Coriolis Rig Heat Loss Scheme Verification and New Test Section Design

Prepared For:

Solar Turbines, Inc.
2200 Pacific Highway
P.O. Box 85376
San Diego, CA 92186-5376

&

Southwest Research Institute (SwRI)
Mechanical Engineering Division
6220 Culebra Road
San Antonio, TX 78238-5166

Prepared By:

Robert G. Murphy
Graduate Research Assistant
Department of Mechanical and Aerospace Engineering
West Virginia University
Morgantown, WV 26506-6106

Supervised By:

Industrial Mentor: Dr. Yong Kim
Senior Heat Transfer Engineer
Heat Transfer Group
Solar Turbines
San Diego, CA 92186-5376

Academic Advisor: Dr. Andrew C. Nix
Research Assistant Professor
Department of Mechanical and Aerospace Engineering
West Virginia University
Morgantown, WV 26506-6106
1.0 Introduction

Solar Turbines, Inc. has developed a facility to study the effects of rotation on the heat transfer on internal cooling channels. The “Coriolis Rig” which will be described in detail below was operated at 0, 350, 600 and 750 RPM cases. Three different temperature cases (120F, 140F, 180F) were also examined. Two different setups for the internal cooling passage were used in the rig. One setup had a smooth channel and the other setup had a channel lined with trip strips on the pressure and suction side. The data was compared to see the heat transfer advantage of trip strips instead of a smooth channel in a rotating test section. During the data reduction the endwall heater output was found to be very low for the trip strip case. The responsible engineer believed the heater would need to produce almost twice the output of its counterpart during the smooth channel test. The reason for this is the trip strips should create higher heat transfer coefficients on the end wall than the smooth channel case.

An investigation was started to look into possible endwall issues. The engineers began by looking into the possibility of a slip ring error but the results showed the same trend at 0 RPM case. This proved that the slip ring could not be the cause of the error. The next step was to examine the instrumentation in the test section. No errors were found during the examination of the test section. Since there were no obvious errors the assumptions for the heat loss of the system were reexamined.

It was decided that an analysis of the conduction at the endwall of the test section may help discover the source of the error. This report will describe the process that was used to analyze the heat loss assumption and the conclusions that were reached from the results. The author performed more test cases in ANSYS Workbench than what is reported here but decided to exclude them to reduce the overall length of the report. These other cases do not directly relate to the objectives of this report.

The Coriolis Rig will be used to examine two new cooling channels. The two channels are part of a new cooling scheme that is being implemented into future industrial gas turbine engines. The second half of this report will explain all of the steps to reach the current state of the design for a new test section.

2.0 Coriolis Rig Setup

The Coriolis rig was constructed to study the effects of rotation on heat transfer in internal cooling channels. The rig consists of a cage that house two wing-shaped arms. In one arm is a test section with a 3-pass serpentine cooling scheme. The other arm has a counter weight. Thermocouples are attached to the test section and pass through a slip ring to be recorded into the DAQ system. The rig is capable of having a rotational speed of 750 RPM. Air enters from the root of the wing and passes through the 3-passes before exiting out of the tip of the wing. Heaters are used on the top and bottom of the entire channel area. There are also heaters on the endwalls at the bend regions. These heaters are manually adjusted to try and match the thermocouple readout of the temperature of each plate to the desired temperature range for the present case. Figure 1 displays an image of the Coriolis Rig set up.
3.0 3D Heat Transfer Analysis

A 3D heat transfer analysis was performed using ANSYS Thermal Workbench. The objective of this analysis is to determine whether the heat loss assumption used in data reduction is valid. It will also be used to analyze the endwall region. The following sections will describe the setup of the model in ProE, the thermal analysis setup in ANSYS and the produced results.

3.1 ProE Setup

A 3D model of the current Coriolis test section was provided by Solar. The model consists of two end caps, two side pieces and two pieces that hold the channel plates. This model can be seen below in Figure 2.
The model needed to be trimmed down so that the endwall region could be examined. The model was split in half so that the symmetry condition could be applied. Figure 3 displays the final model after all of the cuts had been made.

To make the analysis accurate, the model needed to have all of the components that the actual test section contains. This list includes: copper plates, silicon spacers,
insulation and aluminum to simulate the wing. Figure 4 displays the final model with all of the components that would be imported into ANSYS.

![Final Model of Endwall Region](image)

**Figure 4: Final Model of Endwall Region**

### 3.2 ANSYS Model Setup

The model was imported into ANSYS to begin the setup of the thermal analysis. The first step required inputting the material properties for each of the materials in the model. Table 1 displays the materials and their coefficient of thermal conductivity, k, value.

<table>
<thead>
<tr>
<th>Material</th>
<th>k (BTU/ft-hr.^°F)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Aluminum</td>
<td>137.22</td>
</tr>
<tr>
<td>Copper</td>
<td>231.11</td>
</tr>
<tr>
<td>Delrin Black</td>
<td>0.2083</td>
</tr>
<tr>
<td>Polystyrene, Expanded</td>
<td>0.0173</td>
</tr>
<tr>
<td>Silicon</td>
<td>85.51</td>
</tr>
</tbody>
</table>

Meshing was completed using the ANSYS Workbench auto-mesher. A finer mesh was tested to see if it would produce different results. The finer mesh did not change
the results by more than a degree °F and took a significant amount of time to solve. Therefore it was decided that the auto-mesher was sufficient for the current analysis.

Heaters were used to maintain the channel temperature in the actual test section. These heaters were found to not heat evenly so a correction factor was created. The heaters were split into six different sections representing the copper plates. An average of each of the six sections was calculated. Each averaged section was then divided by the total average of the total heater to create a correction factor to account for the varying heat coming out of the heater. The method allowed a correction factor to be created for each individual copper plate. A similar process was used for the end plates but instead of having six sections there were only two sections. The correction factors would be applied to the heat flux for each plate in a method described in section 3.4.

3.3 Applying Global Boundary Conditions

Boundary conditions that were not going to be changed regardless of the test case were applied to the model next. In the figures above several internal passages can be seen. These passages are used to house wiring and instrumentation in the actual test section. To simulate the low heat transfer taking place in these channels a free or natural convection boundary condition was applied. The other global condition that would not change is the application of the law of symmetry. This was applied using a perfectly insulated boundary condition to all surfaces on the top, right and front of the model. Figure 5 displays the model with the natural convection in the internal channels (highlighted yellow) and the perfectly insulated condition (highlighted in blue) on the top, right and front of the model.
The final condition that would not change for each test was inside the channel. All of the plates, spacers and walls were given a perfectly insulated condition to simulate the heat loss test performed with insulation placed in the channel. After all of these conditions were set the outer boundary condition had to be set for each rotational case.

3.4 Heat Loss Boundary Conditions

The heat loss tests were conducted at 0, 350, 600, and 750 RPM. They were also performed at three different temperatures: 120F, 140F, and 180F. Newton’s law of cooling was used to calculate the outer heat transfer coefficient. A rearranged version of the equation that was used is seen below in Equation 1.

\[
h = \frac{\dot{Q}}{A_{\text{surface}} \left( T_{\text{wing,avg}} - T_{\text{cage,avg}} \right)}
\]  

(1)

Where \( h \) is the heat transfer coefficient, \( Q \) is the wattage from the heaters, \( A_{\text{surface}} \) is the total outer surface area, \( T_{\text{wing,avg}} \) is the average wing temperature and \( T_{\text{cage,avg}} \) is the average cage temperature. The values were imported from the raw data files that were recorded during the tests. For each RPM case the three temperature cases were averaged together. This was done because the differences were negligible at different temperature ranges. Table 2 displays the values at all four RPM cases.

<table>
<thead>
<tr>
<th>RPM</th>
<th>( h ) (BTU/hr-ft(^2)-°F)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>6.78</td>
</tr>
<tr>
<td>350</td>
<td>7.64</td>
</tr>
<tr>
<td>600</td>
<td>9.29</td>
</tr>
<tr>
<td>750</td>
<td>10.27</td>
</tr>
</tbody>
</table>

The 0 RPM case had the same \( h \) value on the entire outer shell of the model. However, for the rotating cases the large side of the shell was split to simulate the effect of the wing on the side of the model. Figure 6 displays the two faces that the \( h \) value listed above was applied. The remaining value was given a low value to simulate the low heat transfer occurring due to the wing extension coming from it.
The heat flux values that were applied to each copper plate were calculated by taking the total heat coming from each heater and dividing it by the plate area. This value was then "corrected" using the correction factors that were calculated above. The endwall heater power was split in half since only half of the heater was being simulated.

3.5 Heat Loss Results

The cases were run to try and match the plate temperatures with the experimental data. Contour maps of each case were created and from these an average plate temperature for each plate could be calculated. The results are shown below in Figures 7, 8, 9 and 10.
Figure 7: Heat Loss Test 0RPM Case ANSYS vs. Experimental

Figure 8: Heat Loss Test 350RPM Case ANSYS vs. Experimental
The results match up well with the experimental data. Most of the plate temperatures are within 10 degrees of the recorded temperature. Plate 15 was the least accurate of all the plates. It was between 10-15% different than the recorded temperature. ANSYS over predicted the temperature of Plate 15 on all but one test case.
3.6 Endwall Flow Cases

There were several cases performed at different flow rates for both smooth and trip strip setups. Those cases provided results that matched the temperatures and trends of the experimental data well. They will not be presented in this report to cut down the size of the report. The endwall region needed to be examined to look into the problem recorded in the experimental results.

The experimental setup was the same until the channel section. The plates, spacers and walls had a convective heat transfer coefficient (HTC) applied to them to simulate the flow going through the channel. The HTC values were pulled from a data reduction spreadsheet created by Solar engineers. The HTC value was calculated using the plate temperature and bulk temperature at each plate as the temperature difference. The heat flux at each plate was calculated using the wattage of the heater divided by the area of the individual plates. The wall values were created by averaging the HTC values of the plates that were in contact with them. Figures 11 and 12 display the results of the smooth channel and trip strip cases, respectively.

Figure 11: Low Flow 0RPM Smooth Channel Case ANSYS vs. Experimental
The results show there is a distinct difference in the trip strip case. The results match the trend of the data and are within a difference of 10% on all data except the last few plates of the smooth channel results. The reason the data does not match the data better at the end plate is speculated to be due to the heat that would be coming through the wall from the third channel. This effect could not be modeled in the current ANSYS model. The data for the trip strip case shows the data matches within 10% on the channel plates. However, the data at the endwall plates shows a large difference in the temperatures and a completely different trend.

4.0 New Test Section Design

A new test section was needed to incorporate a new cooling scheme that Solar will be implementing. This test section will have two straight channels. One will be lined with trip strips while the other will be a pin-fin arrangement. The actual channels are curved but the model will simulate straight channels with the geometry taken at a cross section approximately one inch from the tip of the blade. The following sections will discuss the objectives of this project, the stages of the design process and the current state of the design.

4.1 Objectives of New Design

The interior channel is being changed from a 3-pass serpentine channel to a two-channel design with trip strips and pin-fins. The change is meant to test a new cooling scheme that Solar will be implementing into future industrial gas turbine engines. The test sections overall diameter will be reduced by a ½ inch to allow for a ¼ inch of insulation to be added around the heated section. The reason this will be done is to try
and mitigate the heat loss through the material to the aluminum wing. The material of the test section will be changed to accommodate higher operating temperatures and pressures. The heaters will be changed to account for the change from a flat channel to a curved faced channel. The test section will be extended to fit further down the sleeve of the arm. The reason this will be done is to help house the thermocouple wires better. It will also allow an entrance length to be installed to keep the flow conditions as close to the actual flow conditions.

4.2 Design Stages

The first step in the design process was to construct a core that would be used as the base for the model. This core was created from the actual core of the new blade design. A cross section was taken and the trip strips and pin-fins were patterned onto the straight passages. Figure 13 displays the core setup.

![Figure 13: New Test Section Core Model](image)

This core was used to make a molding of the trip strips and pin-fins. From this point the individual copper plates were created for each channel. Figures 14 and 15 display a trip strip and pin-fin plate, respectively.
These plates were assembled together into halves of the test section. Figure 16 and 17 display the pressure side and suction side of the new test section. The white strips seen in the figure are silicon spacers that are positioned between the individual plates.
Figure 16: Pressure Side of New Test Section Concept

Figure 17: Suction Side of New Test Section Concept
5.0 Conclusions

The results for the heat loss tests show the current heat loss assumption is valid. The model setup insulated everything except the outer surface. This setup only allowed the heat to escape out of the outer surface. The figures show the plate temperatures are within a few degrees on most of the test runs. Two exceptions are the 160°F on the 0RPM case and Plate 15 on most of the cases. The 160°F case appears to be an experimental error due to the fact that the trend does not match the rest of the experimental data. Plate 15 was over predicted by the model in every case. The author examined the raw data to see if the thermocouple was reading incorrectly but could not find an error in the data. The reason for Plate 15 experimental reading are concluded to be an instrumentation error due to no evidence of any other issues that can be examined.

The results from the endwall region disagreed with the experimental results. The smooth channel results matched up well (less than %11) on all of the first channel and endwall plates. The second channel plates showed a divergence from the experimental data. The experimental data shows a trend where the channel plates get warmer as they travel down the flow path. The reason for the discrepancy is due to the heat transfer through the dividing wall between channels 3 and 2. Channel 3 is hotter than channel 2 and there would be heat passing through the wall. This aspect was not modeled in the current ANSYS model due to the application of the law of symmetry in that section.

The trip strip data showed good correlation (less than 10%) between the experimental and ANSYS channel plates. However, the endwall region was different by as much as 27%. The closest temperature at the endwall had a difference of 19%. Not only were the values but the trends were completely different also. The experimental data shows a trend where the endplates get hotter than the plate temperatures in the bend. This was contradicted by the ANSYS results that show the endplates decrease in temperature. Since the heat loss assumption was proven to be correct and the instrumentation was proven to be accurately set up the author is left to conclude that the calculated values for the heat transfer coefficients in the bend region are incorrect. More tests should be conducted varying the trip strip setup to try and ascertain what flow patterns and secondary flows are being created in the channel that are not currently being considered in the data reduction.

The copper plates are completely designed for the new test section so the next step will be to start modifying the exterior of the new test section. There are several things that are being worked on but are not complete at the time of this report was due. Creating channels to run thermocouples and pressure gauges will be required to allow the testing engineers easier access to the test section during the runs. Calculating the required heater output for the new test section will be required. Once the heater power output value is calculated, vendors for the new test section material and heater setup will be contacted.

6.0 Acknowledgements

The author would like to acknowledge several people and institutions for contributing to his education and experience. The author would like to thank Dr. Klaus Brun and his team at the Southwest Regional Institute in San Antonio, Texas for managing the University Turbine Systems Research (UTSR) Fellowship. Specifically, Andrea Barnett for all of her help at every step of the process. The Department of Energy and National Energy
Technologies Laboratory in Morgantown, WV for allowing the author to pursue this great opportunity. Solar Turbines, Inc., John Mason, Dr. Hee-Koo Moon and the entire Aero, Thermal and Performance group for providing an extremely rewarding experience to develop skills in the heat transfer field and enjoy the amenities of San Diego, CA. Thanks to the authors mentor at Solar, Dr. Yong Kim, for all of his guidance and instruction throughout the summer. Finally, thanks to Dr. Andrew C. Nix at West Virginia University for being a great friend and mentor. Without his guidance and support being selected as a 2011 UTSR Fellow would not have been possible.