



Features

Mesh Methods

- ▶ Tetrahedral meshing methods
 - Patch conforming
 - Patch independent
 - CFX-Mesh
- ▶ Hexahedral meshing methods
 - General sweep
 - Thin sweep
 - MultiZone
 - Hex-dominant
- ▶ Surface meshing
 - Default quad, quad/tri or tri
 - Uniform quad or quad/tri
- ▶ Beam meshing

Mesh Controls

- ▶ Global
 - Physics preference settings
 - Relevance settings
 - Inflation settings
 - Curvature-based refinement settings
 - Proximity-based refinement settings
 - Smoothing settings
 - Transition/growth settings
 - Pinch (defeaturing) settings
 - Quality settings
 - Element midside node settings
 - Rigid-body behavior settings
- ▶ Local controls
 - Automatic contact detection
 - Contact sizing
 - Body mesh method controls
 - Body, face, edge, vertex sizings
 - Body, face, edge, vertex sphere of influence
 - Body, face, edge curvature based refinement
 - Body of influence
 - Solver-based refinement controls
 - Mapped-face meshing controls
 - Match mesh controls

Comprehensive Multiphysics Meshing Tools

Meshing is an integral part of the computer-aided engineering (CAE) simulation process. The mesh influences the accuracy, convergence and speed of the solution. Furthermore, the time it takes to create a mesh model is often a significant portion of the time it takes to get results from a CAE solution. Therefore, the better and more automated the meshing tools, the better the solution.

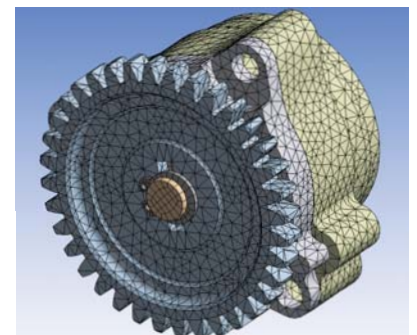
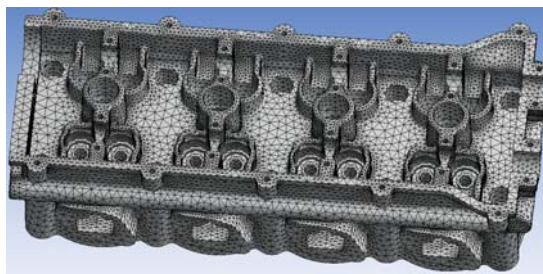
From easy, automatic meshing to a highly crafted mesh, ANSYS provides the ultimate solution. Powerful automation capabilities ease the initial meshing of a new geometry by keying off physics preferences and using smart defaults so that a mesh can be obtained upon first try. Additionally, users are able to update immediately to a parameter change, making the handoff from CAD to CAE seamless and aiding in up-front design.

Once the best design is found, meshing technologies from ANSYS provide the flexibility to produce meshes that range in complexity from pure hex meshes to highly detailed hybrid meshes; users can put the right mesh in the right place and ensure that a simulation will accurately validate the physical model.

ANSYS has a range of meshing tools that cater to nearly all physics. While the meshing technologies were developed to meet specific needs in the areas of solid, fluid, electro-magnetic, shell, 2-D and beam models, access to these technologies is available across all physics.

Solid Modeling

For solid models, meshing technologies from ANSYS provide robust, well-shaped quadratic tetrahedral meshing on even the most complicated geometries. With automatic contact detection and setup, a user requires little training to perform sophisticated analysis.



In addition, users can generate pure hex meshes using one of several mesh methods, depending on the type of model and whether the user wants a pure hex mesh or a hex-dominant mesh.

Features

- Virtual topologies
- Pinch controls
- Inflation controls
- Gap tool
- Winding-body meshing
- Wire-body meshing
- Rigid-body meshing
- Gasket meshing
- CAD instances meshing

Bi-Directional CAD Connections

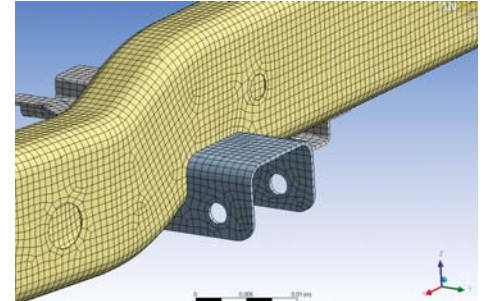
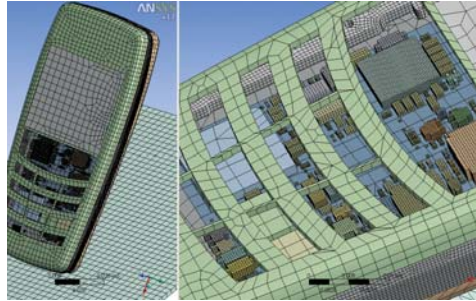
- ▶ CATIA® V5
- ▶ Unigraphics® NX™
- ▶ Autodesk® Inventor®
- ▶ Autodesk® MDT
- ▶ CoCreate Modeling™
- ▶ Pro/ENGINEER®
- ▶ SolidWorks®
- ▶ Solid Edge®

CAD Readers

- ▶ CATIA® V4
- ▶ ACIS
- ▶ IGES
- ▶ Parasolid®
- ▶ STEP
- ▶ STL

Mesh Export

- ▶ ABAQUS®
- ▶ ANSYS® Mechanical™
- ▶ ANSYS® CFX®
- ▶ ANSYS® AUTODYN®
- ▶ CGNS
- ▶ ANSYS® ICEM CFD™
- ▶ ANSYS® FLUENT®
- ▶ ANSYS® LS-DYNA®
- ▶ NASTRAN®
- ▶ ANSYS® POLYFLOW®
- ▶ SAMCEF



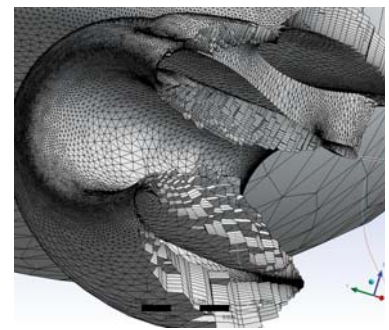
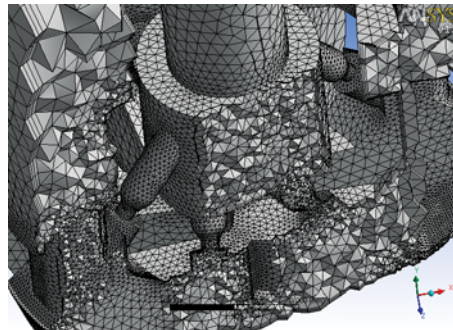
ANSYS meshing technologies provide physics preferences that help automate the meshing process. For an initial design, a mesh can often be generated in batch with an initial solution run to locate regions of interest. Further refinement can then be made to the mesh to improve the accuracy of the solution. There are physics preferences for structural, fluid, explicit and electromagnetic simulations. By setting physics preferences the software adapts to more logical defaults in the meshing process for better solution accuracy.

Other physics-based features that help with structural and explicit dynamics include:

- ▶ Automated contact handling
- ▶ Editable contact definitions
- ▶ Rigid-body contact meshing
- ▶ Gasket-element meshing
- ▶ Periodic mesh matching
- ▶ Automated beam and shell meshing
- ▶ CAD instance modeling/meshing
- ▶ Solver-based refinement
- ▶ Thin solid-shell meshing

Fluid Modeling

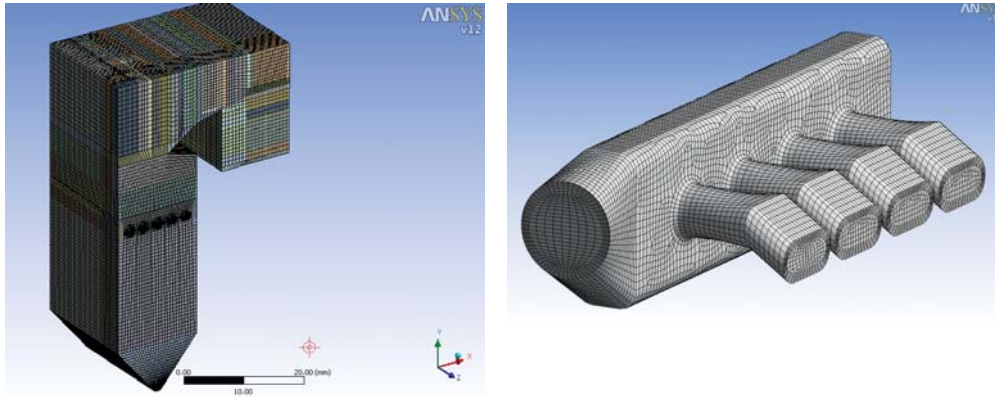
Meshing solutions from ANSYS for fluid models provide unstructured tri- and quad-surface meshing driven by curvature, proximity, smoothness and quality, in combination with a pinch capability that automatically removes insignificant features. The combination of automated surface meshing, boundary layer technology (including automatic proximity handling) and an Advancing Front tet mesh algorithm ensures high-quality, push-button meshing for fluid flow analysis. Extended sizing, matching, mapping and sweep controls provide additional flexibility, if needed.



Tet/Prism meshing of a complex drill bit (left) and the nacelle/wing/fuselage of an aircraft (right) using meshing technology from ANSYS

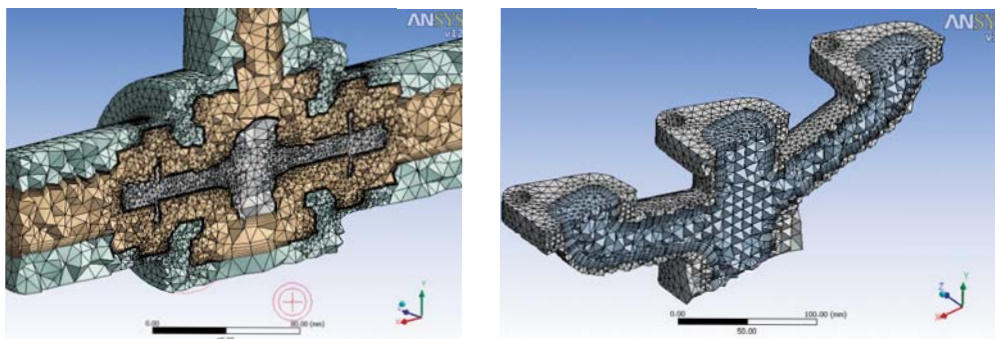
As with the solid modeling, users can also generate pure hex meshes using one of several mesh methods. Inflation can be added to the hex-meshing methods to capture boundary layers.

In addition to tet and hex meshing, there are extensive tools for hybrid meshing with hexahedral and tetrahedral regions or bodies. These models can be meshed with conformal or nonconformal meshes at common interfaces. Inflation layers can be generated across bodies.



Examples of all-hex meshing in the power and automotive industries

The ability to control the interface with conformal or nonconformal meshes is particularly useful in handling large changes in mesh size, or to handle an interface between different physics. For example, when modeling a fluid structure interaction (FSI) problem, the solid and fluid regions share common interfaces but the mesh can be conformal or nonconformal.



Meshing technology from ANSYS allows you to create suitable meshes for structural and fluid flow simulations in the same model

For users who want control over every step in the meshing process to create highly crafted brick meshes or to mesh directly on scan data, extended meshing capabilities are available in ANSYS ICEM CFD and TGrid software products.

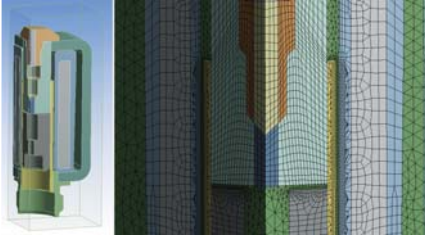
Related Products

The ANSYS meshing platform provides the foundation for leveraging meshing technologies from ANSYS Mechanical, ANSYS ICEM CFD, ANSYS CFX, GAMBIT®, TGrid™ and CADOE™, and takes advantage of the ANSYS® Workbench™ 2.0 framework so users can access these meshing tools in a unified environment tailored to their experience. ANSYS also offers extended meshing products that allow more interactive meshing needed by advanced users or for difficult geometries.

The ANSYS ICEM CFD suite of tools supports a wide range of CAD and faceted data imports including STL. Its patch-independent meshing technologies allow for meshing of very poor or badly connected data that would not be possible with other meshing tools. It includes a variety of mesh methods and types including: block structured hex mesh; Octree, Delaunay and Advancing Front tet meshes; prism meshing; body-fit Cartesian, stairstep Cartesian, hex-dominant and hex-core meshing; and autoblocker and quad-paved surface meshing. These mesh methods can be interchanged and combined with legacy meshes and/or handcrafted meshes. It also offers advanced mesh editing which, in addition to being extremely powerful and flexible, maintains projection with the geometry for greater accuracy. ANSYS ICEM CFD also provides output to a number of structured and unstructured, fluid and structural mechanics solvers.

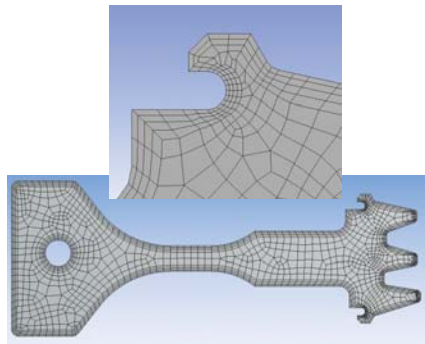
TGrid is a specialized pre-processor for fluid flow analysis. It is used to create large unstructured tetrahedral and hex-core meshes for highly complex geometries such as automotive underhood and cabin models. The advanced surface wrapper technology is equipped with a fully automatic leak/hole detection and fixing algorithm; this eliminates tedious manual cleanup and at the same time ensures the creation of a closed domain. A single surface recovery technique for proper simulation of thermal shields is also available. TGrid's tetrahedral meshing algorithm combines the speed and stability of a Delaunay approach with the quality of an Advancing Front approach. In addition, there are several quality-enhancing tools that lead to improved accuracy of fluid flow analysis. TGrid's prism layer technology includes fully automatic proximity and baffle handling, and its cavity remeshing module allows users to swiftly replace parts and components without remeshing the full model.

Features

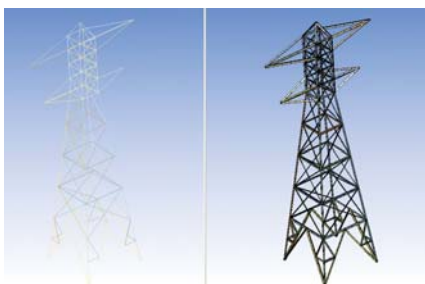


Solenoid model with refined mesh of air gap

Courtesy Parker Hannifin, Fluid Control Division



Axisymmetric model with inflation layers



Simplified beam model representing an electrical tower

Courtesy Hughes Christensen

ANSYS® Emag™ Modeling

ANSYS Emag meshing software for electromagnetic models provides automated yet well-controlled meshing to accurately capture the field (dielectric) regions with tet, hex or hybrid meshes.

Electromagnetics models typically include narrow gaps between parts, such as rotors and stators. It is important to have a refined mesh in these gaps, and ANSYS Emag meshing software provides full control over this “air gap” meshing. ANSYS® Icepak® and Ansoft products, along with other solutions from ANSYS, deliver customized applications with integrated meshing capabilities for electromagnetic problems for component-level, board-level and cabinet-level design.

Shell and 2-D Modeling

Shell modeling and meshing solutions from ANSYS offer numerous approaches that provide meshes that best meet the physics. In general, this consists of two approaches that use common tools:

- ▶ 2-D axisymmetric or planar models can be used to simplify 3-D physics in a 2-D fashion. 2-D models can be meshed with quad meshes, quad-dominant meshes or all-triangle meshes.
- ▶ Shell models can be used to simplify 3-D models to a set of sheets with a defined thickness. This is particularly useful for modeling sheet metal or thin structural parts. Shell parts can also be meshed with quad meshes, quad-dominant meshes or all-triangle meshes.

Beam Modeling

ANSYS meshing solutions provide easy methods to simplify geometry to beam models or create beam models that help users easily construct simplified models for quick analyses.

The ANSYS Advantage

With the unequalled depth and unparalleled breadth of our engineering simulation solutions, companies are transforming their leading edge design concepts into innovative products and processes that work. Today, 97 of the top 100 industrial companies on the “FORTUNE Global 500” invest in engineering simulation as a key strategy to win in a globally competitive environment. They choose ANSYS as their simulation partner, deploying the world’s most comprehensive multiphysics solutions to solve their complex engineering challenges. The engineered scalability of our solutions delivers the flexibility customers need, within an architecture that is adaptable to the processes and design systems of their choice. No wonder the world’s most successful companies turn to ANSYS — with a track record of almost 40 years as the industry leader — for the best in engineering simulation.