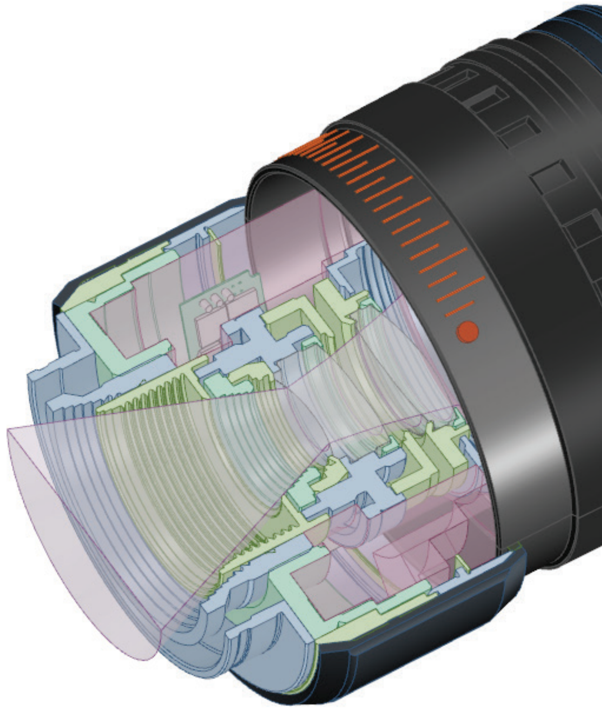
ANSYS SpaceClaim

ANSYS SpaceClaim is the fastest platform for working with geometry, whether to design a part, prepare a model for downstream jobs, or reverse engineer an STL file. With the release of ANSYS SpaceClaim 2016, we deliver 10-times faster 3-D modeling than any other product on the market today.



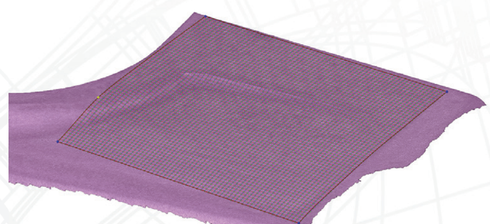
ANSYS SpaceClaim customers will speed up their time to market with 10-times performance improvements, more tools for faster geometry creation and editing, and expanded file import and editing capabilities. ANSYS SpaceClaim empowers customers to realize efficiency gains across their entire product workflow.

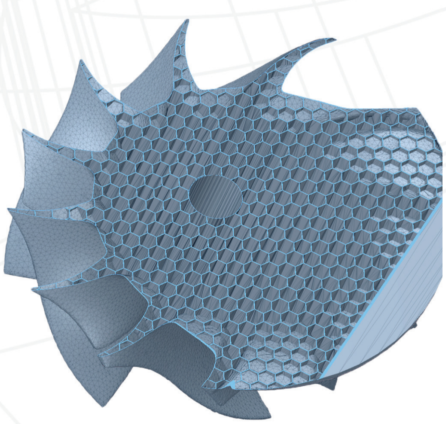
Multiple Performance Enhancements

- Includng 60% reduction in startup time; 10-50x speed up time for large model save and load time; and 200x speed improvement to tools like Imprint
- Import Enhancements such as: ECAD (Cadence, GDSII, ODB++, etc) with variable fidelity import; Sketchup 2015; Solidworks 2015; Creo 3.0; ST 7; CATIA V6
- Graphical support for DirectX11 and OpenGL
- New language support: Russian, Dutch, Hungarian

Design Enhancements

- Live linking with Keyshot is now available to easily update edits to a model in Keyshot without redesigning the rendering scene
- Users will be especially excited about the new ability to flag groups for locking/ monitoring dimensions and the new scripting option which will be out in Beta
- In addition, there will be enhancements such as: Patch blend preview; Interactive curve, drag, snap, and copy; Ability to wrap points





New Reverse Engineering tool

Our Skin Surfacing tool allows ANSYS SpaceClaim users to more easily reverse engineer complicated geometry quickly by surface fitting to faceted models. Users can direct edit the patch boundaries, connect neighboring patches, create 4-sided, 3-sided, domed, and periodic patches, and automate smoothing

3-D Printing Enhancements

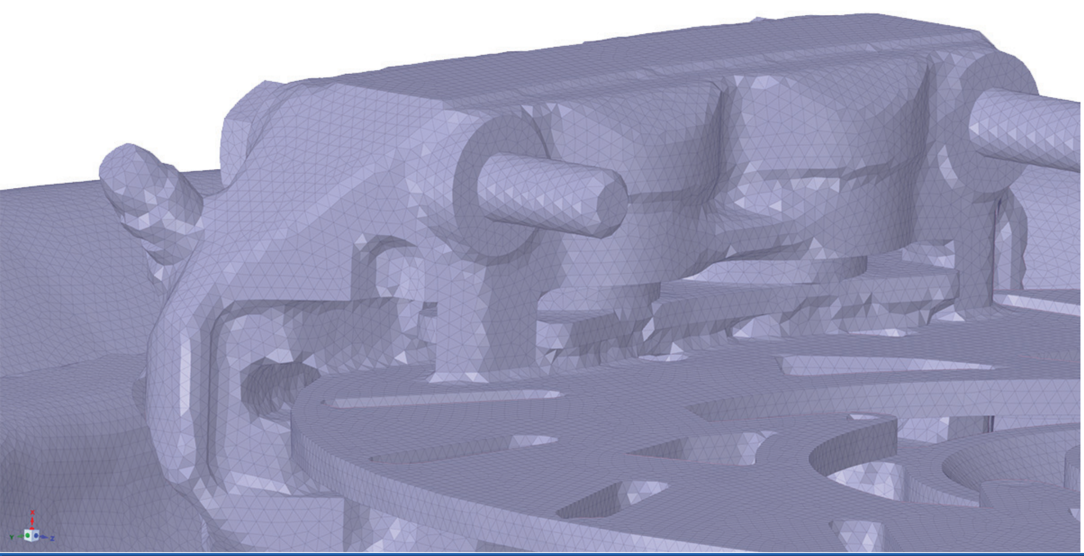
- Another exciting new feature is our new Shrinkwrap Tool for 3-D printing, which makes it easier and faster to repair STLs and create watertight faceted files from complicated geometry
- Import PLY files and Import/Export VRML files
- We improved the sensitivity in our Thickness Detection Tool and improved our Autofix Tool to correct over-connnected angles
- Customers will see a 50-100% performance improvement for many of our 3-D printing tools

Manufacturing Additions

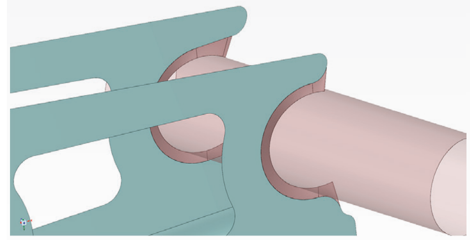
- Turn profile MFG-centric improvements like protrusion handling; improved orientation setup, and feed rate input
- Other manufacturing enhancements such as: Curve table improvements; QIF inspection format import/export; and SAT PMI export
- New Sheet Metal Tools: Flange Tool; Create/Unfold watertight corners; Create/Unfold complex forms

Simulation Enhancements

- We created an Extract Geometry command for sub-modeling
- There are Improvements to the Imprint tool such as: tolerant imprint added; support for beam/face/edge intersections; and a larger/no item limit
- We expanded the Power Selection Tool capability for beams, midsurfaces by profile type, and thickness
- Repair tools are improved to include surface split edges and surface short edges
- We've added a Library of standard beam x-sections
- There is a new Weld tool to ensure the mesh includes all welded connections
- New capability for continuous round chain splitting



SpaceClaim 2016: The fastest solution to create, edit, and repair 3-D data.



- Enhanced 3-D Printing Preparation
- Design
- Manufacturing, including Sheet Metal
- Reverse Engineering
- Simulation
- Import/Edit
- Performance

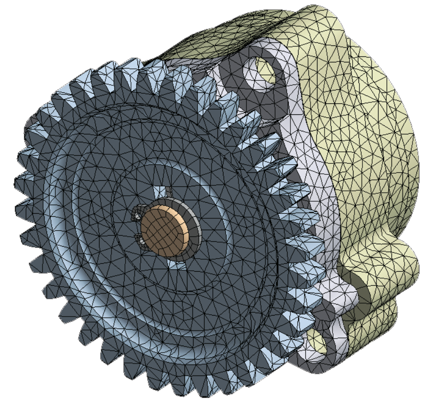
ANSYS, Inc.
www.ansys.com
ansysinfo@ansys.com
866.267.9724

© 2015 ANSYS, Inc.
All Rights Reserved.

Realize Your Product Promise®

ANSYS®

Meshing Solutions

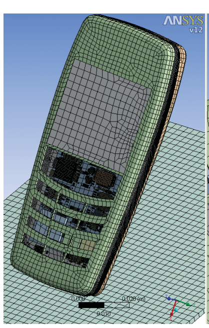


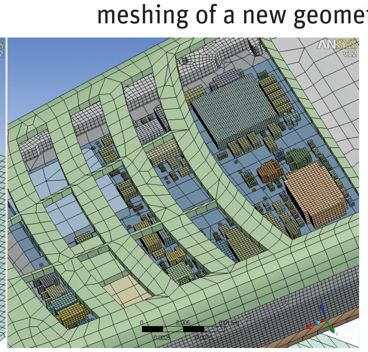
ANSYS Meshing Solutions

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 handoff from CAD to CAE seamless and aiding in upfront design.





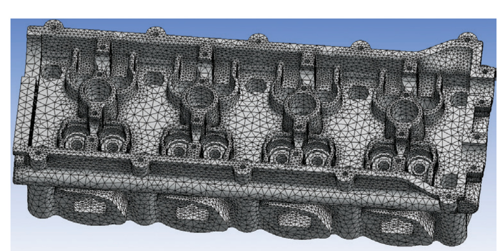
Once the best design is found, meshing technologies from ANSYS provide the flexibility to produce meshes that range in complexity from pure hex to highly detailed hybrid; a user can put the right mesh in the right place and ensure that a simulation will accurately validate the physical model.

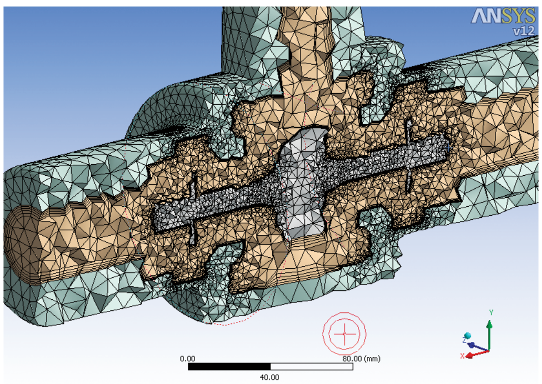
ANSYS has a range of meshing tools that cater to nearly all physics. While the meshing technologies were developed to meet needs in a specific area — solid, fluid, electromagnetic, shell, 2-D or beam models — access to these technologies is available across all physics.

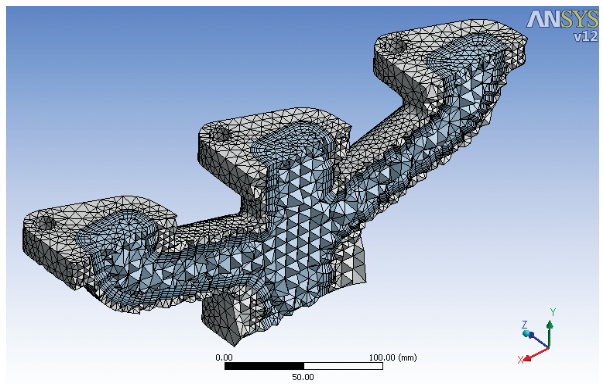
Structures Modeling

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.

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







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

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

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.

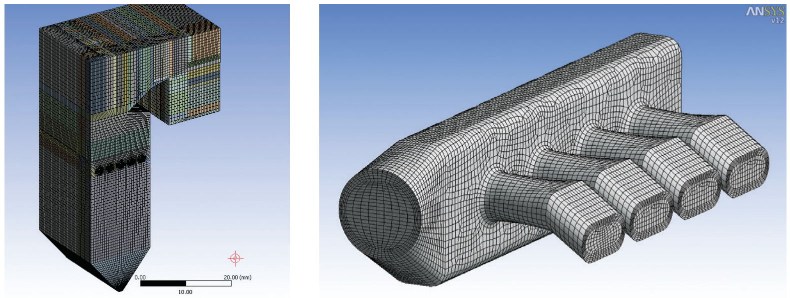
Beyond tet and hex meshing, there are extensive tools for hybrid meshing with hexahedral and tetrahedral regions or bodies. These models can be meshed with conformal or nonconformal meshes at common interfaces. Inflation layers can be generated across bodies.

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.

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
  - Solver-based refinement controls
  - Mapped-face meshing controls
  - Match mesh controls
  - Virtual topologies
  - Pinch controls
  - Inflation controls
  - Gap tool



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.

Features *continued*

- 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

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.

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, quaddominant 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

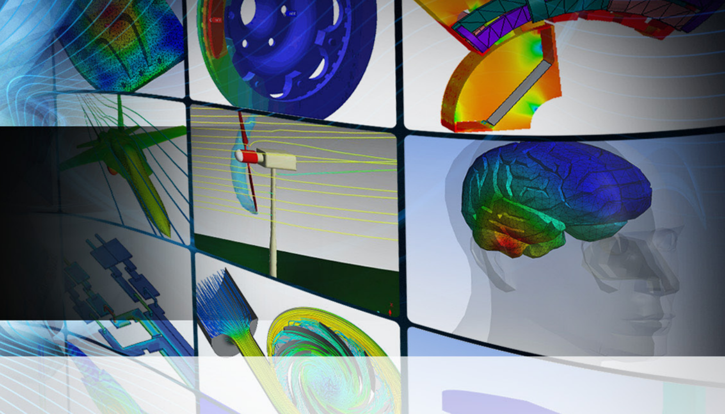
With the unequalled depth and unparalleled breadth of ANSYS engineering simulation solutions, companies are transforming their leading-edge design concepts into innovative products and processes that work. Today, almost all of the top 100 industrial companies on the "FORTUNE Global 500" invest in engineering 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, Inc.
www.ansys.com
ansysinfo@ansys.com
866.267.9724

ANSYS is dedicated exclusively to developing engineering simulation software that fosters rapid and innovative product design. Our technology enables you to predict with confidence that your product will thrive in the real world. For more than 40 years, customers in the most demanding markets have trusted our solutions to help ensure the integrity of their products and drive business success through innovation.

ANSYS and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. All other brand, product, service and feature names or trademarks are the property of their respective owners.

© 2014 ANSYS, Inc. All Rights Reserved.



ANSYS® RELEASE 13.0

ANSYS ICEM CFD

Extend your ANSYS Meshing Capabilities with ANSYS® ICEM CFD™ Meshing Software

- Mesh from dirty CAD and/or faceted geometry (STL, etc.)
- Efficiently mesh large, complex models
- Hexa mesh (structured or unstructured) with advanced control
- Extended mesh diagnostics and advanced, interactive mesh editing
- Output to a wide variety of computational fluid dynamics (CFD) and finite element analysis (FEA) solvers and neutral formats

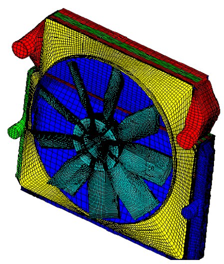
Geometry Import

ANSYS ICEM CFD software supports a wide range of direct CAD interfaces and geometry readers, including faceted geometry and mesh readers. Components from different formats can easily be combined within one meshing session. Flexible geometry support, combined with the ANSYS geometry — tolerant meshers, can reduce or eliminate the need for CAD repair or NURBS surfacing of faceted geometries.

Complete Meshing Suite

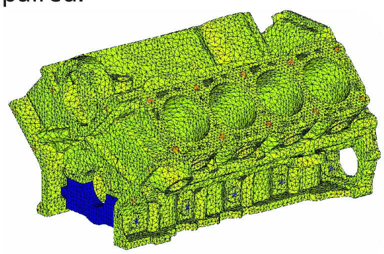
The flexible mesh-generation tools within ANSYS ICEM CFD offer the capability to parametrically create volume or surface meshes from geometry or mesh in multi-block structured, unstructured hexahedral, Cartesian, tetrahedral, tetra/prism hybrid, hexa hybrid and unstructured quad/tri shell formats.

ANSYS ICEM CFD includes surface-patch independent mesh methods for generating shell, tetrahedral or hexahedral meshes. Patch independence allows the user to selectively capture important geometric features yet ignore flaws such as slivers or gaps. These meshers can walk over significant defects in the geometry without requiring it to be repaired.



Courtesy Modine Manufacturing.

ANSYS ICEM CFD Hexa uses a primarily top-down blocking approach to efficiently hex mesh complex models without the need to subdivide the geometry. It offers interactive and automated tools that provide a high degree of quality and control for hex meshing. The blocking is scriptable and parametric and can be associated with topologically similar geometries, saving work on successive models and empowering simulation to lead design.



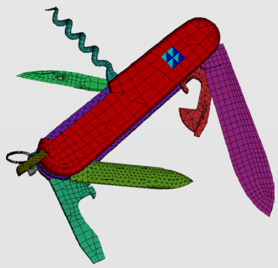
Courtesy Ford Motor Company.

A wide variety of meshing tools to support a wide variety of physics

Product Features

Geometry

- Direct CAD interfaces: Pro/ENGINEER®, UGS™ NX™, SolidWorks®, ANSYS® DesignModeler™
- Geometry readers: CATIA®, Parasolid®, ACIS®, CAPRI, DXF/DWG, I-DEAS® Viewer XML (IDI), IGES, STEP, Plot3D, Rhino3D, ANSYS® Workbench™ readers and more
- Faceted geometry: STL, VRML, formatted point data, third-party mesh formats
- Tools for geometry creation, repair and simplification



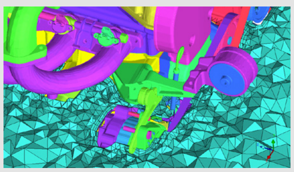
ANSYS ICEM CFD software is the Swiss Army® Knife of meshers

Meshing Capabilities

- Robust algorithms for creating a wide variety of mesh types: hexa, tetra, prism, pyramid, quad, tri or bar elements (linear or quadratic) for use with a range of physics and solvers
- Extended, parametric and persistent mesh controls can be applied globally or specifically as needed
- Tolerant of imperfect CAD data containing sliver surfaces, gaps, holes and overlaps

Mesh Editing and Diagnostics

- Check mesh, wide variety of metrics, histogram, subsets and other display tools



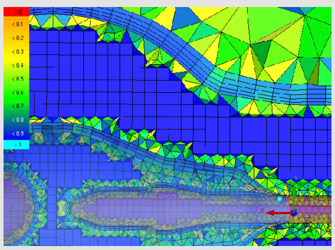
Shrinkwrap over complex geometry

Product Features

- Smooth, coarsen, refine, extrude, remesh, stitch, merge nodes or meshes, split, move, align, redistribute, change projection, transform, change element types, renumber, adjust thickness or normal, delete and much more

Solver Output Interfaces

- Output more than 100 CAE formats including ANSYS products, third-party solvers and neutral formats
- Advanced mechanical setup including contact, material and element properties, loads, constraints and advanced solver options
- Option to run certain solvers in batch mode

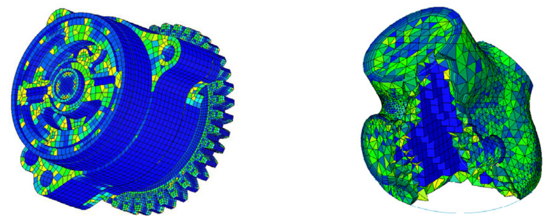


Miscellaneous

- A unified and customizable GUI environment
- Fully scriptable with automated batch processing
- Tetra/prism parallel meshing on shared memory computers
- Multi-level undo and redo functionality for geometry, meshing and mesh editing actions
- Supported on 32- and 64-bit Windows® XP, Vista® and Windows 7®, 32- and 64-bit Linux® SuSE™ and Red Hat®, and UNIX® brands SGI® Altix®, IBM® AIX and HP® IA64 UNIX

Mesh Diagnostics and Editing

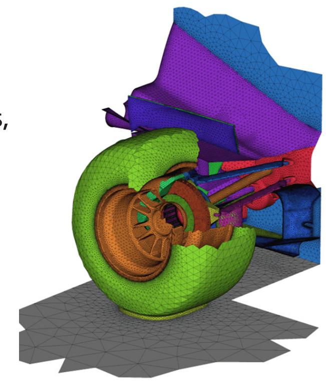
ANSYS ICEM CFD technology includes industry-leading mesh diagnostic and repair capabilities. In addition to providing comprehensive mesh checks, it offers a wide variety of quality metrics and other tools for mesh diagnosis and repair. Interactive mesh editing functions allow for precise control of node location, splitting of edges, creation of elements and much more. Automatic operations include smoothing, coarsening/refining, remeshing, and merging hexa and tetra meshes. All of the mesh editing functions respect the geometry features, maintaining geometric integrity.



Easy hybrid meshing

Output Interfaces

ANSYS ICEM CFD offers output interfaces to more than 100 different CAE formats, including support for all major CFD solvers, neutral formats such as CGNS, and FEA solvers such as ANSYS® Mechanical™, ANSYS® LS-DYNA®, ABAQUS® and NASTRAN®.



ICEM CFD octree tetra is ideal for conjugate heat transfer in complex assemblies

The ANSYS Advantage

With the unequalled depth and unparalleled breadth of engineering simulation solutions from ANSYS, companies are transforming their leading edge design concepts into innovative products and processes that work. Today, almost all 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 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.

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

Contract Holder

ANSYS, ANSYS Workbench, Ansoft, AUTODYN, CFX, FLUENT, and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. All other brand, product, service and feature names or trademarks are the property of their respective owners.

© 2010 ANSYS, Inc. All Rights Reserved. Printed in U.S.A. MKT0000497 10-10