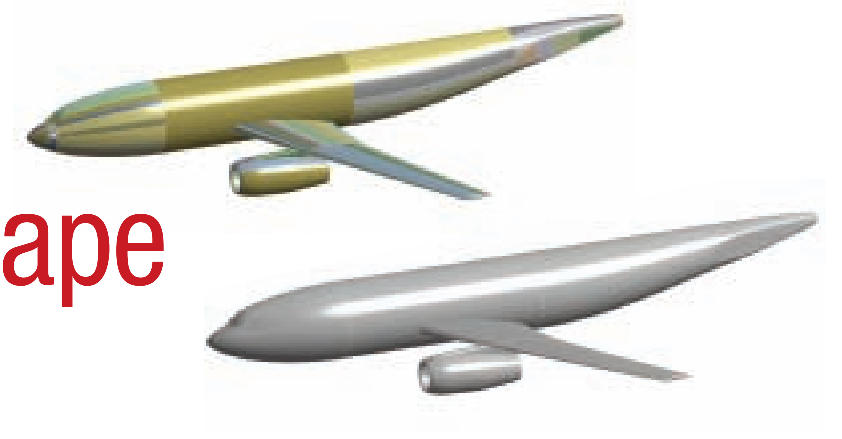


Automated cleanup and repair of imported geometry: New tools automatically detect and fix typical problems, such as small edges, sliver faces, holes, seams and faces with sharp angles. Geometry models can now be prepared for analysis at a much faster pace. These images show an aircraft model before (top) and after (bottom) cleanup.

Taking Shape in 12.0

ANSYS combines depth of simulation with industry experience to provide geometry and meshing tools that realize simulation results faster.

Engineering simulation software users have been known to spend up to 90 percent of their simulation-related time working on pre-processing tasks. By targeting developments in capabilities to increase ease of use, simplifying pre-processing tasks, and increasing the capabilities of pre-processing tools, ANSYS has systematically delivered exciting advances to increase the efficiency of simulation.

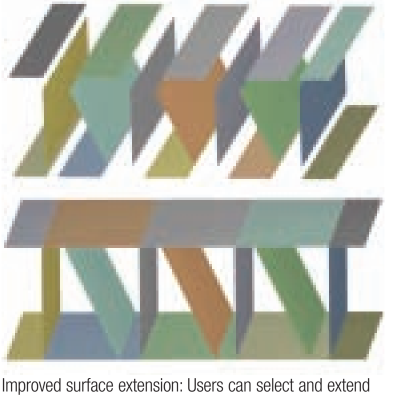
ANSYS has combined rich geometry and meshing techniques with its depth of knowledge and experience, and the end result is products capable of harnessing integrated geometry and meshing solutions that share core libraries with other applications. At releases 10.0 and 11.0, ANSYS introduced robust, new meshing capabilities from ANSYS ICEM CFD and ANSYS CFX tools into the ANSYS meshing platform — which provides the foundation for unifying and leveraging meshing technologies, making them interoperable and available in multiple applications. Taking advantage of the enhanced ANSYS Workbench 2.0 framework, the company provides further significant improvements for ANSYS 12.0 geometry and meshing applications.

CAD Connections

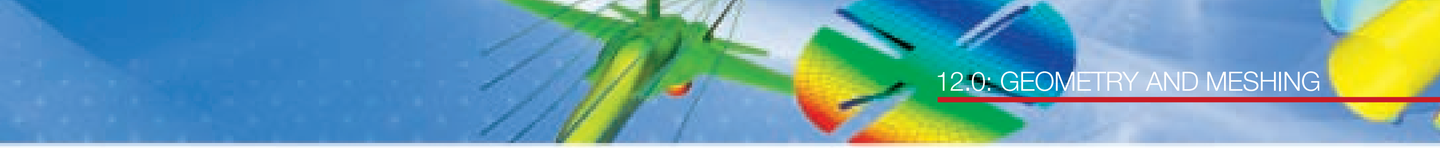
ANSYS continues to deliver a leading CAD-neutral CAE integration environment, providing direct, associative and bi-directional interfaces with all major CAD systems, including Unigraphics®, Autodesk® Inventor®, Pro/ENGINEER®, CATIA® V5, PTC CoCreate® Modeling, SolidEdge®, SolidWorks®, and Autodesk® Mechanical Desktop®. Software from ANSYS also supports file-based readers for IGES, STEP, ACIS®, Parasolid®, CATIA® V4 and CATIA V5. At ANSYS 12.0, geometry interfaces have been enhanced to import more information from CAD systems, including new data types such as line bodies for modeling beams, additional attributes such as colors and coordinate systems, and improved support for named selections created within the CAD systems.

For pre-processing larger models, release 12.0 includes support for 64-bit operating systems, and smart and selective updates of CAD parts. The newly introduced ability to selectively update CAD components allows users to update individual parts instead of an entire assembly, thus making geometry updates much faster and more targeted.

> "ANSYS 12.0 will set the stage for major improvements in our design processes. Two of Cummins' core tools, ANSYS FLUENT and ANSYS Mechanical, are coming together in the ANSYS Workbench environment. I am also very pleased to see that geometry import continues to improve, and we have several more meshing options."
>
> — Bob Tickel
> Director of Structural and Dynamic Analysis
> Cummins, Inc.



Improved surface extension: Users can select and extend multiple groups of surfaces in a single step, a procedure that greatly simplifies the process of closing gaps between parts after mid-surface extraction. The images show a sample model before and after surface extension.



Geometry Handling in ANSYS DesignModeler

Geometry modeling in the ANSYS Workbench environment is greatly improved to provide increased automation, greater flexibility and improved ease of use for the task of preparing geometry for analysis. The feature-based, parametric ANSYS DesignModeler tool, which can be used to create parametric geometry from scratch or to prepare an existing CAD geometry for analysis, now includes automated options for simplification, cleanup, repair and defeaturing.

`Merge`, `Connect` and `Project` features have been added for improved surface modeling in ANSYS 12.0. Face and Edge merge operations can be used to easily simplify models by eliminating unnecessary features and boundaries, leading to improved mesh and solution quality. The `Connect` operation can be applied to ensure proper connectivity in models with gaps and overlaps.

Automated cleanup and repair capabilities have been improved in the 12.0 release. New tools automatically detect and fix typical problems, such as small edges, sliver faces, holes, seams and faces with sharp angles. Geometry models can now be prepared for analysis at a much faster pace. As always, analysis settings remain persistent after performing these operations and are updated automatically in response to changes in geometry.

Shell modeling has been enhanced in several ways, including improved surface extensions. The ability to select and extend groups of surfaces greatly simplifies the process of closing gaps between parts after mid-surface extraction. The result is easier modeling of welds, for example.

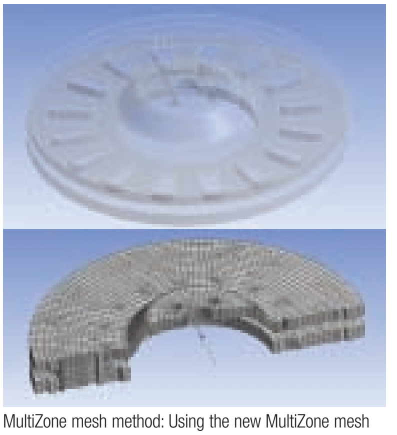
Analysis-specific tools within the ANSYS DesignModeler product now include an automated option to extract flow volumes for fluid dynamics analyses. In addition, several new features, including user-defined offsets, user-defined cross sections and better orientation controls, are available for improved beam modeling for structural analyses.

Improved attribute support is available with ANSYS DesignModeler 12.0. This includes options to create attributes within ANSYS DesignModeler as well as to import additional attributes from external CAD, including named selections, coordinate systems and work points.

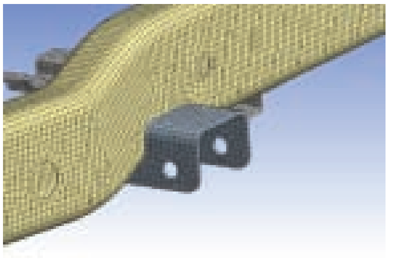
ANSYS Meshing Platform

A primary focus for ANSYS 12.0 has been to provide an automated meshing solution that is best in class for fluid dynamics. With the addition of capabilities from GAMBIT and TGrid meshing applications, major improvements have been made in the automatic generation of CFD-appropriate tetrahedral meshes with minimal user input. Advanced size functions (similar to those found in GAMBIT), prism/tet meshing (from TGrid) and other ANSYS meshing technologies combine to provide improved smoothness, quality, speed, curvature and proximity feature capturing, and boundary layer capturing.

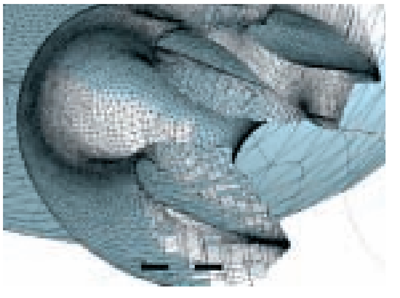
In the area of hex meshing, the traditional sweep and thin sweep methods have seen evolutionary improvements. A new method called MultiZone has been integrated into the ANSYS meshing platform. By combining existing ANSYS ICEM CFD Hexa technology with improvements in automation, MultiZone allows the user to automatically create hex meshes for many complex geometries without requiring geometry decomposition.



MultiZone mesh method: Using the new MultiZone mesh method, a user can mesh complicated models with a pure hex mesh without the need for geometry decomposition. This brake rotor example can be meshed with a pure hex mesh in a single operation.



Thin solid sweep method: Using the thin solid sweep mesh method, complicated sheet metal parts can be easily hex meshed without the need for midsurfacing or welding. The mesh can be generated to conform to the shared interface to increase the accuracy and speed of the solution.

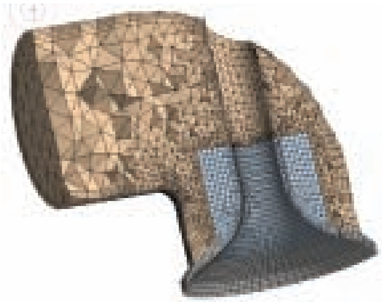


Patch conformal tet method with advanced size functions: With minimal input, ANSYS size function–based triangulation and inflation technology can handle advanced CFD meshing challenges, such as this benchmark aircraft model.

In the area of hybrid meshing, the MultiZone method allows for complicated regions to be meshed with a hybrid mesh (tet, hex-core, hex-dominant), further improving the flexibility and automation of this meshing approach. For more control in key areas of concern, the Sweep and Patch Conforming methods can be employed with conformal inflation layers throughout.

Though many of these enhancements were driven by fluid dynamics needs, they also benefit users of other types of simulation. For example, users performing structural analyses will benefit from the improved automation and mesh quality. Additional meshing enhancements for structural analyses include:

- Physics-based meshing improvements
- Rigid body meshing for contact
- Automated meshing of gaskets
- Improved handling of beams
- Thin solid meshing improvements
- Support for multiple elements through the thickness



Hybrid mesh: Using a combination of sweep and tetrahedral mesh methods, a user can quickly control the mesh in regions of interest to improve the accuracy of the solution without the need for a pure hex mesh (and the time required to generate it).

- Generation of conformal meshes in multi-body parts
- Enhanced and new mesh controls
- Pinch features to help in defeaturing models
- Improved smoothing
- Improved flexibility in size controls and mesh refinement
- Arbitrary mesh matching to improve node linking and solver accuracy

These improvements, though driven by structural analysis needs, provide benefits to the entire spectrum of ANSYS users.

ANSYS ICEM CFD

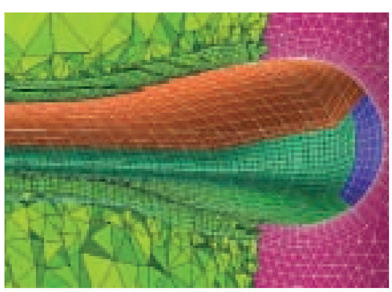
For ANSYS 12.0, ANSYS ICEM CFD meshing development focused on two primary tasks: improved implementation of ANSYS ICEM CFD meshing technology within the ANSYS meshing platform and continued development to enhance the ANSYS ICEM CFD product for interactive meshing customers. Because the ANSYS ICEM CFD integration involves the sharing of core libraries, improvements made for the ANSYS meshing platform also enhance the ANSYS ICEM CFD meshing product (and vice versa).

MultiZone meshing is an example of a crossover technology that has received special attention in both ANSYS meshing and the stand-alone ANSYS ICEM CFD meshing product. This hybrid meshing method combines the strengths of various meshers, such as ANSYS ICEM CFD Hexa and TGrid, in a semi-automatic blocking framework. Within the ANSYS Workbench environment, multizone automation provides multi-source, multi-target and multi-direction sweep capabilities reminiscent of the GAMBIT Cooper tool. In the stand-alone ANSYS ICEM CFD product, this is an excellent way to mesh for external aerodynamics in a semi-automated way that provides rapid hybrid meshing with a high degree of control and quality.

Improvements for ANSYS ICEM CFD 12.0 include process and interface streamlining, new hexa features, BFCart mesher enhancements, mesh editing advancements, output format updates and more. ■

Ben Klinkhammer, Shyam Kishor, Erling Eklund, Simon Pereira and Scott Gilmore of ANSYS, Inc. contributed to this article.

Named selection manager: This new feature allows a user to create and save named selections within CAD systems and then to use them within ANSYS applications. This example uses the named selection manager within Pro/ENGINEER.



ANSYS ICEM CFD: MultiZone meshing that combines the strength of various meshing tools, automatically generated this hybrid grid for a tidal turbine.

New developments in the ANSYS TurboGrid software are used to create high-quality meshes for bladed components with minimal user input.
Geometry courtesy PCA Engineers.

Enhancements to Turbomachinery Tools

With release 12.0, a number of enhancements have been incorporated into ANSYS BladeModeler, the design tool tailored to bladed geometries for rotating machinery. Within the BladeGen component, the integrated tools for determining initial blade shape and size (which were developed in conjunction with partner PCA Engineers Limited) have been expanded to cover centrifugal compressors and axial fans in addition to radial turbines and centrifugal pumps. The other component of ANSYS BladeModeler, BladeEditor, includes new blade geometry modeling capabilities to create and modify one or more bladed components. As an add-in to ANSYS DesignModeler, ANSYS BladeModeler provides access to ANSYS DesignModeler's extensive functionality to create non-standard geometry components and features.

ANSYS TurboGrid software includes a number of evolutionary improvements in release 12.0, and introduces a completely new meshing technology. This tool fully automates a series of topology and smoothing steps to largely eliminate the need to manually adjust mesh controls, yet still generates high-quality fluid dynamics meshes for bladed turbomachinery components.