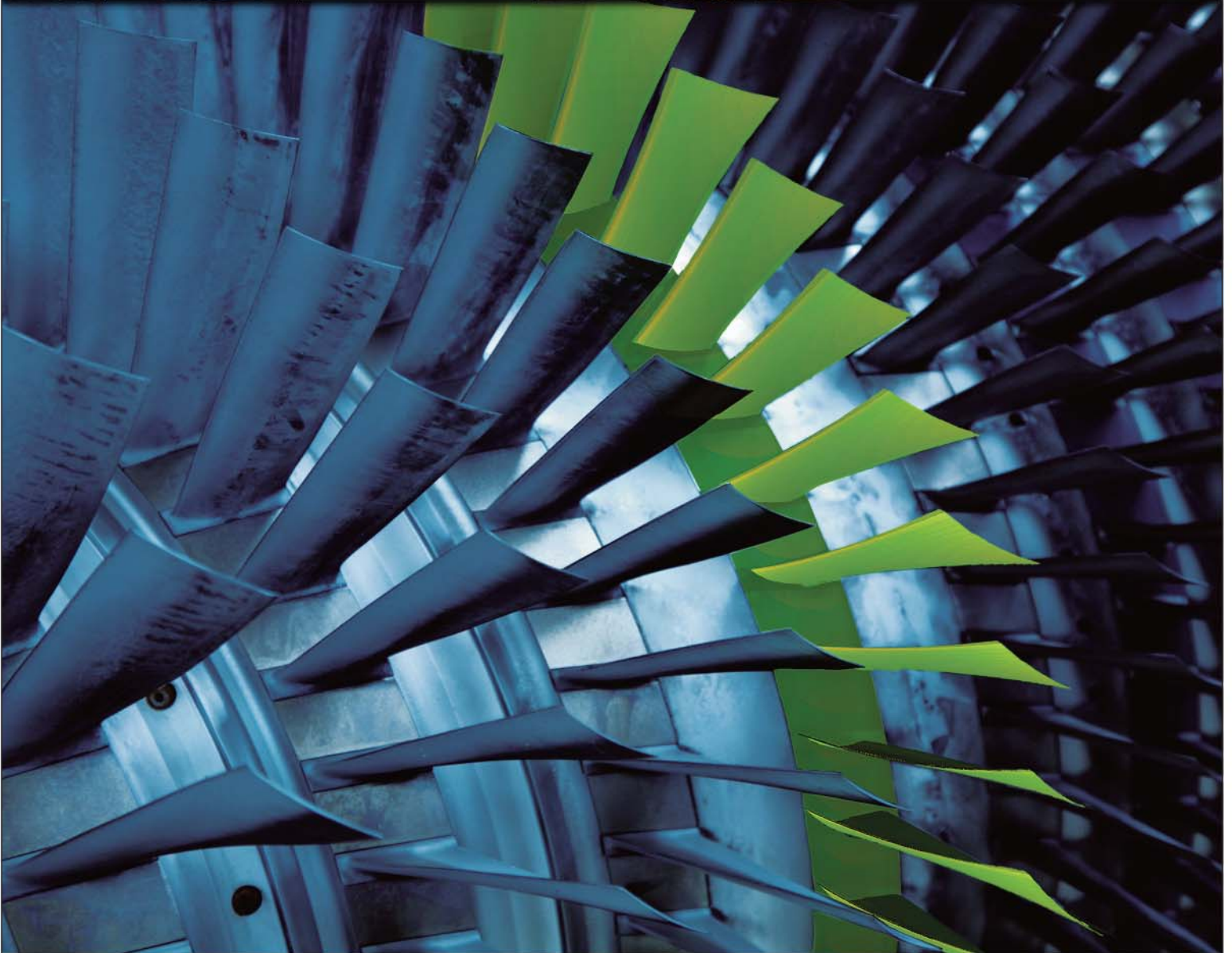
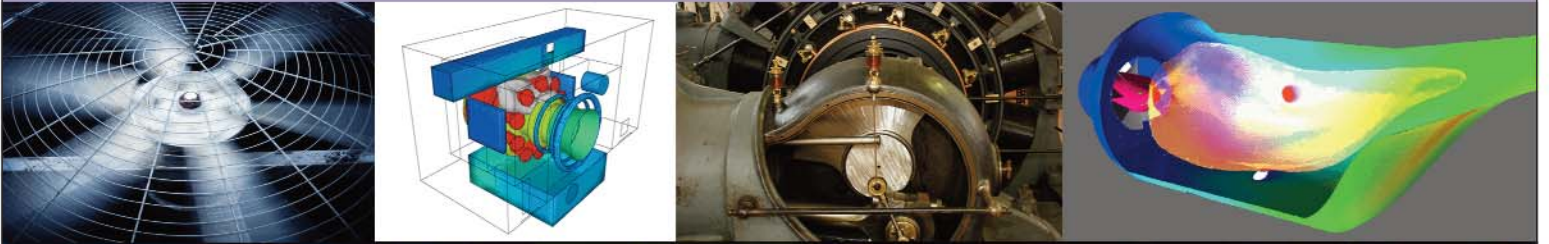




Engineering Simulation Solutions for
turbomachinery



With the unequalled depth and unparalleled breadth of engineering simulation solutions from ANSYS, companies that develop turbomachinery are transforming their leading-edge design concepts into innovative products and processes that work. Today, 97 of the top 100 industrial companies on the "FORTUNE Global 500" invest in engineering simulation as a key strategy to win in a globally competitive environment. They choose ANSYS as their simulation partner, deploying the world's most comprehensive multiphysics solutions to solve their complex engineering challenges. The engineered scalability of our solutions delivers the flexibility customers need, within an architecture that is adaptable to the processes and design systems of their choice. No wonder the world's most successful companies turn to ANSYS — with a track record of almost 40 years as the industry leader — for the best in engineering simulation.

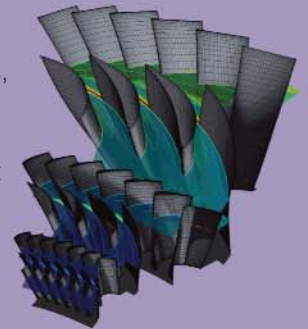


Challenges and Solutions

Turbomachinery plays a vital role in almost all industry sectors. These rotating machines change the state of working fluids (as in pumps or compressors), convey or transport fluids (fans and pumps), extract energy (turbines) and create propulsion (aircraft engines and propellers). Performance, efficiency, reliability and rapid delivery have always been important, but today's world conditions intensify the pressures designers face. As the provider of Simulation Driven Product Development™ tools, ANSYS, Inc. is helping turbomachinery providers effectively position themselves in a dynamic and highly competitive market.

High-Fidelity Simulations

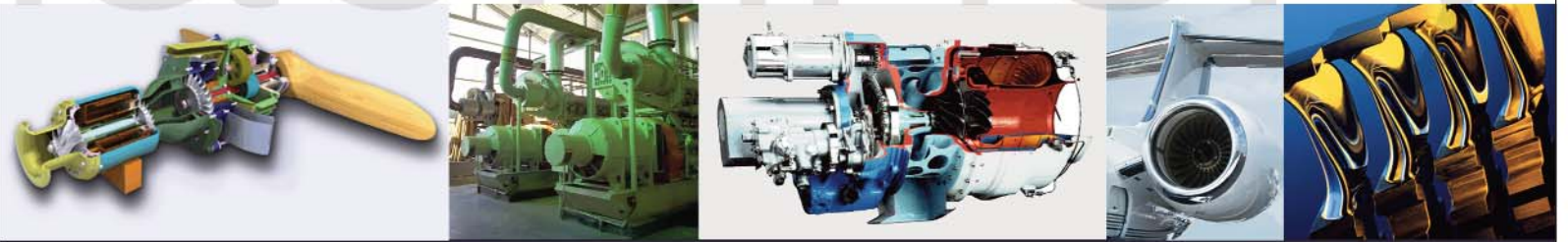
Turbomachinery developers have long used engineering simulation software to increase performance and reduce the cost of testing. Today, many companies rely on software from ANSYS to accurately simulate internal or external aerodynamics and hydrodynamics, thermal and structural state, combustion, and other mission-critical behaviors. Integrated and coupled comprehensive multiphysics capabilities from ANSYS account for the interactions of different physical phenomena in the industry's highly integrated, interdependent machine components and systems. Engineered scalability enables individual users, departments or entire business units with various requirements for analysis to perform serial or parallel computations on individual laptops, compute clusters or enterprise-wide computing resources.



Customized Tools for Design Efficiency

Turbomachinery developers require a design system consisting of a broad range of tools that enable rapid changes and the ability to quickly investigate a multitude of operating conditions and parameters. ANSYS provides a suite of special-purpose, high-productivity tools for tasks such as blade row geometry definition, flow path meshing, 1-D performance estimation, accelerated case setup, design of experiment (DOE) analysis and component-specific post-processing. These tools are configurable within the ANSYS® Workbench™ environment, which provides complete and scalable multidisciplinary design and analysis capabilities as well as easy communication and interaction with other tools and within existing design systems.

machinery



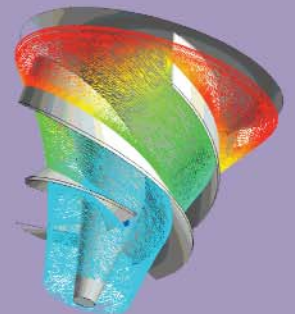
Capabilities

“Simulation was a critical tool in successfully developing the TEI-TJ-1X microjet engine and in the follow-on work in designing the TEI-TP-1X turboprop engine. In the early stages of the development cycle, simulation provided valuable insight for quickly identifying potential problems and evaluating alternative solutions. This prevented large numbers of costly and time-consuming late-stage design changes and enabled us to verify the design with the minimum number of physical tests.”

Bulent Acar
Design Engineering and
Technologies Leader
Tusas Engine Industries, Inc.
Istanbul, Turkey

- ▶ **Pre-Processing Solutions:** Blade row specific geometry and meshing; 1-D aero design; 3-D parametric modeling; wrapping technology; bi-directional CAD connectivity; geometry creation and editing; decomposition; structured; unstructured; polyhedra; CAD import and export; automation; scripts; legacy data handling

Key Products: ANSYS® BladeModeler™, ANSYS® DesignModeler™, ANSYS® ICEM CFD™, ANSYS ICEM CFD Hexa CAA V5, ANSYS® Mesh Morpher™, ANSYS® TurboGrid™, GAMBIT®, TGrid™



- ▶ **Mechanical Simulation:** Static, modal and harmonic analyses; transient and spectrum analyses; buckling and fatigue analyses; automated contact analyses; composites; deformable geometry; geometric nonlinearity; linear and nonlinear materials; component mode synthesis; rigid and flexible multi-body dynamics; rotordynamics; explicit dynamics; topological optimization; variational sensitivities; customizable loads; full suite of multiphysics element technology; iterative solvers; direct solvers; parallelized

Key Products: ANSYS® AUTODYN®, ANSYS® DesignSpace®, ANSYS® Fatigue™, ANSYS® LS-DYNA®, ANSYS® Mechanical™, ANSYS® Professional™, ANSYS® Structural™, ANSYS® Multiphysics™, ANSYS® Rigid Dynamics

- ▶ **Electromagnetic Simulation:** Parasitic extraction; electrostatic and magneto-static analysis; low- and high-frequency electromagnetic analysis; low-frequency electric and magnetic analysis; antenna radiation patterns; near- and far-field extension analysis

Key Products: ANSYS® Emag™, ANSYS Multiphysics, ANSYS® Icemax®

Capabilities (continued)

- ▶ **Thermal Solutions:** Steady and unsteady; conduction, convection and radiation; phase change; mass transport; fluid elements; parallelized

Key Products: ANSYS® CFX®, ANSYS Mechanical, ANSYS Multiphysics, ANSYS® FLUENT®, FLUENT® for CATIA® V5

- ▶ **Fluids Solutions:** Application-specific, CAD-embedded and easy-to-use general interfaces; moving and deforming geometry; steady and unsteady flow; 2-D, axi-symmetric, axi-symmetric with swirl and 3-D flow; incompressible and compressible flow; inviscid and viscous flow; laminar, transitional and turbulent flow; LES/DES/SAS; aero-acoustics; convective, conductive, radiative and conjugate heat transfer; species transport and reactions; dispersed and mixed phases; combustion; rotating machinery with stage interfaces; customizable physics and user interface; segregated and coupled solvers; parallelized; blade-row specific post-processing

Key Products: ANSYS CFX, ANSYS CFX-FlO, ANSYS FLUENT, FLUENT for CATIA V5

blade row aerodynamics • compressors • rotordynamics •
fluid structure interaction • bird strike • turbines • diffusers •
cavitation • exhaust systems • seals • emissions control •
composites • gearing • pumps • **turbomachinery**
dynamics of rigid and flexible bodies • fatigue • fans •
aero-acoustics • combustors • turbochargers • blowers •
transition ducts • bearings • inlets • secondary flow path

About ANSYS, Inc.

ANSYS, Inc., founded in 1970, develops and globally markets engineering simulation software and technologies widely used by engineers and designers across a broad spectrum of industries. The Company focuses on the development of open and flexible solutions that enable users to analyze designs directly on the desktop, providing a common platform for fast, efficient and cost-effective product development, from design concept to final-stage testing, validation and production. The Company and its global network of channel partners provide sales, support and training for customers. Headquartered in Canonsburg, Pennsylvania, U.S.A., with more than 60 strategic sales locations throughout the world, ANSYS, Inc. and its subsidiaries employ over 1,600 people and distribute ANSYS products through a network of channel partners in over 40 countries.

Visit www.ansys.com for more information.

ANSYS®
www.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

ANSYS, ANSYS Workbench, Ansoft, 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. ICEM CFD is a trademark used by ANSYS, Inc. under license. All other brand, product, service and feature names or trademarks are the property of their respective owners.

Images courtesy iStockphoto.com, Alden Research Laboratory, Alstom Hydro Canada Inc., Florida Turbine Technologies, Inc., Honeywell Aerospace, Tusas Engine Industries, Inc., Siemens AG Power Generation and Gerber, A.G.; Sigg, R.; Volker, L.; Casey, M.V.; Surken, N. "Predictions of non-equilibrium phase transition in a model low pressure steam turbine", Journal of Engineering for Power and Energy, September 2007, Vol. 221, No. A6, pages 735-744.

© 2008 ANSYS, Inc. All rights reserved.

MKT0000260



GSA Contract Holder