ANSYS TurboGrid

ANSYS TurboGrid provides automated, high-quality hexahedral meshing for all blade shapes and sizes.
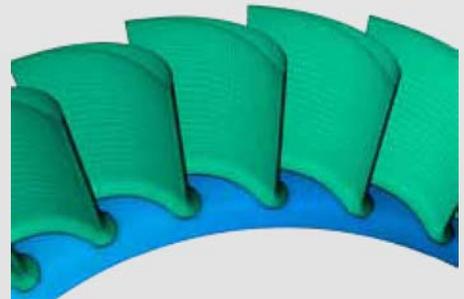
ANSYS® TurboGrid™ software enables designers and analysts of rotating machinery to produce scalable, repeatable meshes in an automated, high-productivity environment. Superior technology is used to create high-quality, feature-resolving meshes, ideal for fast and accurate CFD analysis.

ANSYS introduces automated topology and mesh (ATM) technology, a meshing breakthrough that produces high-quality hexahedral meshes in an automated manner. With minimal user input, the ATM method adapts the block topology for the given blade geometry, yielding a smooth mesh with high orthogonality. Scalability, repeatability and consistency are important inherent characteristics: These are essential requirements for confident comparison of performance predictions between designs, minimizing solution mesh dependency. The ATM method complements the traditional control point topology method, which gives the user full control over all aspects of the mesh.
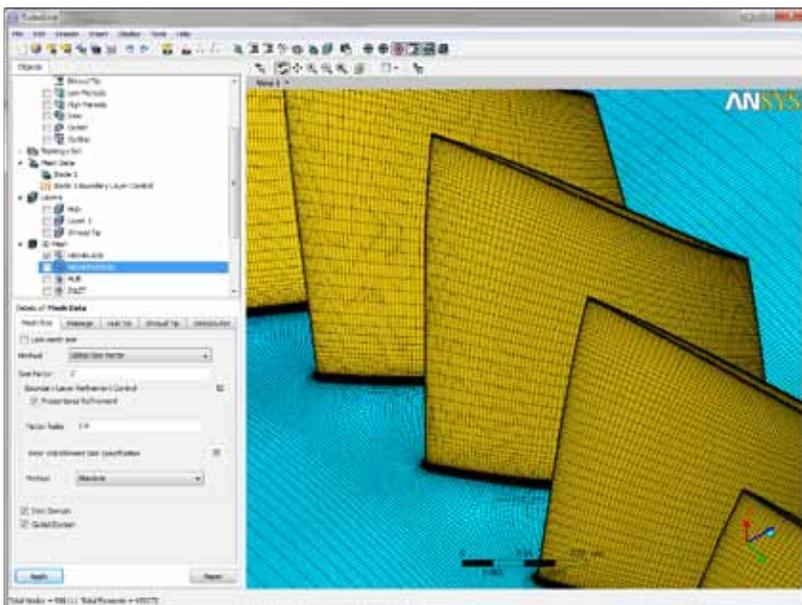
ANSYS TurboGrid is one component of the ANSYS suite of rotating machinery-specific tools, all integral within the ANSYS® Workbench™ environment. ANSYS TurboGrid runs within a modern graphical user interface or in batch mode, enabling parametric, full-physics analysis with a wide range of high-fidelity fluids, structural and dynamics tools from ANSYS.
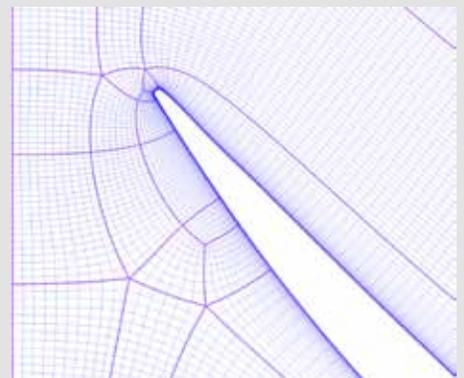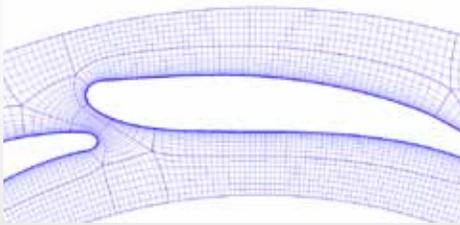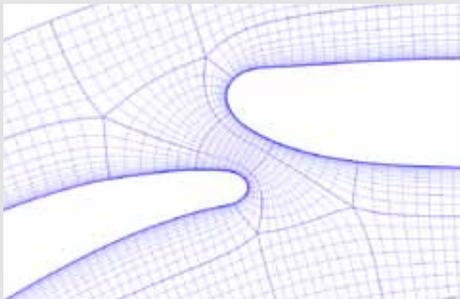
ANSYS TurboGrid provides three views for inspecting the geometry and mesh: blade-to-blade, meridional and 3-D (shown above).

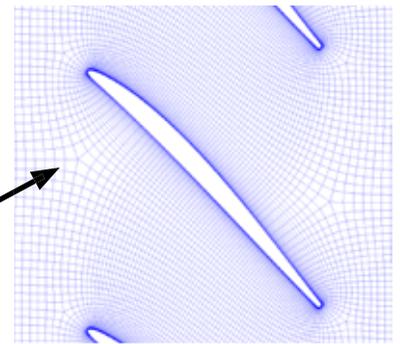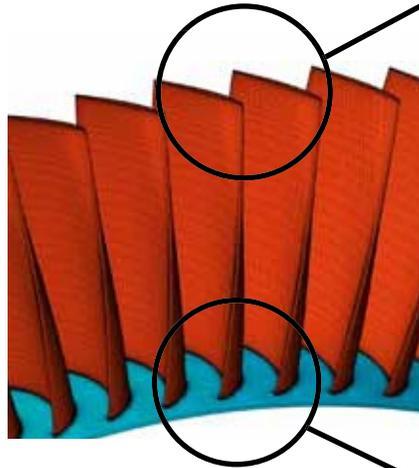Axial (below) and radial (above) turbomachinery
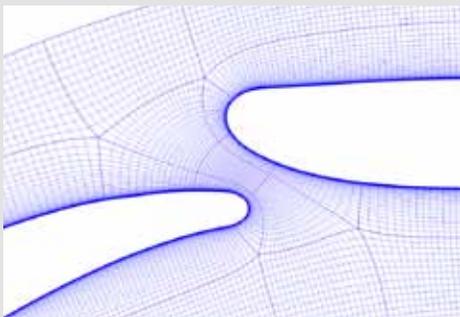
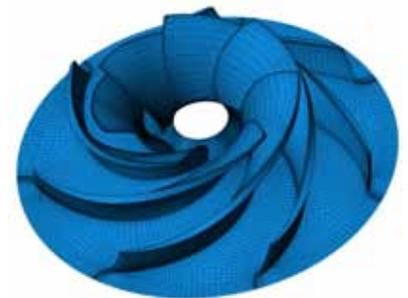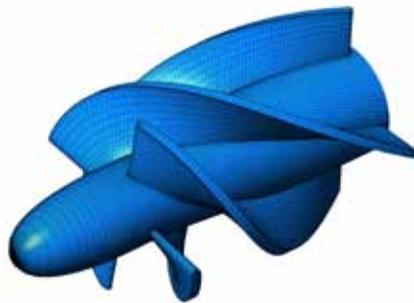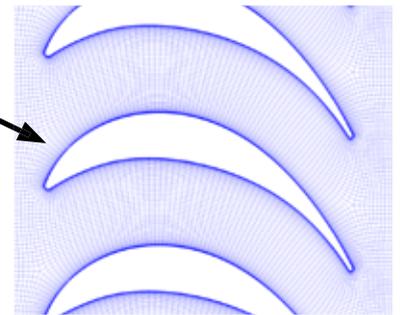High-quality meshes for blades with fillets

User-customizable topology

The ATM method easily handles blades in close proximity

Quality and shape remain consistent with mesh refinement

The ATM method adjusts to maintain mesh quality as blade shape changes along the span

High-quality meshes for hydraulic machinery

GSA Contract Holder