Small improvements to rotating machinery design can translate into large operating savings. For these improvements to be identified very early in the design process, each component and any interaction needs to be considered. Engineers worldwide turn to ANSYS integrated solutions to design reliable and efficient rotating machinery systems that meet and exceed market needs.

ANSYS provides a complete product portfolio of structural, thermal, fluid and electromagnetic analysis capabilities that have the ability to simulate your products under real-world conditions.

The design optimization cycle for rotating machinery can be reduced greatly by a combination of turbomachinery-targeted CAD and simulation technologies. ANSYS software incorporates extensive turbomachinery design and analysis expertise and advanced fluid dynamics, body dynamics, stress analysis and thermal simulation technology into an environment accessible to both designers and analysts.

Applicable to both new designs and redesigns, the ANSYS® Workbench™ environment provides an integrated geometry design and analysis system that links all elements of the development process. The system includes turbomachinery optimized geometry creation, hybrid or hexahedral grid generation, analysis and results generation. All rotating machinery primary flow passages for axial, radial or mixed flow machines can be considered in 3-D in the preliminary design stage as well as for the final design using a suite of high fidelity CFD physics models.

ANSYS opens the door to new ideas by providing an application-specific CAE environment for parametric design of all of these components. Now complete systems can be easily considered together during the preliminary design phase.
Computational fluid dynamics software along with high-quality meshing tools from ANSYS have long been market and technology leaders in this demanding industry. These industry solutions can be combined with ANSYS® Mechanical™ software for structural and stress analysis, including the simulation of high fidelity effects such as fluid-structure interaction and the first commercial laminar to turbulent flow transition model.

ANSYS, Inc. works with customers, suppliers, and manufacturers of rotating machinery, providing CAE analysis and design software and technical services that meet their requirements. ANSYS® CFX® software enables designers to accurately model fluid and heat transfer processes in all rotating machinery components and the ANSYS Mechanical suite allows designers to perform linear and non-linear structural, fatigue and vibration analysis.

In the gas turbine industry typical flow analysis applications include: the primary passages of axial compressors and turbines, flow inside turbine blades, secondary flow passages, film cooling, radial compressors, combustors, gas turbine enclosures, inlet and exhaust ducting, silencers, and flow around the cowl of airplanes. Typical mechanical system applications include: thermo-mechanical analysis of gas turbine components, structural linear dynamics (modal/forced and random vibration) of rotating structures, rotodynamics of rotating structures, non-linear analysis with contact of large assemblies, fluid structure interaction, containment analysis and bird strike using ANSYS® LS-DYNA™ software and design optimization.

Some of the reasons why companies in the gas turbine industry trust ANSYS simulation include:

**Single Geometry Model with Multi-CAD Associativity**

ANSYS® BladeModeler™ software is a specialized, easy-to-use tool for the rapid 3-D design of rotating machinery components. The ANSYS® DesignModeler™ product, included with ANSYS BladeModeler, provides a direct link to geometry models created in a variety of CAD packages. As the geometry portal for all ANSYS products DesignModeler proves a single source of geometry for a complete range of engineering simulation tools.

**Comprehensive Suite of Mesh Creation Tools**

The mesh influences the accuracy and convergence of the simulation. From automatic hybrid meshing to specialized hex mesh creation tools, ANSYS, Inc. provides the ultimate meshing solution. Mesh creation products include specialized tools such as the ANSYS® TurboGrid™ product for bladed components, CFX®-Mesh™ software and ANSYS tools for general CFD and mechanical applications, as well as the ANSYS® ICEM CFD™ meshing product for demanding CFD applications.

ANSYS TurboGrid provides designers and analysts of rotating machinery with an automated mesh creation tool that increases productivity and enables design optimization. TurboGrid creates high-quality hexahedral mesh while preserving the underlying geometry based on a number of predefined mesh style templates and building-blocks. Very high quality hex meshes with highly refined meshing in all boundary layer and tip clearance
regions can be created with minimal interactive input, as well as batch driven for repeatable meshing of design variations. TurboGrid provides fast turnaround of the grid generation process to ensure maximum utility from the CFD analysis process. This is critical in gas turbine analysis where meeting rigorous specifications is paramount and small increases in efficiency can lead to millions of dollars profit.

**Specialized CFD Pre-processing for Efficient Setup of Turbomachinery Applications**

User workflow for turbomachinery pre-processing has been improved to include simplified presentation, more dynamic user interaction and feedback, and component mesh or case reload. Users are guided through physics definition by a turbo-specific preprocessing wizard. The wizard simplifies the creation of multiple passage and/or multistage turbomachinery analyses.

**Parallel Performance**

Time is Money! At ANSYS we understand how much time means to you and that multiprocessing is one means to reduce analysis time. Multiprocessing computer environments (consisting of multiprocessor servers or networked workstations or clusters) may be employed to generate simulation and analysis results much more quickly. Both the Parallel Performance for ANSYS and the CFX® Parallel Computing™ products allow larger cases to be solved in a fraction of the time that the serial operation requires.

**Fluid-Structure Interaction**

ANSYS, Inc. provides the world’s most advanced fluid-structure interaction software, permitting combined fluid and solid physics analysis. Our FSI approach preserves the individually validated specialized software components in the computational fluid dynamics and stress analysis disciplines, while at the same time permitting state-of-the-art interaction between the fluid and solid. The advanced FSI capability utilizes the ANSYS multi-field solver to provide true bi-directional FSI capability for time transient or steady state analysis with moving and/or deforming geometry. See the ANSYS FSI solution brochure for more information.

**Turbulence**

CFX offers advanced turbulence models, from eddy viscosity to Second Moment Closure (SMC), and stochastic turbulence models such as LES, DES and SAS models. The robust and accurate implementation of the Shear-Stress Transport (SST) model in CFX is recommended for accurate predictions of decelerating and separating flows, a critical issue in modeling the flow in bladed components. The SST model thus enables prediction of design and off-design performance. In addition ANSYS, Inc. offers the world’s first predictive transition model.

**Rotor/Stator Interactions**

CFX offers three different Multiple Frame of Reference algorithms to simulate rotor-stator interactions: Frozen Rotor, Stage and Transient Rotor-Stator. All are implemented in a fully conservative and implicit manner with no restrictions on grid topologies. For example, hex meshes along bladed passages can be combined with unstructured mixed element meshes in complex casings.
Combustion & Radiation

Combustion models include single and multiple step eddy breakup models, as well as advanced Flamelet models for diffusion flames and the Zimont model for premixed/partially premixed combustion. With the multi-step Eddy Dissipation Combustion model, users can select from a wide range of pre-defined reaction mechanisms, or customize the reaction library to their own specific needs. CFX can solve all species as a coupled system, greatly accelerating convergence, especially for complex reaction mechanisms. Post-flame analysis has been extended through the addition of a Thermal and Prompt NOx models and the Magnussen Soot Model. In addition, CFX can now solve radiative heat transfer in complex three-dimensional geometries for a broad range of radiation phenomena including transparent, gray and multi-band systems using P1, Discrete Transfer and Monte Carlo methods.

Particle Tracking

The Particle Tracking model allows the solution of one or more discrete particle phases within a continuous phase. Particle properties, size distribution, boundary interaction and injection patterns are all controllable by the user. Particle Tracking can be used to investigate a variety of particle and droplet effects including particle separation, blade erosion and spray distribution for inlet fogging. Particle tracking can be applied in a steady or transient mode, is fully supported for parallel computation, and is also permitted in a multiphase Eulerian model. An extensive list of particle related models and abilities is available including particle breakup models, particle mixture models (with intra-particle reactions), built-in coal particle models, droplet evaporation and boiling models, and particle quantitative operations.

Specialized CFD Post-processing for Efficient Setup for Turbomachinery Applications

Turbomachinery CFD post-processing involves a combination of qualitative and quantitative functions, both general purpose and turbo-specific; ANSYS® CFX®-Post™ provides both. In addition to general purpose CFD visualization, turbo-specific views of the data are available in a turbo post-processing mode. Meridional and unrolled blade-to-blade views are available using turbo coordinates. In addition to a variety of relative and absolute frame quantities and coordinates, meridionally-averaged data can be evaluated.

Quantitative calculations can be performed using built-in and user programmable functions, allowing flexible definition of formulas for machine performance, efficiency, etc. These formulas operate on the flow simulation results using the same data resolution used inside the flow solver, so quantities like massflow-weighted averages at machine inlet and outlet regions are as accurate as possible. Attention to this level of detail in quantitative calculations ensures the accuracy expected by turbomachinery analysts is also achieved in post-processing. Equally important is that all post-processing functions are easily scriptable for operation in a batch environment. ANSYS CFX also allows users to easily integrate automatic CFD post-processing into a turbomachinery design process tailored to meet their specific requirements.