Fluid Flow Validation of a Turbine Inlet Duct

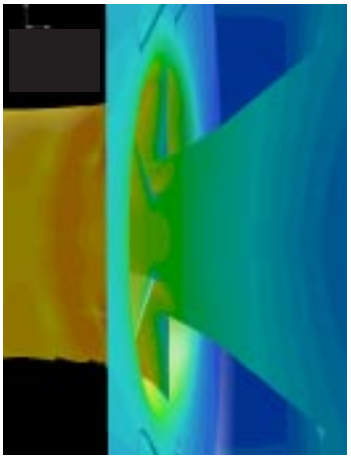
EX11

Features Demonstrated

- 3D unstructured tetrahedral mesh
- Inviscid flow modeling
- Pressure boundary conditions

A turbine inlet duct plays a crucial role in determining the performance and efficiency of power generating turbine systems. In a properly designed inlet, the flow achieves desirable velocity and temperature profiles at the inlet which will enhance the turbine performance and minimize losses. The component geometry is complex and thus CFD helps in the vital study of fluid behavior in it.

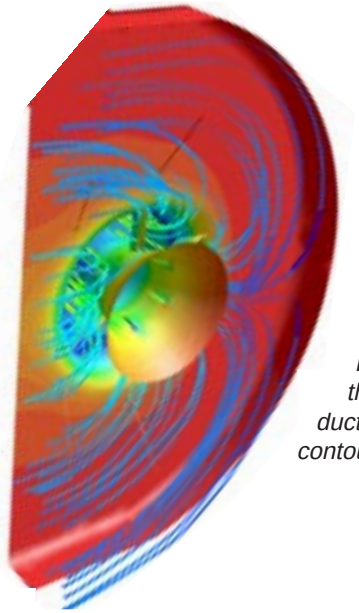
RAMPANT's CFD analysis of one such inlet duct in a stationary gas turbine engine helps to demonstrate this. An unstructured tetrahedral mesh of about 150000 cells is used to resolve the intricate features of the duct such as the struts and the support bars. Symmetry in the problem geometry allows analysis of only one-half of the duct. Fluid enters the sideways inlet of the duct and exits through the circular exit after passing by the support struts. The analysis is done using the inviscid flow model offered in RAMPANT. RAMPANT's ability to handle pressure boundary conditions is utilized for prescribing the flow conditions at the inlet and the exit, with the total pressure defined at the inlet and the static pressure given at the exit.



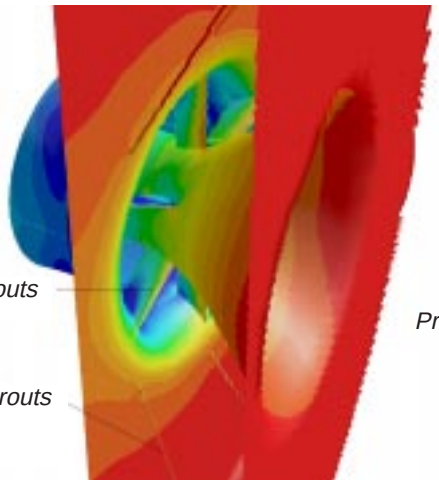
Mach number contours

The flow ribbons show a smooth and gradual transition of the fluid from the casing into the duct. RAMPANT's results also show high speed flow motion near and around the ends of the supporting struts. Such results can help designers optimize the strut design, enhance it's mechanical strength and help minimize losses caused by the strut in the flow. RAMPANT generated pressure contours help designers locate the more mechanically critical regions in the component.

RAMPANT's ability to capture complex flow details without costly physical modeling provides designers with a valuable tool for more cost effective optimization of a turbine design. The effect of geometrical constraints such as the struts and supports, as in this case, can be studied on the overall performance of the component. RAMPANT results can help engineers and designers optimize the geometrical configuration of a component and study its performance at different flow conditions without actually fabricating the component.



Fluid motion in the turbine inlet duct with pressure contours on the wall



Geometry created with ICEM

FLUENT
WWW.FLUENT.COM