University Turbine Systems Research (UTSR)
Industrial Fellowship Program
2012

Design of Test Section for Coriolis Rig and Numerical Simulation of Cooling Flow Through Test Section

Prepared For:
Solar Turbines, Inc.
Mentor: Dr. Yong Kim
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:
Timothy Repko
Graduate Research Assistant
Department of Mechanical and Aerospace Engineering
West Virginia University
Morgantown, WV 26505-6106

Academic Advisor: Dr. Andrew C. Nix
Assistant Professor
Department of Mechanical and Aerospace Engineering
West Virginia University
Morgantown, WV 26505-6106

1.0 Introduction

The primary method for cooling a turbine in industrial gas turbine engines has been internal, forced convection, air-cooling. This cooling flow is taken from a high-pressure section in the compressor and used to cool the turbine section with 1-5% of the mass flow of air through the engine being used. By using cooling flows, the blade temperature can be lowered by 200-300 Kelvin thus allowing the rotor inlet temperature to be hotter and the overall efficiency of the engine to increase. Internal cooling passages have evolved over time from simple single pass, internal cooling schemes to much more complex schemes that are used today; utilizing multi-pass serpentine channels, turbulators, pin fins, and other techniques to enhance the heat transfer effects. An example of a first stage blade that uses a multi-pass serpentine channel is shown below in Figure 1.

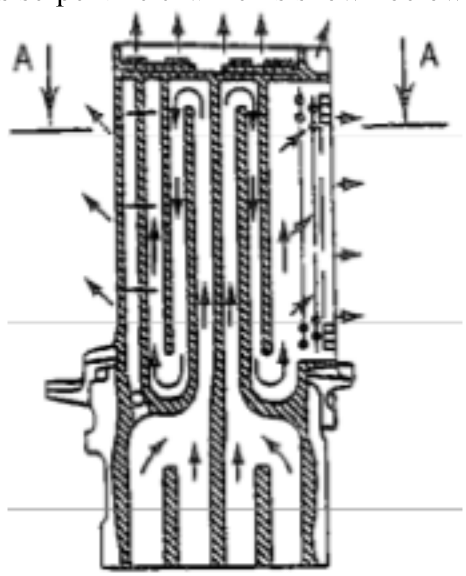

Figure 1: First Stage Blade - Multi-Pass Serpentine Channel

Solar Turbines is developing a new internal cooling passage design to improve the efficiency of one of its engines. This new cooling passage is being designed to substantially lower the cooling flow required to keep the first stage turbine blades within an acceptable temperature range. Lowering the cooling flow can increase the efficiency of the engine by allowing the turbine to run hotter as previously stated; or the turbine can be allowed to run with the same rotor inlet temperature as before but with a better-cooled blade. As a general rule of thumb, lowering the surface temperature by 25 degrees Celsius can double the operational life of a component. With durability being paramount in industrial gas turbine engines, improving the cooling is a key design consideration for industrial gas turbine design engineers.

2.0 Coriolis Rig and Effects of Rotation

2.1 Effects of Rotation

An effect that has been observed in internal cooling passages is the effect of rotation on internal flow. The coolant flow through the rotating blade is subjected to Coriolis forces and rotational buoyancy forces that alter the flow through the blade creating secondary flows that asymmetrically distribute the flow through the cooling passage and must be considered when designing a turbine blade. Significant research has been done on this effect at numerous universities as well as previous experiments at Solar. The asymmetric distribution of the flow in a rotating channel has significant effects on the local heat transfer coefficient as well as the overall heat transfer in the cooling passage. Since convective heat transfer is dependent on the flow field, any alteration to the flow field by the rotational effects significantly alters the heat transfer. Thus, accurate prediction of the flow field is needed to accurately predict the heat transfer inside the cooling passage. Figure 2 below illustrates the secondary flow vortices that occur due to rotation.

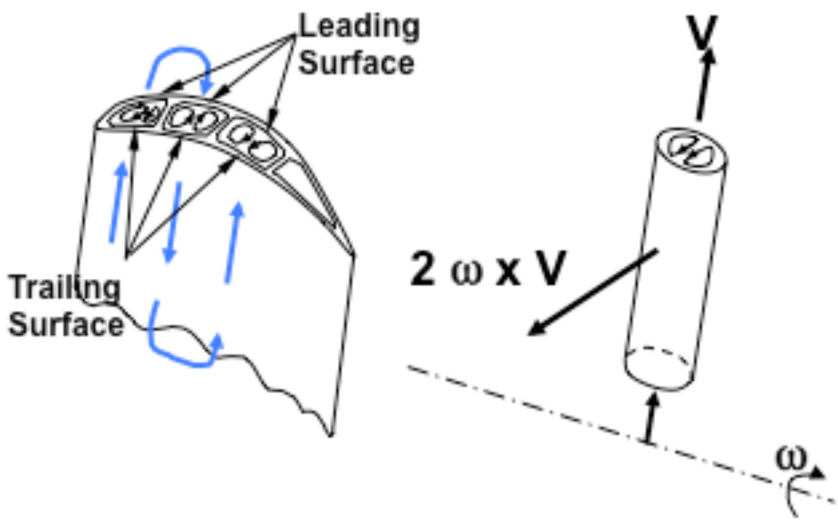

Figure 2: Secondary Flow Vortices due to Rotation Effects

On the right side of Figure 2 the flow with velocity, V, through a tube is illustrated. As the tube is rotated at angular velocity, ω, the flow field is altered from that of a stationary case to the rotational case. On one side of the tube the convective heat transfer would be aided by the secondary flow vortices whilst the other side would have a decreased effect. This effect is very important in predicting the performance of the turbine and the heat transfer distribution for the blade and is a primary concern in this study.

2.2 Test Rig Setup

A test cell deemed the Coriolis rig was first commissioned 4 years ago for rotating passage heat transfer investigation. It was designed to simulate the rotational effects on internal cooling passages. These rotational effects can then be studied and applied to full-scale engine designs through the use of various dimensionless similarity parameters. The Coriolis rig is a large rotating airfoil, enclosed in a thick steel case that is powered by a 100 horsepower motor. The total diameter of both arms of the rig is 8 feet and one of them is shown in Figure 3. One of the arms of the airfoil holds the test sections while the other is a balancing weight to keep the rig stable. The rig will be used to experimentally test the rotational effects on the new internal cooling passage design as opposed to the numerical simulation analyzed in Section 4.0.

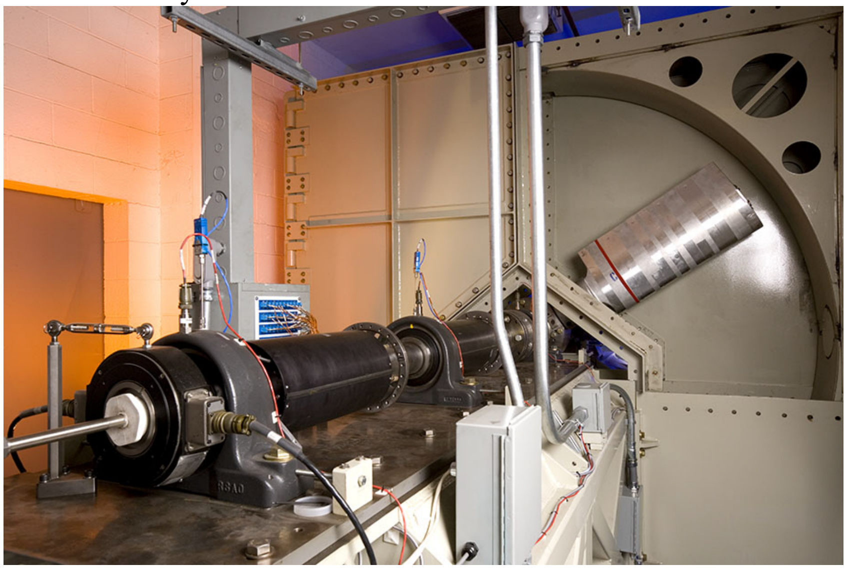

Figure 3: Coriolis Rig

Testing of the new internal cooling passage design at full engine conditions is not practical so this rig is scaled using dimensionless similarity parameters, namely the rotational number, Ro, and the Reynolds number, Re; both of which are given in Equation 1and Equation 2, respectively.

$$\mathrm{Ro} = \frac{V}{\Omega D} \qquad \text{Equation 1}$$

$$\mathrm{Re} = \frac{VD}{\nu} \qquad \text{Equation 2}$$

The characteristic length, D, was dependent on the inlet to the test section designed in Section 3.0. From other known conditions the flow was matched through theses similarity parameters.

3.0 Design of Test Section

3.1 Internal Cooling Passage Design

As mentioned in the introduction, Solar is working to design a new cooling passage to improve the efficiency of its engines. This new cooling passage design is similar to the cooling approach taken in aircraft gas turbine engines but is new concept at Solar. This cooling passage was designed to allow for a very high effectiveness and can lower the cooling flow by almost 2%. This can translate into a substantial increase in power output.

For the new internal cooling passage the flow enters through the base of the blade the same as a more traditional turbine blade but this design differs in that the flow splits into two separate flows; one going to the pressure side of the core and the other going to the suction side of the core. The new internal cooling passage has a metal 3D core that has four walls adjacent to the flow to increase the surface area for the convective cooling of the blade, thus increasing the total heat transfer from the blade to the cooling flow.

The core of the blade acts as a cold sink and surrounding the core is the cooling flow field that transfers heat from the hot outer wall of the blade through convection. Located along the length of the core are ribs that provide structural support to the core as well as separating it into 5 separate channels or “legs”. Conductive heat transfer between the hot outer surfaces of the blade to the core, acting as a cold sink, through the ribs is an additional benefit of the design. Throughout the cooling section channels are pin fins, which are heat transfer enhancement technique that place cylindrical shaped fins perpendicular to the flow to increase the forced convection cooling. Due to vortex shedding of the cylinders as well as other effects, turbulence is increased; which in turn better mixes the flow and aids in the cooling. The increase in cooling comes at the cost of the pin fin section having a high-pressure drop across the section and requiring a higher pressure at the inlet to the section to pump the fluid through.

Generally the high-pressure drop across the pin fin section would require the cooling flow to be taken from a later stage in the compressor, and thus at a higher pressure, for the case of a multi-pass serpentine channel. In the case of the new cooling passage the high-pressure drop across the pin fin section is mitigated by pumping action caused by the rotational acceleration. An additional benefit from pin fins in this cooling scheme is the conductive heat transfer acting in the same manner as the support ribs from the hot outer surface of the blade to the core.

3.2 Area of Interest for Test Section

Due to the effect of rotation, the mass flow is not distributed evenly between the pressure side of the core and the suction side of the core although the initial design at static conditions did have an even flow split. The degree that the rotation affects the flow separation varies between each of the legs with some legs having a higher sensitivity to rotation than others. Since the mass flow rate through the two sides of the cooling passage are not equal at rotating conditions, the heat transfer is not equal and can pose a problem if not addressed. Thus, the primary area of interest to be considered when designing the test section is the section where the channel splits the flow diverting some of it to the pressure side and the rest to the suction side. The leg with the most deviation from static case is the one considered in the analysis.

3.3 Design of Test Section

A test section was developed in collaboration with senior engineers to test the effects of rotation on the new internal cooling scheme in the Coriolis rig. As previously stated, the main area of interest was to test how sensitive the flows split between the pressure and suction side is due to the effects of rotation. The test section was scaled up to the 4.5 times the actual size of the blade. This scaling was not arbitrary but instead was the maximum size that the test section could be and still fit into the Coriolis rig. A plenum section was designed and added on the outlet of the flow through the cooling passage to provide a backpressure similar to the pressure that would be seen in the actual cooling passage. A three-dimensional view of the CAD drawing of the test section is shown in Figure 4.

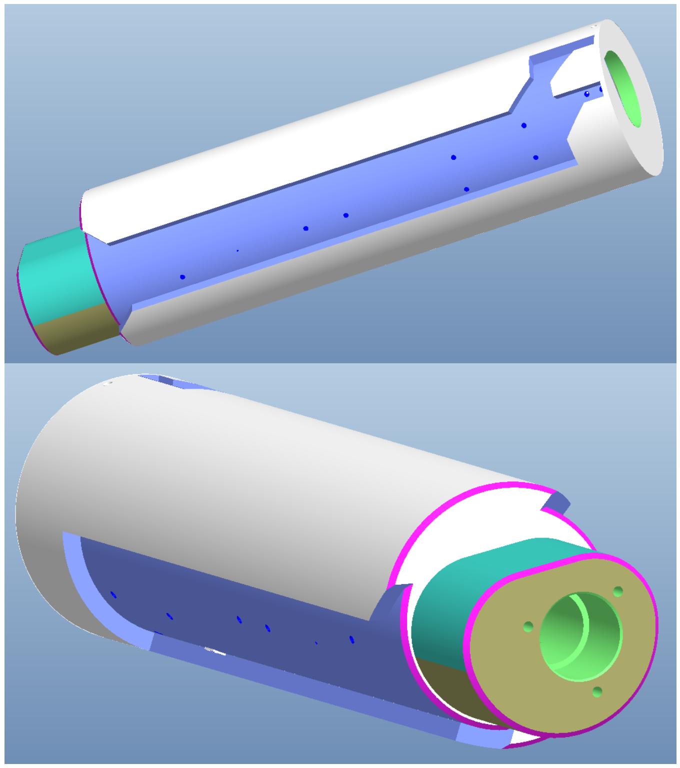

Figure 4: CAD Drawing of Test Section

Static pressure taps and thermocouples were placed throughout the test section so that measurements could be taken to understand the flow within the test section. Recessed areas around the outside of the test section were included to be able to easily rout the static pressure lines and thermocouple wires to the DAQ system. Also included in the design was a flow straightener before the inlet of the cooling passage to provide a more uniform flow to the area of interest. The location of this flow straightener can be seen in the bottom imagae of Figure 4 where the green section has a small lip for the straightener mesh to be located.

4.0 CFD Analysis of Test Section

Star-CCM+ is robust and versatile computational package used to pre-process, solve, and post-process fluid dynamics simulations. It was used for all aspects of the analysis of this test section from grid generation to visualizing the results.

4.1 Mesh Generation

Two different polyhedral meshes were created in Star-CCM+ for the geometry of the test section. Both meshes were created using the polyhedral meshing algorithms in Star-CCM+. The meshes were created using prism layers to better resolve the mesh density near to the walls in the boundary layer region to more accurately capture the detail in the boundary layer. The first mesh contained 2.36 million cells and was the mesh used to produce the results for the analysis. The second mesh was much finer mesh with 10.3 million cells. The second mesh decreased the base size by approximately 25% and added prism layers to test grid independence of the solution computed using the coarse mesh. It was found that there was less than a 2% difference at all static pressure probe locations between the coarse and fine grids for pressure and temperature.

4.2 Flow Scaling

To have the data be relevant for the actual engine conditions the flow was scaled using the Reynolds number and the Rotational number. The Mach number is not a factor since the flow in both the engine and the rig is less than 0.3 and is assumed incompressible. The buoyancy effects due to heat transfer were assumed to be negligible for the purposes of this test and only the fluid mechanics were considered in the analysis. Future simulations may include theses effects but their effect would not be noticeable in the experimental validation of this test. The Coriolis rig operates using room temperature air and rotates at a much lower speed than the actual engine conditions and it has a rotational arm that is over 3 times larger than the effective rotation arm of the engine. From known operating conditions of the engine, the mass flow rate through the internal cooling passage can be determined through simple algebraic manipulation of the continuity equation, Equation 1, and Equation 2.

4.3 Boundary Conditions and Models

A steady, 3D, turbulent solver was used in the numerical simulation. The full RANS were solved with density being found using the ideal gas law. A realizable k-epsilon turbulence model was used to resolve the turbulence. The dynamic viscosity and thermal conductivity were found based on Sutherland's Law. The molecular weight, specific heat, and turbulent Prandtl number were all held constant and set to the default setting for air.

At the base of the test section a mass flow inlet was specified with value of the mass flow being specified. The pressure at the outlet of the plenum was set to atmospheric conditions. To resolve the turbulence in the k-epsilon turbulence model the length scale and turbulence intensity were specified. A length scale that with the same order of magnitude as the width of the inlet section was used and the turbulence intensity was set to 1% based on real flow conditions of the coolant flow.

Numerical simulations of the flow were computed using both a rotational reference frame and a static reference frame. For the rotational case a new reference frame was created in Star-CCM+ to have the test section rotating in the negative x-direction at 750 revolutions per minute.

4.4 Results

It was found that the secondary flow vortices change the velocity profile of the flow through the inlet for the rotation case and the effect propagates as the flow moves downstream. This effect is illustrated in Figure 5.

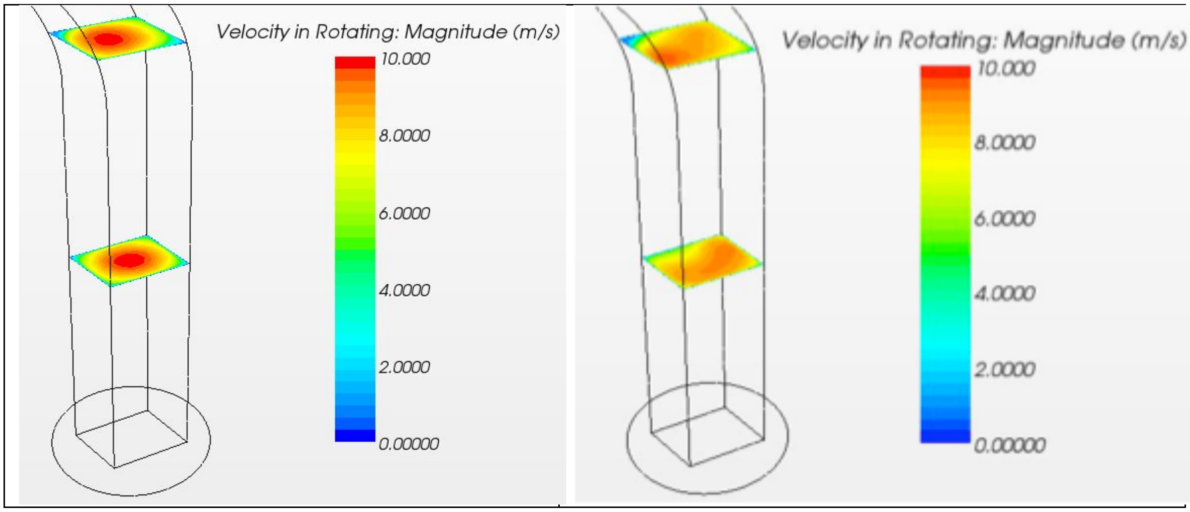

Figure 5: Velocity Contours in Inlet Section of Test Section for Static Conditions (Left) and Rotating Conditions (Right)

As the flow moves downstream, shown as upwards in the figure, in the static case it can be seen that there is predictable velocity profile that is comparable to any flow through a duct. The flow does not change in a qualitative sense as it moves through the inlet section. Contrarily, the flow is much more complicated in the rotational case. The secondary flow vortices induced from the rotational effects cause a sort of flattening of the peak velocity and a higher velocity in the region nearer to one wall whilst a lower velocity in the region to the opposite wall. This effect is more pronounced at the plane further downstream where the low velocity flow causes an area of recirculation that can be seen in Figure 6.

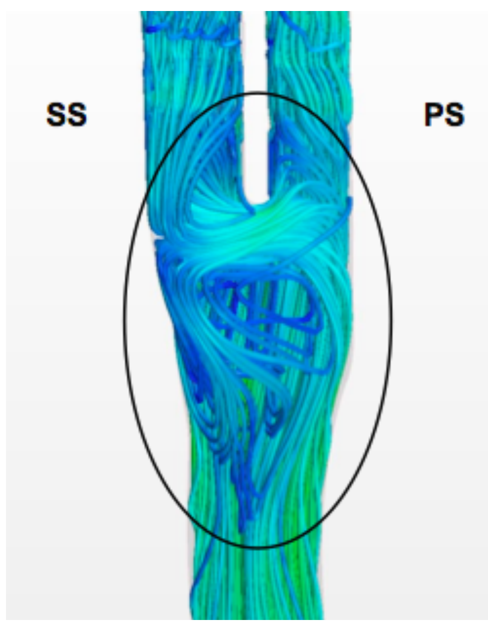

Figure 6: Streamlines – Flow Split

This area of recirculation is likely to be a primary factor in the tendency of the flow to favor the pressure side. The flow tends to be forced towards the pressure side and has to overcome the effect of rotation to reach the suction side. This can been seen in the streamlines in Figure 6 that are directed from the pressure side to the suction side in the circled region.

The rotational effects tend to split the flow in favor of the pressure side channel with 54% of the mass flow going to the pressure side. This trend is the same trend observed as the one in the full CFD model completed in a previous study at Solar. The percentage of flow through each channel was a few percent off from the full engine model and further work could be done to refine the Coriolis rig models. Due to the inherently unsteady nature of flow through pin fin sections, unsteady simulations may be considered in the future.

5.0 Transient Liquid Crystal Test

5.1 Thermocouple Mapping

A static transient liquid crystal test was designed to validate the internal heat transfer coefficients for the new 3D core cooling passage. This test was not designed to incorporate the rotational effects, as was the intent for the previous test. For this test a SLA model was designed by engineers at Solar to be able to disassembled and have the inside painted with a liquid crystal material that changes color with respect to temperature. Figure 7 shows a previous liquid crystal test done at Solar to map out the convective heat transfer distribution for a multi-pass serpentine channel design.

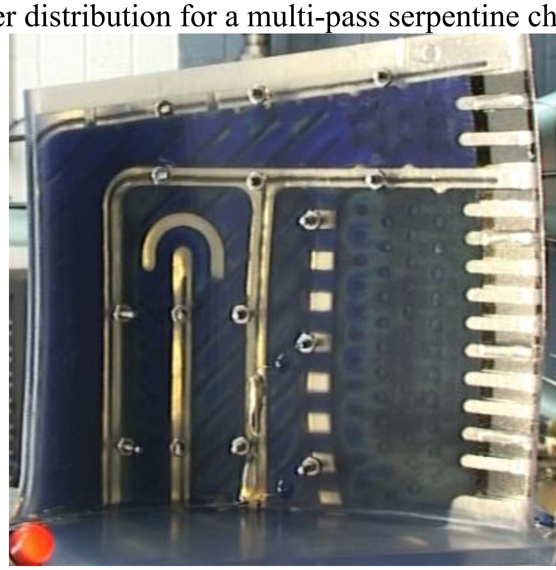

Figure 7: Transient Liquid Crystal Test of Multi-Pass Serpentine Blade

To be able to accurately predict the heat transfer coefficients from the test, a series of thermocouples must be placed throughout the test section to obtain temperature measurements at discrete locations. For areas between the thermocouples the heat transfer distributions can be determined using existing correlations.

For this static test redundant thermocouples were placed at the inlet of each leg of the cooling passage near the flow split location. Thermocouples were also placed at any bends in the legs and also evenly spaced sections near the outlet close to the trailing edge. A total of 40 thermocouples were needed to be able to obtain temperature data at the necessary locations. From the thermocouples and the data acquired from the liquid crystal test, the heat transfer distribution can be obtained for the entire cooling passage.

5.2 Pressure Tap locations

For this test, pressure taps were of a secondary concern. A previous test was conducted earlier in the design phase to map out the entire pressure distribution. In turn, only 6 pressure taps were used near the outlet of the legs close to the trailing edge of the blade as a check to compare to the previous data.

6.0 Conclusions

The results from this study were compared to results from a full-scale engine conditions CFD model and same trends were established for the flow to favor the pressure side of the channel. The percentage of flow through each side of the channel did not match the full engine model and this may be due to the vast difference in rotational speeds from the full-scale to the rig scale. The numerical simulation of the flow will provide useful data to compare to the experimental results. The recirculation region shown in Figure 6 is present in both the static and rotational case but much more pronounced in the rotational case. This may lead to possible changes in the geometry of the cooling passage to mitigate these effects. Also, due to the unsteady nature of the flow and the turbulent nature of the flow, more CFD analysis may be warranted with a different turbulence model and possibly an unsteady solver to better resolve the effect of rotation on this cooling passage design.

7.0 Acknowledgements

Being able to work at Solar Turbines gave me a great deal of insight into the development of industrial gas turbine engines. The experience gained at Solar helped bring together previous knowledge of gas turbines and greatly expanded my understanding of how a gas turbine engine is designed and operated. This knowledge and understanding will stay with me as I continue my education and develop as an engineer.

I would like to thank John Mason for his role in selecting me to work in the Aero, Thermal, and Performance group and for his guidance and help by way of answering questions that arose pertaining to my understanding of turbines. I would also like to thank my mentor Dr. Yong Kim for all of his guidance and patience with me throughout the fellowship and Dr. Hee-Koo Moon and the rest of the heat transfer group for sharing their knowledge. A great deal of thanks also goes out to Southwest Regional Institute for organizing the UTSR Fellowship program and specifically Andrea Barnett for helping in each step of the process and answering all questions that arose regarding the fellowship.
Also I would like to thank Ken Ridler for all of his help and patience in designing the CAD models.

Last but certainly not least, I owe much thanks and gratitude to Dr. Andrew Nix for be a great advisor and his constant guidance and encouragement to pursue knowledge in gas turbine engines.