

University Turbine Systems Research
2012 Fellowship Program
Final Report

Prepared for:



General Electric Company

Gas Turbine Aerodynamics
Marion Building
300 Garlington Rd
Greenville, SC 29615, USA

Prepared by:

Shelby E Nelson

Department of Mechanical Engineering
University of Nevada, Las Vegas
4505 S. Maryland Parkway
Las Vegas, NV 89154

Date Submitted:
August 9, 2012

Contents

Abstract.....	3
Introduction	3
Part-Span Shroud Cascade.....	4
<i>ICEM Meshing</i>	5
<i>Pressure Ratio Sweep</i>	6
Next Generation Gas Turbine Phase I Rig	7
<i>Tip Clearance Study</i>	8
<i>Squealer Tip</i>	10
<i>Reynolds Number Sensitivity</i>	12
MATLAB Script	13
Conclusion.....	14
Acknowledgements.....	15

Abstract

Engineers on the Gas Turbine Aerodynamics team at GE Energy create state-of-the-art technology that goes into high performing industrial gas turbines. To create such advanced machines, computational fluid dynamics tools are used to analyze the inner workings of the turbine. These tools model the turbine quite well; however, they must be validated with actual test data. Engineers create turbine rigs with different experimental configurations to test certain aspects that go into a design. These rigs are important because engineers need to understand if their use of computational fluid dynamics is true to the physics of what is actually happening.

Introduction

Computational fluid dynamics (CFD) is used by engineers in Turbine Aero to understand how the fluid moves through the turbine which allows for a better design. To perform CFD, meshing the fluid domain of the problem is required. After meshing, boundary conditions are applied to the model, such as total pressure at the inlet and exit. Then a CFD program solves fluids equations, such as the Navier-Stokes equations, over a period of specified iterations until a final value is reached. Once this final value is reached, the problem is converged and the case is ready for post-processing.

Post-processing involves examining the results from CFD. Different programs can be used for post-processing cases which allow for visualizing the flow by either a graphical contour plot of the airfoil or through basic line plots made in Excel or MATLAB. CFD results can be compared to rig test data in order to validate the model used for CFD. An accurate representation of the CFD model is important because engineers must understand the fluid mechanics of the turbine to design for the optimum efficiency.

Three major projects are explained in this report. For the first project, the Part-Span Shroud Cascade is meshed in ICEM to show proper meshing techniques. A pressure ratio sweep was performed using CFD to analyze the effects of having a part-span shroud in steam turbines. The cases are post-processed and the bladerow efficiency is plotted. For the second project, a study is conducted on the Next Generation Gas Turbine Phase I Rig to show how changing the tip clearance and Reynolds number affects the performance. The CFD results are compared to actual rig test data. For the third project, a MATLAB script was created to plot the loading distribution at one span of a transient case.

Part-Span Shroud Cascade

A part-span shroud is placed in the middle in the latter stage buckets in steam turbines to provide extra mechanical damping. Although this provides a structural advantage, the engineers concerned with aerodynamics have to optimize the design of the shroud to minimize loss to the aerodynamic performance. Figure 1 shows two buckets joined together with a part-span shroud.

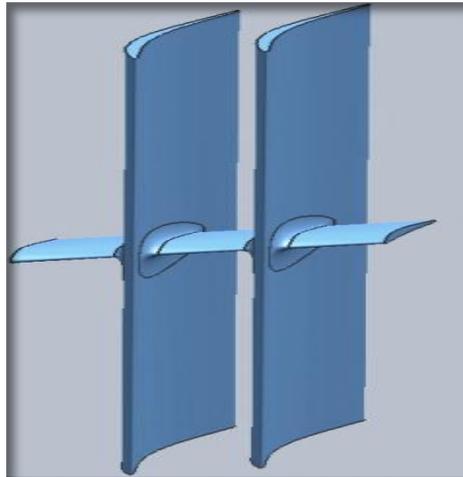


Figure 1 - Example of a Part-Span Shroud

To understand how the aerodynamics is affected in steam turbine buckets with part-span shrouds, a cascade test rig is being built. Figure 2 shows a cross-section of a typical cascade test rig. The blades shown above are in the green section of the diagram. The boundary conditions going into the inlet will be carefully matched to the conditions in an actual turbine. After this test is conducted, the results will be compared to the CFD pre-test predictions. This analysis is important because it is important to understand part-span shroud applications in steam turbine last stage buckets.

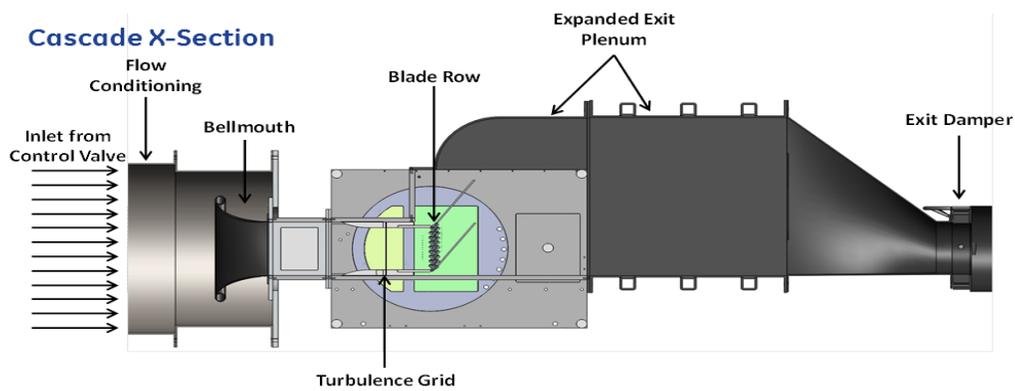


Figure 2 - Test Rig Configuration

ICEM Meshing

Meshing is an important aspect of CFD analysis. Ansys ICEM was used to mesh a two-dimensional cross-section of the cascade. To mesh the cascade, ICEM uses an unstructured 2D tetra grid to randomly fill the shape of the geometry. As seen by Figures 3 and 4, the mesh resolution is dense enough to allow for a proper representation of the fluid mechanics. There is higher resolution around the airfoil to properly capture its curved geometry.

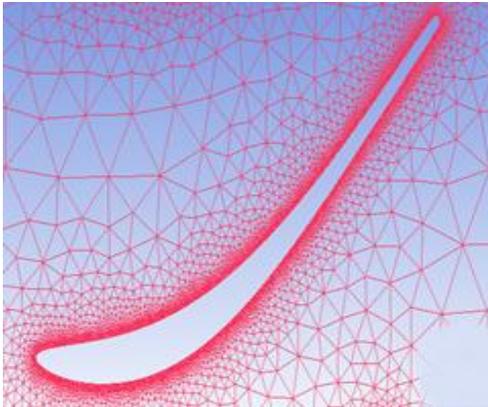


Figure 3 - Fine mesh around airfoil

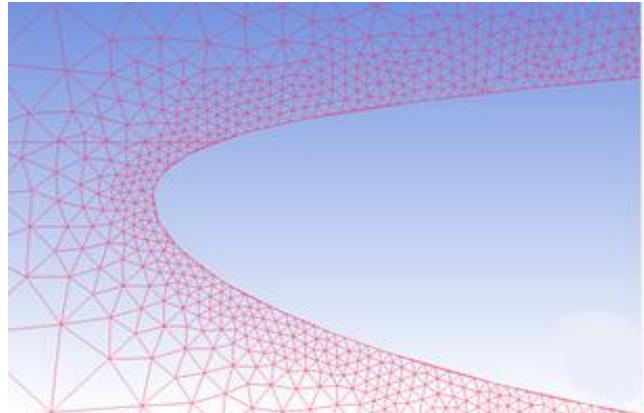


Figure 4 -Close-up of mesh around leading edge of an airfoil

Boundary layers must also be considered when meshing. A prism mesh is used on the border of the geometry to account for the boundary layer. As opposed to a tetra mesh, a prism mesh is a structured grid where the cells are stacked on top of each other. Figure 5 shows a prism mesh on the border of the cascade casing geometry. The cells of the prism mesh seem to decrease size as the cells get closer to the border of the geometry. This is another technique used to capture the boundary layer profile on the surface.

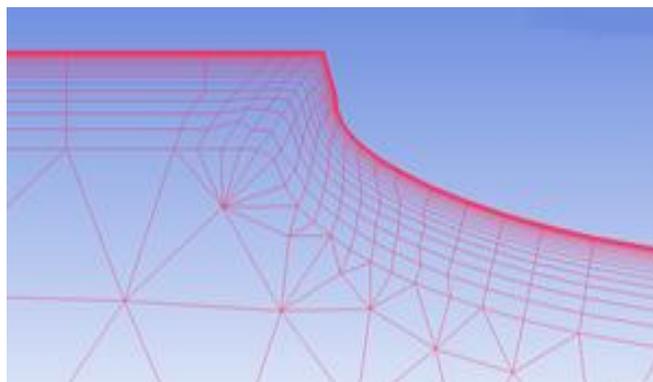


Figure 5 - Prism mesh example

Pressure Ratio Sweep

To simulate different engine operating conditions, a pressure ratio sweep was done in CFD on the part-span shroud cascade. An airfoil with a part-span shroud is compared to an airfoil without a part-span shroud. Figure 6 is a contour plot of the Mach number of the two different cases. There is a strong shock on the suction side of the airfoil that can be seen by the red-orange color which indicates high Mach numbers. There is also a shock that comes off of the part-span shroud. The curvature of the shroud causes the flow to be accelerated creating another shock. The two shocks are also visible in Figure 7 which is a 2D axial cut downstream of the trailing edge for the case with and without the shroud.

The cascade was also tested to see how it behaves at various pressure ratio conditions. Figure 8 shows a low pressure ratio on the left airfoil, the design point in the middle, and the high pressure ratio on the right airfoil. The lower pressure ratio case has significantly lower Mach numbers than the higher pressure ratio case. The effect of the shock from the part-span shroud seems to be less prominent on the higher pressure ratio case. The shock near the trailing edge is the dominating loss mechanism.

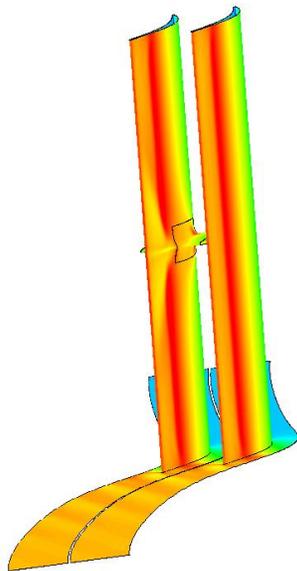


Figure 6
Case with PSS and without PSS

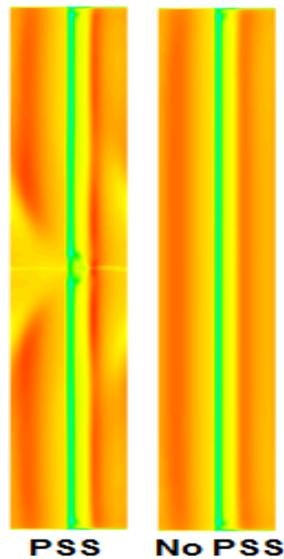


Figure 7
2D Profile Aft of PSS

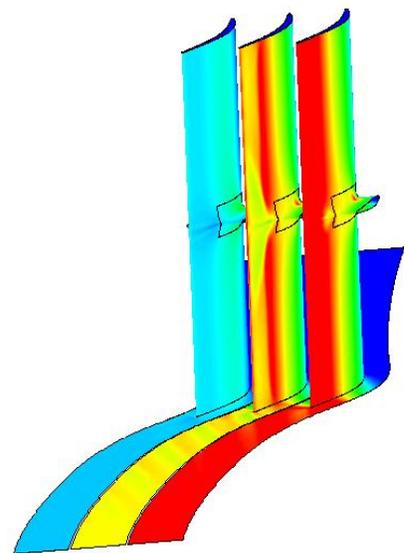


Figure 8
Pressure Ratio Sweep

Next Generation Gas Turbine Phase I Rig

The Next Generation Gas Turbine Phase I Rig is a test rig used to validate CFD code that the Turbine Aero team use when designing airfoils. The Phase I Rig is a 1/5 scaled model of the 7FB industrial gas turbine engine. Figure 9 shows the three stage turbine rig set up. This rig only tests the turbine section of the complete gas turbine. The compressor section was used at the test facility to provide the specified turbine inlet conditions.

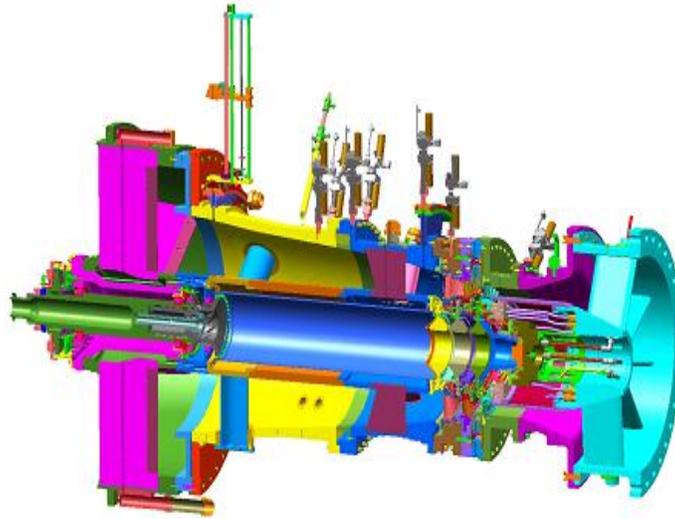


Figure 9

As seen by Figure 10, the test rig is an accurate scaled down representation of the actual turbine section of an engine. There are cooling flow circuits to represent the secondary flow that occurs within an engine.

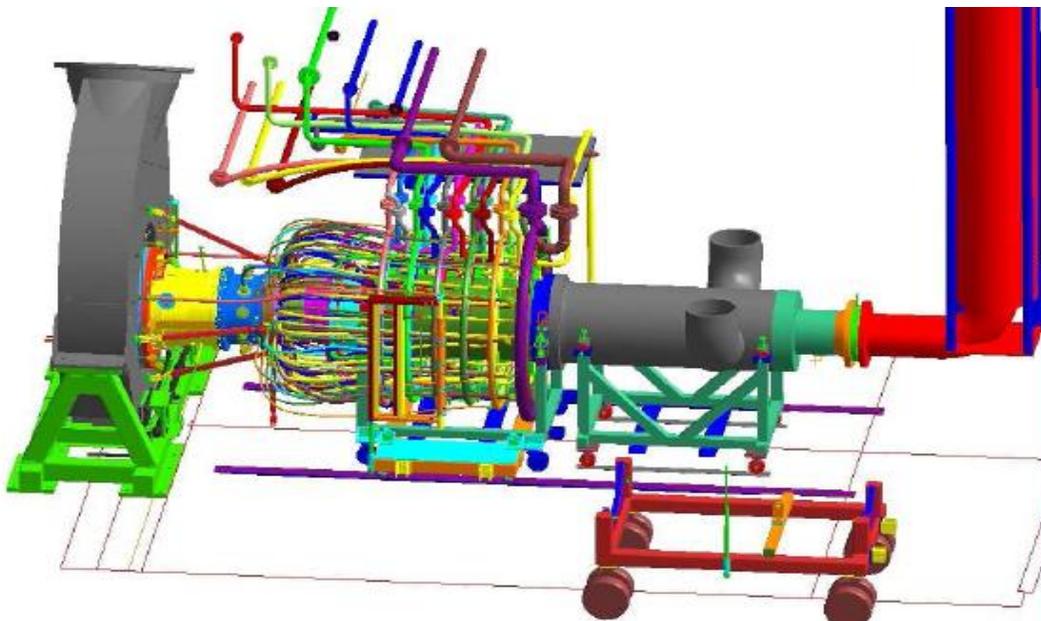


Figure 10

Tip Clearance Study

Much of the loss that is attributed to a turbine is seen in leakage over the tip of unshrouded blades. Since there must be a clearance from the top of the first stage bucket so that the tip of the blade does not rub the casing, flow leaking over the tip of the blade is inevitable. The flow that leaks over the tip is considered a loss mechanism because the bucket is not extracting work from this flow.

There were two measurement devices in the Phase I Rig that are used to compare to the CFD: a traverse probe and kiel heads. A traverse probe is a device that measures total pressure and temperature between two nozzles. It traverses the span of the nozzle and measures data continuously from the hub to the tip. A kiel head measures total pressure and temperature at discrete points on the leading edge of the nozzle.

CFD was used to analyze three separate cases: closed clearance, nominal clearance, and open clearance. The data from the traverse probe was compared to the CFD results. The kiel head total pressure test data was also compared with the results from CFD seen in Figure 11. The points on the graph are the test data points. The dashed lines represent the CFD at different clearances. The CFD is represented as a line instead of discrete points because it is an average taken at the location of the kiel heads. Although the CFD and the data are not a perfect match, the CFD seems to follow the overall trend of the kiel head data.

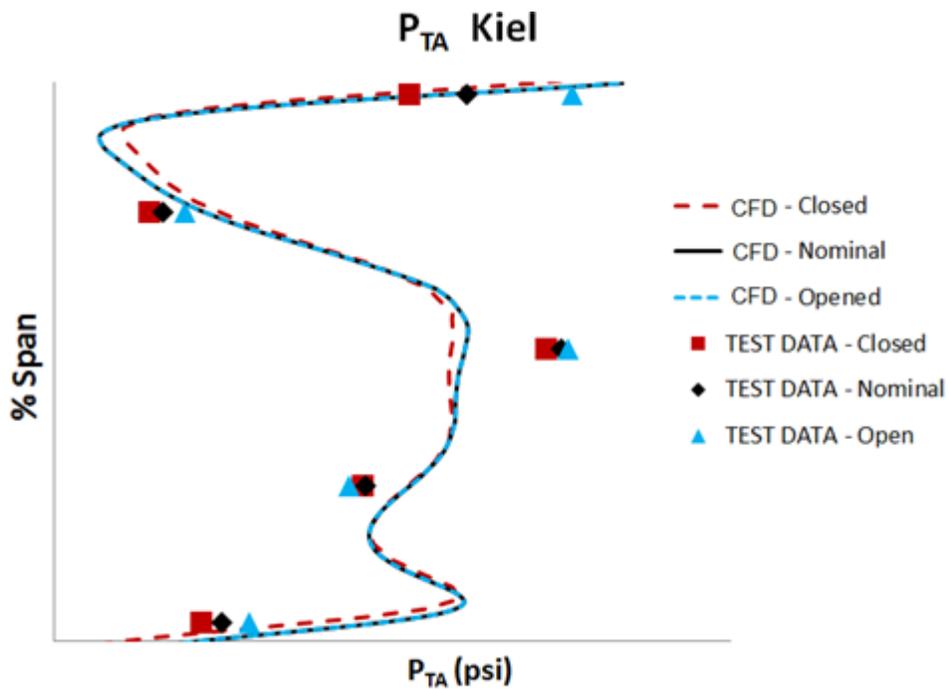


Figure 11

Figure 12 shows the overall efficiency of the different clearances compared with test data where the middle point is the nominal clearance. The CFD curve has a more pessimistic trend than the actual test data. This can be seen in the slope of the two curves. The CFD predicts a more drastic effect in efficiency as the tip gap increases. A larger tip gap will decrease turbine efficiency.

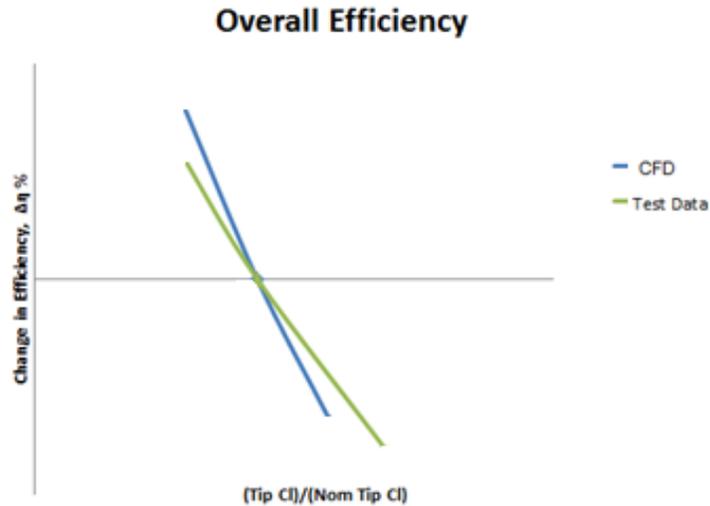


Figure 12

To get a visual representation of the flow leaking over the tip, streamlines were plotted using the GE in-house post-processing software. Figure 13 shows the streamlines over the tip with three different planes. The colors represent total pressure with blue being the lowest pressure and orange being the highest pressure. Plane 1 is the most upstream and Plane 3 is the most downstream. As shown by the figure, the streamlines coming over the tip create a loss vortex where no work is being done on this flow. This is seen by the blue region of low pressure on the plane cuts. As the vortex travels downstream it mixes with the flow that the bucket has done work on.

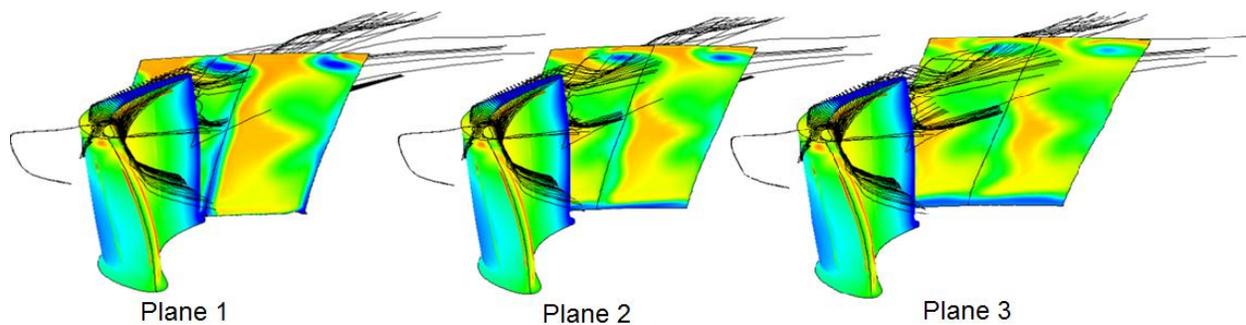


Figure 13

Squealer Tip

In the previous section, a flat tip was modeled for simplicity during CFD calculations. However, a squealer tip, a cavity on the top of a bucket, exists on the actual geometry to reduce over tip leakage. The squealer tip traps the fluid leaking over the tip in the squealer cavity making it a more torturous path for the fluid which decreases the overall leakage.

The squealer tip geometry was modeled in CFD. A contour plot of the loss in the bucket is seen in the figures below. Figure 14 shows the loss with the squealer tip geometry, and Figure 15 shows the loss with the simplified flat tip geometry. Streamlines are added to give a visual representation of how the flow travels over the tip. The flow leaking over the tip creates a vortex seen by the red high-loss circular region near the upper right corner of the 2D cut. This red region is greater on the bucket with the flat tip geometry implying that the squealer tip geometry decreases the over tip leakage.

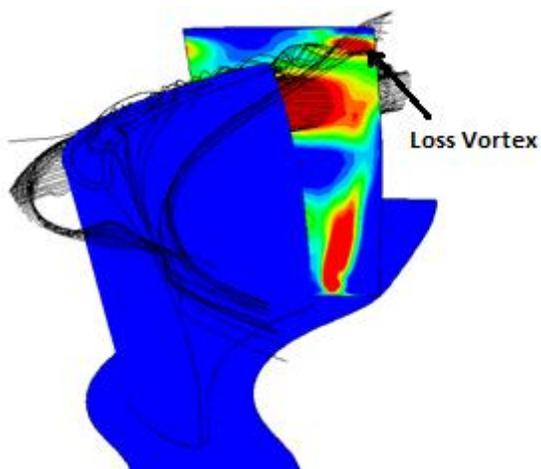


Figure 14 - Squealer Tip Loss

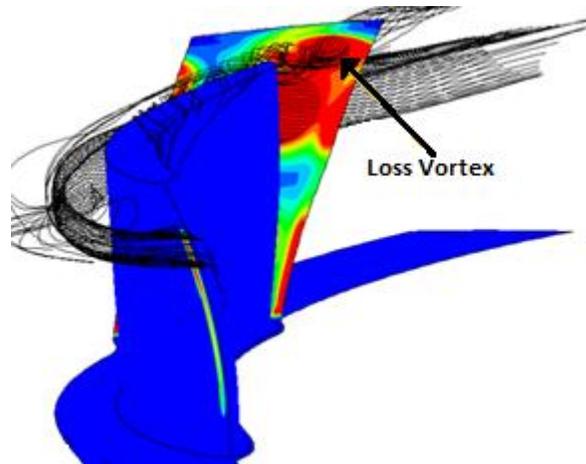


Figure 15 - Flat Tip Loss

The overall efficiency trend of the squealer tip was also compared with the flat tip and test data. Figure 16 shows that the performance of the squealer tip is better than the simplified flat tip model, but it has a steeper slope than the data and the flat tip model which means that the squealer tip is more sensitive to change in clearance than the flat tip or test data. Since the test data was taken from the rig which has the squealer tip geometry, it was expected that the squealer tip geometry should better match the trend of the test data. This unexpected result could be grid related. Further investigation must be done to account for this counter-intuitive result.

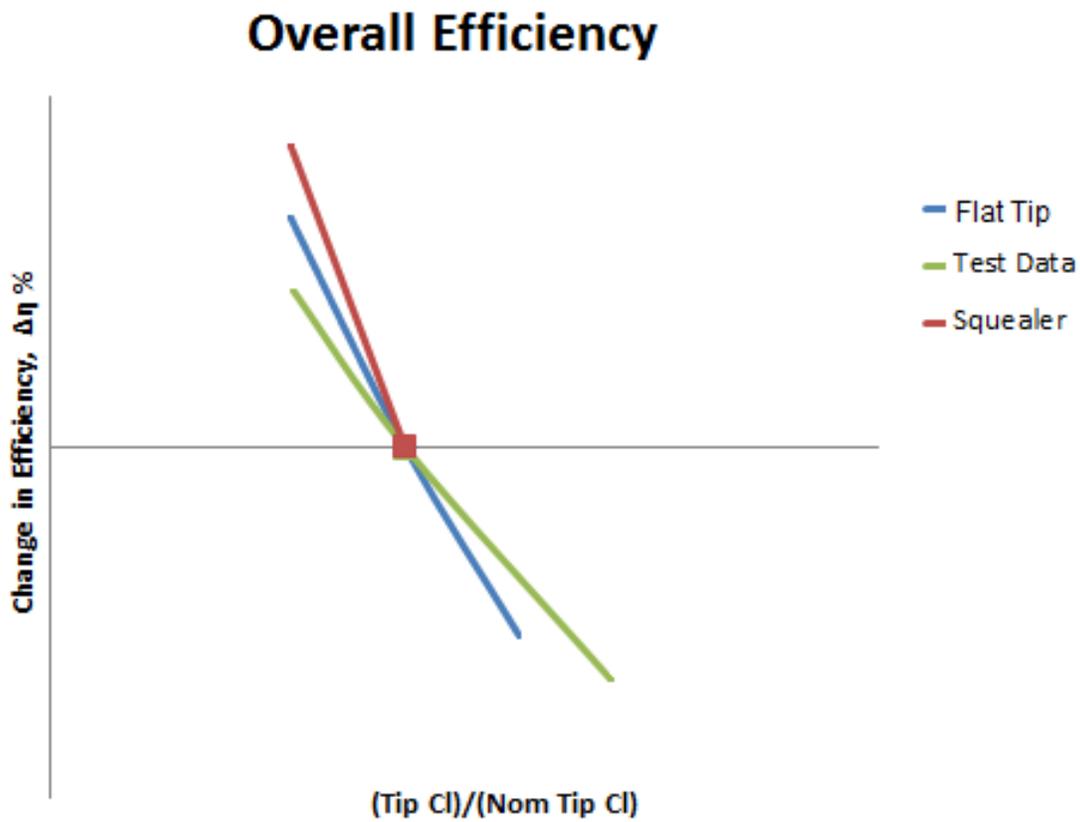


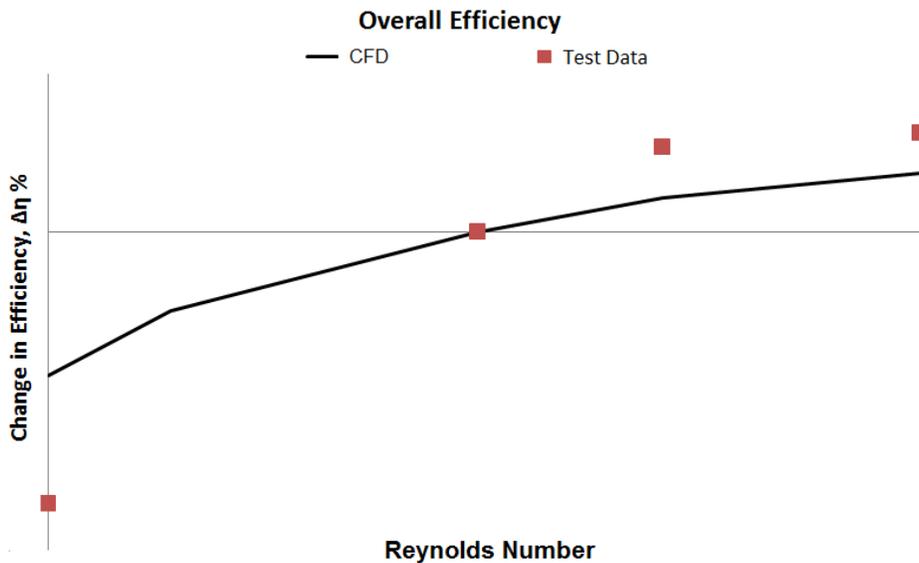
Figure 16

Reynolds Number Sensitivity

A Reynolds number sensitivity sweep was conducted by changing the inlet pressure and keeping the same pressure ratio across the turbine. The goal was to compare the data acquired in the rig test to the results from the CFD cases and to plot the overall efficiencies. During the test, the rig was operated at a range from a low Reynolds number, or a low inlet pressure, to a high Reynolds number, or high inlet pressure. To compare the results from the test, the four cases were modeled with CFD. The CFD was performed using a fully turbulent model.

The loading diagrams were compared for each of the cases to make sure that the pressure ratios and corrected speed are the same with only Reynolds number varying between each case. Due to its proprietary nature, these results cannot be discussed any further.

Figure 17 shows the efficiency trend of the data compared with the results of CFD. The red dots are the test points and the black line represents the CFD. The CFD matches the rig test data fairly well. A possible reason for the CFD not matching the data perfectly is because the initial assumption that that the rig is fully turbulent may not be entirely correct. A transition model might be a better assumption. The next step would be to model the different cases using a transition model and plotting the overall efficiency with the rig test data.



Reynolds Number
Figure 17

MATLAB Script

The traditional way of post-processing cases is to take data from the CFD results and plot the data in Excel. Creating loading diagrams, for example, is time consuming because each data file has to be imported into Excel and plotted. This can be especially cumbersome for transient cases where there are multiple moments in time at different spans of the airfoil. For a transient case, there is a directory for each span. Within each directory, there are different data files for each different moment in time for that span. Within each data file, there are columns of data that must be plotted. A tool that will allow the user to choose which columns from the data file to plot and which spans to plot is needed.

A script was written in MATLAB to plot the loading diagram at each time step on one graph. This is useful to see how the surface loading at each time step. As can be seen in Figure 18, the loading varies over time due to the effects of the upstream and downstream bucket. Currently, this program can only read in files from the directory where the code is executed. In the future, the code will be generalized to read in multiple time step files from a certain number of directories specified by the user. The code will also have user inputs so the user will be able to plot other information besides loading plots.

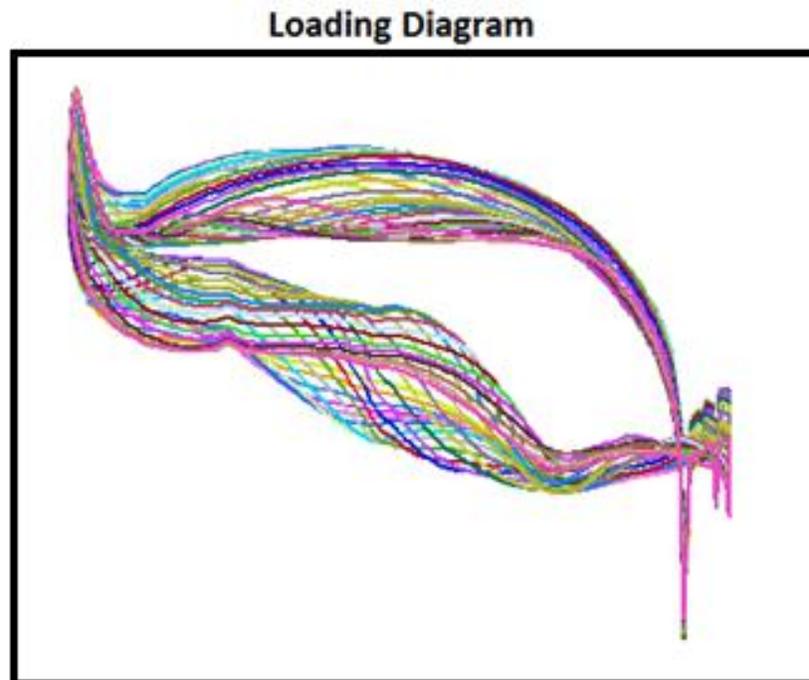


Figure 18

Conclusion

The three major projects that were reported on offered different conclusions. For the part-span shroud cascade, it was possible to get a decent mesh. The next step would be to perform a CFD analysis and compare the results with test data after the test is conducted. The pressure ratio sweep that was performed using CFD offers a valid pre-test prediction to compare with actual results from the test. These results will help the engineers understand the effects of having a part-span shroud in steam turbines.

The tip clearance and Reynolds number sensitivity study done with CFD on the Next Generation Gas Turbine Phase I Rig compares relatively well to the rig test data. Some adjustments can be made in the CFD model to better match the test data. Modeling the actual bucket geometry in CFD gave an expected higher performance than the flat tip, but lead to a steeper slope that will need to be further investigated. Also instead of running the cases with a fully turbulent assumption for the Reynolds number sweep, a transition model assumption can be used and compared with the data.

The MATLAB script successfully plotted the transient loading distributions at one span, but will later need to be generalized to be a suitable tool to use for a variety of cases. There will be inputs for the user to choose which particular parameters to plot.

Overall, the results from the three projects performed during the UTSR fellowship will help engineers continue to understand the fluid mechanics inside the turbine. These results will also help perfect the design tools used to design better performing gas turbines.

Acknowledgements

I would like to thank the University Turbine Systems Research fellowship program and GE Energy for giving me the opportunity to work in the gas turbine field. Before I came to GE Energy, I had a basic understanding of gas turbines that I gained in my turbomachinery class that I took at school. Working with the engineers on the Turbine Aero team has allowed my classroom knowledge to become less abstract theory and more practical. Actually experiencing engineering outside of a classroom through my UTSR fellowship allowed me to gain a deeper understanding and appreciation of the gas turbine field. Inspired by the knowledge I have gained at GE Energy on the Turbine Aero team, I am looking forward to starting a career in the gas turbine field.

I would also like to thank Gunnar Siden for allowing me to work on the Turbine Aero team, my mentor, Sylvain Pierre, for giving me industry and graduate school advice, Neil Ristau for his guidance through my assignments, and the entire Turbine Aero team for answering all my questions and helping me understand gas turbines beyond what I would learn in the classroom.