# Fluid Structure Interaction Makes for **Cool Gas Turbine Blades**

## An integrated simulation process improves performance without sacrificing longevity.

*By Michel Arnal, Christian Precht and Thomas Sprunk, Wood Group Heavy Industrial Turbines AG, Switzerland*
*Tobias Danninger and John Stokes, ANSYS, Inc.*

The blade geometry

The internal features of the blade geometry include the plenum (blue) and the cooling channels (gold).

In gas turbines, hot gas from the combustion system flows past the rotating turbine blades, expanding in the process. In order to reach desired levels of efficiency and power output, advanced gas turbines operate at very high temperatures. As a result, the components subjected to these high temperatures often require cooling. One method of cooling the turbine blades involves extracting air from a compressor and forcing it through a plenum and into channels inside the blade. While effective cooling of the blades can increase their lifespan, it can also reduce the thermal efficiency of the engine. It is therefore important to develop designs that extend component life while having a minimal effect on engine thermal efficiency. Numerical simulations that accurately capture the interaction between the fluid and thermal effects can play an important role in the design process.

Wood Group Heavy Industrial Turbines provides a comprehensive range of support solutions, including re-engineered replacement parts and maintenance, repair and overhaul services for industrial gas turbines and related high-speed rotating equipment used in the global power generation and oil and gas markets. One example of the work done by Wood Group is a recent project involving the re-engineering of the blade from the first stage of a gas turbine. The goal of the project was to optimize the blade design and improve its longevity. The numerical simulation process coupled ANSYS CFX software for the fluid flow, ANSYS Mechanical software for the structural response of the blade, and the 1-D thermal and fluid flow simulation package Flowmaster2. This set of simulation tools provided an efficient virtual prototype that was used to assess the performance of the turbine blade under actual operating conditions.

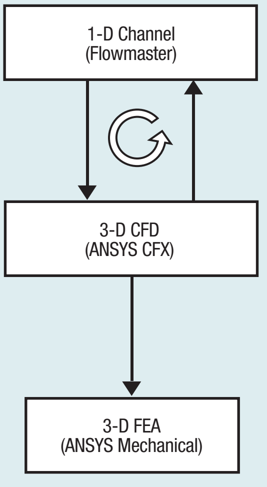
## CFD Model

The original 3-D CAD geometry, which is intended for manufacturing, was extended for the purpose of the simulation using ANSYS DesignModeler. The extensions served to better represent the true operating conditions of the rotor. For example, gaps not present under normal operating conditions were closed. This extended CAD model then served as the basis for the CFD mesh.

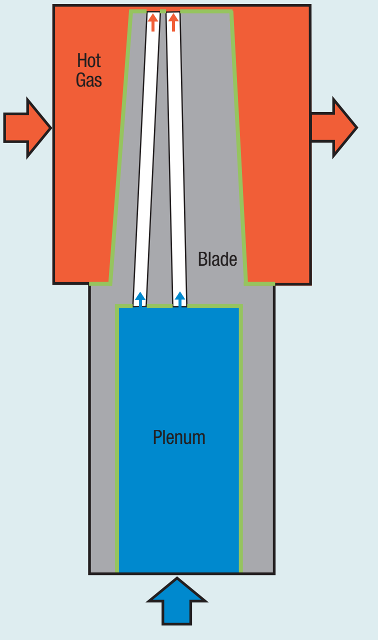
The two fluid domains (the hot gas flow around the blade and the coolant airflow in the plenum) and one solid domain (the blade itself) were meshed independently using ANSYS ICEM CFD meshing software. Generalized grid interfaces (GGIs) were used to connect the non-matching mesh topologies of the individual domains. The cooling channels were modeled using Flowmaster2, and the result of this 1-D simulation was connected to ANSYS CFX using the standard CFX Expression Language (CEL), which requires no user programming. Taking advantage of CEL callback functions, the coolant air flow in the plenum, the hot gas around the blade and the heat conduction through the solid blade can be solved for in a single ANSYS CFX simulation. At the same time, the CFD simulation can use the unique ANSYS CFX model for laminar to turbulent transition, a key feature that properly captures heat transfer rates from the hot gas to the blade surface as the boundary layer develops. The temperature field in the solid blade as computed by ANSYS CFX software was then directly written out in a format appropriate for the subsequent ANSYS Mechanical calculation.

## FE Model

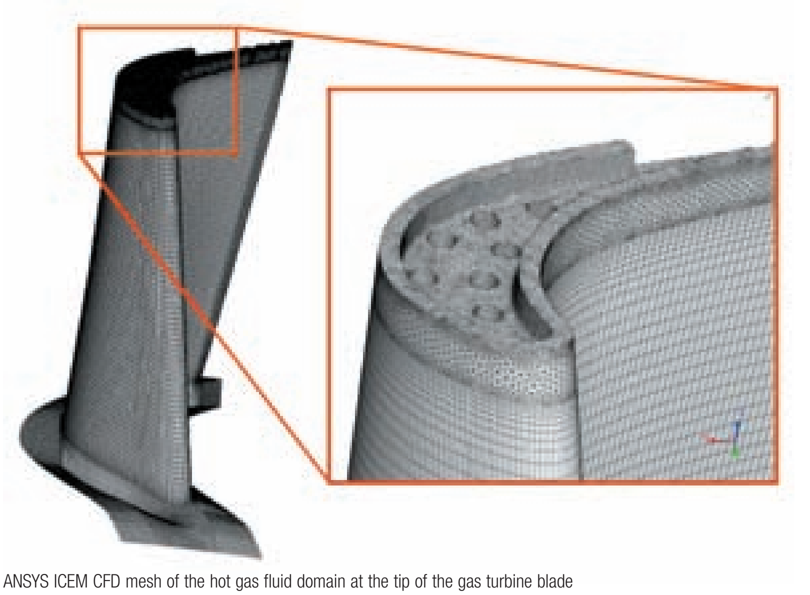
For the simulation using ANSYS Mechanical software, the 3-D temperature field in the solid blade, calculated in




Overview of the interaction between the simulation tools used to provide a virtual prototype of the gas turbine blade.




A schematic of the two fluid (blue and red) and one solid (gray) domain used to perform the analysis. Boundary conditions are defined at locations indicated with arrows. The green lines indicate generalized grid interface connections. The inflow to the hot gas domain from the cooling channels is described by CFX Expression Language callbacks to 1) the flow out of the plenum domain and 2) the heat transfer from the solid domain to the cooling channels.

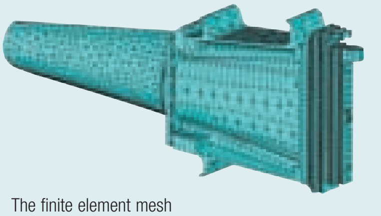


ANSYS ICEM CFD mesh of the hot gas fluid domain at the tip of the gas turbine blade the ANSYS CFX conjugate heat transfer analysis, was used as input for the thermal load. This, along with the rotational load on the blade at operating conditions, determined the stress distribution. Together, the resulting thermal and mechanical stress distributions in the blade were used to determine component life. Applying these loads, life-limiting elements of the blade design could be determined and new design alternatives evaluated.

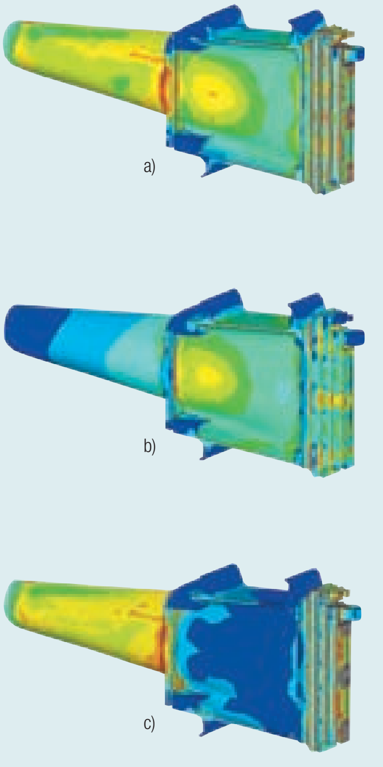
The ability to combine the entire fluid and thermal analysis through the use of standard functionality, especially the powerful CFX Expression Language and its callback functions, are key to making simulations such as this feasible. By combining both CFD and structural analysis with a 1-D thermal simulation, this virtual prototype has provided a more complete understanding of the performance of each blade design in a given set of operating conditions. This allows modifications to be made early in the design process, and therefore is essential in the efforts to help improve efficiency and increase longevity. ■

**Suggested Reading**

Arnal, Michel; Precht, Christian; Sprunk, Thomas; Danninger, Tobias; and Stokes, John: Analysis of a Virtual Prototype First-Stage Rotor Blade Using Integrated Computer-Based Design Tools. Proceedings of ESDA2006 8th Biennial ASME Conference on Engineering Systems Design and Analysis, Torino, Italy, July 2006.



The finite element mesh

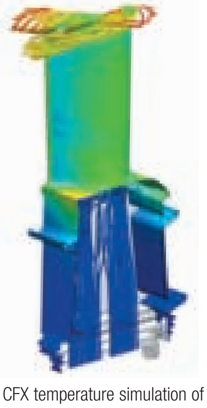


a)

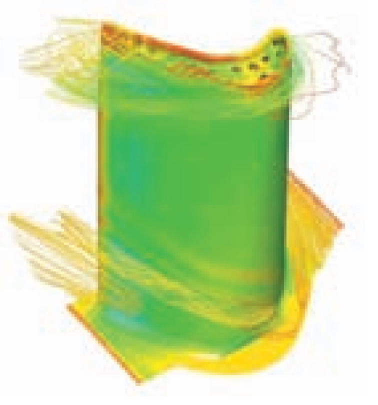
b)

c)

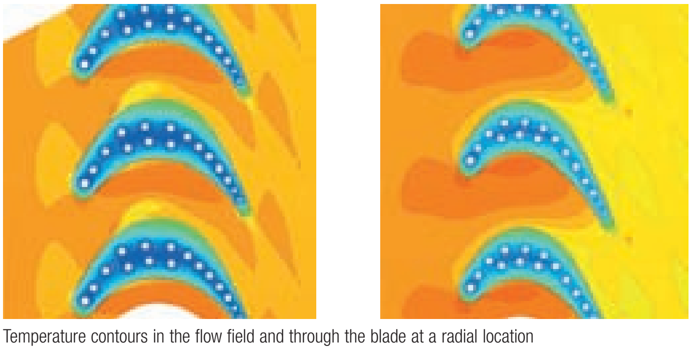
Stress distribution in the directionally solidified blade due to a) temperature variations (but not including rotational effects), b) centripetal forces (assuming a constant temperature) and c) the combination of temperature variations and centripetal forces



ANSYS CFX temperature simulation of the blade surface. Streamlines show flow from the inlet into the plenum and from the cooling channel outlets into the hot gas. Where the internal cooling channels are close to the blade surface on the suction side of the blade near the trailing edge, areas of lower temperature are shown in blue.



CFD simulation of heat flux distribution. The heat transfer is from the hot gas to the blade surface in most areas, but in the tip region the heat transfer is positive corresponding to where the cooling air from the cooling holes comes into contact with the blade surface.



Temperature contours in the flow field and through the blade at a radial location near the blade platform (left) and outer casing (right)