

Using New Meshing Features in ANSYS Workbench Simulation

Knowing when and how to apply key features of the latest structural meshing tools can result in greater efficiency.

By Sheldon Imaoka, Technical Support Engineer, ANSYS, Inc.

As solvers become faster and computers more powerful, the solution portion of finite element analysis shortens and a larger portion of overall simulation time is spent on pre-processing, including generating the mesh — the fundamental element-based representation of parts to be analyzed. Recognizing this trend, ANSYS, Inc. has addressed the need for faster and more reliable structural meshing with new technologies in ANSYS Workbench *Simulation* 11.0 (also known as ANSYS Workbench Meshing 11.0). These new capabilities result in very robust meshing and save considerable amounts of time (especially for complex geometries) with features that automate many routine tasks while providing users high levels of control of their model.

During the past several years, meshing in ANSYS Workbench *Simulation* has not only grown to encompass typical meshing algorithms available in traditional structural software from ANSYS but has also included many features requested by its large base of users worldwide. This wealth of new meshing capabilities includes:

- Physics-based meshing and element shape checking
- Higher degree of mesh sizing controls
- Patch independent surface and volume meshing
- Flexible sweep and hexahedral meshing, including automated generation of SOLSH190 solid-shell elements

Physics-Based Meshing and Element Shape Checking

Traditionally, software from ANSYS requires users to select the appropriate element type first; meshing algorithms and conservative shape-checking criteria are typically independent of the physics of the problem. On the other hand, ANSYS Workbench *Simulation* provides users with the ability to set default global meshing options under the **DETAILS** view of the **MESH** branch that is dependent on the analysis physics.

The ANSYS Workbench platform can generate meshes for structural, thermal, electromagnetics, explicit dynamics or computational fluid dynamics (CFD) analyses, but the meshing considerations vary for each. For example, lower-order elements with a finer mesh density tend to be used in CFD analyses, whereas higher-order elements with a coarser mesh density may be preferred in structural analyses.

For each physics, different criteria are used for element shape checking to ensure that the elements provide accurate results for that particular analysis. For mechanical analysis users, **STANDARD** and **AGGRESSIVE** shape checking are also available. **STANDARD** shape checking is suitable for linear analyses, while **AGGRESSIVE** checking provides more



Figure 1. In an analysis of 2-D contact between gears, a **SPHERE OF INFLUENCE** (in red) is defined with a user-specified radius (left). This generates a fine mesh within this area for studying von Mises stresses at the point of tooth contact as well as within the geometry of the gear body (right).

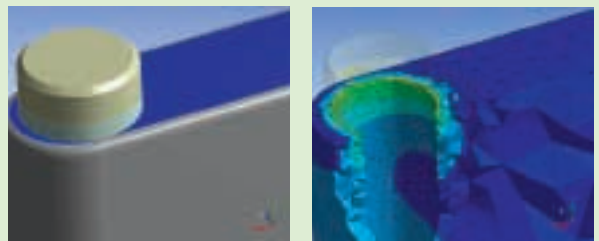


Figure 2. In modeling a fastener in a part (left), **CONTACT SIZING** is used to automatically create a fine mesh density in the area of initial contact. Section planes show the resulting detailed stress distribution inside the part (right), providing an optimal mesh in capturing contact behavior.

conservative element shape-checking criteria to account for possible distortion of the elements during nonlinear analyses.

Higher Degree of Mesh Sizing Controls

Mesh sizing controls are available in ANSYS Workbench *Simulation* under the MESH branch, allowing users to specify element size on vertices, edges, faces or bodies (parts), with number of divisions and mesh biasing available on edges. Two features introduced in ANSYS Workbench *Simulation* 10.0 are SPHERE OF INFLUENCE and CONTACT SIZING.

Under a surface or body mesh sizing branch, instead of specifying a uniform mesh density for the entire geometric entity, a user can use a defined COORDINATE SYSTEM and a radius to designate a sphere where elements will have a certain size. This is helpful in specifying a smaller mesh density without requiring existing geometry to identify that region. The sphere of influence mesh control is also useful in propagating a mesh density inside the geometry, rather than concentrating a finer mesh only on the surface of the model. In analyzing the stress distribution of 2-D contact between gears, for example, a user may create a sphere of influence to generate a fine mesh on the edges of the gear teeth as well as within the geometry of the gear body (Figure 1).

Many users already know that contact regions are associated with the geometry in ANSYS Workbench *Simulation*, so remeshing an assembly does not require re-creating contact pairs. Another useful related feature is CONTACT SIZING, which allows users to define a more uniform, finer mesh density in a contact region to provide a better distribution of contact pressure. Instead of having users manually select surfaces on which to define element sizes, however, the CONTACT SIZING control allows users to specify mesh densities that are applied only in areas of initial contact for the defined contact regions. Thus, a user only has to drag-and-drop a contact region from the CONNECTIONS branch to the MESH branch and specify an element size — so only the actual areas that are in initial contact will have that finer mesh (Figure 2).

Meshing and Defeaturing

The default volume mesher in ANSYS Workbench *Simulation* automatically includes some defeaturing, unlike the mesher in traditional software from ANSYS, which meshes all surfaces, including any sliver areas present in the model. The user can control the percentage of defeaturing by specifying the DSMESH DEFEATUREPERCENT variable in the VARIABLE MANAGER located under the TOOLS menu. Consequently, instead of meshing small slivers (which would generate more nodes and elements), ANSYS Workbench *Simulation* internally ignores these surfaces, providing a more robust, efficient mesh that requires little cleanup of CAD geometry (Figure 3).

While automatic defeaturing is helpful, this built-in defeaturing is not meant to compensate for larger

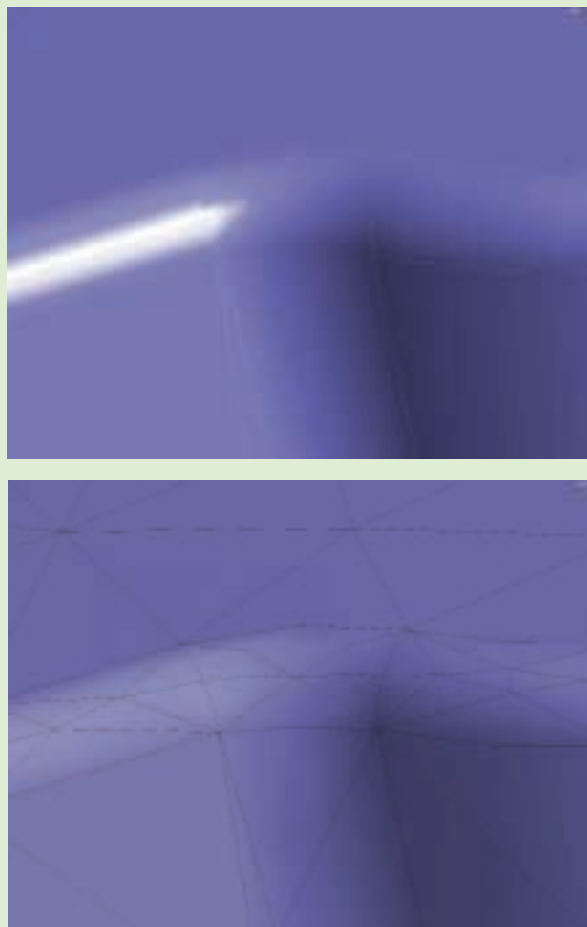


Figure 3. Instead of modeling the small sliver in this model (top), the automatic defeaturing capability in ANSYS Workbench *Simulation* intentionally ignores this surface (bottom) in generating a more robust, efficient mesh.

surfaces the user wants to ignore. Instead, either VIRTUAL TOPOLOGY or the PATCH INDEPENDENT MESHER can be used for this purpose.

A VIRTUAL TOPOLOGY branch can be inserted from the MODEL branch. Once added, virtual cells can be defined that effectively merge surfaces and edges for meshing and pre-processing purposes. Although the functionality is similar to concatenation for mapped meshing in ANSYS (ACCAT and LCCAT commands), virtual cells are used for tetrahedral meshing in ANSYS Workbench *Simulation*, thus giving the user greater flexibility.

Using the METHOD mesh control, a user can also utilize the patch independent meshing algorithm (also named UNIFORM QUAD/TRI or UNIFORM TRI for surface meshing). The patch independent meshing algorithm takes a different approach, as it does not start off with a surface mesh but uses an Octree algorithm instead, so the mesher is not constrained by all of the surfaces present in the model. This algorithm is useful when a user may want to perform gross defeaturing of a very complex part or if a user wants to

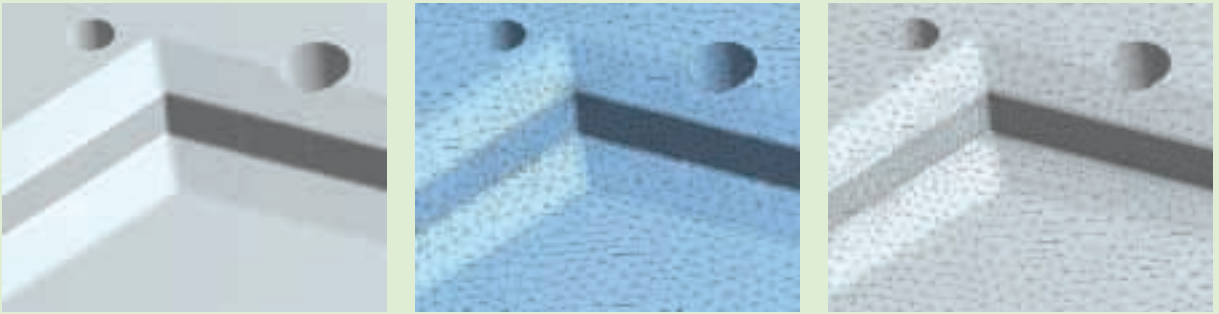


Figure 4. In modeling a part with various surfaces and fillets (left), the default mesher honors all surface boundaries, resulting in mesh size variations (middle) in the highlighted area. The patch independent mesher tends to produce a more uniform-sized mesh (right) in this area by not honoring all boundaries of unscoped surfaces.

generate a uniform mesh. Although the mesher can skip over small features, any scoped surfaces (that is, loads applied on certain faces) will have their boundaries respected.

Although **VIRTUAL TOPOLOGY** can be used in conjunction with the **PATCH INDEPENDENT MESHER**, the author recommends using either of the following procedures for meshing (Figure 4):

- Use the default patch-conforming meshing algorithm for surfaces/parts, and use virtual cells to group small surfaces to adjacent ones, if needed. This method provides a mesh that conforms to the geometry, although the user can merge together unimportant, small surfaces to reduce the node/element count. This technique is helpful if a lot of manual defeaturing (via **VIRTUAL TOPOLOGY**) is not required. The user can also set the **DEFEATUREPERCENT** variable for global defeaturing of very tiny geometric features.
- For bulky, complex geometry whose surfaces the user may not need to mesh in detail, add a **METHOD** branch to specify the patch independent meshing algorithm and any defeaturing or curvature/proximity refinement that is desired. Scope (via **NAMED SELECTIONS**, **CONTACT REGIONS**, **LOADS & SUPPORTS** and **RESULTS**) all geometric entities whose features should be kept. This technique is useful if gross defeaturing is required, since manual specification of regions is not required.

Thin Solid Volume Sweeping

The SOLSH190 solid-shell element is a specially formulated eight-node hexahedral element that has special shape functions to prevent locking, even when the thickness is very small. The SOLSH190 element provides a straightforward way to account for variable shell thickness and allows for a natural transition to regular solid elements.

Because of the special shape functions in the thickness direction, the user needs to pay special attention to the

SOLSH190 node numbering, the generation of which can sometimes be a cumbersome task in some software from ANSYS. ANSYS Workbench *Simulation*, however, provides a convenient meshing capability to automatically generate SOLSH190 elements with the proper orientation. For sweep-meshable parts, the **METHOD** mesh control provides automatic or manual sweep meshing. There is an additional option for **THIN MODELS**, which generates SOLSH190 elements. Besides automatic detection of source and target surfaces, users can specify multiple source areas and not be limited to a single source area (Figure 5). ■

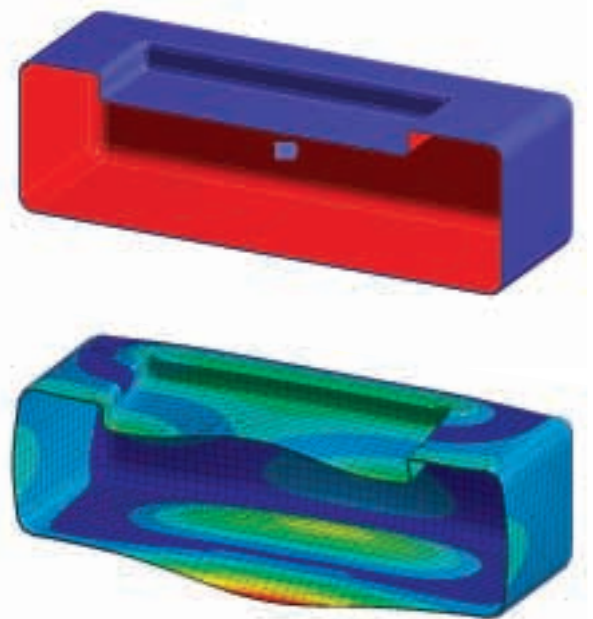


Figure 5. In this example of thin solid volume sweeping capability, interior red faces are designated as source surfaces, and ANSYS Workbench *Simulation* automatically detects the purple companion target surfaces (top). The resulting swept mesh then is used in calculating an accurate mode shape (bottom) for the thin solid part.