

**T**he business of putting a man on the moon or getting someone from one side of the world to the other is a complex challenge. The aerospace industry must design products for the harshest of environments, and the products need to work. Otherwise, people's lives are in danger. With this in mind, engineering requires testing and re-testing to make sure a product will endure throughout its lifetime. The aerospace industry relies on physical and CAE testing to make sure products can hold up to gravity, for example. ANSYS helps by providing CAE tools to reduce the need for expensive physical prototypes and to improve up-front design before any prototypes are made.

### ANSYS Simulation Solutions

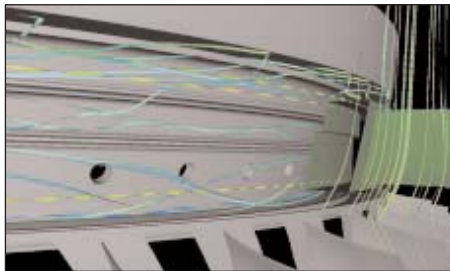
- ▶ ANSYS® DesignXplorer™
- ▶ ANSYS® Emag™
- ▶ ANSYS® CFX®
- ▶ ANSYS® Mechanical™
- ▶ ANSYS® Multiphysics™
- ▶ ANSYS® ICEM CFD™

*"The investigation showed that ANSYS CFX can reliably simulate complex phenomena such as cavity flows."*

– **Olivier Seite**  
Engineer  
MTU Aero Engines, Germany

*"ANSYS ICEM CFD grid generation products are extremely efficient and accurate and, by far, the most advanced and user-friendly tools currently available. Our unique applications in computational electro-magnetics truly benefit from the versatility of ANSYS ICEM CFD Hexa and Tetra tools in producing accurate solutions and reducing cycle time. The timely response of the ANSYS customer support staff also adds significant value to the ANSYS ICEM CFD products."*

– **Touraj Sahely**  
Member, Technical Staff  
HyPerComp, Inc.



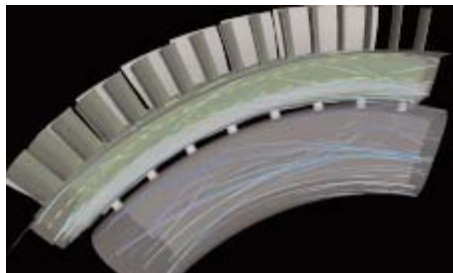
Streamlines in the stator wells and rotor drum, axial view (The geometry has been duplicated over 60°; the actual computational domain covered only 7.5°)

importance of precise strain and rotor tip clearance predictions, which in turn demand accurate knowledge of the thermal behavior of rotor discs. For this, the flow and heat transfer mechanisms inside the rotor drum must be known as precisely as possible.

MTU Aero Engines chose ANSYS CFX for computing such challenging flows, which are usually difficult to simulate due to their complexity and the lack of information about their characteristics. The investigation with ANSYS CFX showed that it can reliably simulate phenomena as complex as these cavity flows. More importantly, the extent of insight provided by the simulations is impossible to obtain with experimental methods. This positive experience will pave the way for further applications of CFD, including cavity optimization studies.

### CASE-IN-POINT

At MTU Aero Engines, a supplier of sub-systems that include high-pressure compressors and high- and low-pressure turbines for commercial and military engines, CFD is commonly employed in areas such as blade design. An increasingly crucial application of CFD is the computation of flow and heat transfer in the multiple rotating cavities of the engine's secondary air system. Indeed, growing performance requirements emphasize the impor-



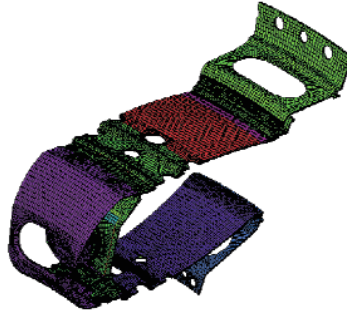
Top view of the streamlines in the stator wells

### ANSYS provides solutions to:

- ▶ Enable design-level CAE for concept design of mechanical systems, from air flaps to landing gear
- ▶ Defeature and simplify assembly-level models to quickly create CAE models for analysis
- ▶ Reduce amount of physical tests by allowing more CAE simulations
- ▶ Help correlate physical tests to CAE simulations, through which an engineer can learn additional information from the CAE model. For example, in a wind tunnel test, results are obtained through sensors that might sometimes obstruct flow. With CAE simulation, results can be probed without the need for sensors. Thus a general flow pattern can be found without introducing artificial objects that might obstruct flow.
- ▶ Create in-cabin modeling for aircraft customer comfort
- ▶ Optimize wing design by comparing multiple design alternatives

### CASE-IN-POINT

To meet aggressive new product introduction (NPI) analysis, GE Global Research Center collaborated with GE Aircraft Engines to develop advanced combustor design technologies. In particular, a multidisciplinary design optimization was conducted for an aircraft engine combustor structure, a multidisciplinary process involving aero CFD, combustion, heat transfer CFD, dynamics, thermal, mechanical and life-prediction factors. Mesh generation plays a critical role in an automated design process. Meshing strategies need to be developed to generate high-quality meshes in an automatic fashion (without human intervention). To ensure analysis accuracy for this project, an all-hex mesh was required for the full combustor sector model. ANSYS ICEM CFD was used for the automated meshing strategies because its structured Hexa mesher offers unique features with the capability and flexibility that GE required.



### About ANSYS, Inc. Solutions

ANSYS designs, develops, markets and globally supports engineering simulation solutions used to predict how product designs will behave in manufacturing and real-world environments. Its integrated, modular and extensible set of solutions addresses the needs of organizations in a wide range of industries. ANSYS solutions qualify risk, enabling organizations to know if their designs are acceptable or unacceptable — not just that they will function as designed. ANSYS helps organizations achieve:

- Innovative and high-quality products and processes
- Fewer physical prototypes and test setups
- Faster return on investment due to reduced development time
- A more flexible and responsive information-based development process, enabling the modification of designs at later stages of development
- A front-end simulation strategy that offers a superior method for bringing products to market in less time and with fewer costs

### 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-conscious product development, from design concept to final-stage testing and validation. Headquartered in Canonsburg, Pennsylvania, U.S.A., with more than 25 strategic sales locations throughout the world, ANSYS, Inc. and its subsidiaries employ approximately 600 people and distribute ANSYS products through a network of channel partners in over 40 countries.

### The ANSYS Advantage

ANSYS software provides customers with a competitive advantage:

- ▶ Lowering the cost to do CAE analysis through wizards and process-based tools that reduce training time and analysis time
- ▶ High accuracy CAE hex models with boundary layers to provide better correlation with physical tests
- ▶ ANSYS® Workbench™, which provides a unified product development environment offering integration across a wide range of design processes — ranging from geometry modeling and editing, meshing and pre-processing, advanced analysis (structural, thermal, electro-magnetics, CFD, etc.) and robust design optimization



[www.ansys.com](http://www.ansys.com)

ANSYS, Inc.  
Southpointe  
275 Technology Drive  
Canonsburg, PA 15317  
USA  
724.746.3304  
[ansysinfo@ansys.com](mailto:ansysinfo@ansys.com)

Toll Free USA/Canada:  
1.866.267.9724  
Toll Free Mexico:  
001.866.267.9724  
Europe:  
44.870.010.4456  
[eu.sales@ansys.com](mailto:eu.sales@ansys.com)

Publication subject to change without prior notice.  
ANSYS, ANSYS Workbench, CFX, AUTODYN and any and all ANSYS, Inc. product and service names are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries located in the United States or other countries. ICEM CFD is a trademark licensed by ANSYS, Inc. All other trademarks or registered trademarks are the property of their respective owners.