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 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. 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

**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.

**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
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.