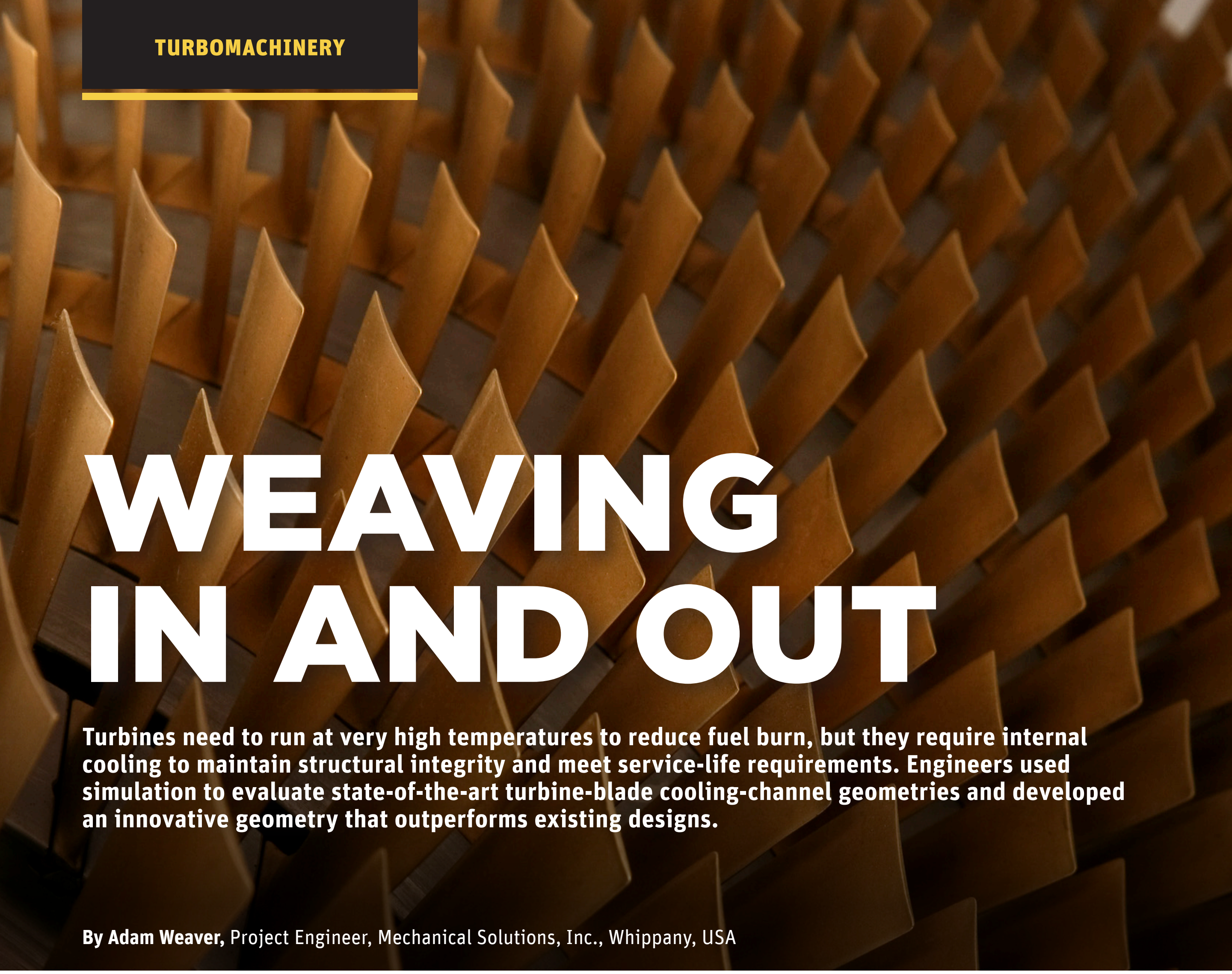


WEAVING IN AND OUT

Turbines need to run at very high temperatures to reduce fuel burn, but they require internal cooling to maintain structural integrity and meet service-life requirements. Engineers used simulation to evaluate state-of-the-art turbine-blade cooling-channel geometries and developed an innovative geometry that outperforms existing designs.

By Adam Weaver, Project Engineer, Mechanical Solutions, Inc., Whippany, USA

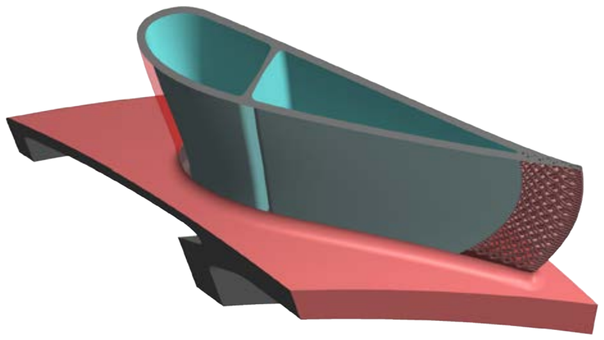
Natural gas–fueled turbines operate more efficiently at high temperature. Gas temperatures in the latest generation of turbines can reach 2,700 F, but blade materials can typically only withstand temperatures of about 1,700 F, so cooling is required. However, because the trailing edges of turbine-blade airfoils are very thin, there is little opportunity to cool them without endangering the structural integrity of the blade. Another challenge is that the amount of flow required to cool the trailing edge, as well as pressure losses within the cooling circuit, must both be minimized because the power required to generate this flow detracts from the efficiency of the engine.

Trailing edges with internal cooling channels are usually manufactured using investment casting, in which ceramic cores are used to create internal channels. This has limited designers to developing cooling channels with very simple geometries. However, new manufacturing technologies can now create more complex cooling channel shapes. But what shape is the best? The author and his colleagues at Purdue University used ANSYS CFX to investigate the performance of some new geometries and to create an innovative shape that delivers significantly higher performance.

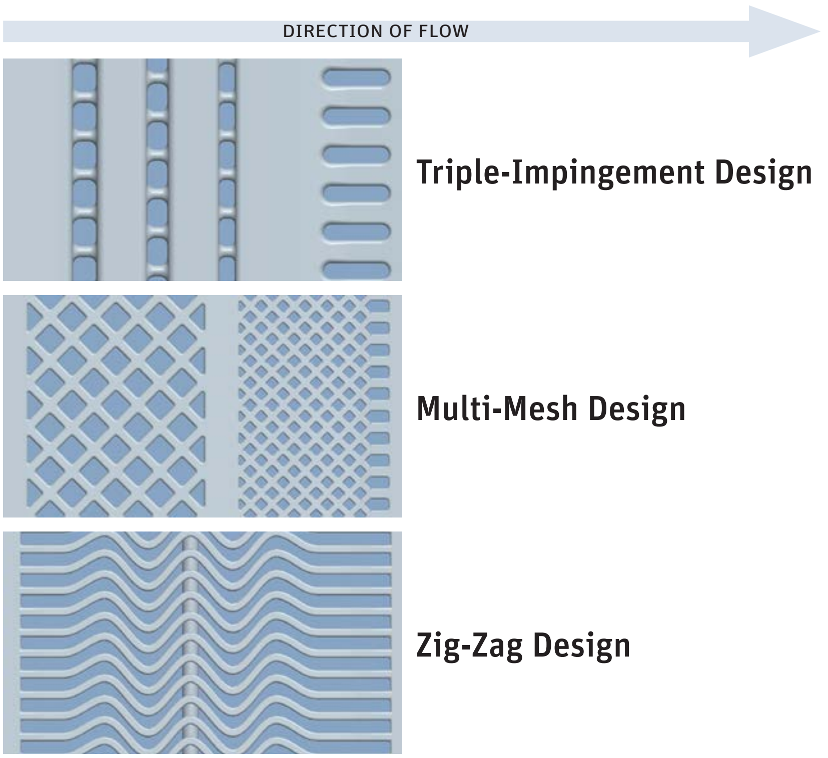
PRODUCING CERAMIC CORES FOR COOLING CHANNELS

Investment casting has been the technology of choice for producing gas turbine components with complex airfoil shapes and internal cooling passage geometries. For this type of casting, the lost wax process creates a ceramic shell with an interior that corresponds to the airfoil shape. One or more ceramic cores are positioned within the shell to form the internal cooling passages. Molten alloy is introduced into the shell and allowed to cool and harden. Then the shell and core are removed by mechanical or chemical means to leave the finished component. Until recently, design choices for cooling

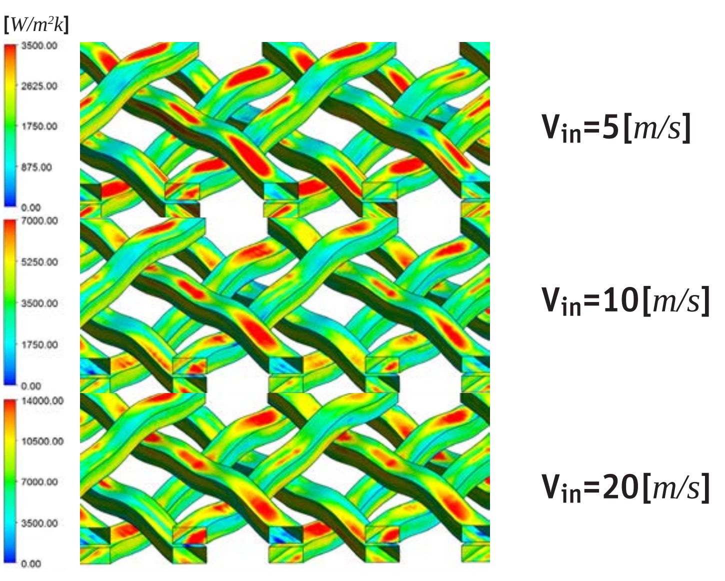
Because trailing edges of turbine-blade airfoils are very thin, there is little opportunity to cool them without endangering the structural integrity of the blade.



Position of new channels within the blade



Three new cooling channel geometries enabled by new manufacturing methods



Heat transfer coefficient contours for the weave design at different flow velocities

channels have been limited by manufacturers' abilities to produce ceramic cores with geometric features of about 1 or 2 mm. So only very simple cooling channel shapes could be used for turbine blade cooling passages.

These simple channel geometries are not very efficient at transferring heat from the surrounding metal. If the cooling channel is straight with a square cross-section, for example, a hot thermal boundary layer will form near the solid surface, but the flow in the center of the channel will absorb little heat. Much of the energy expended to cool the airfoil is wasted.

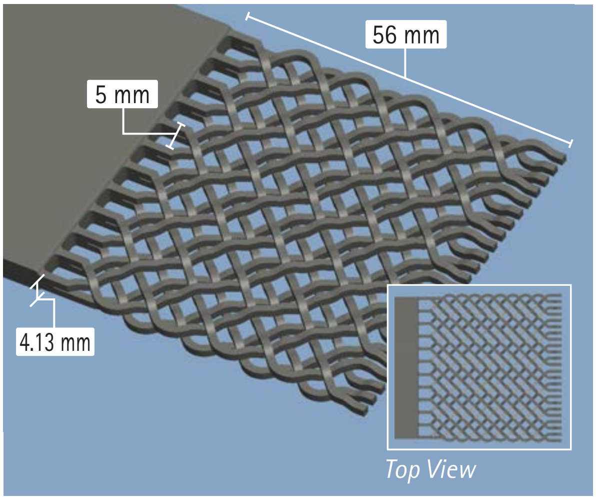
New manufacturing methods introduced in the last couple of years have broadened the scope for possible cooling channel geometries. One new manufacturing approach involves slicing a computer model of the cooling channels into slivers about 25 microns thick, which are used to create photomasks. The photomasks are in turn used to etch metal foils that are laminated together to make an extremely accurate 3-D cavity. The master pattern produces a mold from a material such as silicone, which can be used to cast ceramic material to very high levels of accuracy.

SIMULATING NEW COOLING CHANNEL GEOMETRIES

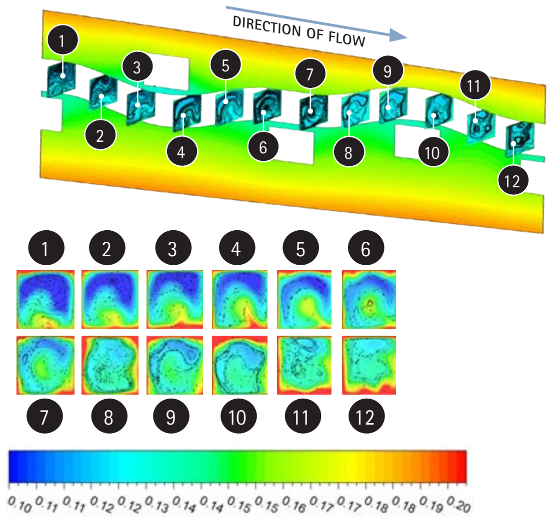
These new manufacturing methods led the team at Purdue to explore several cooling channels with more complex geometries that can break up the boundary layer and increase heat transfer.

- The triple-impingement design causes high-velocity jets to form through area contractions and directs the jets at perpendicular solid surfaces.
- The multi-mesh design uses square posts angled at 45 degrees to the flow direction to deflect and mix the flow with jet-to-jet interactions.
- The zig-zag design constricts the flow to channels and bends these channels to cause secondary flow structures and drives cool fluid to the walls.

FLUID STRUCTURE INTERACTION MAKES FOR COOL GAS TURBINE BLADES — ARTICLE
ansys.com/weaving101



▲ The weave design has a flow divider at the upstream end and aligned ducts at the trailing edge of the airfoil.



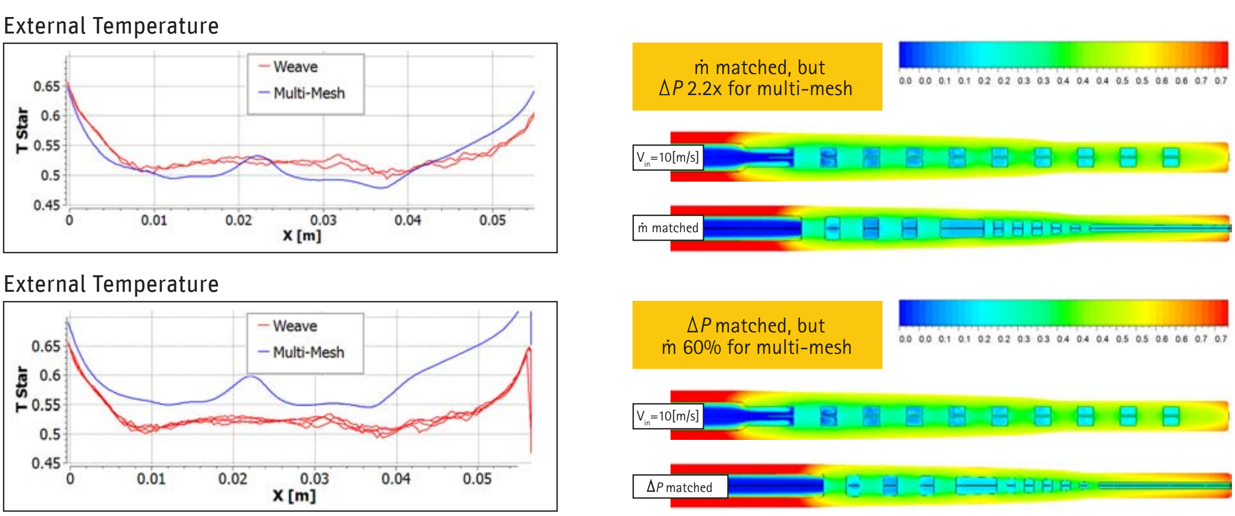
▲ Streamlines and temperature contours projected on cross sections of the weave cooling channel show how it breaks up boundary layers.

The Purdue team used CFD to evaluate the performance of these designs along with the author's own design, which has two layers of 45-degree–angled channels that undulate at intersections to avoid contact and constantly disrupt the flow direction. The team performed conjugate heat transfer analysis with the shear-stress transport (SST) turbulence model for all four designs. The temperature of the hot gas surrounding the trailing edge was maintained at 1,755 K, and the external heat transfer coefficient was set at 2,000 W/m^{2}K. The temperature of the cooling flow at the inlet passage was 673 K, and the pressure at the exit of the duct was maintained at 25 bars. Inlet coolant mass flow or pressure was varied to investigate a range of performance. The passages were 2 mm tall.

ANSYS meshing and fluids capabilities allowed the team to create models that consistently converged to a solution with node spacing as small as 0.001 mm, although more commonly the spacing was 0.01 mm near the wall where mesh density is most critical. Other codes that were tried in this application often did not converge at this node spacing. The researchers validated the CFD results against a test problem consisting of a planar jet impinging on a flat plate and achieved very good correlation.

IDENTIFYING THE BEST GEOMETRY

The researchers simulated the three existing designs to select equivalent points on the performance map so that they could directly compare any two designs. The results show that, for a given mass flow rate, the multi-mesh design performs about 5 percent better than zig-zag and 10 percent better than triple-impingement in terms of the amount of heat transferred. For a given pressure drop, zig-zag provides about 30 percent more heat transfer than triple-impingement and multi-mesh, but requires about twice the cooling flow rate compared to



▲ Temperature distribution using matched mass flow (top) and matched pressure drop (bottom) shows the superiority of the weave design over the multi-mesh design.

The weave design produces a higher flow rate of cool gas and thus provides 20 percent greater heat transfer, resulting in a relatively cooler blade.

multi-mesh and triple-impingement methods. The researchers concluded that multi-mesh is the best design among these three because it achieves maximum cooling with minimum cooling fluid flow.

The researchers then compared the multi-mesh design to the new weave design. With the same mass flow rate, multi-mesh performs similarly in terms of heat transfer, and even provides locations where external temperatures are lower than those of the weave design. However, multi-mesh's strenuous flow path resulted in more than two times the amount of pressure loss when compared with the weave design, demonstrating a much less efficient use of the cooling airflow. When pressure drop is matched between the two designs, the weave design produces a higher flow rate of coolant gas and thus provides 20 percent greater heat transfer, resulting in a relatively cooler blade.

The increase in heat transfer makes it possible to operate at a higher combustion temperature while maintaining the same blade temperature, thereby providing a higher thermal efficiency cycle for the engine. The amount of efficiency that can be gained will depend greatly on the specific engine used and the owner's preferred operating limits. The Air Force has awarded a Small Business Innovation Research (SBIR) grant to integrate the weave design into the Air Force Research Laboratory's research turbine at Wright-Patterson Air Force Base with the goal of ramping up to full-scale testing of the new design in 2016.

References

[1] Weaver, A.M.; Liu, J.; Shih T., I-P. A Weave Design for Trailing-Edge Cooling. The American Institute of Aeronautics and Astronautics (AIAA) SciTech Forum and Exposition. 2015.

[2] Weaver, A.M.; Liu, J.; Shih T., I-P; Klinger, J.; Heneveld, B.; Ames, R.; Dennis, R.A. Conjugate CFD of Three Trailing-Edge Cooling Designs. *Proceedings of ASME Turbo Expo 2013 Power for Land, Sea and Air.* 2013.

Keeping Turbines Cool: A Multi-faceted Challenge

The high and increasing operating temperatures of contemporary gas turbines present "hot section" engineers (including those who work with high-pressure turbines) with many challenges. Both materials and fluids engineers must contribute to overcoming heat transfer challenges. One part of the fluids solution is predicting the flow in the complex passages inside blades; a related concern is understanding the main flow path physics.

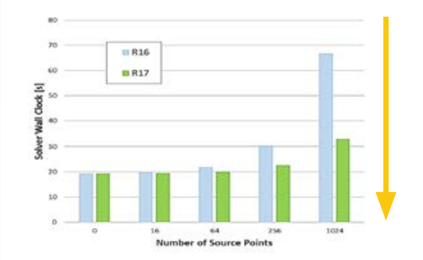
Much of the cooling air supplied by the compressor is ultimately delivered to the main flow path, providing a thermal cushion between the hot main flow and the metal surfaces that enclose that flow. The flow field is quite complicated because the interaction of the cooling jets and the main flow results in flow features that are much smaller than the main flow path scales. Thus, a full and detailed computational resolution is still a challenge. In addition, the flow is transient due to the relative motion between the rotor and stator blades.

One useful simplifying approach is to model the cooling jets as sources of mass and energy. Such sources could number in the hundreds. In the most recent release of ANSYS CFX, the workflow is streamlined so that any additional computational cost for inclusion of hundreds or thousands of source points is minimized.

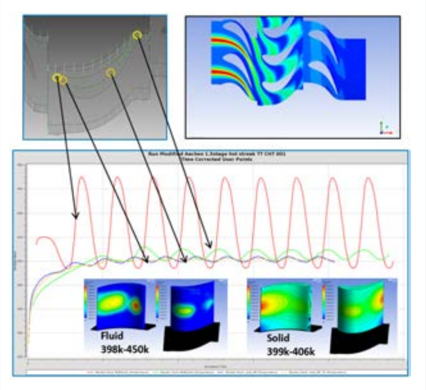
Engineers would also like to simultaneously solve for the heat transfer in blades and in the flow path. Again a problem of scale complicates matters, because the time scale for conduction is several orders of magnitude different than in the fluid. ANSYS provides a solution for transient blade row simulations to overcome this taxing problem.

With each release, ANSYS works to improve the fidelity, efficiency and scalability of its turbomachinery tools.

— **Brad Hutchinson,** Global Industry Director, Industrial Equipment and Rotating Machinery, ANSYS



A new capability reduces total CPU time by as much as 70 percent when a large number of source points are involved.



A transient blade row simulation with conjugate heat transfer shows the effect of thermal hot streaks and their temporal variation.