Study of Heat Transfer Effects in Cooling Passages in Gas Turbine Blades

Faculty Advisor: Ryoichi Amano    Author: Michael Stefik  Co-Author: Mandana Saravani & Nicholas DiPasquale

University of Wisconsin-Milwaukee, Department of Mechanical Engineering

Motivation

Thermal efficiency has become a crucial factor when designing a gas turbine. To increase thermal efficiency, the gas turbine blade cooling channels need to be optimized. This optimization will allow the turbine blades to withstand hotter temperatures for extended periods of time while reducing the amount of air needed to achieve this cooling effect.

Conclusions

1. Heat transfer rate increases as the air inlet velocity increased due to an increasing Reynolds Number.
2. Heat transfer rate becomes highest at a larger rotational speed because of more air turbulence and buoyancy forces.
3. Experimental results followed the same trends as the experimental measures, which validated the experimentally measured heat transfer rates.

Future Work

1. Future cooling channels will utilize design modifications to reduce the overall airflow while increasing the air turbulence and heat transfer using the heat transfer correlations found in this study.
2. Conduct more experiments at different rotational speeds to increase knowledge of the effect rotational speed on heat transfer.
3. Add Rib Turbulators to understand how this modification affects heat transfer.

Acknowledgments

Special thanks to Dr. Ryoichi Amano and the UWM Mechanical Engineering Department for advice and direction with the completion of this study. Special thanks to Mandana Saravani & Nicholas DiPasquale for their knowledge and contributions toward this study.

For Further Information

For further information on this study please feel free to contact me at mstefik@uwm.edu

Computation Fluid Dynamics (CFD) Setup

Goal: Understand how the Reynolds Number and the rotational speed of the channel affect the overall heat transfer

Computational Parameters

- Computational Flow Dynamics code for analyzing both flow and heat transfer were employed.
- Large Eddy Simulation (LES) used for simulation analysis
- Bottom wall heat flux maintained at 2.7 MW/m²
- Varying inlet velocities and rotational speeds are used to analyze the affects on heat transfer within the test channel

Experimental Setup

Goal: Validate CFD data to determine the trends associated with increasing the Reynolds number and Rotational Speed

Test Equipment

- Air blower regulates air velocity through test channel
- 7.5 hp, three-phase induction AC motor regulates rotational speed
- Thermocouple probes used to measure bulk fluid temperature
- High-Resistance Heating strips introduce heat into test-section
- Data measured through data loggers (MWTC-D-K-915) and a wireless receiver (MWTC-REC1-915)

Experimental Results

1. Heat transfer rate increases as the air inlet velocity increased due to an increasing Reynolds Number.
2. Heat transfer rate becomes highest at a larger rotational speed because of more air turbulence and buoyancy forces.
3. Experimental results followed the same trends as the experimental measures, which validated the experimentally measured heat transfer rates.

For Further Information

For further information on this study please feel free to contact me at mstefik@uwm.edu

Acknowledgments

Special thanks to Dr. Ryoichi Amano and the UWM Mechanical Engineering Department for advice and direction with the completion of this study. Special thanks to Mandana Saravani & Nicholas DiPasquale for their knowledge and contributions toward this study.

For Further Information

For further information on this study please feel free to contact me at mstefik@uwm.edu