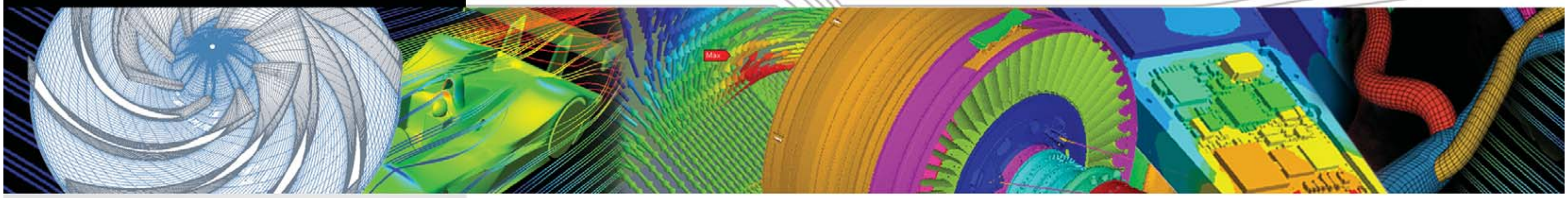
ANSYS® CFX® for Free Surface Flows



TECHNICAL BRIEF

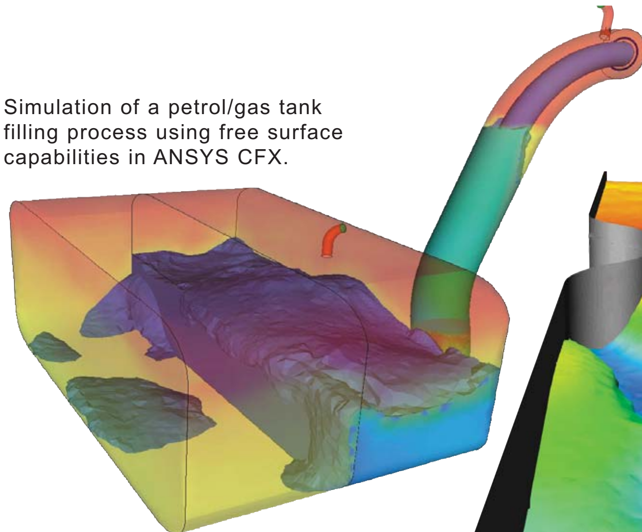
Robust and accurate simulation of free surface flows

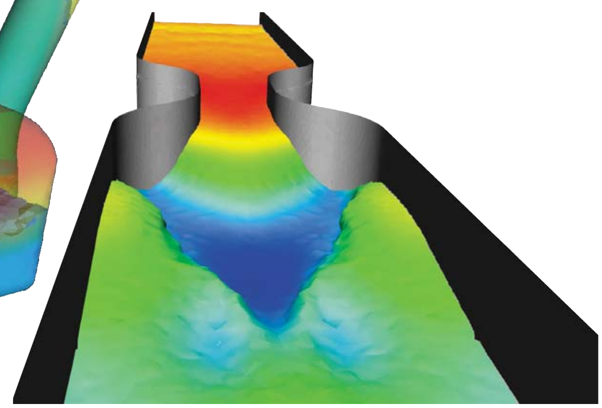
ANSYS CFX's flow solver accuracy and robustness offer significant benefits in all free surface flow problems. Applications include drag prediction around ships and submarines, water levels in open-channel flows and reservoirs, sloshing forces in fuel tanks and ballasts, automotive fuel tank filling interruptions, performance of Pelton turbines, optimizing mould filling processes, and individual droplet calculations.

www.ansys.com

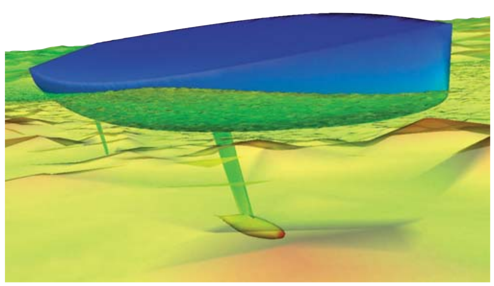
Accurate computational simulation of free surface flows requires CFD software that can capture the sharp interface between two fluids in an efficient and precise manner. ANSYS CFX software provides users with these essential requirements, and much more, by combining a unique compressive discretization with its coupled multigrid solver. When further combined with CFX's outstanding parallel capabilities, CFX provides a powerful way of obtaining free surface solutions in an efficient, robust manner.

Simulation of a petrol/gas tank filling process using free surface capabilities in ANSYS CFX.





ANSYS CFX captures the hydraulic jump in a canal flow with contraction, showing the air/water interface shaded with the height of the liquid.



ANSYS CFX simulation of the free surface over a yacht traveling at 10 knots.

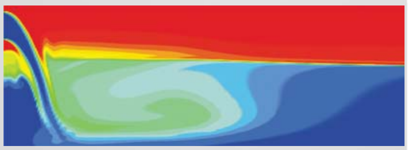
The model

The free surface flow model is a homogeneous extension of CFX's powerful Eulerian-Eulerian multiphase model. By homogeneous we mean that all fluids share common velocity, pressure, and (if applicable) turbulence fields. The computational effort required to solve free surface flow problems is consequently reduced considerably compared to that of a full Eulerian-Eulerian multiphase flow simulation.

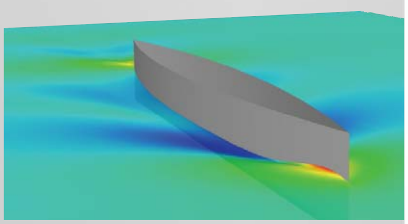
Compressive discretization

CFX uses a unique 'compressive' discretization scheme for the calculation of phasic volume fractions. The anti-diffusive characteristics of this scheme provide the user with much sharper, and hence more accurate, interfaces than possible with conventional bounded second order schemes. A significant advantage of the compressive scheme is that it provides crisp interfaces for both steady-state and transient simulations.

Technical Brief



CFX simulation of a weir overflow, combining the free surface model with the full Eulerian-Eulerian multiphase model to allow air and water to separate into a sharp interface.



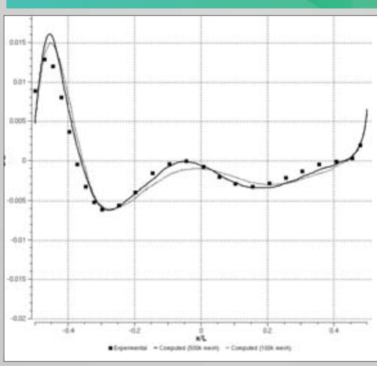


Chart: Comparison of measured and predicted wave profiles for the Wigley experimental hull at Fr=0.267. Top: free surface along the hull.

In some instances, the homogeneous free surface model is not appropriate. For example, where splashing is present, the assumption of a shared velocity field does not allow the fluids to separate. In such situations, the free surface model can be combined with ANSYS CFX's full Eulerian-Eulerian multiphase model to provide accurate solutions.

Stable solutions

The free surface flow model includes additional numerical enhancements to provide accurate, stable solutions. For example, the global continuity equation is cast in its volumetric form which provides improved stability at the interface where a large density ratio can occur. Pressure-velocity-body force coupling is also carefully implemented to avoid spurious velocity wiggles at the interface.

Physics flexibility

The free surface model is fully compatible with all other CFX modeling capabilities, including turbomachinery frame change models, mesh motion, fluid-structure interaction, fluid compressibility, general grid interface and surface tension effects (including wall adhesion).

Automated mesh adaption

ANSYS CFX's adaptive mesh refinement can also counteract numerical diffusion to allow further sharpening of the free surface interface. CFX can adapt all types of mesh elements, including tetrahedral, hexahedra, pyramids, and prisms. Grids can be imported in a wide variety of formats and adaption can be immediately applied.

Sometimes the location of the free surface interface changes as the solution develops. As a result, a refined mesh generated in an early stage of the solution may not be necessary later. The mesh refinement algorithm in CFX automatically coarseness the grid in these locations.

Applying the model

As models with large numbers of computational cells (order of millions) are becoming the norm, CFX's excellent convergence characteristics, linear scalability and superb parallel performance provide a most efficient modeling tool for free surface flow calculations.

ANSYS direct CAD access and automatic meshing tools have extended the accessibility of free surface flow simulations to even the most complex geometries found in industry. From fuel/ballast tanks in ships to submerged entry nozzles in steel casters, automatic meshing offers maximum flexibility and high quality grids with the minimum of user effort.

References

P. Zwart. Numerical Modelling of Free Surface and Cavitating Flows. *VKI Lecture Series: Industrial Two-Phase Flow CFD*, 2005.

P. Zwart. Industrial CFD Applications of Free Surface and Cavitating Flows. *VKI Lecture Series: Industrial Two-Phase Flow CFD*, 2005.

www.ansys.com

GSA Contract Holder
FSS Contract 0123456789

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

TB05Aug_CFX_Freesurface