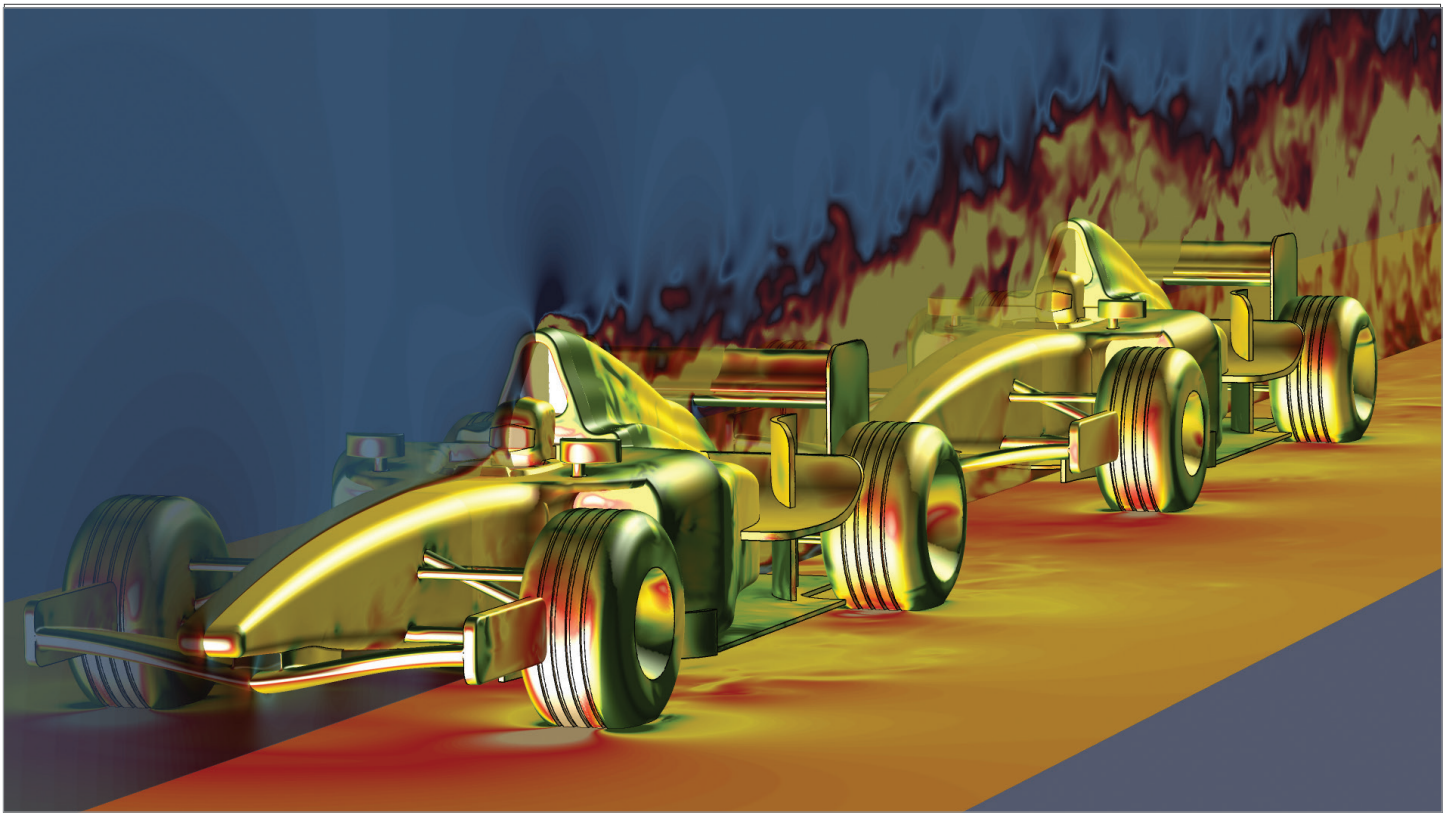
Altair AcuSolve™

Computational Fluid Dynamics

Altair | HyperWorks



Altair AcuSolve is a powerful Computational Fluid Dynamics (CFD) tool, offering a full range of physical models. Simulations of flow, heat transfer, turbulence, and non-Newtonian materials are all handled by AcuSolve's robust scalable solver technology. The well validated physical models are delivered with unmatched accuracy on fully unstructured meshes: less time building meshes means more time to explore designs.

Product Highlights

- Efficient and flexible workflow
- Full set of physical models: flow, turbulence, miscible and immiscible multiphase, and heat transfer simulations
- Accurate and stable even on highly skewed meshes
- Fast efficient solutions for both transient and steady-state simulations
- Parallel scalability on thousands of computing cores
- Advanced, stable multiphysics capability: rigid or flexible body coupling to Altair's solvers or third-party applications

Learn more:
altair.com/acusolve

Benefits

With AcuSolve's philosophy there is no need to sacrifice robustness and convenience to achieve accurate results. Days or weeks iterating on mesh quality are unnecessary to attain a stable and accurate solution: build your mesh, run the solver, then interrogate the solution.

Advanced Physics? No problem!
Challenges such as eddy-resolving turbulent simulations, miscible or immiscible multiphase, and fluid-structure interaction are simplified by AcuSolve's single solver technology. When investigating complex physics there is no need to struggle with differencing schemes, time integration settings, CFL-based stability limits, or any other solver setting. All supported flow regimes are handled by a single solver without any tuning for specific applications.

High Speed, Parallel Performance
To leverage CFD to its fullest extent, interrogation of multiple design candidates early in the design process is key which needs the fast solution process, delivered by Altair AcuSolve:

- Solution of the fully-coupled pressure/velocity equations for rapid nonlinear convergence
- Efficient parallel architecture providing distributed, shared, or hybrid parallel operation
- Proven parallel scalability on thousands of compute cores

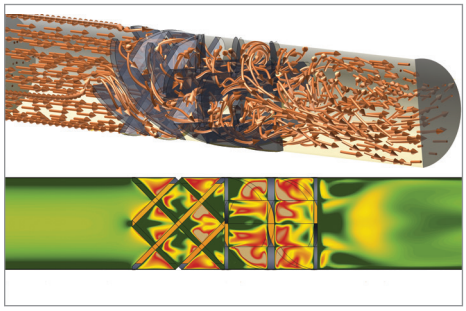
Simulation Features

Flow Modeling
AcuSolve's flow simulation focuses on incompressible and subsonic compressible flow, offering a full set of material models for investigating Newtonian and non-Newtonian flow fields. Specialized models, such as Stokes flow, are also available where the full Navier-Stokes equations are not needed.

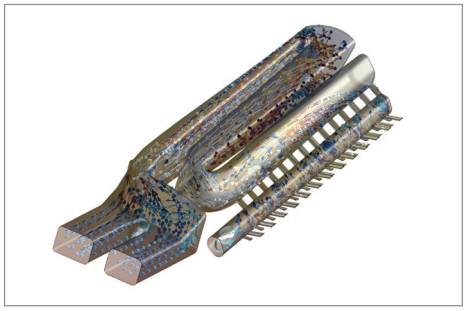
Heat Transfer and Radiation Modeling
For analyzing heat transfer in both solid and fluid mediums, AcuSolve supports a full set of features:

- Conjugate heat transfer
- Natural convection
- Enclosure radiation
- Radiation through participating media
- Solar radiation

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

- Humidity condensation and evaporation
- Thermal shells for modeling thin solids
- Simplified heat exchanger models

Turbulence Modeling
A complete selection of turbulence modeling capabilities in AcuSolve fulfills higher resolution transient simulations by supporting these 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 turbulence transition, AcuSolve supports the following transition models (compatible with Spalart-Allmaras and SST RANS/DES models):
- γ one-equation model
- γ-Reθ two-equation model

Multiphase Modeling
AcuSolve's multiphase flow simulation capability enables the simulation of two immiscible, incompressible fluids or one miscible, incompressible carrier fluid with N dispersed phases. The multiphase capabilities 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. For more sophisticated multiphase applications, species transport can also be included.

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 boundary condition tools to define how the motion of a boundary surface propagates through the model.

User-Defined Functions (UDF)
With AcuSolve material models, boundary conditions, source terms, and many other features within the solver can be customized by writing your own functions. The standard set of data access functions within the UDF, are enhanced by client-server programming capabilities, enabling CFD simulations to couple with external applications, such as control system codes.

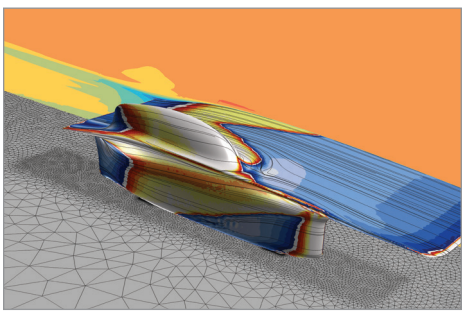
Multiphysics Capabilities
When its strength in simulating transient flows and deforming meshes, AcuSolve offers multiphysics computations and support without coupling to external codes:
- Rigid body dynamics of non-interacting bodies
- Linear structural deformations

Additional capabilities, by coupling to products in the Altair HyperWorks™ suite include:
- Non-linear structural deformations (coupling with Altair OptiStruct™)
- Multi-body dynamics (coupling with Altair MotionSolve™)
- Low frequency electromagnetics (coupling with Altair Flux™)
- Finite mass particle tracing (coupling with Altair AcuTrace™)

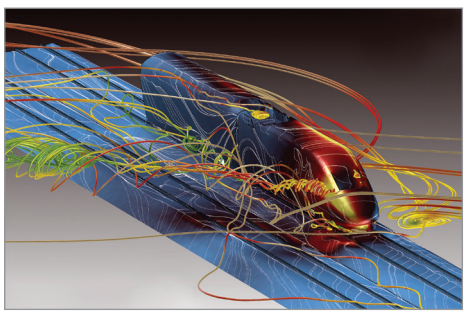
Integrated Optimization
Extending Altair's rich history with optimization, AcuSolve offers a variety of optimization capabilities contained entirely within the solver execution. AcuSolve's integrated response surface-based optimization is available for:
- Parameter optimization
- Shape optimization

Pre-processing Features
HyperWorks CFD is Altair's pre-processing Graphical User Interface (GUI) for general-purposed CFD and has a full set of options to build CFD models quickly and efficiently. SimLab is Altair's GUI for multiphysics and gives access to basic flow and heat transfer simulations, along with preparing thermal fluid structure interaction simulations from a single interface. HyperMesh has a significant portion of the most frequently used commands for general usage of AcuSolve.

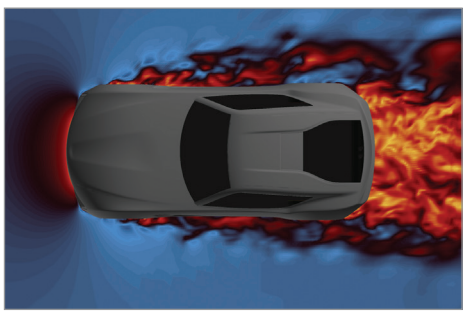
Post-processing Features
Post-processing of AcuSolve results is handled by AcuFieldView™, an OEM version of FieldView post-processor in which client-server based parallel operation is used and comes with a full set of tools to automate interrogation of solutions.



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