ANSYS RELEASE 13.0

ANSYS Meshing Solutions

Comprehensive Multiphysics Meshing Tools

Meshing is an integral part of the computer-aided engineering simulation process. The mesh influences the accuracy, convergence and speed of the solution. Furthermore, the time it takes to create and mesh a 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.

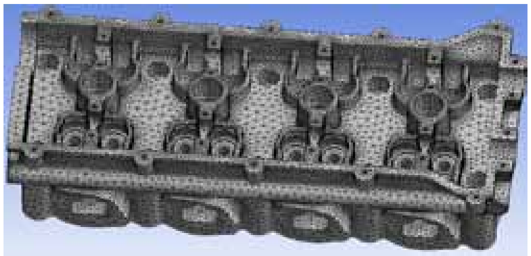
From easy, automatic meshing to a highly crafted mesh, ANSYS provides the ultimate solution. Powerful automation capabilities ease the initial meshing of a new geometry by keying off physics preferences and using smart defaults so a mesh can be obtained upon first try. Additionally, a user can update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

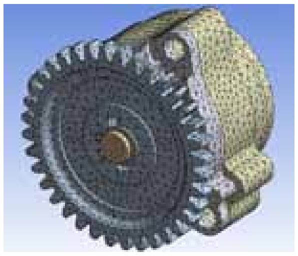
Once the best design is found, meshing technologies from ANSYS provide the flexibility to produce meshes that range in complexity from pure hex to highly detailed hybrid; a user can put the right mesh in the right place and ensure that a simulation will accurately validate the physical model.

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

Structures 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 or hex-dominant mesh.





Features

Mesh Methods

- Tetrahedral meshing methods
  - Patch conforming
  - Patch independent
  - ANSYS CFX-Mesh
- Hexahedral meshing methods
  - General sweep
  - Thin sweep
  - MultiZone
  - Hex-dominant
  - Cut cell Cartesian
- 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

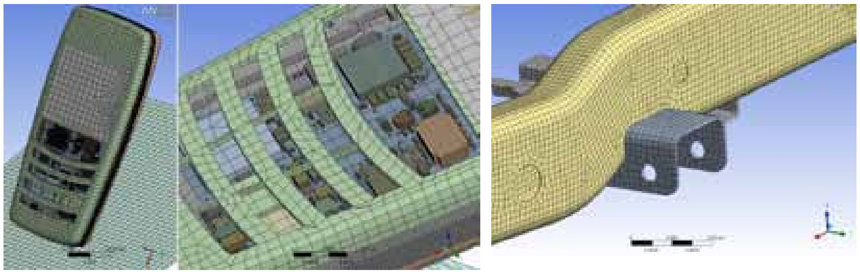
Features

- Solver-based refinement controls
- Mapped-face meshing controls
- Match mesh controls
- 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®

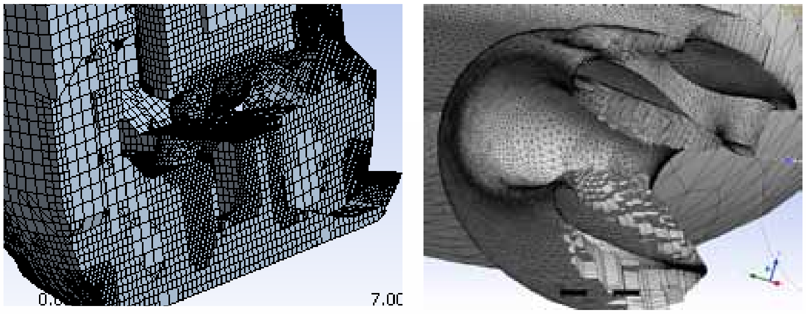
CAD Readers
- CATIA® V4
- ACIS®
- IGES
- Parasolid®
- STEP
- STL

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 to 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.

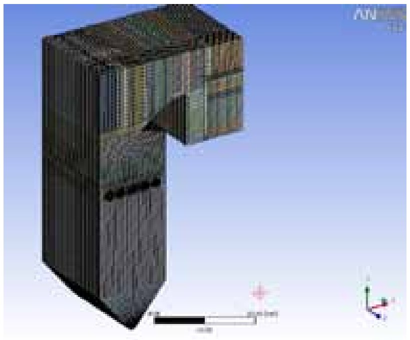
Other physics-based features that help with structural and explicit dynamics include:
- Automated contact handling
- Automated beam and shell meshing
- Editable contact definitions
- CAD instance modeling/meshing
- Rigid-body contact meshing
- Solver-based refinement
- Gasket-element meshing
- Thin solid-shell meshing
- Periodic mesh matching

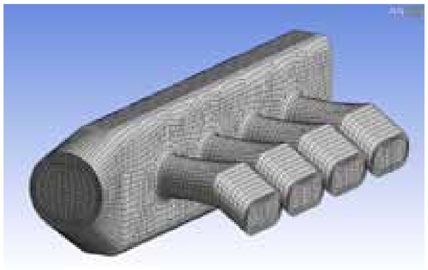


CutCell meshing of a complex drill bit (left) and Tet/Prism meshing of nacelle/wing/fuselage of an aircraft (right) using meshing technology from ANSYS
Courtesy Hughes Christensen.

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.

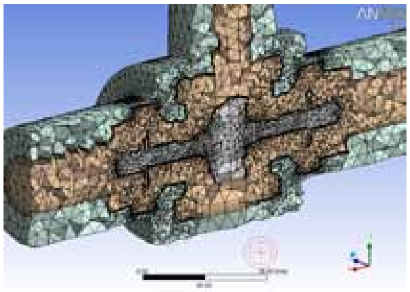
As with the solid modeling, a user can generate pure hex meshes using one of several mesh methods. Inflation can be added to the hex-meshing methods to capture boundary layers.

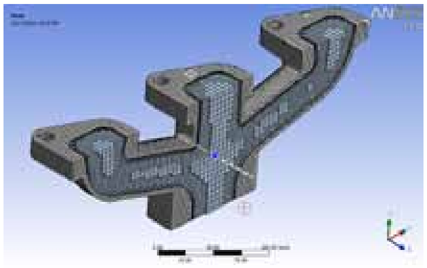




Examples of all-hex meshing in the power and automotive industries

Beyond 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.





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

The ability to control the interface with conformal or nonconformal meshes is particularly useful in handling large changes in mesh size or 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.

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.

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.

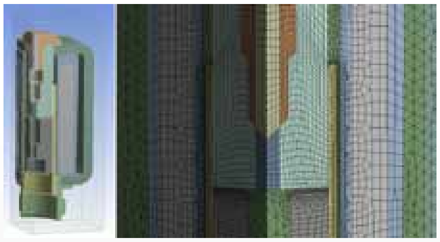
Electromagnetics models typically include narrow gaps between parts, such as rotors and stators. It is important to have a refined mesh in these gaps, and ANSYS Emag meshing capabilities provide full control over this "air gap" meshing.

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™. It takes advantage of the ANSYS® Workbench™ 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.

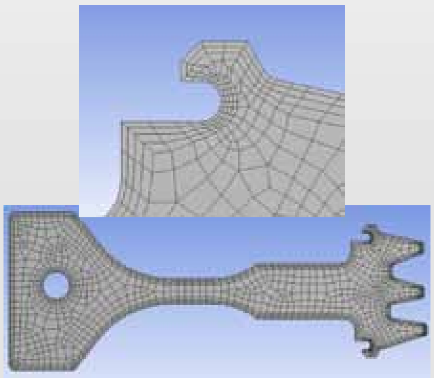
The ANSYS ICEM CFD suite of tools supports a wide range of CAD and faceted data imports including STL. Its patch-independent meshing technologies allow for meshing of very poor or badly connected data that would not be possible with other meshing tools. It includes a variety of mesh methods and types including block structured hex mesh; Octree, Delaunay and advancing front tet meshes; prism meshing; body-fit Cartesian, stairstep Cartesian and 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. The suite 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 provides output to a number of structured and unstructured, fluid and structural mechanics solvers.

TGrid technology 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 auto-matic 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. The TGrid 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. The prism layer technology of TGrid includes fully automatic proximity and baffle handling, and its cavity remeshing module allows the user to swiftly replace parts and components without remeshing the full model.



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

ANSYS® Icepak® and the Ansoft family of products, along with other solutions from ANSYS, deliver customized applications with integrated meshing capabilities for electromagnetic problems, whether for component-level, board-level or cabinet-level design.

Shell and 2-D Modeling

Shell modeling and meshing solutions from ANSYS offer numerous approaches in providing 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 provides easy methods to simplify geometry to beam models or to create beam models that help users easily construct simplified models for quick analyses.

The ANSYS Advantage

With the unequalled depth and unparalleled breadth of ANSYS engineering simulation solutions, companies are transforming their leading-edge design concepts into innovative products and processes that work. Today, almost all 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 multi-physics solutions to solve their complex engineering challenges. The engineered scalability of solutions from ANSYS 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 40 years as the industry leader — for the best in engineering simulation.

ANSYS®

www.ansys.com

ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com

Toll Free U.S.A./Canada:
1.866.267.9724
Toll Free Mexico:
001.866.267.9724
Europe:
44.870.010.4456
eu.sales@ansys.com

GSA Contract Holder