



Product Features

CFX-Flo Add-Ons : Geometry Modeling and Interfaces

ANSYS® DesignModeler™ software has native, bi-directional, integration with the most popular CAD systems including:

- ▶ Autodesk Inventor® / MDT
- ▶ Autodesk Inventor® Professional Stress
- ▶ CATIA® v4 and v5
- ▶ Pro/ENGINEER®
- ▶ Solid Edge®
- ▶ SolidWorks®
- ▶ Unigraphics®

The software supports neutral formats such as IGES, Parasolid and SAT.

Need CFD? Quick-Start your Designs with CFX-Flo

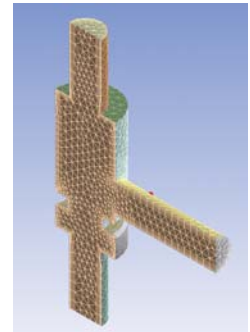
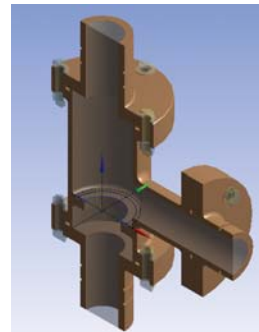
Demanding product development cycles require innovative CAE solutions. ANSYS is at the forefront of simulation-driven product development. Simulation and analysis utilized in the early stages of product design enable fast, efficient and cost-effective product creation. This includes providing computational fluid dynamics (CFD) as an indispensable tool for up-front product design.

ANSYS customers can achieve new heights in product innovation no matter what level of CFD is required in their design processes.

One Environment

The ANSYS® Workbench™ environment provides a single environment for simulation from start to finish, enabling users to perform more product development tasks faster. ANSYS Workbench delivers the basis for a full CAE solution from ANSYS, providing access to a wide variety of simulation technologies.

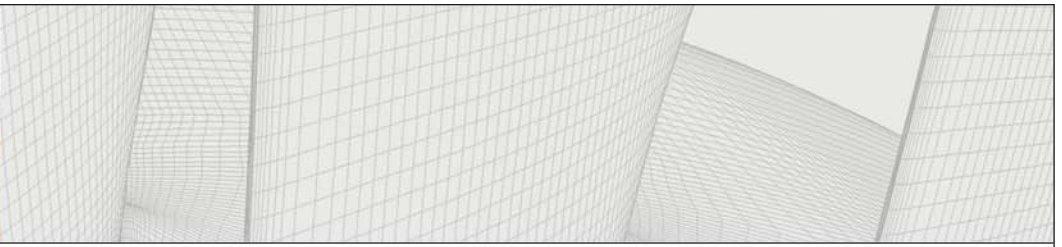
All settings are persistent, and connected back to the parametric CAD model, from analysis-specific modifications made to the geometry through the application of physics, solver control parameters, graphic objects created during post-processing, and quantitative expressions evaluating performance. This makes it easy to investigate the entire operating range and/or multiple design variations without the need for tedious re-work.



ANSYS Workbench provides a consistent easy-to-use environment for importing CAD models, simplifying the model defining the fluid volume and rapidly generating high-quality CFD meshes. Geometry courtesy CADFEM GmbH.

Geometry Modeling with Bi-directional Associative CAD

CFX-Flo CFD software has direct access to CAD through the geometry interfaces of ANSYS Workbench. The same CAD model generated by design engineers can be used for simulation of fluid dynamics or stress analysis. ANSYS DesignModeler software contains the tools necessary to modify an



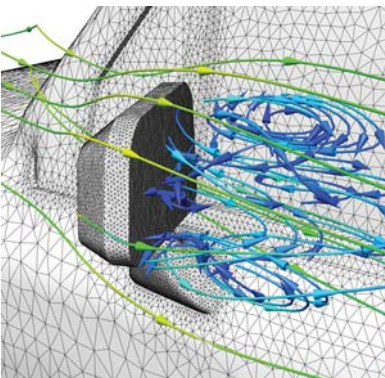
Product Features



Ball valve



Automotive cooling fan



Automotive mirror

Graphics, quantitative information and even animations in CFX-Flo provide the information required to develop better designs.

attached or imported model to suit the requirements of a desired simulation. This link is bidirectional, which allows the engineering analyst to send parametric changes back to the CAD system without having to learn the CAD package.

Quick Setup and Fast Design Iteration

A wizard guides the user through the simulation physics setup quickly and easily. The wizard steps through the process of boundary condition specification and automatically checks that all necessary information has been entered. Subsequent design iterations are easily created simply by modifying the geometry and updating the mesh used with the pre-processor. Persistence ensures that boundary conditions do not need to be re-entered when the parametric geometry changes.

Fast and Accurate CFD Solver

CFX-Flo contains the power of the ANSYS CFX computational fluid dynamics software package. For more than 20 years, in a broad range of industries and applications, CFX has been trusted to deliver accurate results quickly. All the models work together and in parallel, leading to accurate and fast solutions. The ANSYS CFX solver includes leading-edge technology that provides the fast run times and the quick project turnaround required to efficiently integrate CFD into a design system.

Understanding Designs

ANSYS CFX provides a wealth of data that can't be extrapolated from bench tests. Vector, streamline and contour plots give rapid insight into the three-dimensional nature of the flow. Easily-generated animated movies help to effectively communicate these results. Quantitative calculation capabilities provide the hard engineering numbers needed to evaluate and compare various design iterations so that the final product meets requirements.

Communicating Designs

ANSYS CFX software includes a freely-distributed 3-D viewer that can be used to share 3-D design images with colleagues and customers and then embed in presentations or reports. CFX-Flo includes automated report generation that quickly documents important aspects of design iterations in formats such as HTML.

Scalable

With ANSYS CFX software as the foundation, CFX-Flo provides access to a scalable suite of CFD products and features that ensure the ability to include higher fidelity models, physics interactions and parallel processing as demands on simulation increase. It is simple to add more functionality while continuing to seamlessly support and leverage the investment in existing CAE assets.