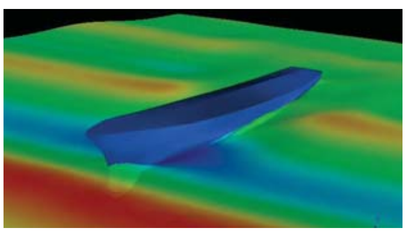
ANSYS® CFX®
Flexible CFD Solutions

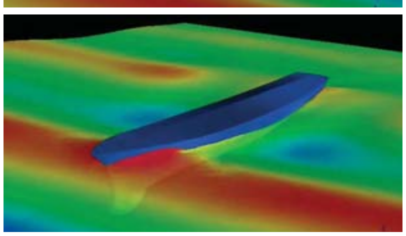
Technical Brief

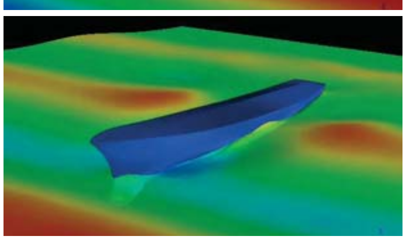
Open Structure for Better Design in Less Time

Through implementation of CFX Command Language™, CFX Expression Language™, user FORTRAN and a junction box, ANSYS CFX CFD software offers unprecedented integration access and control to users.

The open structure of the ANSYS CFX product means that you can modify the standard models, develop completely new ones or couple to your own FORTRAN, C or C++ software. This software is designed for the future, in which CFD will be closely integrated into ever larger multi-domain, multi-physics simulations.

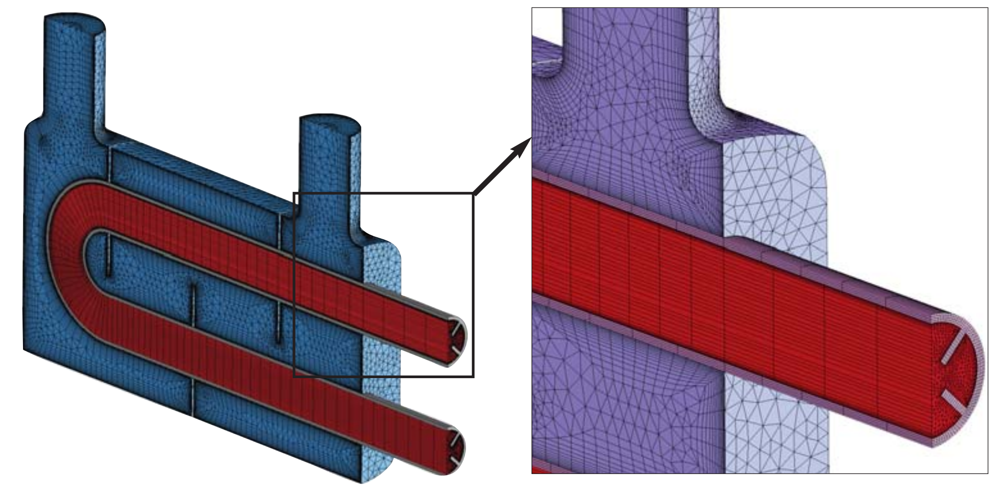




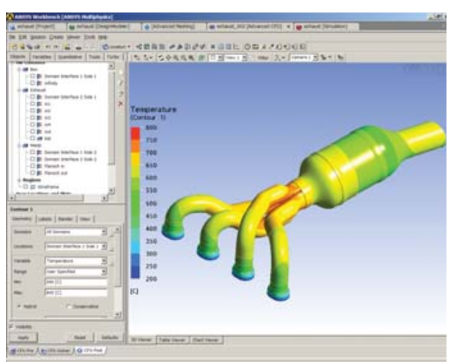


The above images show a transient ship hull-wave interaction calculation that was performed using ANSYS CFX software and custom FORTRAN code. ANSYS CFX determines the forces acting on the boat, FORTRAN relates the forces to the hull motion.

ANSYS CFX computational fluid dynamics (CFD) software offers an open architecture that encourages customization, on all levels. Both input and results are in accessible formats that allow easy customization enabling full integration within your existing software environment and assisting you in maximizing return on your investment. Every component of this software can be customized to your requirements. From simple parameterization to implementation of complex physical models, the ANSYS CFX product offers all the tools you need.



ANSYS CFX software allows any combination of meshes to be used. This hybrid mesh allows for the use of automated meshing approaches where possible, but maintains excellent near-wall mesh spacing to give good resolution of boundary layers. For this example, extruded hex and hex-prism meshes were used for the pipe solid walls and fluid interior, while the shell side mesh is a combination of prism and tet elements.



Specialized interfaces in the ANSYS CFX product provide direct access to many of the parameters, all within the ANSYS Workbench environment.

Integrated in ANSYS Workbench — A Complete CAE Environment

ANSYS CFX software can be used on its own or within the ANSYS® Workbench™ environment. This interface provides a common tool set for geometry management (including bi-directional parametric connections to major CAD systems), parameter management and file management across multiple engineering disciplines including CFD, structural mechanics, electromagnetics and coupled physics simulation.

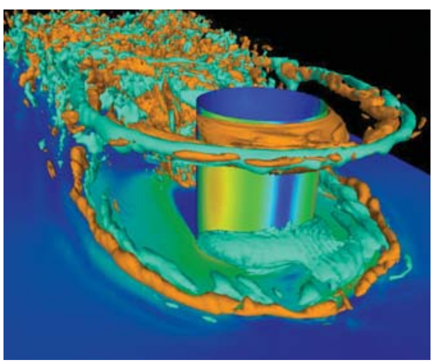
ANSYS®

Technical Brief

Robust and Accurate CFD

From multiphase mixing to high-speed flows, ANSYS CFX software delivers fast, robust and accurate CFD solutions through a unique combination of:

- Advanced coupled multigrid solver technology
- Superb parallel efficiency
- Unmatched meshing flexibility
- Excellent pre- and post-processing capabilities
- Industry-relevant models.



ANSYS CFX software contains advanced turbulence models such as LES, DES and SAS. When fine features of the flow must be resolved in complex geometries, it is only feasible with the efficient ANSYS CFX solver. Picture courtesy FIAT Research Centre and the European Project Alessia.

www.ansys.com

Comprehensive Parameterization

The dedicated CFX Command Language (CCL) allows swift and effective access to the solver, and it allows you to implement physics, boundary conditions, and solver parameters through an intuitive text command file as an alternative to the graphical user interface (GUI). You can run parametric studies quickly by editing command files and changing the appropriate values. This also enables you to run ANSYS CFX software in batch mode or to integrate it in optimization and design systems. CCL can be wrapped around other software to create customized interfaces.

Specific Purpose Models

CFX Expression Language (CEL)

With this powerful definition language you can incorporate your own physical models quickly from within the front end, as well as add new variables, define property relationships and boundary condition profiles. For example, a radiation model and a chemical kinetic model for the nucleation and growth of aerosols have been implemented entirely using CEL. Customized versions of ANSYS CFX software's multiphase models can also be implemented in this way. CEL is very easy to learn. It uses a syntax very similar to the way you'd write equations on your engineering notepad. An expression builder helps you learn how to form expressions useful for physics definition and/or post-processing.

User FORTRAN

CEL allows FORTRAN™ routines to be called, immediately providing more advanced functionality, and it allows you to couple your software to the ANSYS CFX code. Very advanced models can be implemented in this manner. Examples include multiphysics problems or boundary conditions requiring special data or interchange of data with other software. Other programming languages such as C or C++ may be used as well.

Easy Data Exchange

User subroutines can be called at any point or repeatedly while the solver runs using a facility called a junction box. This makes data exchange with other software easy, and provides advanced functionality for model development and customization. The mesh and results application program interfaces allow you to programmatically access the ANSYS CFX input and output files for custom import or export of mesh or CFD results to or from any desired format.

Custom Pre- and Post-Processing

ANSYS CFX pre- and post-processing tools can be driven by user-defined macros to automate commonly repeated problem definition or reporting tasks in CFD analysis. Macro capabilities include full access to the Perl scripting language for maximum user programmability. A macro can also define its own simple graphical user interface, to allow customized input to the automation process. This powerful capability allows experts to capture all the details of complex problem creation or reporting, then create a simple user interface within pre- or post-processing that exposes only the necessary inputs and parameters, and deploy the simplified interface to the rest of the engineering organization.

ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
USA
724.746.3304
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

cfx-flexible_Dec06

GSA Contract Holder