Vane Total Pressure (1)	36
Impeller Exit – Modular Vane Total Pressure (2)	298

Experimental data at the stage exit was available for two total temperature rakes and three total pressure rakes. These rakes had four probes distributed across the span of the flow passage at 12.64%, 37.81%, 62.80%, and 87.64% span. The circumferential locations for these rakes are shown in Table 2.

Table 2: Circumferential locations of stage exit probes

Rake	Circumferential Location [degrees]
Stage Exit – Total Temperature (1)	108
Stage Exit – Total Temperature (2)	252
Stage Exit – Total Pressure (1)	36
Stage Exit – Total Pressure (2)	171
Stage Exit – Total Pressure (3)	324

Figures 9 and 10 shows the spanwise location of the rakes' probes within the compressor's meridional view. Figure 9 presents the entire meridional view of the compressor with a red box denoting the area of zoom shown in Figure 10. The probes are positioned in the flow passage such that they encompass the span of the passage.

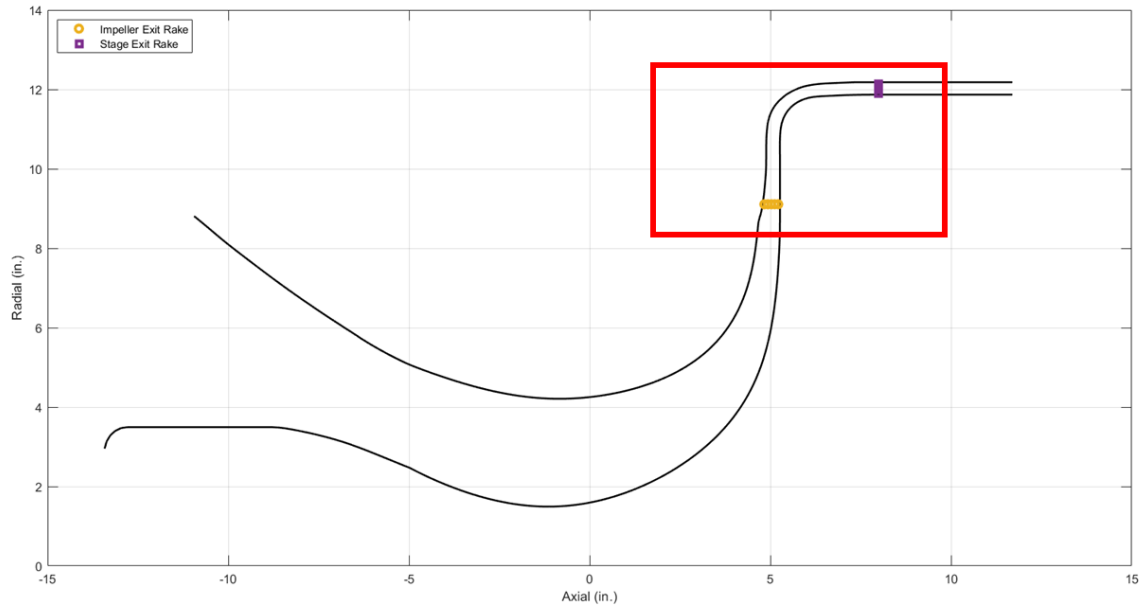


Figure 9: Meridional view of HECC with impeller and stage exit probes denoted

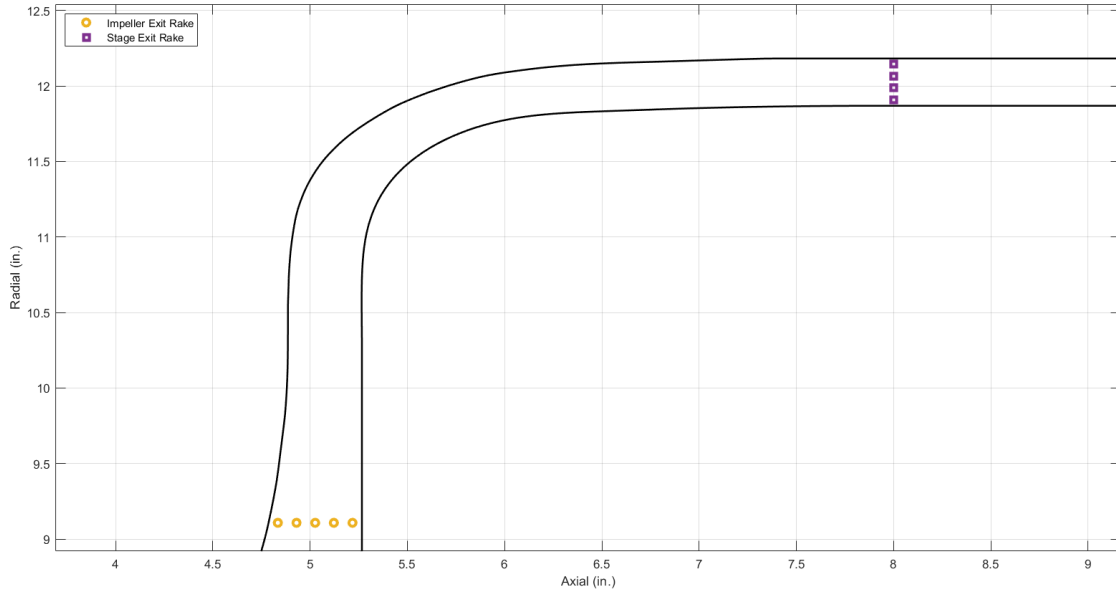


Figure 10: Meridional view of HECC zoomed into diffuser with location of impeller and stage exit probes denoted

Additionally, survey data is collected at the impeller exit station using actuators that cross the flow passage in the spanwise direction [8].

2.2 Mesh Development

2.2.1 Impeller Mesh Development

The impeller mesh consists of a single main and splitter blade pair. A periodic boundary was used in the impeller model to conserve computational resources. The geometry of the impeller mesh was based on the CAD model of HECC. This model was provided by the manufacturer of the compressor, Agilis Engineering. To capture the blade geometry, cuts were made along the span of the blade at 5% span, between 7.5-8% span, and then in increments of 10% starting at 10% span to expose the airfoil profile. Since there is intent for this numerical model to be high-fidelity, three additional cuts were also made along the fillet at the hub to

define the fillet geometry, with the first blade section being at the hub. These profiles are shown in Figure 11, compared to the blade's CAD geometry.

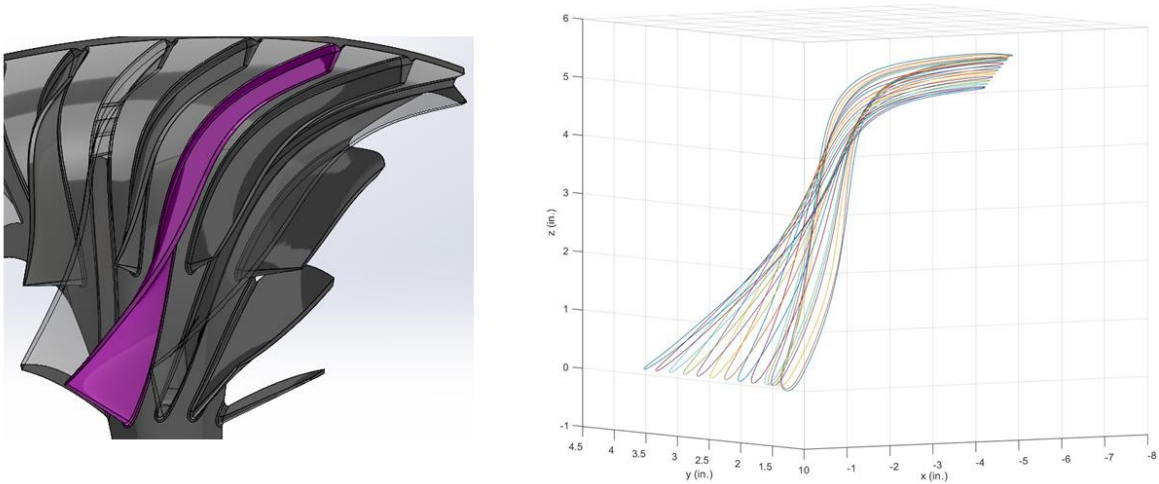


Figure 11: HECC impeller geometry with main blade colored pink (left), compared to extracted main blade profiles (right)

Points were placed along the perimeter of the airfoil, with the concentration of points increased in the leading and trailing edges of the profile to fully define their shape. Each profile was defined by 400 points. The Cartesian coordinates of these points were then recorded and converted into axial-radial coordinate pairs. The CAD model of the stationary impeller, so the profile data pulled from the CAD model were cold coordinates and had to be converted to hot coordinates. These coordinates define the impeller geometry while it is imparting work onto the flow. The endwall data available were hot coordinates, so the blade section coordinates were shifted axially so that the section at the hub overlaid the endwall hub coordinates. The difference

between the cold and hot coordinates are presented in Figure 12. The geometry of the splitter blade was captured in the same manner.

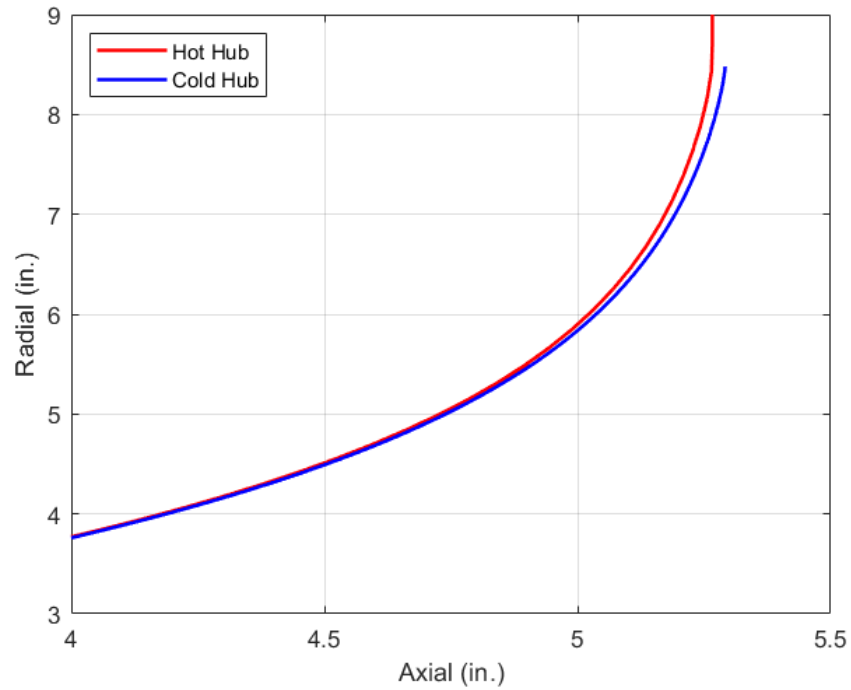
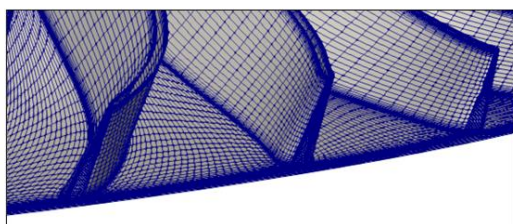


Figure 12: Meridional view of HECC hub at impeller comparing hot and cold coordinates

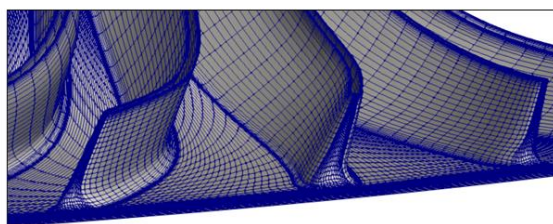
The mesh was generated using the GPU version of code WAND from ADS. Figure 13 shows the resulting impeller mesh. Figure 14 demonstrates the difference in geometry between a surface mesh excluding fillets and a surface mesh including fillets.



Figure 13: Computational domain for impeller



Mesh with no fillets



Mesh with fillets

Figure 14: Comparison between surface mesh excluding (left) and including (right) fillets

Though the general shape of the fillet was generated in the impeller mesh, a distortion can be seen on the fillet at the trailing edge of both main and splitter blades. This is exhibited in Figure 15. The fillet transitions into a flatter surface in this region. After consultation with ADS, it was concluded that the mesh generation code struggled with the abrupt transition. This distortion was not expected to have a significant effect on the simulated flow, however.

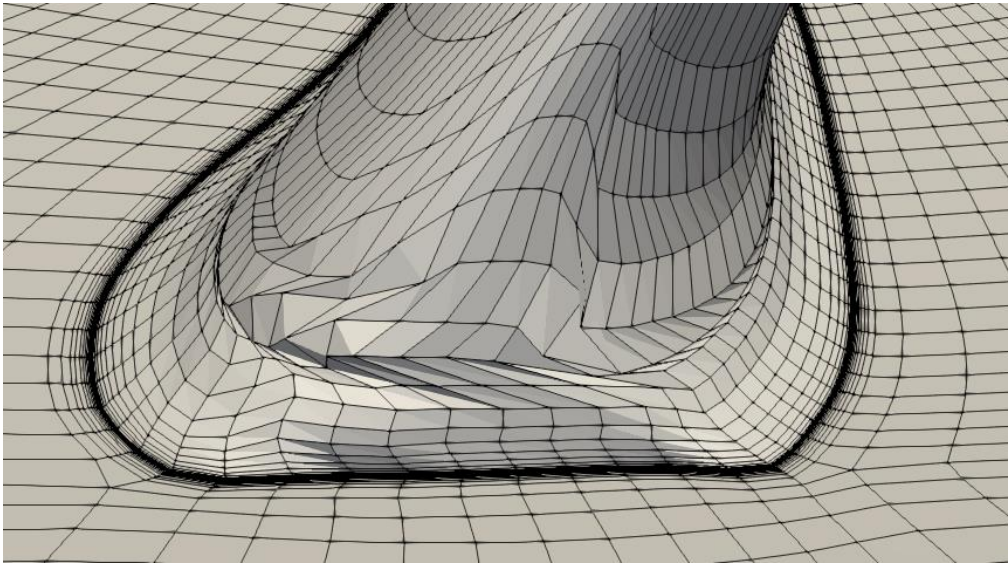


Figure 15: Mesh distortion at main blade hub trailing edge

2.2.2 Duct Mesh Development

The other components of the compressor, the inlet and vaneless diffuser, were meshed using the radial-axial endwall coordinate data. A mesh was also created for an exit duct located after the stage exit measuring plane, even though one did not exist in the experiment. The purpose of the exit duct in the numerical model was to prevent the influence of any boundary effects on flow predictions at the stage exit. Figure 16 presents the full stage mesh.

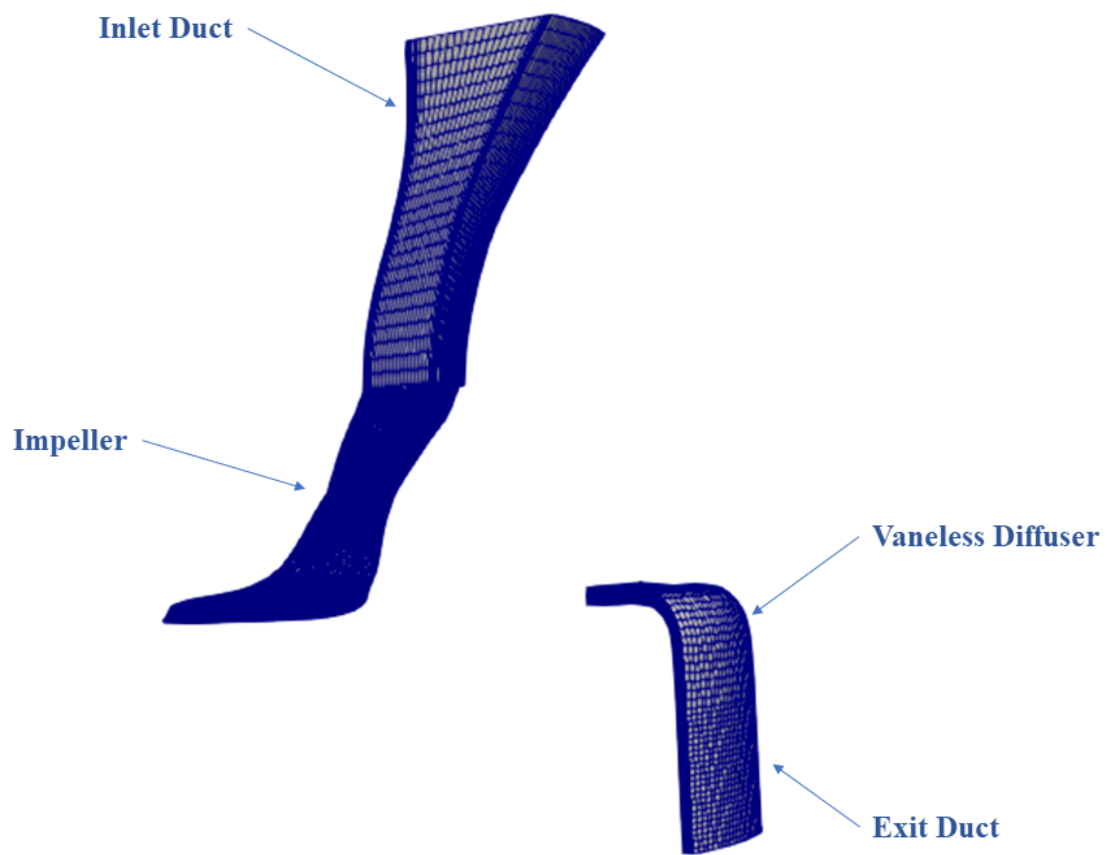


Figure 16: Full stage computational domain

2.3 Model Initialization

Prior to the grid convergence study, the model must be initialized. Figure 17 shows the inputs and outputs of the model.

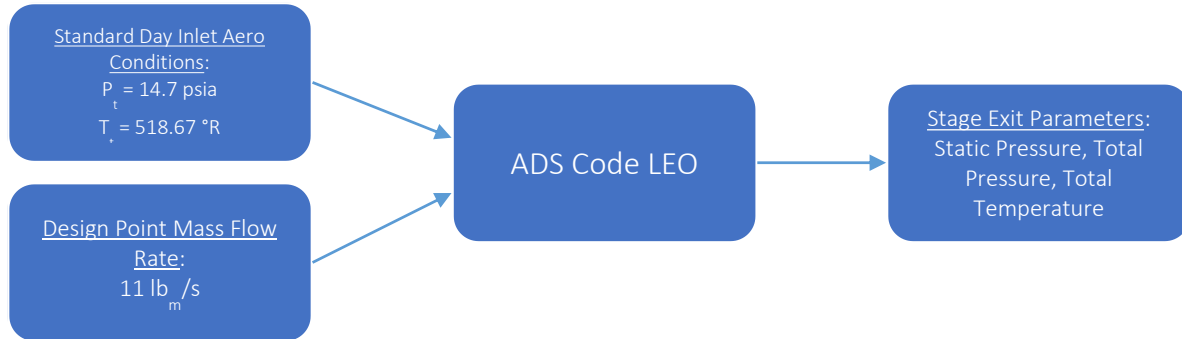


Figure 17: Inputs and outputs of CFD model

Other inputs were inlet flow angles and flow Mach number. Flow angles were defined by three points: one at the hub, one at 50% span, and one at the shroud. Tangential flow angles were set to 0°, while radial flow angles were specified to be approximately the slope of the hub and shroud endwalls and was linearly interpolated between those values at 50% span. The flow Mach number was left as ADS Workbench's default value of 0.1 and was set to be the same for all three points in the inlet.

The selected turbulence model was Wilcox k- ω 98. The tip clearance was set to the design value of 0.012 inches. In the interest of developing a high-fidelity model, γ was varied in computations and sliding planes were used between mesh interfaces. The flow fields for all compressor components were set to rotate at the design rotational speed of 21,789 RPM.

As mentioned previously, the GPU version of ADS' code LEO was used as the solver. The Target Mass Flow tool, included in ADS Workbench, was used to initialize the model.

Given a mass flow value, the stage exit static pressure was iterated until the model achieved the prescribed mass flow.

2.4 Grid Convergence Study

A grid convergence study was conducted to ensure that the solution of a CFD model does not depend on the resolution of the mesh. Five meshes of increasing granularity were used in this study. Table 3 shows the quantity of nodes in each mesh. The original mesh that was developed was labeled as Medium.

Table 3: Grid counts for each grid resolution

Grid Resolution	Grid Points
Coarse	481,255
Medium	770,391
Fine	1,131,203
Very Fine	1,557,955
Finest	2,200,267

The grid convergence study was performed for the 100% corrected rotational speed speedline as this includes the design operating point. The results of this study can be seen in Figures 18-20.

The meshes were able to converge on a solution for stage total pressure ratio (TPR), with the Very Fine and Finest meshes overlaying one another.

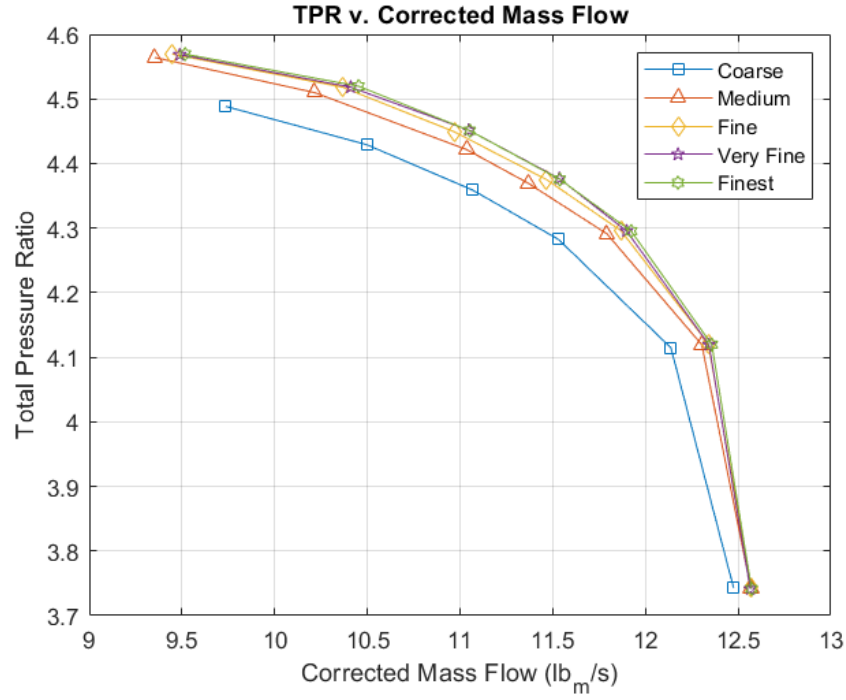


Figure 18: Grid convergence study results for stage total pressure ratio

Stage total temperature rise (TTR) is computed using the following equation:

$$TTR = \frac{T_{t7} - T_{t0}}{T_{t0}} \quad (1)$$

The solution for stage total temperature rise ratio began to converge between the Coarse and Very Fine meshes but diverged between the Very Fine and Finest meshes.

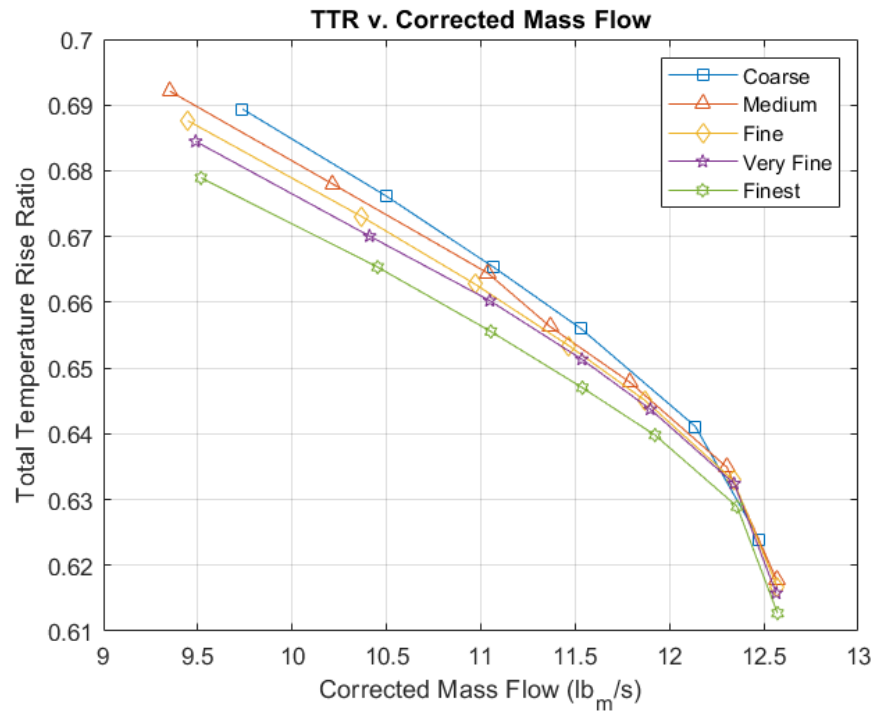


Figure 19: Grid convergence study results of stage total temperature rise ratio

Because adiabatic efficiency is a function of inlet and stage exit total temperature, this phenomenon can also be seen in the study's efficiency results.

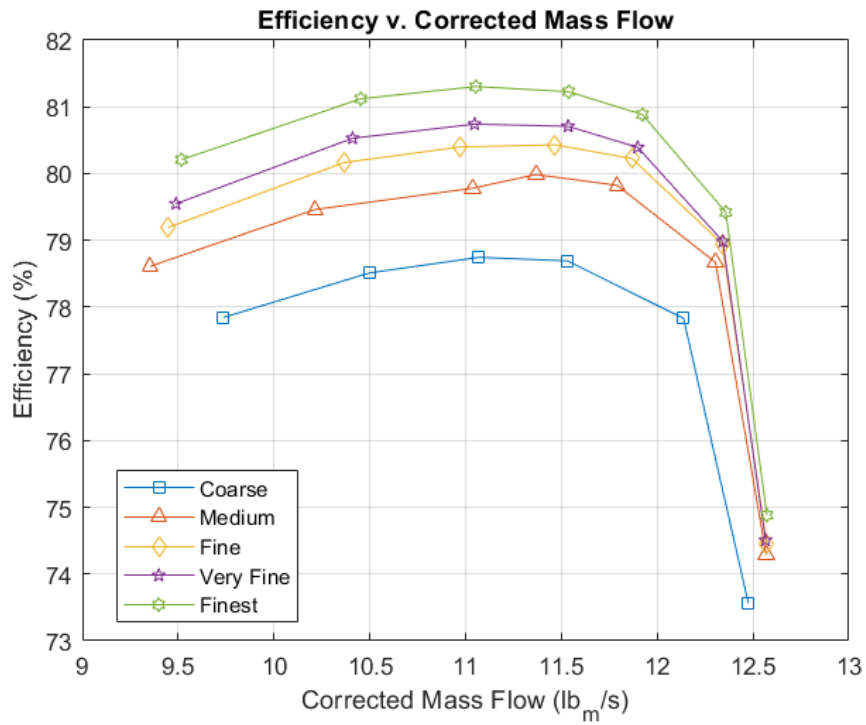


Figure 20: Grid convergence study results for adiabatic efficiency

Since there was a converged solution for stage TPR, the Finest mesh was used for analysis.

However, it is recommended that the grid sensitivity study be continued with one more mesh of a higher resolution to confirm convergence.

3. Numerical Model Comparison

3.1 Post-Processing Method for Overall Performance Assessment

Comparisons were made probe-to-probe between an equivalent experimental case and the CFD model. While flow behavior is predicted for the entire flow passage in the model, information about flow behavior in the experiment is only available where there were probes. The data from the model used in analysis were collected from grid points in a line across the width of the flow passage. A line located at a constant span was used rather than a point to capture any circumferential variation. These lines were located as close as possible to the spanwise location of the physical probes in the experiment. For each measuring plane, this data was first averaged lengthwise, then spanwise. The experimental data was also post-processed with the same method, with the exception of lengthwise averaging since the physical probes are only one point in the flow passage, and with the addition of circumferential averaging to account for circumferential variation recorded in the experiment. Flow parameters of interest for this study are total temperature, total pressure, and adiabatic efficiency, as these three parameters help define the compressor's performance.

To calculate a spanwise average, the weight of each probe was calculated using the following equation, where k is the probe of interest and $Span_k$ is the location in the span of probe k . k is between 1-4 for rakes at the stage exit and between 1-5 for rakes at the impeller exit.

$$Weight_k = \frac{1}{2}(Span_{k+1} + Span_k) - \frac{1}{2}(Span_{k-1} + Span_k) \quad (2)$$

The weighted spanwise average is calculated using Equation 3:

$$\bar{x}_{span} = \sum_{k=1}^{k_{end}} (Weight_k * Probe Value_k) \quad (3)$$

Probe values are from total temperature or total pressure rakes.

Circumferential averages are computed using Equation 4:

$$\bar{x}_{circ} = \frac{\sum_{j=1}^{j_{end}} \bar{x}_{span_j}}{j_{end}} \quad (4)$$

j serves as a counter for the rakes and j_{end} is the quantity of the type of rake of interest. For example, a circumferential average of the total pressure at impeller exit is an arithmetic average of the spanwise averaged values for each total pressure rake located at the impeller exit.

A third parameter of interest is adiabatic efficiency. The calculations for efficiency values utilize real gas and dry air properties from REFPROP [9].

The plots presented in this section compare the model to circumferentially averaged values from the experiment. Per rake comparisons are available in Appendix A. Contour plots from the model are also shown throughout this section. Full passage contours are available in Appendix B.

3.2 Stage Exit Comparisons

3.2.1 Overall Performance

Table 4 presents a comparison of circumferentially averaged TPR, TTR, and adiabatic efficiency values observed at the stage exit for the numerical simulation and the experiment.

Table 4: Comparisons between model and experiment for performance prediction at stage exit

	CFD	Experiment	Percent Error
TPR	4.649	4.540	2.345%
TTR	0.6495	0.6469	0.4019%
Efficiency	0.8424	0.8264	1.936%

Comparisons of overall performance show that TTR may have been best predicted out of the three parameters of interest, but a spanwise analysis is necessary to further investigate the differences between the numerical model and the experiment.

3.2.2 Spanwise Analysis: Stage Total Pressure Ratio

The comparison for stage TPR and model total pressure contour can be seen in Figure 21. The model follows the same trend observed in the experiment but underpredicts the total pressure near the hub and overpredicts for the remaining probes. This overprediction grows with increasing span, with the prediction closest to the shroud having a percent error of 11.46%.

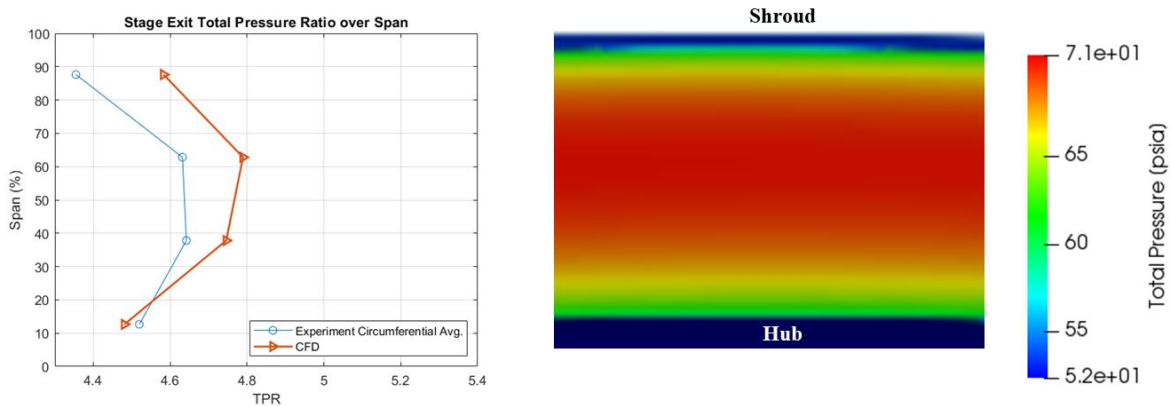


Figure 21: Stage TPR comparison and total pressure contour

The simulated total pressure distribution at the stage exit is circumferentially uniform. Figure 33 in Appendix B shows the contour for the entire passage.

3.2.3 Spanwise Analysis: Stage Total Temperature Rise Ratio

Though there may seem to be good agreement between the CFD model and the experiment for overall TTR, the two cases are in disagreement with the pattern of total temperature variation across the span. The model exhibits increasing stage exit total temperature with increasing span, while the opposite occurs in the experiment, as seen in Figure 22. However, the model makes a better prediction of the magnitude of TTR through the stage compared to the magnitude of stage TPR, with the largest percent error present for stage TTR being 2.654% at the probe closest to the shroud.

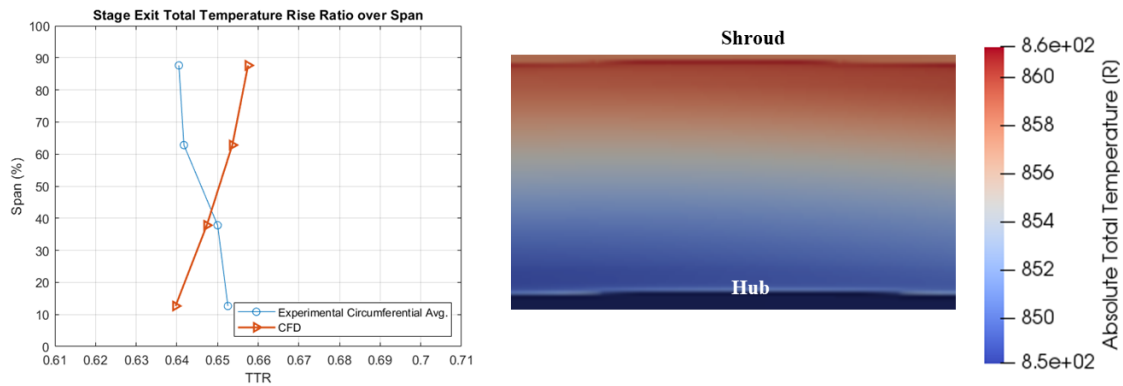


Figure 22: Stage TTR comparison and total temperature contour

Figure 22 also presents the model's total temperature contour at the stage exit. Unlike the total pressure distribution, there is circumferential variation in stage exit total temperature. However, as seen in Figure 23, the total temperature distribution follows the same general pattern of

increasing total temperature with increasing span. The red box denotes the area of the passage displayed in Figure 22.

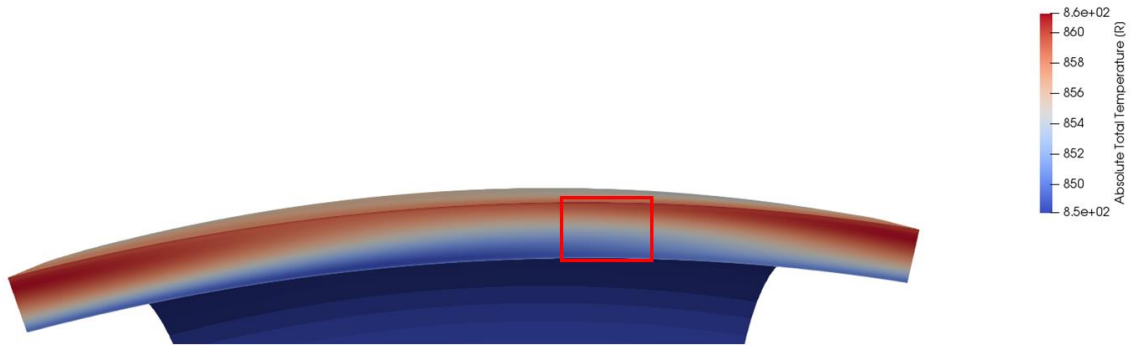


Figure 23: Full stage exit total temperature distribution

3.2.4 Spanwise Analysis: Stage Adiabatic Efficiency

Figure 24 displays the comparison for stage adiabatic efficiency. Efficiency peaks closer to the center (50% span) of the flow passage and is minimized at the endwalls. Though it can be observed that the two cases follow a similar trend, the model consistently overpredicts the

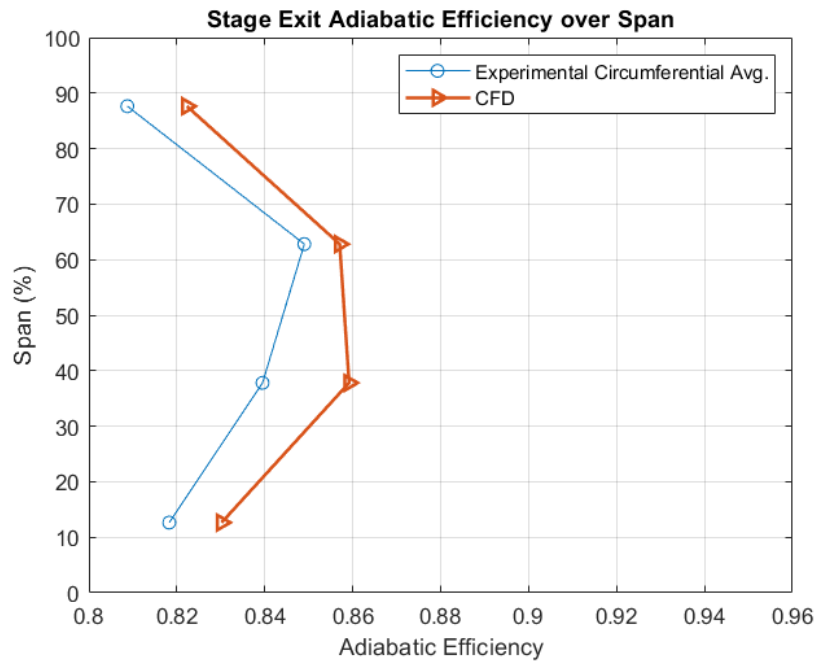


Figure 24: Stage exit adiabatic efficiency comparison

compressor's efficiency. The probe located at 37.8% span has the largest difference between model and experiment by an efficiency of 2%.

3.3 Impeller Exit Comparisons

3.3.1 Overall Performance

Comparisons of overall performance parameters between the model and experiment are presented in Table 5. In terms of overall performance, the model tends to predict the behavior of the compressor at the impeller exit better than at the stage exit, as percent error values at this location are lower.

Table 5: Comparisons between model and experiment for performance prediction at impeller exit

	CFD	Experiment	Percent Error
TPR	5.163	5.117	0.8990%
TTR	0.6569	0.6567	0.03046%
Efficiency	0.9040	0.8946	1.051%

3.3.2 Spanwise Analysis: Impeller Total Pressure Ratio

Figure 25 compares predicted and experimental spanwise impeller TPR. Like at the stage exit, the model and experiment follow similar trends. The model overpredicts impeller exit TPR for the first three probes starting at the hub, with the overprediction decreasing with increasing span. Impeller performance closer to the shroud is underpredicted. This misprediction is also the largest out of all probes, with a percent error of 3.276%.

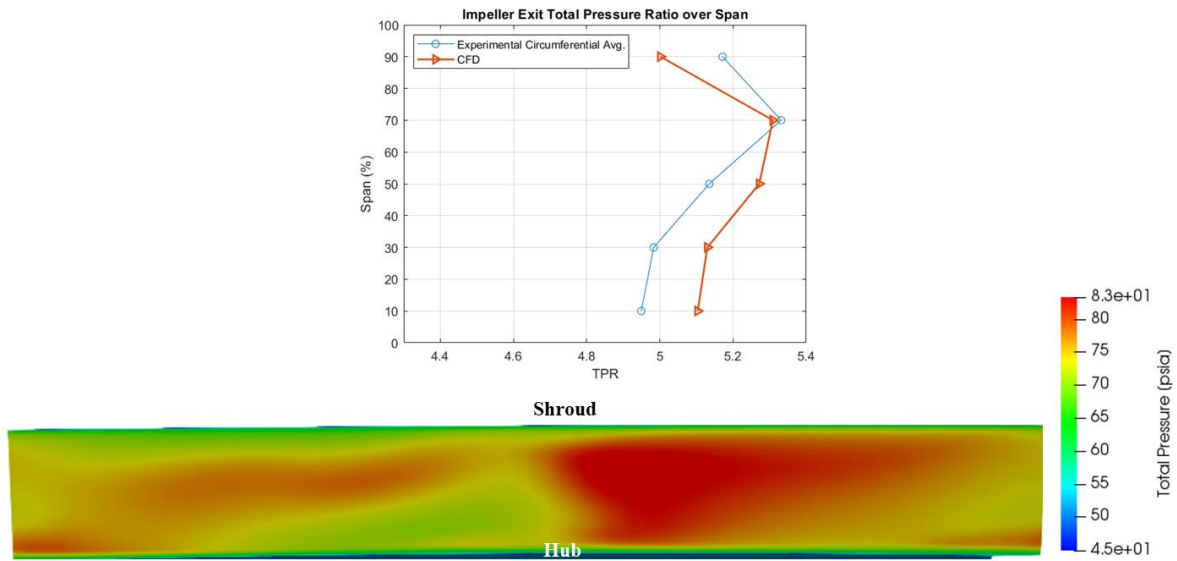


Figure 25: Impeller TPR comparison and full passage total pressure contour

Survey data provides additional detail about flow behavior as it captures data for a larger area of the span. Figure 26 presents survey data for impeller TPR. The largest discrepancy in TPR between the simulation and experiment can be seen at a span of ~80%.

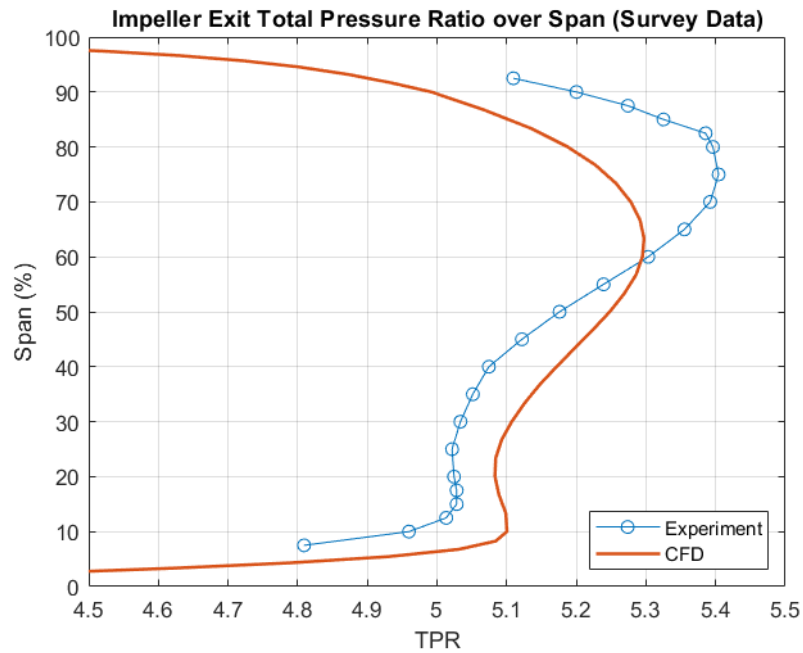


Figure 26: Impeller TPR comparison with survey data

3.3.3 Spanwise Analysis: Impeller Total Temperature Rise Ratio

The model and experiment are in good agreement regarding impeller TTR, as seen in Figure 27. However, the model's prediction differs the most from the experiment at the probe closest to the hub. Between the spanwise TPR and TTR plots, it is evident that the most work is imparted on the flow by the impeller in the top half of the span, as the TPR and TTR are highest in this area.

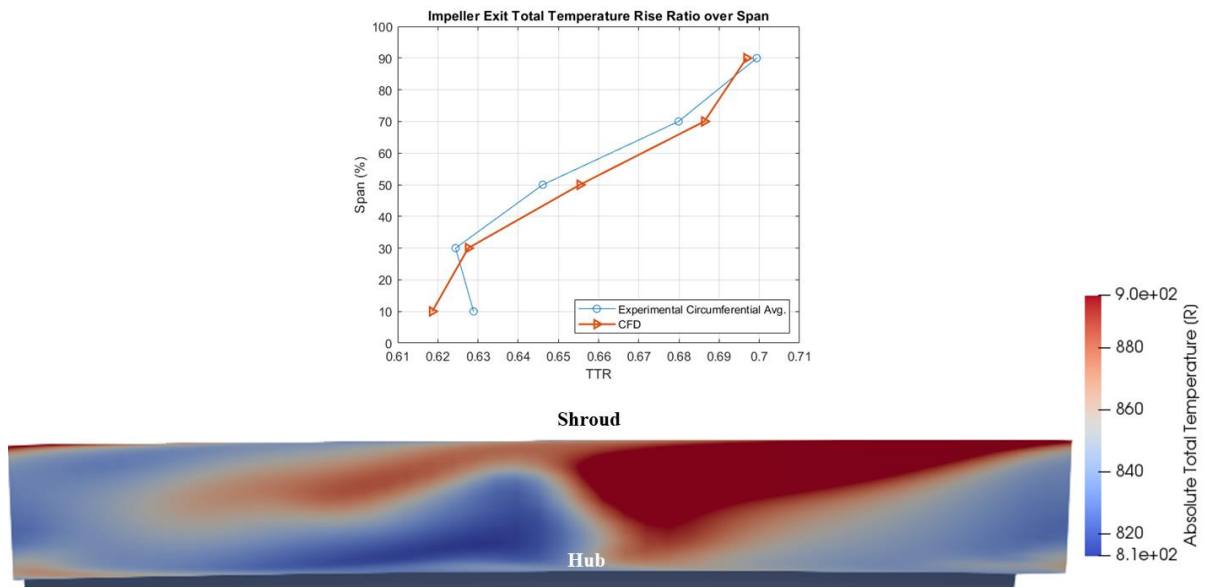


Figure 27: Impeller exit TTR comparison and full passage total temperature contour

3.3.4 Spanwise Analysis: Impeller Adiabatic Efficiency

The adiabatic efficiency across the span of the impeller exit varies more compared to the range of efficiencies in the stage exit, as evident in Figure 28. There is a general pattern of decreasing efficiency with increasing span in both cases. The model significantly overpredicts efficiency near the hub – the difference in efficiency is approximately 4%.

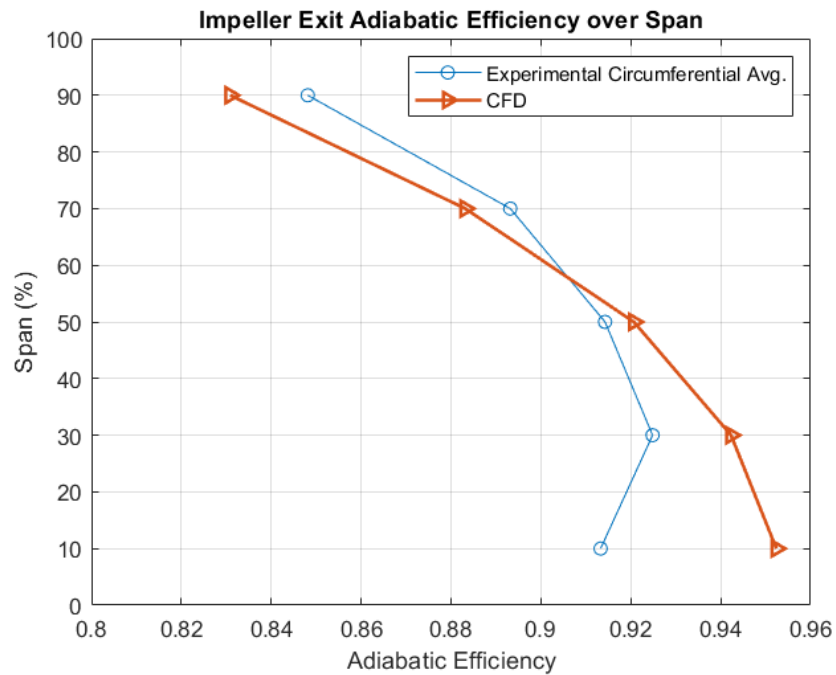


Figure 28: Impeller exit adiabatic efficiency comparison

4. Conclusions and Future Work

4.1 Conclusions

Computational fluid dynamics is a tool used to simulate fluid flows. CFD is used in the context of turbomachinery to aid in the design and analysis of components such as centrifugal compressors. Standing issues with numerical models include its difficulty of simulating complex flows and its slow computational speed. Both problems lead to inaccurate models. The High Efficiency Centrifugal Compressor was developed to yield an open case for the turbomachinery community and to investigate the difficulties that prevent the improvement of centrifugal compressor technology. A CFD model for HECC based at the design operating point was developed as part of this mission.

The high-fidelity mesh was based off the compressor's CAD model and endwall coordinate data. The mesh was generated using the GPU version of AeroDynamic Solutions' code WAND. The model was initialized at the design operating point with the GPU version of ADS' code LEO as the solver. Its inputs included standard day inlet aerodynamic conditions and the design mass flow rate. Additionally, it used a Wilcox $k-\omega$ 98 turbulence model and had sliding planes between mesh interfaces. A grid convergence study was conducted using five meshes of increasing granularity. The study resulted in the convergence of a solution for stage total pressure ratio, and the finest grid in the study was used for comparison to an equivalent experimental case.

Overall, the model overpredicted HECC performance. Predictions were more accurate at the impeller exit compared to the stage exit. Spanwise analyses were also conducted. The model generally followed flow behavior trends seen in the experiment, with the exception of stage exit total temperature rise.

The simulation in its current state contributes to general performance analyses for HECC's vaneless configuration. Valid inferences about performance cannot be made with experimental data alone, as the availability of information on flow behavior is limited to probe locations. Adjustments are still required for the model to be used alongside experimental data for analysis, especially for predictions regarding stage performance. At NASA Glenn, this model of HECC is the first model to be developed completely in-house, which will allow for more thorough analyses. Finally, the results from overall and spanwise comparisons demonstrate the multi-dimensional nature of CFD models. Overall values only define the magnitude of the compressor's performance, and for a simulation to be considered high fidelity, it must accurately replicate the flow behavior through the passage as well.

4.2 Future Work

This model will be further developed to undergo experimental validation. A significant change to the model is the use of the turbulence model of $k-\omega$ SST with a direct normal distance calculation. Another modification is the conversion of cold-to-hot coordinates. The next iteration of the model will likely use a method similar to one described by Holder et. al. in [10]. The vaned configuration will also be modeled for use with any experiments involving the vaned diffuser and exit guide vanes. After this step, it can be used to analyze HECC performance for

upcoming experiments. One such experiment is the assessment of performance with varying blade tip clearance.

During the process of developing the present CFD model, it was discovered that the impeller geometry given by Medic et. al. in [3] was inconsistent with the CAD model provided by the manufacturer. An investigation is ongoing to determine the effects of these differences on computational models.

Ultimately, the goal of developing a CFD model of HECC is to provide a greater understanding of the compressor's performance and to give some insight to the technology available today in computational tools for turbomachinery. This can be achieved by pairing results from the validated model with performance and geometry data for the compressor to be easily accessible by the greater turbomachinery community.

Bibliography

- [1] Cumpsty, N. A., 2004, *Compressor Aerodynamics*, Krieger Publishing Company.
- [2] Kunte, R., Schwarz, P., Wilkosz, B., Jeschke, P., and Smythe, C., 2013, “Experimental and Numerical Investigation of Tip Clearance and Bleed Effects in a Centrifugal Compressor Stage With Pipe Diffuser,” *Journal of Turbomachinery*, **135**(1), p. 011005.
- [3] Medic, G., Sharma, O. P., Jongwook, J., Hardin, L. W., McCormick, D. C., Cousins, W. T., Lurie, E. A., Shabbir, A., Holley, B. M., and Slooten, P. R. V., 2017, “High Efficiency Centrifugal Compressor for Rotorcraft Applications,” p. 332.
- [4] Gooding, W. J., Meier, M. A., and Key, N. L., 2021, “The Impact of Various Modeling Decisions on Flow Field Predictions in a Centrifugal Compressor,” *Journal of Turbomachinery*, **143**(10), p. 101006.
- [5] Miura, T., Yamashita, H., Takeuchi, R., and Sakai, N., 2021, “Numerical and Experimental Study on Rotating Stall in Industrial Centrifugal Compressor,” *Journal of Turbomachinery*, **143**(8), p. 081008.
- [6] Kulkarni, S., and Beach, T. A., 2017, “Progress of High Efficiency Centrifugal Compressor Simulations Using TURBO.”
- [7] “Centrifugal Compressor Benchmark HECC Stage,” CFD SUPPORT [Online]. Available: <https://www.cfdsupport.com/centrifugal-compressor-benchmark-hecc-stage.html>.

- [8] Harrison, H. M., McNichols, E. O., and Blaha, M. R., “NASA Small Engine Components Compressor Test Facility: High Efficiency Centrifugal Compressor Vaneless Diffuser and Transition Duct Configurations,” *Accepted Manuscript: ASME Turbo Expo Forum 2023*, American Society of Mechanical Engineers, Boston, MA, GT2023-103128.
- [9] Lemmon, E. W., Huber, M. L., and McLinden, M. O., 2013, “NIST Standard Reference Database 23: Reference Fluid Thermodynamic and Transport Properties— REFPROP.”
- [10] Holder, J. M., RingHeisen, A., Ha, M. J., Ghimire, S., and Turner, M., 2022, “Improved Automated Turbomachinery Hot-to-Cold Transformation With Cold-to-Hot Capabilities For Off-Design Analysis,” *AIAA SCITECH 2022 Forum*, American Institute of Aeronautics and Astronautics, San Diego, CA & Virtual.

Appendix A. Spanwise Analysis Plots

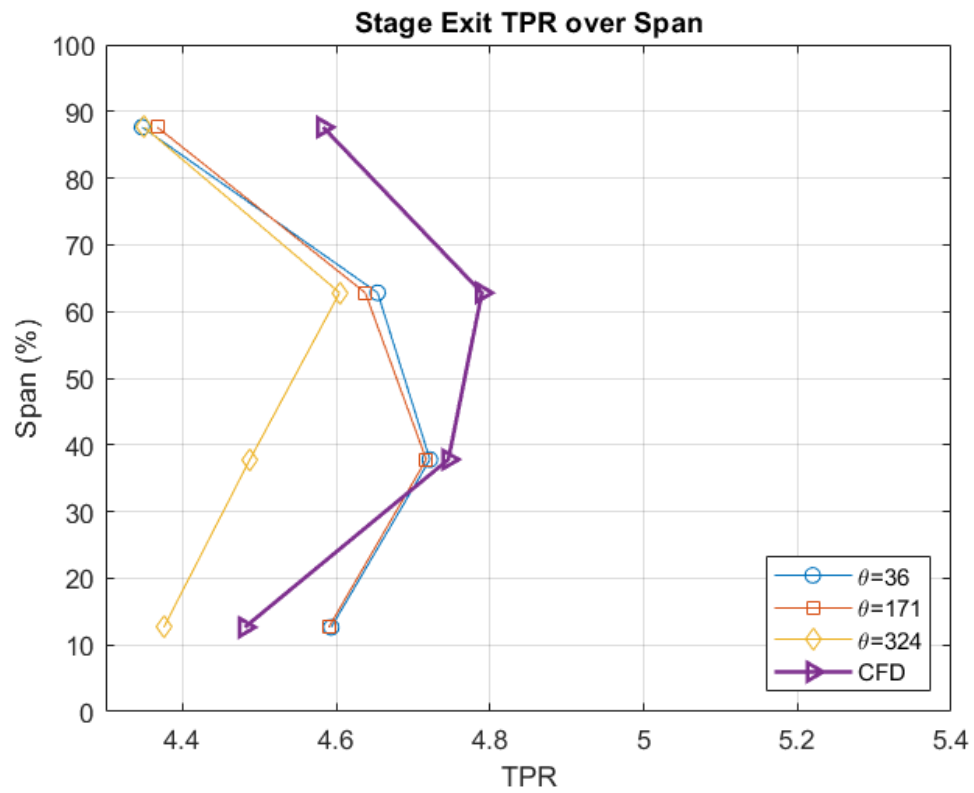


Figure 29: Per rake spanwise comparison for stage TPR

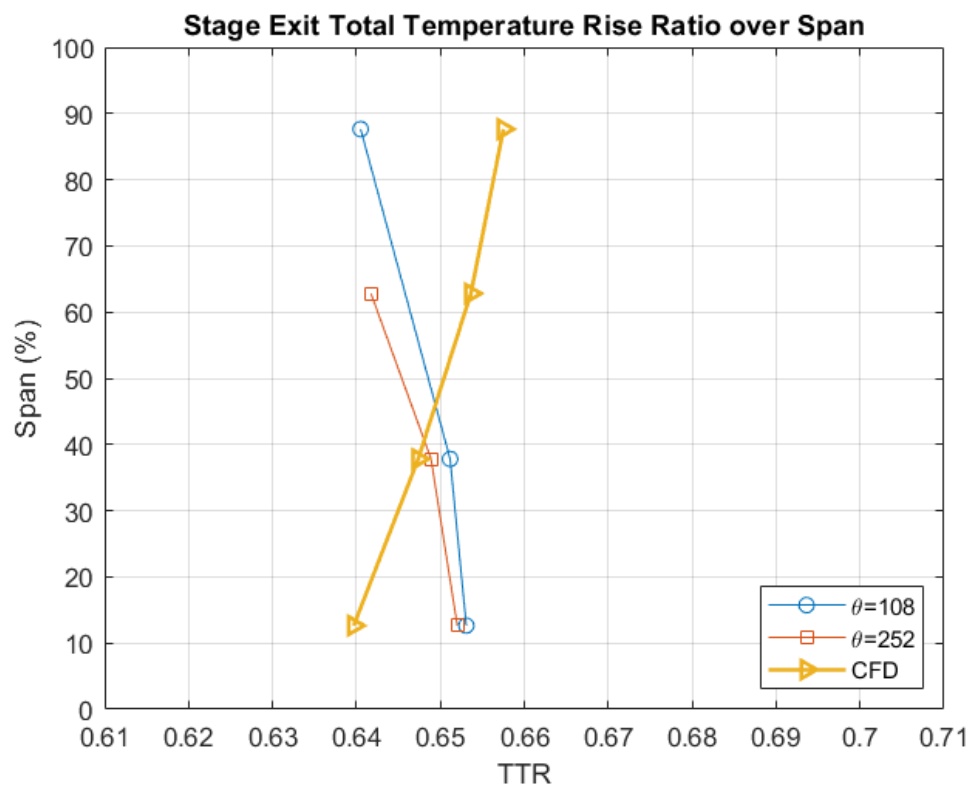


Figure 30: Per rake spanwise comparison for stage TTR

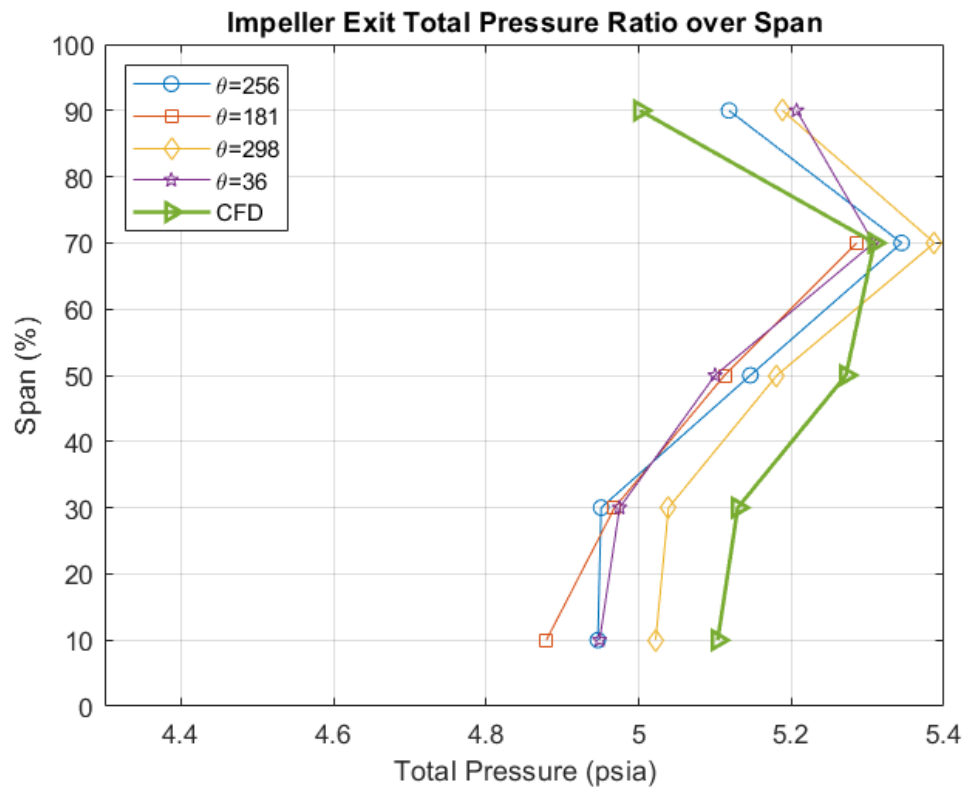


Figure 31: Per rake spanwise comparison for impeller TPR

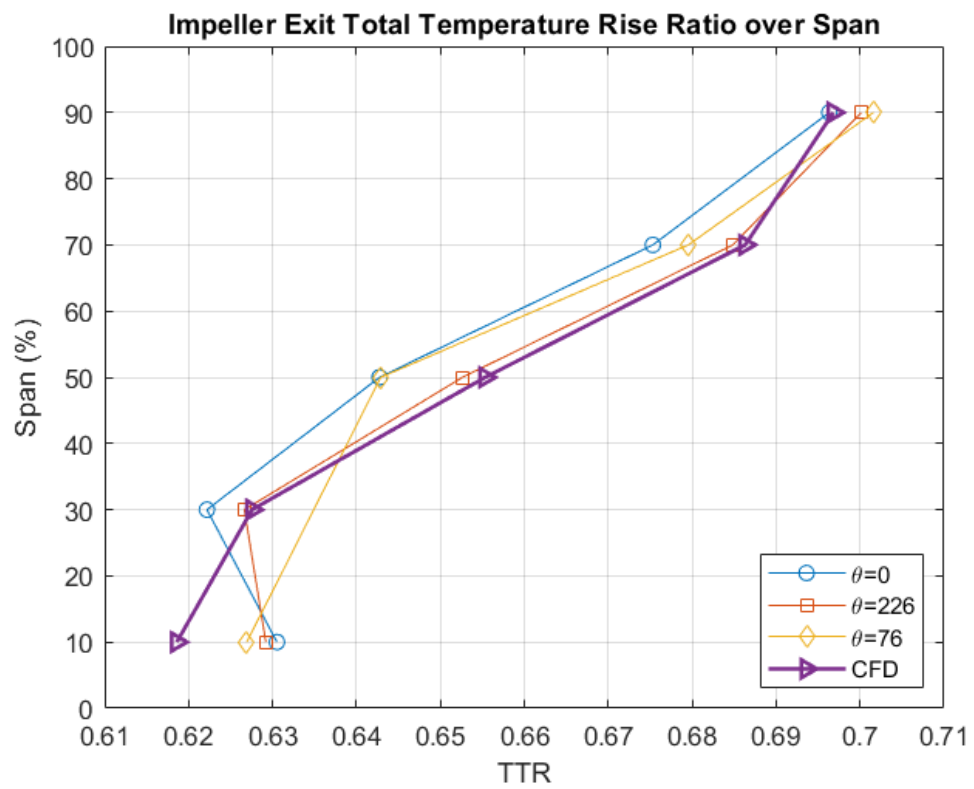


Figure 32: Per rake spanwise comparison for impeller TTR

Appendix B. Full Passage Contours

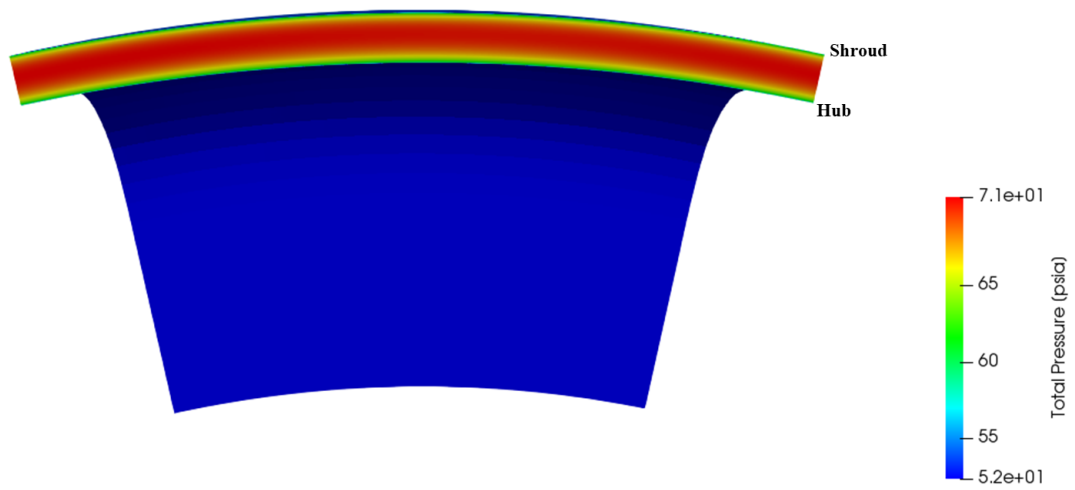


Figure 33: Full passage contour for stage exit total pressure

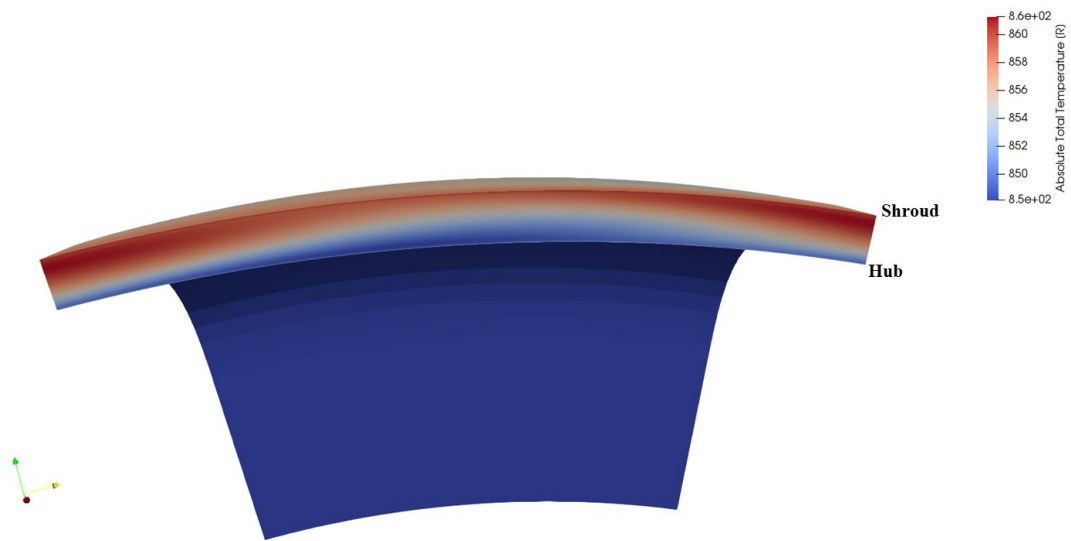


Figure 34: Full passage contour for stage exit total temperature

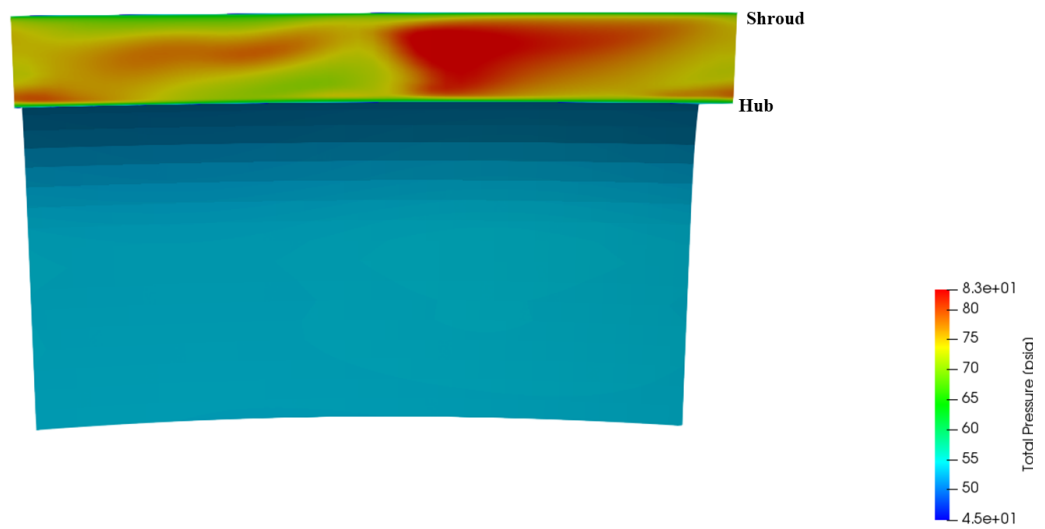


Figure 35: Full passage contour for impeller exit total pressure

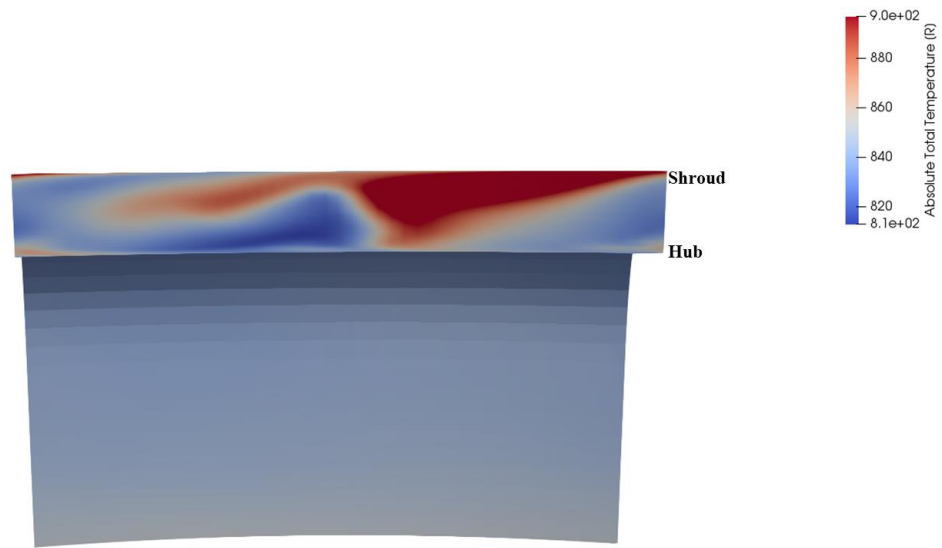


Figure 36: Full passage contour for impeller exit total temperature