



Altair® AcuSolve is a most powerful Computational Fluid Dynamics (CFD) tool, providing users with a full range of physical models. Simulations involving flow, heat transfer, turbulence, and non-Newtonian materials are handled with ease by Altair® AcuSolve's robust and scalable solver technology. These well validated physical models are delivered with unmatched accuracy on fully unstructured meshes. This means less time spent building meshes and more time spent exploring your designs.

Product Highlights:

- Efficient and flexible workflow
- Full set of physical models for flow, turbulence, immiscible multiphase and heat transfer simulations
- Accurate and stable even on highly skewed meshes
- Fast and efficient solutions for both transient and steady-state simulations
- Parallel scalability demonstrated on thousands of computing cores
- Advanced multi-physics capabilities including rigid body and flexible body coupling with Altair®'s solvers as well as third party applications

Learn more:
Altair.com/Altair-cfd/

Benefits

Altair® AcuSolve was designed with a philosophy that you shouldn't have to sacrifice robustness and convenience to achieve accurate results. With Altair® AcuSolve, you aren't forced to spend days or weeks iterating on mesh quality to obtain a stable and accurate solution. Build your mesh, run the solver, and interrogate the solution.

Advanced physics? No problem!

Have applications that challenge your current solver's capabilities? Eddy resolving turbulent simulations, immiscible multiphase, and fluid-structure interaction are a few such examples that come to mind. Altair® AcuSolve's single solver technology simplifies these simulations for you. There's no need to struggle with differencing schemes, time integration settings, CFL based stability limits, or any other solver setting when investigating complex physics with Altair® AcuSolve. All supported flow regimes are handled by a single solver that doesn't require any tuning for specific applications.

High Speed, Parallel Performance

What really allows you to leverage CFD to its fullest extent is the ability to interrogate multiple design candidates early on in the design process. This requires a fast solution process, which Altair® AcuSolve provides via:

- Solution of the fully-coupled pressure/velocity equation system, yielding rapid nonlinear convergence.
- Efficient parallel architecture that provides distributed, shared, and hybrid parallel operation.
- Proven parallel scalability on thousands of compute cores.

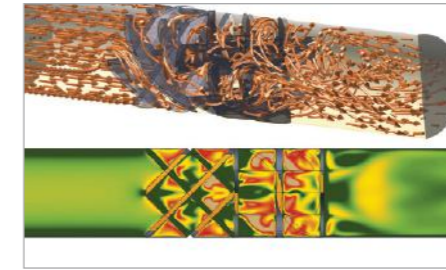
Simulation Features

Flow Modeling

Altair® AcuSolve's flow simulation capability focuses on incompressible and subsonic compressible flows. Within this class of problems, a full set of material models for investigating Newtonian and non-Newtonian flow fields is available. Specialized flow models, such as Stokes flow, are also available for applications in which the full Navier-Stokes equations are not needed.



Streamlines showing flow patterns in a stirred tank



Flow patterns and velocity field in an SMX style mixer



Streamlines through a gas turbine thermocouple cooling passage

Heat Transfer and Radiation Modeling

Altair® AcuSolve supports a full set of features for analyzing heat transfer in both solid and fluid mediums. Supported features include:

- Conjugate heat transfer
- Natural convection
- Enclosure radiation
- Solar radiation
- Thermal shells for modeling thin solids
- Simplified heat exchanger models

Turbulence Modeling

Altair® AcuSolve provides a complete selection of turbulence modeling capabilities to fulfill your simulation needs. Available RANS models include:

- Spalart-Allmaras
- SST
- k-w, BSL k-w
- Realizable k-ε, RNG k-ε, Standard k-ε

For higher resolution transient simulations, Altair® AcuSolve supports the following models:

- Spalart-Allmaras based Detached Eddy Simulation (DES and DDES)
- SST based (DDES and Zonal DES)
- Dynamic coefficient and fixed coefficient Large Eddy Simulation models

For simulations involving turbulent transition, Altair® AcuSolve supports the following transition models (compatible with Spalart-Allmaras and SST RANS/DES models):

- Y one-equation model
- Y-Reθ two-equation model

BETA FEATURE – Multiphase Modeling

Altair® AcuSolve's multiphase flow simu-

lation capability enables the simulation of 2 immiscible, incompressible fluids. The multiphase capability can be used with heat transfer, turbulence, moving and deforming meshes, non-conformal interfaces, and Fluid-Structure Interaction. There is no limit on the density ratio of the two fluids, enabling the simulation of air/water, oil/water, etc.

Moving Mesh Capabilities

Altair® AcuSolve supports two approaches for handling deforming meshes. An Arbitrary Lagrange Eulerian (ALE) mesh motion algorithm provides the most general solution for complex motions. Simpler motions are accommodated by Altair® AcuSolve's boundary condition tools that let you define how the motion of a boundary surface should propagate through the model.

User-Defined Functions (UDF)

Altair® AcuSolve permits customization of material models, boundary conditions, source terms, and many other features within the solver by writing your own functions. In addition to the standard set of data access functions within the UDF, Altair® AcuSolve also provides client-server programming capabilities. This allows you to couple your CFD simulation with external applications such as control system codes.

Multiphysics Capabilities

Altair® AcuSolve's advantages for multi-physics computations result from its strength in simulating transient flows and deforming meshes. Altair® AcuSolve supports the following multi-physics ap-

plications without coupling to external codes:

- Rigid body dynamics of non-interacting bodies
- Linear structural deformation

Coupling to other products in the Altair®, Altair® HyperWorks™ suite provides the following additional capabilities:

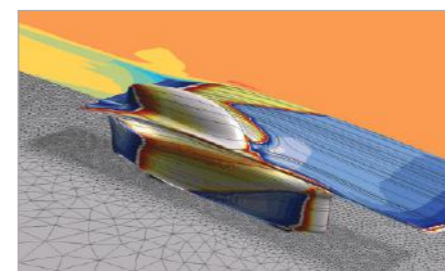
- Finite mass particle tracing (coupling with AcuTrace)
- Multi-body dynamics (coupling with Altair® MotionSolve™)
- Non-linear structural deformations (coupling with Altair® Radioss)

Pre-processing Features

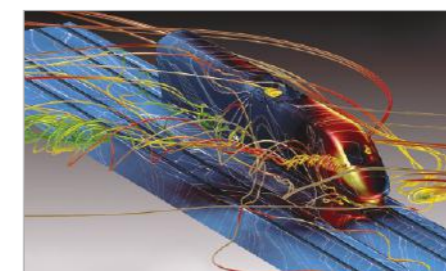
Altair® AcuSolve's dedicated Graphical User Interface, AcuConsole®, contains a full set of options to help you build your CFD models quickly and efficiently. It provides full support for Altair® AcuSolve's features and can be used as an interactive tool or as an automation platform for a batch process.

Post-processing Features

Post-processing of Altair® AcuSolve results is handled by AcuFieldView™, an OEM version of Intelligent Light's FieldView CFD post-processor. AcuFieldView allows you to post-process your Altair® AcuSolve simulations using client-server based parallel operation and comes equipped with a full set of tools to automate the interrogation of your solution.



Surface vorticity contours of a flow transitioning from laminar to turbulent on a solar race car



Pressure field and streamlines over a rail car in cross flow



Detached Eddy Simulation (DES) of flow over a sports car