KEYNOTE LECTURE

FLOW PHYSICS OF FILM COOLING JETS IN CROSSFLOW

Prof. Sumanta Acharya

Ring Companies Endowed Chair and Department Chair
Mechanical Engineering Department,
University of Memphis, USA

Prof. Sumanta Acharya holds the L. R. Daniel professorship and the Fritz & Francis Blumer professorship in the Department of Mechanical Engineering at Louisiana State University (LSU). He is the founding director of the Center for Turbine Innovation and Energy Research which focuses on energy generation and propulsion research. Professor Acharya received his undergraduate degree from the Indian Institute of Technology Kharagpur, and his PhD from University of Minnesota and later joined as a faculty of Mechanical engineering at LSU. He has also developed a multifaceted research program in heat transfer with a focus on gas turbine heat transfer and computational heat transfer. Professor Sumanta has been at the National Science Foundation (NSF) as the Program Director of the Thermal Transport Program since 2010.

ABSTRACT

Gas turbine components downstream of the combustor are exposed to high temperature gases in excess of 1600 ˚C and have to be cooled to maintain material temperatures below the reliability limit. In film cooling, coolant flow is injected through inclined holes on the airfoil and end wall surfaces with the intent of providing a protective coolant film over the metal surfaces. The coolant jet ever rapidly mixes with the hot cross flow and loses surface coverage. Different strategies for keeping the coolant jet close to the surface and reducing mixing have been explored. These include diffusing the holes in the forward and/or lateral directions, introducing pulsations and shaping the hole exit via trenches and craters.

In this talk, the flow physics of the film cooling jet injected into a hot cross flow will be discussed using high fidelity numerical calculations (e.g., Large Eddy Simulation). Flow structures associated with the coolant jet (e.g., the horse shoe vortex structures, the kidney pair vortex, and the shear layer vortices.) will be identified and the mechanisms associated with their origin and evolution discussed. The energetic modes that contribute to the cooling of the surface are identified and analyzed. Modal analysis of the simulation data is performed to understand what modes and associated flow structures play a key role in determining the cooling effectiveness and surface temperature variation. The important role of inlet turbulence and boundary conditions will be presented. The effect of hole shaping, exit hole geometry and jet pulsations are explored via simulations and their impact on the surface heat transfer is outlined.