

Introduction to ANSYS BladeModeler

Length: 1 Day

The Introduction to ANSYS BladeModeler course is a one-day course and consists of lectures and "hands-on" practical examples. This course provides an introduction to creating different types of stationary and rotating blade geometry, such as:

- Centrifugal Compressors, Pumps, and Blowers
- Radial Turbines - Turbochargers, Francis, Expanders
- Axial Fans, Inducers, Compressors or Turbines
- Diffusers, Deswirl Cascades, Guide Vanes

Course Topics Include:

- Hub and shroud design
- 2D Layer generation
- Ruled Element or Sculpted Blades
- Linear or Non-Linear Leading and Trailing Edges
- User Specified Leading and Trailing Edge Shapes
- Shrouded or Unshrouded Designs
- Trim Tool for Splitter Blades

In addition to blade creation, attendees will also learn to use DesignModeler to add fillets, modify a hub, add a rotating shroud, and other common turbo-specific features. Users will also undertake a "hands-on" exercise where they generate a blade geometry, mesh the geometry using ANSYS CFX-Mesh, set up a CFD simulation using ANSYS CFX Turbo Pre, and examine the results in ANSYS CFX Turbo Post.