

ANSYS® BladeModeler™

Geometry Design Tool for All Types of Rotating Machinery

ANSYS BladeModeler software is a specialized, easy-to-use tool for the rapid 3-D design of rotating machinery blading. Incorporating extensive turbomachinery expertise from ANSYS into a user-friendly graphical environment, the software enables the aerodynamic/hydrodynamic and mechanical design of the primary flow path components of axial, mixed-flow and radial machines such as pumps, compressors, fans, blowers, turbines, expanders, turbochargers, inducers and more.

Versatile Design Tool and Integrated Design System

ANSYS BladeModeler can be used as a module within an existing design system or with the comprehensive set of rotating machinery design and analysis tools integrated within the ANSYS® Workbench™ environment. ANSYS provides tools for all functions and all physics, including meanline, throughflow, geometry, meshing, fluid, structural, thermal, modal analysis, rotordynamics, optimization and post-processing. This unique and comprehensive suite of tools couples ease of use with high-fidelity analysis, reducing development time while increasing product confidence.

Revise Existing Designs or Create New Ones

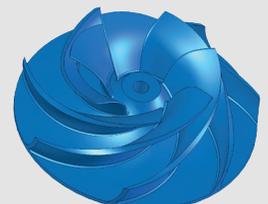
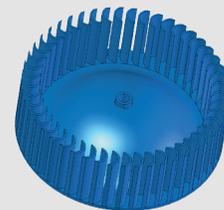
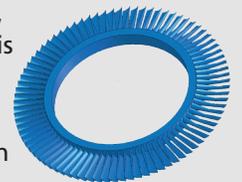
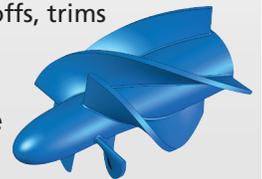
ANSYS BladeModeler technology can import a design produced by an existing tool and then prepare it for detailed design simulation and/or further modification. New designs can be initiated from design rules of choice — or from the integrated Vista suite of meanline sizing tools, developed by PCA Engineers. Basic machine duty information provided to Vista CCD (centrifugal compressors), CPD (centrifugal pumps), RTD (radial turbines) or AFD (axial fans) yields geometry and non-dimensional performance parameters, which are used by ANSYS BladeModeler for the initial design. The final product is a 3-D CAD solid model that may include hub metal, blade fillets, cut-offs, trims and other geometric features.

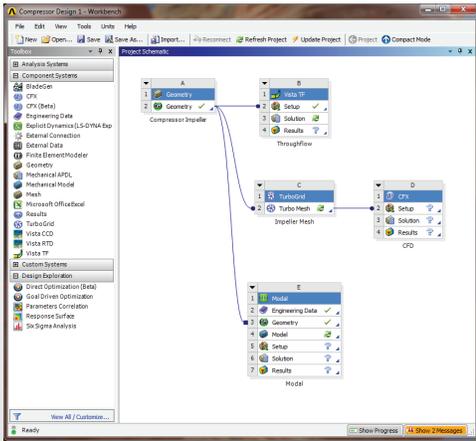
Scalable Geometry, Scalable Analysis

Since the designer is free to select the optimal mode of working, the design process is accelerated. For rapid analysis, the most basic blade sketch can be assessed using throughflow analysis or with relatively coarse CFD or structural meshes. For final design and analysis, more geometric detail can be included, then analysis can proceed with finer meshes as input to the high-fidelity ANSYS® CFD™ or ANSYS® Mechanical™ products, all within a single integrated environment — ANSYS Workbench.

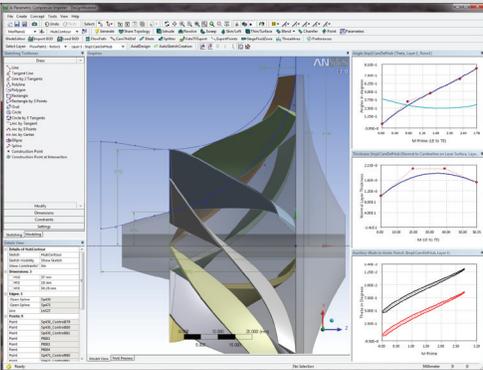
Faster Design

- Aerodynamic/ hydrodynamic and mechanical design in one tool
- General geometry and CAD import and export
- Rich feature set
- Scalable level of geometric detail
- Bladed and non-bladed flow path design
- CAD-based geometry kernel enables fillets, cutoffs, trims and more
- Multiple machine components in one session
- Meanline aero/hydro design tools included
- Coupled to meshing and through-flow analysis tools
- Modern graphical user interface
- Integral with the ANSYS suite of full-physics, high-fidelity analysis tools
- Parameterization speeds optimization





The ANSYS Workbench project schematic clearly shows how geometry created by ANSYS BladeModeler interacts with other tools within the design process.



ANSYS BladeModeler provides the designer with all the necessary views and controls to specify and visualize the design.

Interactive Design, Parameterization and Optimization

ANSYS BladeModeler is a truly interactive blade design tool. Design modifications can be seen clearly and immediately. Relevant design information — such as blade angles, sweep angles, throat area and more — can be parameterized, enabling optimization of the entire process.

Radial and Axial Design Modes

Two modes of blade design are available to cover the needs of both radial and axial machinery designers. Radial machinery designers typically define the meridional profile, blade wrap (blade angle), thickness distributions, and leading/trailing-edge shapes. Axial turbine designers more commonly specify the blade pressure-side and suction-side curves independently. ANSYS BladeModeler can be used in the design mode that is best suited to individual requirements.

Working Environment

ANSYS BladeModeler is divided into several views that break down the blade definition into simpler 2-D components:

- Meridional: allows easy definition of the meridional profile
- Angle distribution: facilitates description of the blade meanline along any layer (meridional streamline)
- Thickness distribution: specifies the normal or tangential thickness distribution of the blade along any meridional streamline
- Section: permits design of the two sides of a blade independently
- Auxiliary: displays a large number of blade geometry diagnostics

These fundamental components are supplemented by a large toolbox of useful functions.

Data Connectivity

The blade geometry designed in ANSYS BladeModeler software can be analyzed immediately within the ANSYS Workbench environment. This includes any 3-D CAD solid modeling operations performed within ANSYS® DesignModeler™. Import and export to third-party software, such as AutoCAD™, IGES, Pro/ENGINEER®, Parasolid™, etc., is also available.

ANSYS[®]
ansys.com

ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com

Toll Free U.S.A./Canada:
1.866.267.9724
Toll Free Mexico:
001.866.267.9724
Europe:
44.870.010.4456
eu.sales@ansys.com



GSA Contract Holder

ANSYS, ANSYS Workbench, HPSS, AUTODYN, CFX, FLUENT and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. ICFM CFD is a trademark used under license. All other brand, product, service and feature names or trademarks are the property of their respective owners. Image Credits: Some images courtesy FluidDA nv, Forschungszentrum Jülich GmbH, Heat Transfer Research, Inc., Riello SPA and iStockphoto.com