Establishing a clean and renewable energy supply is the preeminent engineering challenge of our time. Turbines, in some form, are responsible for more than 98 percent of all electricity generated in the United States and 100 percent of commercial and military air transport. The operation of these engines is clearly responsible for significant consumption of hydrocarbon fuels and, in turn, emission of greenhouse gases into the atmosphere. With such wide-scale implementation, it is understood that even the smallest increase in the operating efficiency of these machines can lead to enormous improvements over the current energy situation. These effects can extend from a reduction in the emission of greenhouse gases to lessening the nation’s dependence of foreign energy sources to lower energy prices for the consumer.

The prominent means of increasing engine efficiency is by raising the ‘Turbine Inlet Temperature’ - the temperature of the mainstream flow after combustion, entering the first stage of the turbine section. The challenge is presented when these temperatures are forced beyond the allowable limits of the materials inside the machine. In order to protect these components, active cooling and protection methods are employed. The focus of this work is the development of more efficient means of cooling ‘hot’ turbine components. In doing so, the goal is to maximize the amount of heat removed by the coolant while minimizing the coolant mass flow rate: by removing a greater amount of heat with a lower coolant mass flow rate, more compressed air is left in the mainstream gas flow for combustion and power generation.

This study is an investigation of the heat transfer augmentation through the fully-developed portion of a narrow rectangular duct (AR=2) characterized by the application of dimples to the bottom wall of the channel. Experimental testing and numerical modeling is performed for full support and validation of presented findings. The geometries are studied at channel Reynolds numbers of 20000, 30000, and 40000. The purpose is to understand the contribution of dimple geometries in the formation of flow structures that improve the advection of heat away from the channel walls. Experimental data reported includes the local and Nusselt number augmentation of the channel walls and the overall friction augmentation throughout the length of the duct. Computational results validate local Nusselt number results from experiments, in addition to providing further insight to local flow physics causing the observed surface phenomena. By contributing to a clearer understanding of the effects produced by these geometries, the development of more effective channel-cooling designs can be achieved.

Major: Mechanical Engineering

Educational Career:
Bachelor’s of Mechanical Engineering, BS, 2009, University of Central Florida

Committee in Charge:
Jayanta Kapat, Sc.D, Chair, Mechanical, Materials, and Aerospace Engineering
Jihua Gou, Ph.D, Mechanical, Materials, and Aerospace Engineering
Ali Gordon, Ph.D, Mechanical, Materials, and Aerospace Engineering
Bijay Sultanian, Ph.D, P.E., MBA, Mechanical, Materials, and Aerospace Engineering

Approved for distribution by Jayanta Kapat, Sc.D, Committee Chair, on May 28, 2010.

The public is welcome to attend.