

# ANSYS®

RELEASE 13.0

## ANSYS CFX

### Powerful Computational Fluid Dynamics Software for Process and Product Design Optimization

For more than 20 years, companies around the world have trusted ANSYS® CFX® technology to provide reliable and powerful computational fluid dynamics (CFD) solutions. ANSYS CFX combines advanced solver technology with a modern user interface and an adaptive architecture to make CFD accessible to both designers with general engineering knowledge and fluid dynamics specialists requiring in-depth model control and options. It is used in a vast array of industries to provide detailed insight into equipment and processes that increase efficiency, improve product longevity and optimize processes.

#### ANSYS CFX and the ANSYS Workbench Environment

ANSYS CFX software is fully integrated into the ANSYS® Workbench™ environment, the framework for the full suite of engineering simulation solutions from ANSYS. Its adaptive architecture enables users to easily set up anything from standard fluid flow analyses to complex interacting systems with simple drag-and-drop operations. Users can easily assess performance at multiple design points or compare several alternative designs. Within the ANSYS Workbench environment, applications from multiple simulation disciplines can access tools common to all, such as geometry and meshing tools.

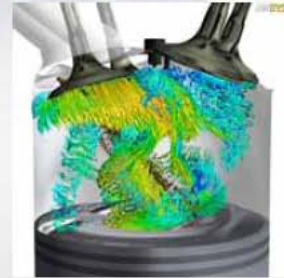
**Geometry:** ANSYS® DesignModeler™ software is specifically designed for the creation and preparation of geometry for simulation. Its easy-to-use, fully parametric environment with direct, bidirectional links to all leading CAD packages acts as the geometry portal for all ANSYS products to provide a consistent geometry source for all engineering simulations.

**Meshing:** Providing accurate CFD results requires superior meshing technology. ANSYS Meshing provides a multitude of meshing technologies in a single application to allow users to select the best option on a part-by-part basis. ANSYS® ICEM CFD™ meshing tools also are available and include unlimited mesh editing capabilities as well as structured hexahedral meshing.

#### CFD Pre-Processing in CFX-Pre

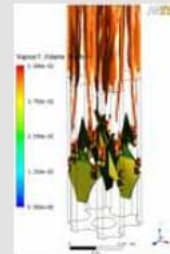
The ANSYS CFX physics pre-processor is a modern and intuitive interface for the setup of CFD analyses. In addition to a general mode of operation, predefined wizards are available to guide users through the setup of common fluid flow simulations. A powerful expression language gives users the ability to customize their problem definition in numerous ways, such as with complex boundary conditions, proprietary material models or additional transport equations. The adaptive architecture of CFX-Pre even allows users to create their own custom GUI panels to standardize input for selected applications, and thereby ensure adherence to established best practices

## Industry Solutions



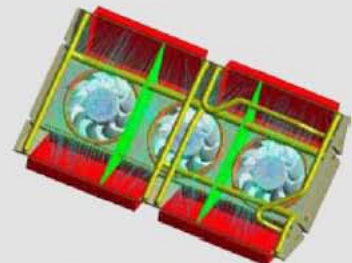
Vortex structures in a four-stroke engine just after injection of fuel and intake valve opening

*Courtesy of BMW Group.*



Nucleate boiling downstream of spacers in a fuel rod bundle assembly

*Courtesy of Dr. E. Krepper, FZ Dresden.*



Bottom of the cooling system showing obstructions

*Courtesy of Voith Turbo.*