

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

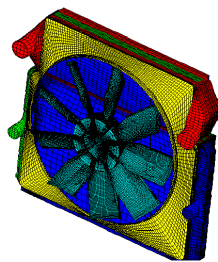
Geometry Import

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

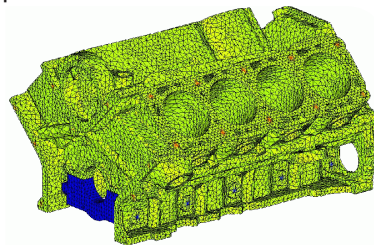
The flexible mesh-generation tools within ANSYS ICEM CFD offer the capability to parametrically create volume or surface meshes from geometry or mesh in multi-block structured, unstructured hexahedral, Cartesian, tetrahedral, tetra/prism hybrid, hexa hybrid and unstructured quad/tri shell formats.

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.



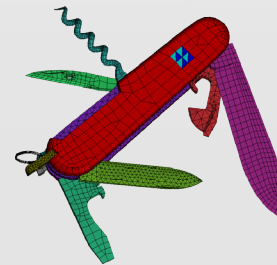
Courtesy Ford Motor Company.

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

Product Features

Geometry

- Direct CAD interfaces: Pro/ENGINEER®, UGSTM 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



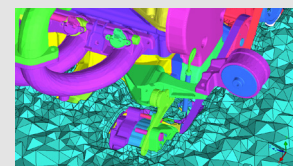
ANSYS ICEM CFD software is the Swiss Army® Knife of meshers

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



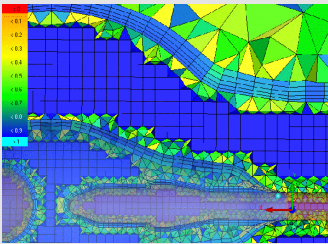
Shrinkwrap over complex geometry

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

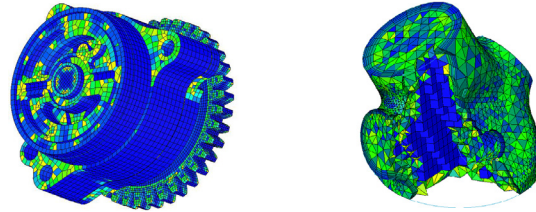


Miscellaneous

- 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.



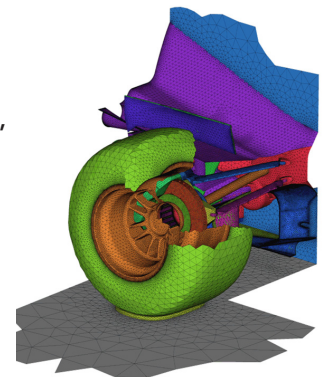
Easy hybrid meshing

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®.

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.



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

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

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.