HyperMesh®

Left-hand Drive

Right-hand Drive

Fixed Roof

Panoramic Roof

Altair HyperMesh is a high-performance finite-element pre-processor that provides a highly interactive and visual environment to analyze product design performance. With the broadest set of direct interfaces to commercial CAD and CAE systems and a rich suite of easy-to-use tools to build and edit CAE models, HyperMesh provides a proven, consistent analysis platform for the entire enterprise.

Product Highlights

- Strong shell and solid meshing algorithms, either fully automatic or with detailed manual control

- Excellent CAD interoperability

- Comprehensive composites modeling support

- Complete interfaces to the industry's most popular solvers

- Management of complex assemblies promoting common model build

Learn more:
altairhyperworks.com/hypermesh

Benefits

Open-Architecture Design
Combining the broadest set of direct CAD and CAE interfaces with user-defined integrations, HyperMesh fits seamlessly within any simulation environment.

High-Speed, High-Quality Meshing
Streamlines the modeling process and provides a suite of tools to model even the most complex geometries.

Model Build and Assembly
HyperMesh provides a part based workflow enabling modular model build. Representation management of different topological variations for the same part speeds up the process to assemble models for different disciplines. Work package distribution for concurrent model generation is available. HyperMesh provides bi-directional communication to PDM for data exchange.

Increases End-User Modeling Efficiency
Using sophisticated batch meshing technology, HyperMesh eliminates the need to perform manual geometry clean-up and meshing, thus accelerating the model development process.

Advanced 3D Model Visualization
3D visualization of all element types (1D, 2D and 3D elements) within an FEA model eases model checking and visual verification.

Model Setup
A variety of sophisticated tools help with efficient model setup. The ID Manager ensures that all entities of a model even across or per include files adhere to a specified numbering scheme of a workgroup or a company to ensure modularity. HyperMesh provides connector technology, a highly automated way for assembly of hundreds of parts with spot- and seam welds, adhesives or bolts. Automated contact management tools are a huge time saver when defining contacts between individual parts.
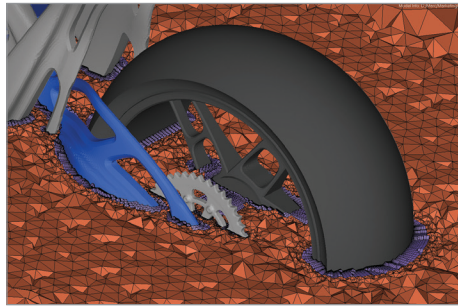
Composites Modeling
Ply and laminate entities facilitate composites modeling in terms of individual layer shapes and their stacking sequence. CATIA CPD and Fibersim readers extract composite data and map it onto FE meshes automatically. Plies and ply angles can be visualized in 3D for easier model verification.
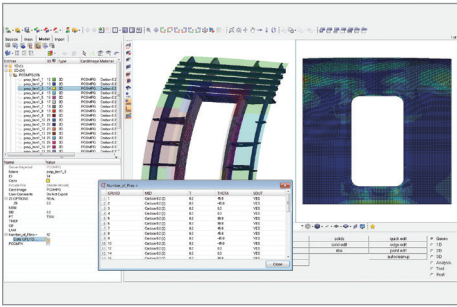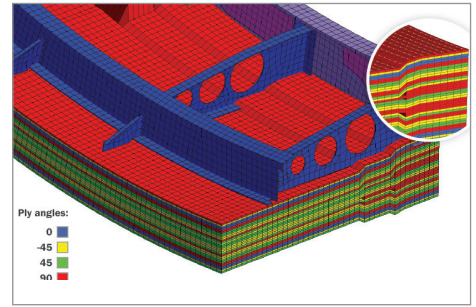
Modern and efficient CAE modeling environment

Efficient generation of high quality meshes

Fast composite modeling process and sophisticated layer visualization

Meshing Capabilities

HyperMesh presents an advanced suite of easy-to-use tools to build and edit CAE models. For 2D and 3D model creation, users have access to a variety of mesh generation capabilities, as well as HyperMesh's powerful automeshing module.

High Fidelity Meshing
- Surface meshing
- Solid map hexa meshing
- Tetra meshing
- CFD meshing
- Acoustic cavity meshing
- Shrink wrap meshing
- SPH meshing

Surface Meshing
The surface meshing module in HyperMesh contains a robust engine for mesh generation that provides unparalleled flexibility and functionality. This includes the ability to interactively adjust a variety of mesh parameters, optimize a mesh based on a set of user-defined quality criteria, and create a mesh using a wide range of advanced techniques.

Solid Meshing
Using solid geometry, HyperMesh can utilize both standard and advanced procedures to connect, separate, or split solid models for tetra-meshing or hexa-meshing. Partitioning these models is fast and easy when combined with HyperMesh's powerful visualization features for solids. This allows less time spent preparing geometries for solid meshing. The solid meshing module quickly generates high quality meshes for multiple volumes.

Mesh Controls
Mesh controls promote meshing automation for surface, batch meshing, adaptive and volume mesh generation. Detailed local and global control of mesh parameters for either the entire model or for individual features and regions of the geometry are possible. Mesh controls can be saved in the database or exported and reused in other models, promoting standardized mesh quality and repeatability.

Batch Meshing
The BatchMesher™ in HyperMesh is the fastest way to automatically generate high-quality finite element meshes for large assemblies. It is available as a standalone application or directly within HyperMesh.

By minimizing manual meshing tasks, this automeshing technology provides more time for value-added engineering simulation activities. BatchMesher provides user-specified control over meshing criteria and geometry clean-up parameters as well as the ability to output to customized model file formats.

Industry Specific Meshing
HyperMesh provides a variety of meshing algorithms for different industries and verticals, such as acoustic cavity meshing and mesh coarsening for NVH applications, and shrink wrap meshing or SPH meshing. HyperMesh also offers a highly-competitive suite of tools for CFD meshing.

CAD Interoperability

HyperMesh includes direct readers to popular native CAD file formats. Moreover, HyperMesh has robust tools to clean-up imported CAD geometry that contain surfaces with gaps, overlaps and misalignments which hinder high-quality mesh generation.

By eliminating misalignments and holes, and suppressing the boundaries between adjacent surfaces, users can mesh across larger, more logical regions of the model. This significantly increases meshing speed and quality. Boundary conditions can also be applied to these surfaces for future mapping to underlying element data.

- ACIS
- CATIA V4/V5
- IGES
- Inspire
- Intergraph
- JT
- PARASOLID
- PTC Creo
- SolidWorks
- STEP
- Tribon
- NX

Customize HyperMesh to Fit Your Environment

Customize your modeling experience through an easy-to-use interface containing drag-and-drop toolbars, configurable pull-down menus and keyboard-controlled shortcuts.

Custom Utilities: Create custom applications that are fully integrated within the HyperMesh interface.

Solver Input Translators: Users can extend HyperMesh's interface support by adding input translators to read different analysis data decks.

Solver Export Templates: Export templates allow the HyperMesh database to be exported to user-defined formats for proprietary and specialized solvers.

CAE Solver Interfacing

HyperMesh provides direct import and export support to the industry's most popular solvers. Additionally, HyperMesh provides a completely tailored environment for each supported solver.

- Abaqus
- Actran
- AcuSolve
- Adams
- ANSYS
- CFD++
- EXODUS
- Femfat
- Fluent
- HyperMath
- LS-DYNA
- Madymo
- Marc
- Moldex3D
- Moldflow
- MotionSolve
- Nastran MSC
- Nastran NX
- nCode
- OptiStruct
- PAM-CRASH
- PERMAS
- RADIOSS
- Samcef
- Simpack
- StarCD