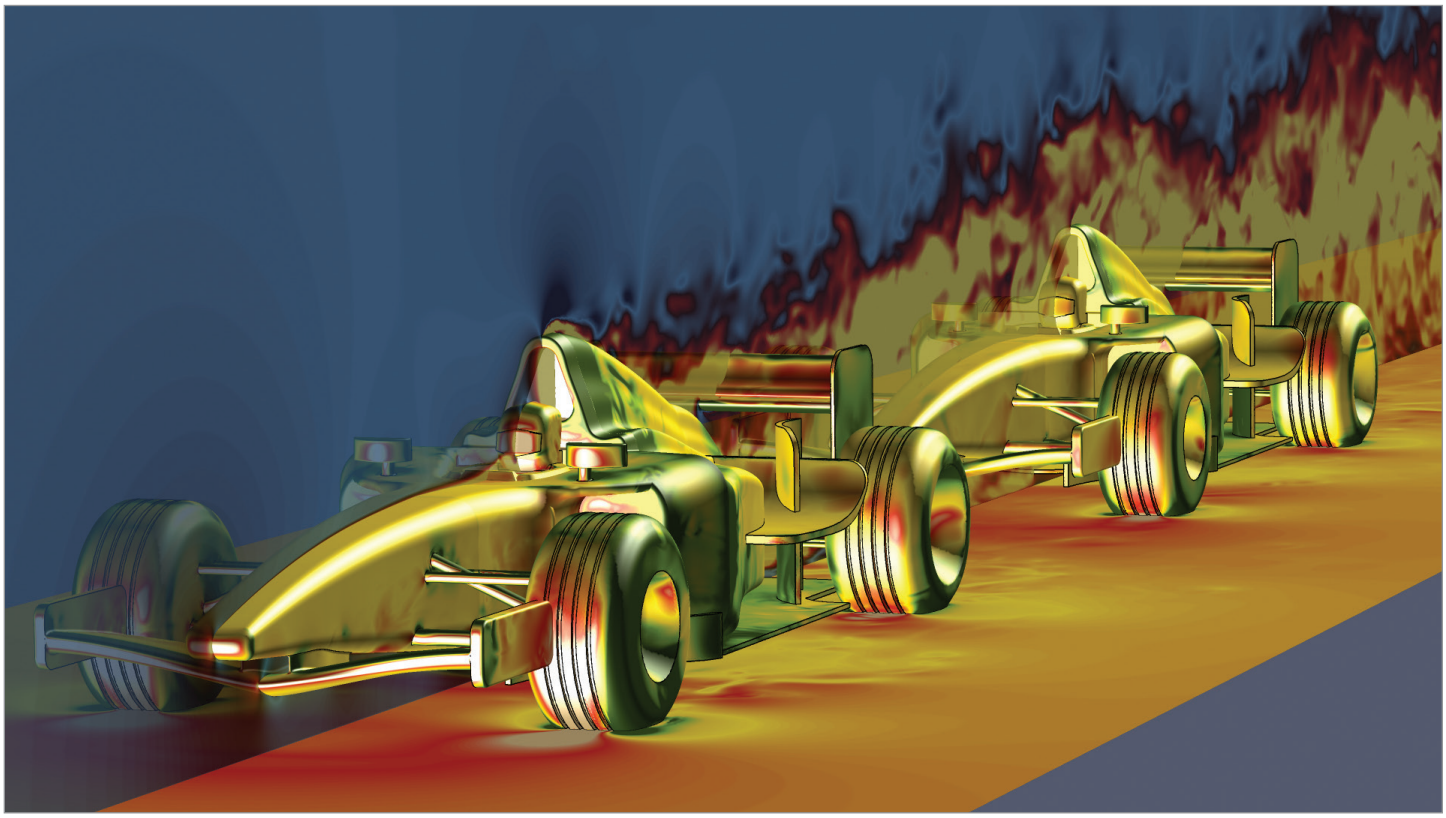
AcuSolve®

Altair | HyperWorks®



AcuSolve is Altair's 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 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, 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

Learn more:
altairhyperworks.com/acusolve

Benefits

AcuSolve was designed with a philosophy that you shouldn't have to sacrifice robustness and convenience to achieve accurate results. With 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, moving mesh simulations, and fluid-structure interaction are a few such examples that come to mind. 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 AcuSolve. All supported flow regimes are handled by a single solver that doesn't require any tuning for specific applications.

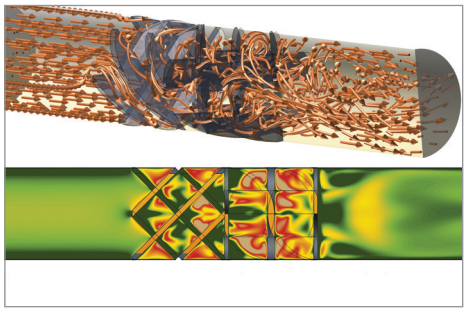
High Speed, Parallel Performance
What really allows you to leverage CFD to its fullest extent is the ability to interrogate multiple design candidates early on in the design process. This requires a fast solution process, which AcuSolve provides via:

- 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.

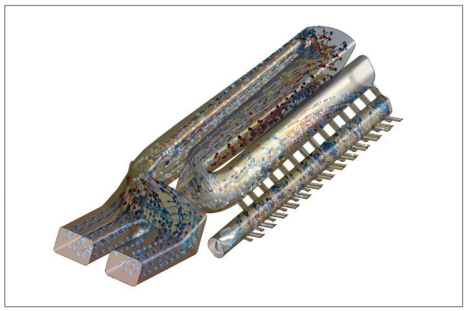
Simulation Features

Flow Modeling
AcuSolve's flow simulation capability focuses on incompressible and subsonic compressible flows. Within this class of

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

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.

Heat Transfer and Radiation Modeling
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-omega

For higher resolution transient simulations, AcuSolve supports the following models:
- Spalart-Allmaras based Detached Eddy Simulation (DES and DDES)
- SST based Detached Eddy Simulation
- Fixed coefficient and dynamic Large Eddy Simulation models

Moving Mesh Capabilities
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 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)
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, 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
AcuSolve's advantages for multi-physics computations result from its strength in simulating transient flows and deforming meshes. AcuSolve supports the following multi-physics applications without coupling to external codes:
- Rigid body dynamics of non-interacting bodies
- Linear structural deformations

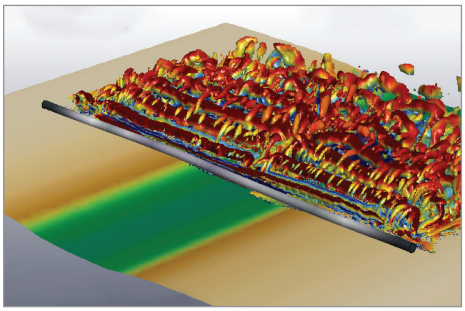
Coupling to other products in the HyperWorks suite provides the following additional capabilities:
- Finite mass particle tracing (coupling with AcuTrace)
- Multi-body dynamics (coupling with MotionSolve)
- Non-linear structural deformations (coupling with RADIOSS)

Pre-processing Features

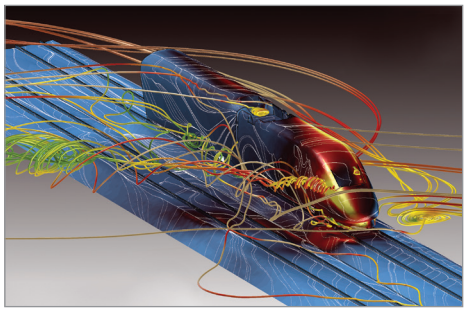
AcuSolve's dedicated Graphical User Interface, AcuConsole®, contains a full set of options to help you build your CFD models quickly and efficiently. It provides full support for AcuSolve's features and can be used as an interactive tool or as an automation platform for a batch process.

Post-processing Features

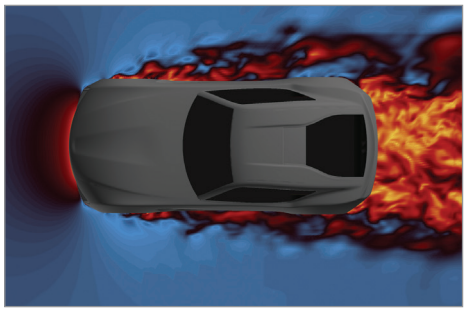
Post-processing of AcuSolve results is handled by AcuFieldView™, an OEM version of Intelligent Light's FieldView CFD post-processor. AcuFieldView allows you to post-process your AcuSolve simulations using client-server based parallel operation and comes equipped with a full set of tools to automate the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



Deflection of subsea pipeline due to vortex induced vibration



Pressure field and streamlines over a rail car in cross flow



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

Altair **Altair Engineering, Inc., World Headquarters** 1820 E. Big Beaver Rd., Troy, MI 48083-2031 USA • (P) +1.248.614.2400 • (F) +1.248.614.2411 • www.altair.com • info@altair.com

03.2016