HIGH-FIDELITY NUMERICAL SIMULATIONS OF TURBINE VANE HEAT TRANSFER UNDER FREESTREAM TURBULENCE

Allocation: Director Discretionary/62.5 Knh
PI: Sumanta Acharya[1]
Collaborator: Yousef Kanani[1]

[1]Illinois Institute of Technology

EXECUTIVE SUMMARY

Designing efficient and reliable gas turbines requires detailed knowledge of the aerodynamics and heat transfer over the turbine surface. Since experiments at realistic engine temperatures (~3000°F), high pressures (20–40 bars) and rotor speeds (~10,000 rpm) are not possible, designers have to rely on data that can be collected over lower parameter ranges or from simulations. Accurate predictions are stymied by the complexities in the flowfield that include freestream turbulence and wakes, high strain rate, high flow acceleration, and the like. Time-averaged equations require modeling and are generally inaccurate. Higher-fidelity simulations, as in the present study, require large computing resources. Rapid advances in the capability of supercomputers such as Blue Waters allow researchers to perform high-fidelity simulations in complex geometries under realistic conditions, and to accurately predict and obtain a fundamental understanding of the flow and heat transfer physics that can then be used for rational design of engineering models.

RESEARCH CHALLENGE

The flow upstream of the turbine airfoil is typically turbulent and characterized by large length scales. The boundary layer on the pressure and suction surfaces of the airfoil experiences this high level of freestream turbulence and can transition into a turbulent boundary layer downstream on the surface. Predicting the transition to turbulence typically requires a very fine grid to resolve the smallest scales in turbulence; this, in turn, requires a very large mesh (on the order of 50–150 million nodes for a realistic Reynolds number) and can make the computations time consuming, even on large supercomputers. Such computations are, therefore, out of reach for most of those in industry, where using Reynolds-averaged turbulence modes is commonplace despite their lower accuracy. The available transitional models are not universally applicable in different geometries and typically perform poorly. To capture the transition to turbulence, the finest scales available in the fluid flow should be resolved, which requires significant computational power.

In the current study, the flow and heat transfer over a turbine vane under realistic engine conditions of high turbulence are simulated using Large-Eddy Simulation (LES). We aim to correctly predict the flow and heat transfer characteristics on both the suction and pressure side of the vane for different levels of elevated freestream turbulence. Experimental observations suggest that the suction-side boundary layer transitions to turbulence, with the transition location moving upstream at higher turbulence levels. Heat transfer also is enhanced on the pressure side of the vane prior to the transition to turbulence.

METHODS & CODES

The vane geometry selected for the computational study is identical to the experiments conducted by Varty and Ames [1]. We studied a linear cascade, which is represented in the computations by a single vane with periodic boundary conditions imposed in the pitch direction. The periodic boundaries are separated by the pitch P=0.773L, where L is the chord length and is selected as the length scale. To capture the secondary flow effects, the numerical domain includes the whole span of the vane between two endwalls with the spanwise length of 0.5L.

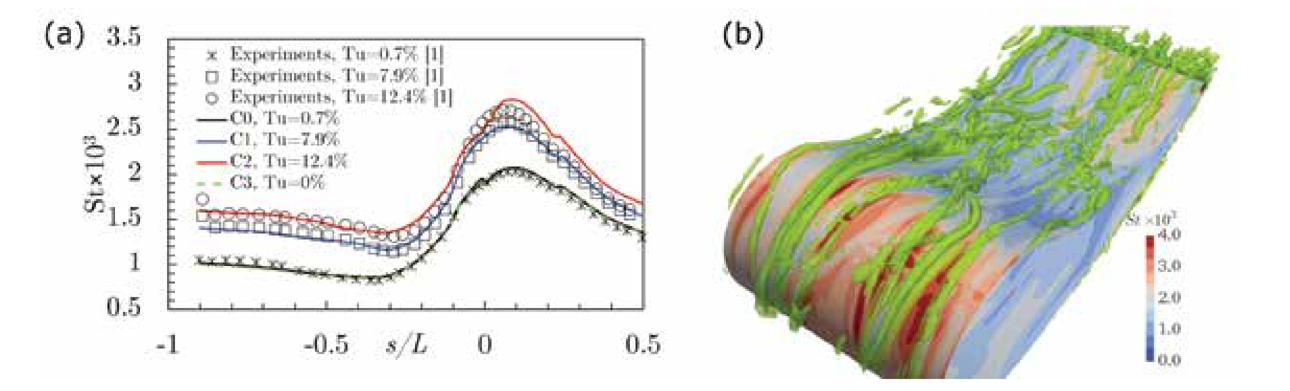

Figure 1: (a) Stanton number distribution near the leading edge and pressure side; and (b) flow structures and instantaneous Stanton distribution over the turbine vane.

We selected the exit chord Reynolds number of Re=510,000 and inflow turbulence levels of 0.7%, 7.9%, and 12.44%, with length scales of 0.16, 0.04, and 0.074 of the chord, also as in the experimental study of Varty and Ames [1]. The inlet boundary is located at x=−0.51L where L is the chord length. We used the Synthetic Eddy Method (SEM) [2] to generate isotropic and homogenous velocity perturbations with the specified turbulence characteristics (i.e., mean velocity, turbulence intensity, and length scale).

We treated the outflow boundary with the homogeneous Neumann condition for both velocity and pressure; it is located 0.4L away from the trailing edge axially, which allows the wake to exit the domain in the periodic direction. All walls are treated with the no-slip condition for the velocity. We imposed a constant heat flux on the vane for the temperature solution.

We solved the incompressible Navier–Stokes equations and the temperature equations using the Pressure-Implicit with Splitting of Operators algorithm with the OpenFOAM flow finite volume code. Air is assumed as the working fluid. We discretized all spatial derivatives with a second-order central scheme. The central difference scheme is blended with a second-order upwind scheme for the convective terms to ensure the stability of the numerical simulations. We used the Quadratic Upstream Interpolation for Convective Kinematics method to approximate the convection term in the passive scalar equation. We used the second-order backward scheme to discretize the temporal derivatives. We conducted the LES using the WALE subgrid-scale stress model.

The heat transfer augmentation prediction over the pressure side of the vane is less computationally demanding than the prediction of the transition to turbulence and requires a mesh with 11 million grid points. Simulations capable of predicting transition to turbulence require large grid sizes in the order of 100 million grid points, which is intensive both computationally and in terms of storage.

RESULTS & IMPACT

Current numerical simulations are capable of accurately predicting the heat transfer augmentation on the leading edge and over the pressure side of the vane (Fig. 1). We have clearly described the mechanism on which this augmentation occurs for the first time. The freestream turbulence generates vortex tubes wrapped around the leading edge (Fig. 1), which perturbs the velocity inside the boundary layer and forms low- and high-speed streaks. These streaks retain their signature downstream up to the trailing edge of the vane. There is a strong correlation between the instantaneous heat transfer and the velocity streaks.

We are using a more refined grid to predict the transition to turbulence on the suction side. Formation of turbulent spots on the suction side of the vane is captured with 88 million grid points

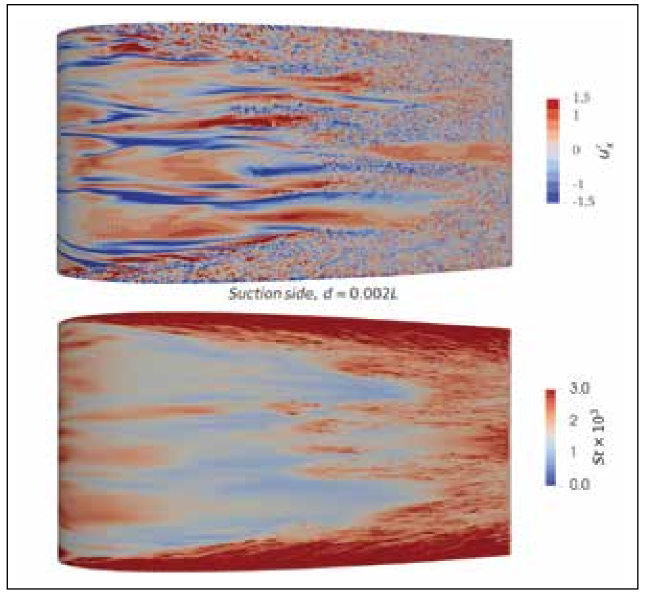

Figure 2: (Top) Formation of turbulent spots visualized by instantaneous velocity fluctuations close to the surface and (bottom) its effect on surface heat transfer on the suction side of the vane.

(Fig. 2). Turbulence spots form and grow in size on the suction side of the vane and eventually trigger the entire boundary layer into a turbulent state. Predictions agree with the measurements and transition locations [1]. We have also correctly predicted the heat transfer enhancement due to the turbulence spots.

WHY BLUE WATERS

Blue Waters provides a powerful tool to conduct high-fidelity simulations. Without accessing the computational resources at the scale that Blue Waters provides, performing these simulations is practically impossible. Further, analyzing the temporal behavior of the flow and heat transfer field requires data storage of terabytes of data, which is made possible by both short- and long-term storage solutions provided by Blue Waters.