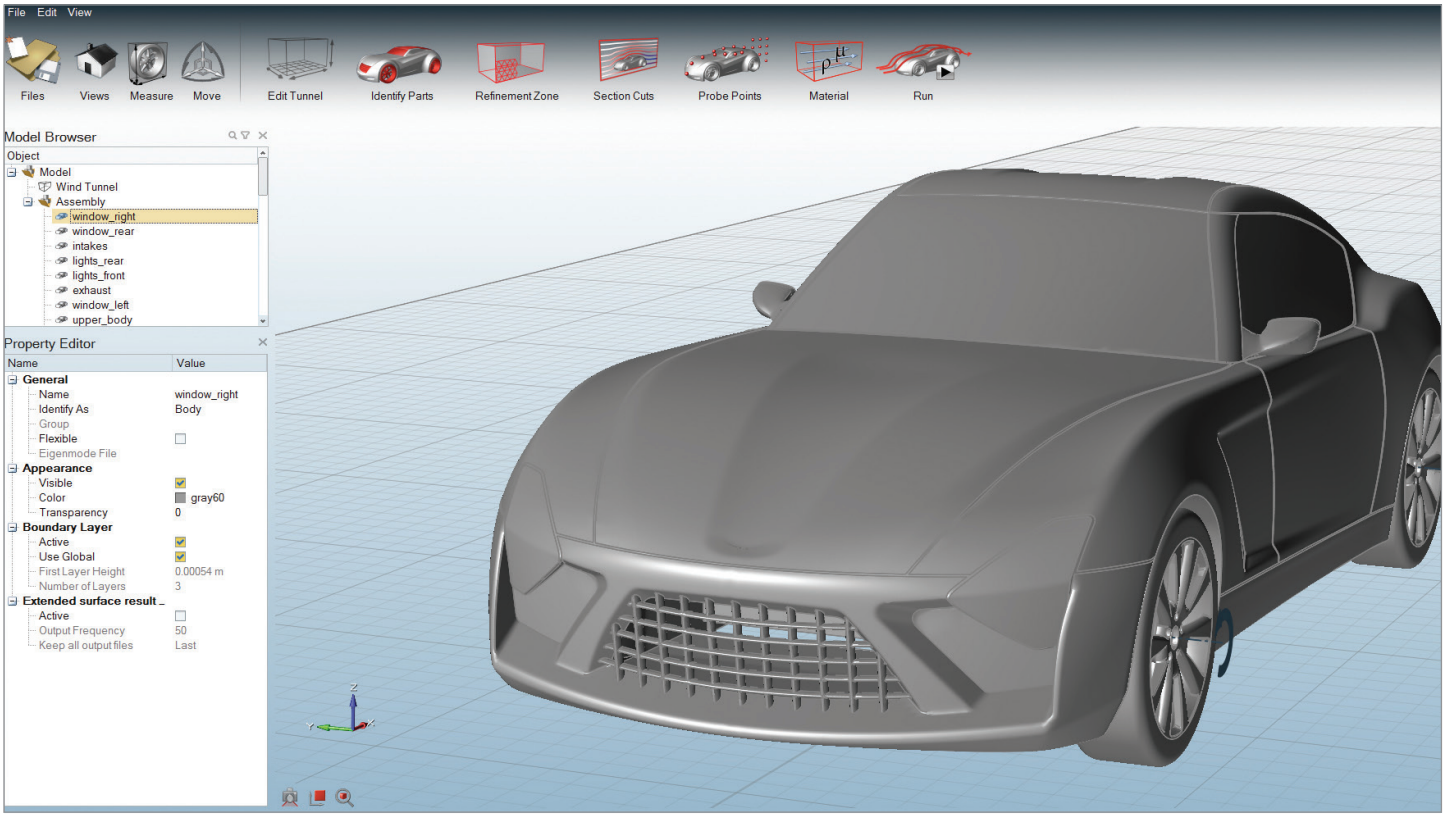
Altair Virtual Wind Tunnel™

External Aerodynamics

Altair | HyperWorks



Altair's Virtual Wind Tunnel (VWT) is a vertical application tailored for external aerodynamic studies. Designed with the users' needs in mind, the graphical user interface provides easy access to problem definition and solution strategies. The automated and customizable report generation after each simulation run provides a consistent method for design evaluation. Altair Virtual Wind Tunnel integrates smoothly with other Altair products.

Product Highlights

- Application to perform external aerodynamic studies
- Accurate, robust, and scalable computational fluid dynamics (CFD) solver
- Streamlined workflow for problem definition and solution strategy
- Automated & customizable report generation

Learn more:
altair.com/vwt

About Virtual Wind Tunnel

Virtual Wind Tunnel (VWT) is a vertical solution providing an efficient environment for external aerodynamic studies. With its automated and streamlined workflow based on Altair's Computational Fluid Dynamics (CFD) solver Altair AcuSolve™, VWT performs simulations of flow around objects, delivering transient or steady state solutions. With a focus on the automotive use case of drag and lift prediction of vehicles, other use cases include aerodynamics of buildings, bicycles, or motor bikes.

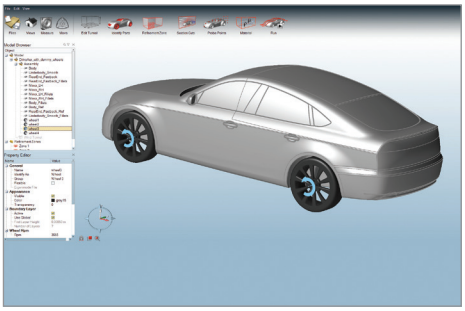
VWT combines several Altair HyperWorks™ technologies, from advanced volume meshing to high-fidelity CFD simulations, rich and powerful post-processing, and an intuitive user interface.

Accurate, Robust and Scalable CFD Solver

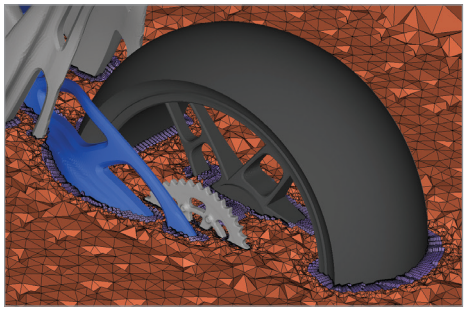
VWT is powered by Altair's computational fluid dynamics solver, AcuSolve, providing fast turnaround times, accuracy, and robustness.

Architected for parallel execution on shared and distributed-memory computer systems, using a hybrid parallelization technique, AcuSolve provides fast and efficient transient and steady-state solutions for unstructured grids. It is capable of scaling over a large number of computing cores.

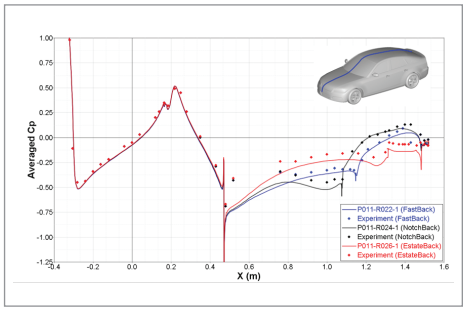
VWT utilizes Reynolds-Averaged Navier-Stokes (RANS) and Detached-Eddy Simulation (DES) technology to model turbulent flows and to predict the flow field. DES technology combines fine-tuned, statistical RANS technology for modeling near walls and attached boundary layers with the ability of large-eddy simulation (LES) to model the separated regions in the wake behind the vehicle. Accurate external aerodynamics results are achieved for both steady-state simulation using the RANS approach when



Dedicated user interface for external aero setup



CFD volume meshing of complex geometry



Proven accuracy with validated solver results

physics allows and transient simulation using the DES approach. AcuSolve's fluid structure interaction (FSI) capabilities are included in VWT to support studies of flexible components in an external flow field, e.g. aerodynamic spoilers.

Rotating parts, e.g. wheels, are modeled by prescribing a tangential wall velocity to include rotational effects into the simulation. For automotive use cases, heat-exchangers or condensers are modeled with a porous material model to consider the pressure drop through the components. The fluid material used during the simulation is defined via density and viscosity, and can be adapted to model for example water or air at a specific temperature.

Advanced Meshing

VWT comes with a fast and efficient unstructured volume mesher including boundary layer generation. User defined volume mesh refinement zones are used to create a locally refined volume mesh to capture important flow phenomena, e.g. the wake of a vehicle or a building. Parameters for boundary layer meshing can be defined globally or on a part basis to have maximum control of the total element count and use refined layers only in regions where it is necessary. Volume meshing for an external automotive aerodynamics analysis (including under body, under hood compartment, and boundary layers) is typically done in a few hours.

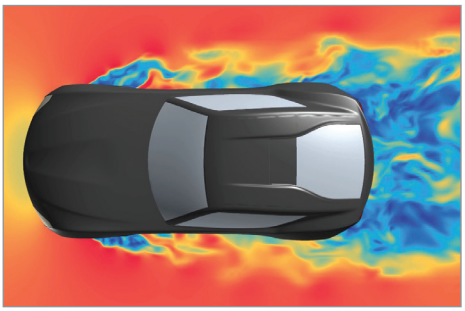
Analysis Report

After each simulation, a report is automatically generated summarizing the results like drag and lift history, providing mesh statistics, and containing the problem definition and solution strategies. User defined probe points can be included into the simulation domain to monitor the evolution of the flow field in a particular location, e.g. pressure fluctuation over time. Furthermore, VWT support user defined planar section cuts for result contour plotting, e.g. velocity contours on the symmetry plane of the vehicle.

Both report entities, the time history plots of the user defined probe points as well as the planar section cuts for contour results plotting, are included in the automatically generated report.

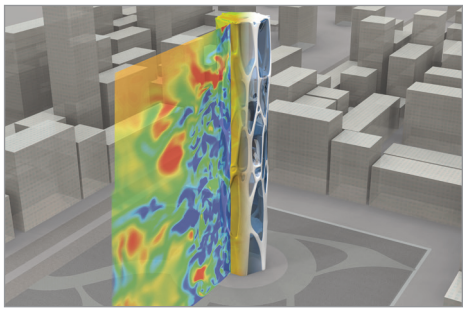
Streamlined Workflow

VWT comes with a light graphical user interface (GUI) containing various automations to reduce the user input and increase efficiency during case setup. The user imports the surface mesh of the aerodynamic study object, e.g. vehicle or building, into VWT, defines the physics and volume meshing parameters, submits the simulation, and obtains a simulation report. Analysis templates can be used to ensure consistent simulation conditions and reporting mechanisms during design evaluation.



Transient & steady solutions

Bicycle aerodynamics



Architectural aerodynamics