LARGE EDDY SIMULATIONS OF FILM COOLING FROM CYLINDRICAL AND SHAPED HOLES

David Houston Leedom
M.S. Candidate
Faculty Advisor: Dr. Sumanta Acharya

ABSTRACT
Large Eddy Simulations (LES) of cylindrical, laterally diffused, and console holes are performed in order to both investigate new cooling hole geometries and give weight to the use of LES instead of RANS for film cooling research.

INTRODUCTION
Modern gas turbine engines operate at high temperatures in order to be more efficient and generate more power. The process gas temperatures often exceed the melting temperature of the materials used by several hundred degrees Celsius. The surfaces of the blades and vanes in contact with the hot gas must be kept cool in order to prevent melting and creep failure. Cooling may be affected internally (by passing cool air through the inside of the blades) or externally via film cooling.

Film cooling is the injection of cool air onto the surface of the blade in order to provide a thermal buffer between the hot process gas and the blade surface. This injection would ideally happen through a uniform slot, but the structural strength of such a slot seldom suffices to support the aerodynamic loads. Alternatively, discrete coolant injection holes are more structurally sound, but the effused jets are prone to blowing off of the surface or otherwise providing non-uniform cooling. In order to combat this, discrete holes are often shaped to provide more slot-like performance [1].

Computational tools are of great use in this area because many hole shapes can be examined in greater detail for less cost than through physical experiments. Validation of the codes used is critical to ensure agreement with the physical case. Difficulties arise because the flow is unsteady and turbulent. The current industry standard is turbulence modeling via Reynolds Averaged Navier-Stokes (RANS). RANS comes from temporal filtering of the Navier-Stokes equations, and models all scales of turbulence within the flow.

As an alternative to RANS, one may fully resolve all of the turbulent eddies down to the Kolmogorov scales (at which the eddies dissipate into thermal energy). This method of simulation is known as Direct Numerical Simulation (DNS). DNS requires a grid much finer than that required in RANS and therefore requires more computational resources. DNS is possible on modern computers, but is too computationally intensive to be practical for widespread use.

LES can be viewed as an intermediate level between RANS and DNS. In LES, the turbulent eddies are resolved (as they are in DNS) down to the scale of the grid. The remaining smaller eddies are accounted for via a sub-grid scale model. This is possible because in LES, the Navier-Stokes equations are filtered spatially instead of temporally as they are in RANS.

NUMERICAL METHOD
The code used in this study is an in-house parallel code called Chem3D which is being developed by LSU’s Turbine Innovation and Energy Research (TIER) group. The code solves the Navier-Stokes equations in their compressible-conservative form using a finite volume based discretization. The flow domain is divided into rectangular blocks of hexahedral cells; each block is given to one processor to solve. The parallel nature of the code allows for computation on large grids (about 4.5 million cells in this case) in a relatively short amount of time. The implementation of LES uses a dynamic Smagorinsky model by which the model coefficient in the eddy viscosity expression is evaluated dynamically via the Germano identity at each physical time step [2].

GRIDS AND BOUNDARY CONDITIONS
The basic layout of a flow domain for this study with the boundary condition enforced at each face is shown in Figure 1. A law-of-the-wall velocity profile with a boundary layer thickness equal to two hole diameters is enforced at the profiled inlet. The feeding plenum chamber’s inlet is set to provide a certain mass flow rate of coolant per unit width of cooled blade surface to account for the pitch of each hole geometry being different.
The three geometries compared in this study are cylindrical, laterally diffused, and console [3] holes. All holes are inclined at an angle of 35 degrees to the surface. The cylindrical hole has a length to diameter ratio of 3.5. The laterally diffused hole spreads out in the spanwise (pitch) direction toward the surface of the blade with the intent of slowing the effused jet down to minimize penetration of the jet into the process air. The console hole begins as a round hole, but smoothly transitions to a uniform slot at the surface. During the transition, the cross sectional area of the hole decreases, which accelerates the coolant. This higher-speed cooling jet is preferable from an aerodynamic loss point of view (as compared to the laterally diffused hole). Sketches of the geometries are shown in Figure 2.

Figure 1: Flow domain and boundary conditions

Figure 2: Sketches of cylindrical (left), console (middle), and laterally diffused (right) hole geometries

RESULTS

Plots of time-averaged midplane nondimensional temperature ($\theta$) are shown in Figure 3 to illustrate the differences in the jet effused from each of the three geometries under investigation. Note that jet penetration (compared to the cylindrical hole) is less in the laterally diffused case and least in the console case.

The results of the study confirmed the known trends toward improved film cooling from shaped holes. Inlet boundary conditions (boundary layer thickness, velocity profile, and turbulence levels) were found to have a huge effect on the effused jet’s film cooling potential. The choice of boundary conditions for the feeding plenum chamber was also found to be important because the low-speed vortices that develop from the inlet are stretched and intensified as they accelerate up the delivery tube (as shown in Figure 4) resulting in asymmetric flow fields. The turbulence generated by the jet was found to be highly anisotropic; this could be a problem for RANS models which assume that all turbulence is isotropic.

Figure 3: Nondimensional midplane averaged temperature

Figure 4: Vorticity in the direction of the delivery tube

ACKNOWLEDGEMENTS

This study was supported by the Clean Power and Energy Research Consortium funded through the State of Louisiana. Financial support in the form of a fellowship from the Louisiana Board of Regents is gratefully acknowledged. The high performance computing resources provided through the LSU Center for Computation and Technology, LSU Information Technology Services, and the Louisiana Optical Network Initiative are greatly appreciated.

REFERENCES