

Date	March 2, 2008	Memo Number	STI:08/01B
Author	Sheldon Imaoka	ANSYS Revision	11.0
Subject	Sheldon's ANSYS.NET Tips and Tricks: Meshing in Workbench		
Keywords	Meshing		

## 1. Introduction:

Mechanical users have long enjoyed access to a variety of powerful meshing tools inside of ANSYS to generate high-quality shell, tetrahedral, and swept meshes. During the past several years, meshing in ANSYS Workbench Simulation has not only grown to encompass traditional meshing algorithms in ANSYS but has also developed many features requested by ANSYS users as well as integrated meshing technologies from ANSYS CFX and ANSYS ICEM CFD.

ANSYS Workbench Simulation 11.0 (a.k.a. ANSYS Workbench Meshing 11.0) offers users a wealth of meshing capabilities<sup>1</sup>, including the following:

- Physics-based meshing & element shape checking
- Higher degree of mesh sizing controls
- Patch-independent surface and volume meshing
- Additional controls for sweep meshing of solid-shell elements

This memo hopes to cover some meshing topics pertinent for mechanical users, although the user should keep in mind that Workbench Simulation offers various meshing tools for CFD, Electromagnetic, and Explicit Dynamics users as well.

## 2. Physics-based Meshing Preferences & Element Shape Checking:

ANSYS users are familiar with the fact that meshing in ANSYS requires that the user select the appropriate element type first, and the 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. ANSYS Workbench can generate meshes for structural, thermal, electromagnetics, explicit dynamics, or 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 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 in order to ensure that the elements provide accurate results for that particular analysis.<sup>2</sup> For mechanical users, “Standard” and “Aggressive” shape checking are also available: “Standard” shape checking is suitable for linear analyses, but “Aggressive” shape checking provides more conservative element shape-checking criteria to account for possible distortion of the elements during nonlinear analyses.

---

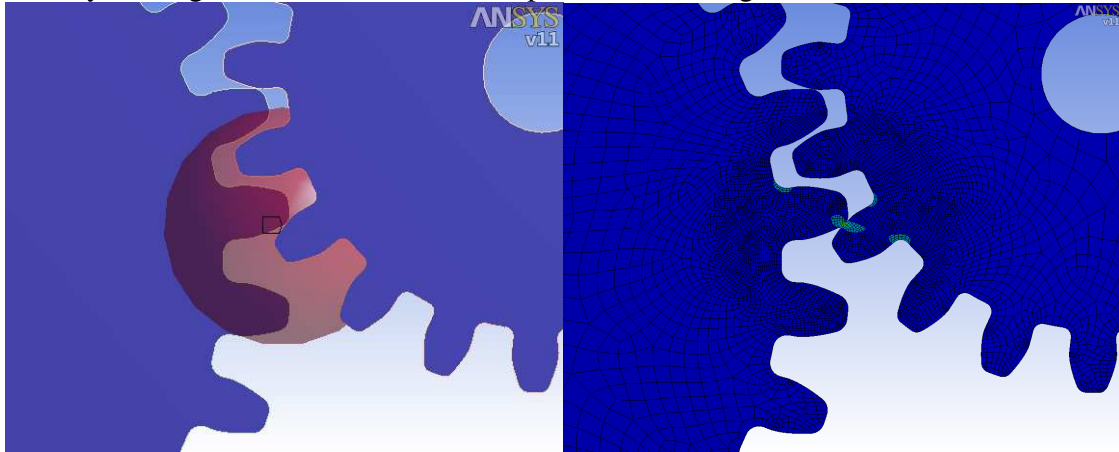
<sup>1</sup> For a more comprehensive discussion on meshing options in ANSYS Workbench, refer to “ANSYS Workbench Help > Meshing Help” documentation

<sup>2</sup> See ANSYS Workbench Help: “Meshing Help | Meshing Capabilities in Workbench | Mesh Controls | Global Mesh Controls”

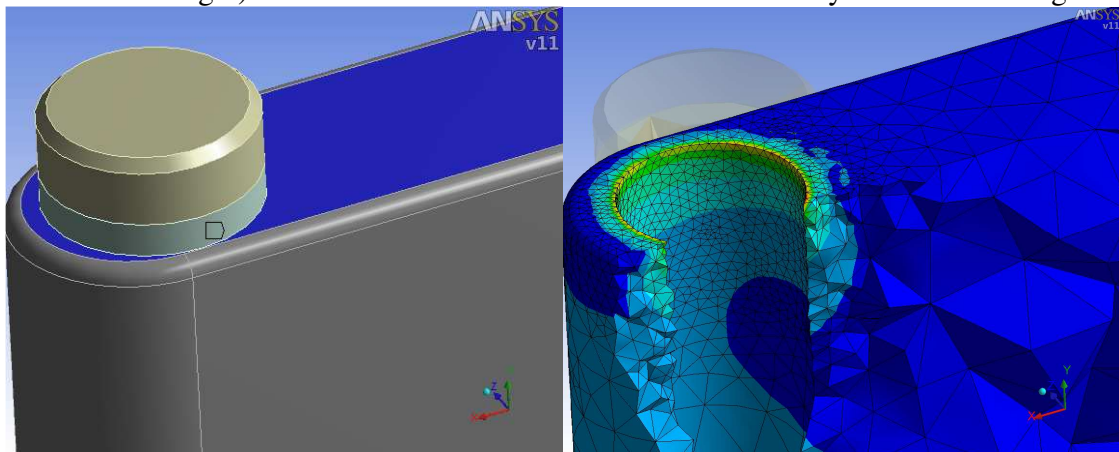
### 3. Higher Degree of Mesh Sizing Controls:

Users will find that typical mesh sizing controls are available in ANSYS Workbench Simulation under the “Mesh” branch – specification of element size on vertices, edges, faces, or bodies (parts)<sup>3</sup> with number of divisions and mesh biasing available on edges.

Two features that may be new to ANSYS users 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, as shown in the example of contact of gear teeth below:



Contact regions in Workbench Simulation are associated with the solid model geometry, so remeshing does not require a user to regenerate contact elements. Also, “Contact Sizing” allows users to define a more uniform, finer mesh density in a contact region to provide a better distribution of contact pressure. A user can drag-and-drop a contact region from the “Connections” branch to the “Mesh” branch and specify an element size – only the actual areas which are in initial contact will have that finer mesh. The figures below illustrate this behavior – note that the entire upper surface (highlighted in blue) is associated with the contact region, but the resulting mesh (a Section Plane shown on the right) reveals that the finer mesh is concentrated only in the contact region.

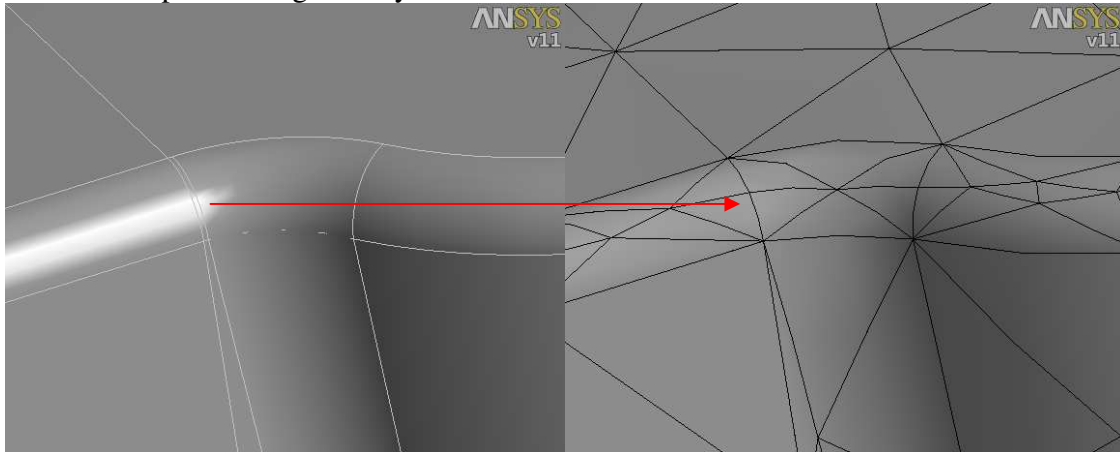


---

<sup>3</sup> A convenient feature is that the mesh size will be shown as a circle next to the cursor in the Graphics Window.

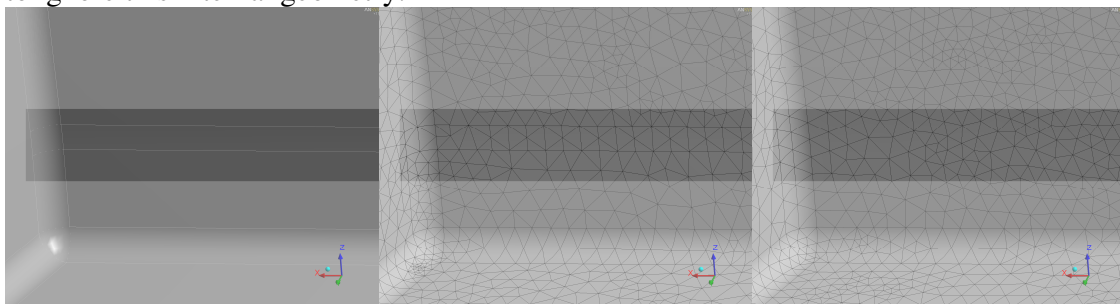
#### 4. Meshing & Defeaturing:

Unlike the mesher in ANSYS that meshes all surfaces, including any sliver areas present in the model, the default volume mesher in Workbench Simulation automatically includes defeaturing. The user can control the percentage of defeaturing by specifying the `DSMESH DEFEATUREPERCENT` variable.<sup>4</sup> Note in image on the left, a very small area exists on the right side. Instead of meshing this small sliver (which would generate more nodes and elements), this surface was internally ignored, resulting in the mesh shown on right. This provides a much more robust, efficient meshing algorithm for users requiring little clean-up of CAD geometry.



While automatic defeaturing is helpful, this built-in defeaturing is not meant to compensate for larger surfaces that the user wants to ignore. Instead, either “Virtual Topology” or the “Patch-Independent Mesher” can be used for this purpose.

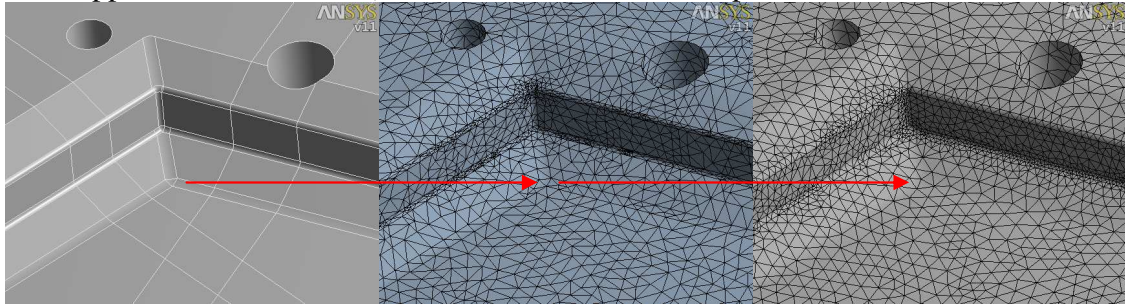
A “Virtual Topology” branch can be inserted from the “Model” branch. Once added, “virtual cells” can be defined, which effectively merges surfaces and edges for meshing and preprocessing purposes<sup>5</sup>. Although the functionality is similar to ‘concatenation’ for mapped meshing in ANSYS (`ACCAT` and `LCCAT` commands), virtual cells are used for tetrahedral meshing in ANSYS Workbench Simulation, thus giving the user greater flexibility. Note in the figure below that there is a highlighted strip of surface – with virtual topology, the strip can be merged with surrounding surfaces, allowing the mesher to ignore this internal geometry.



<sup>4</sup> For details, see the following ANSYS Workbench Help section: “Simulation Help | Simulation Basics | Customizing Simulation | Variables”

<sup>5</sup> Note that edges internal to surfaces merged as a virtual cell are removed, so loads and boundary conditions must be applied to the virtual cell itself, not to its constituent surfaces.

Using the “Method” control, a user can also utilize the patch-independent meshing algorithm (a.k.a. “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 generate a uniform mesh. Although the mesher can ‘skip’ over small features, any scoped surfaces (i.e., loads applied on certain faces) will have its boundaries respected.



In the above example, note that the default mesher (shown in the middle) honors all of the edge boundaries of each surface. Consequently, more variation in the element sizes result if small surfaces are present. On the other hand, the patch-independent mesher (shown on the right) skips over internal edges for unscoped surfaces, resulting in a more uniform mesh, even with proximity/curvature refinement active (as in this sample model).

Although Virtual Topology can be used in conjunction with the Patch-Independent Mesher, the author recommends using *either* of the following procedures:

- Use the default patch-conforming meshing algorithm for surfaces/parts, and use virtual cells to group small surfaces to adjacent ones, if needed.<sup>6</sup> 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 ‘defeating’ (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.

---

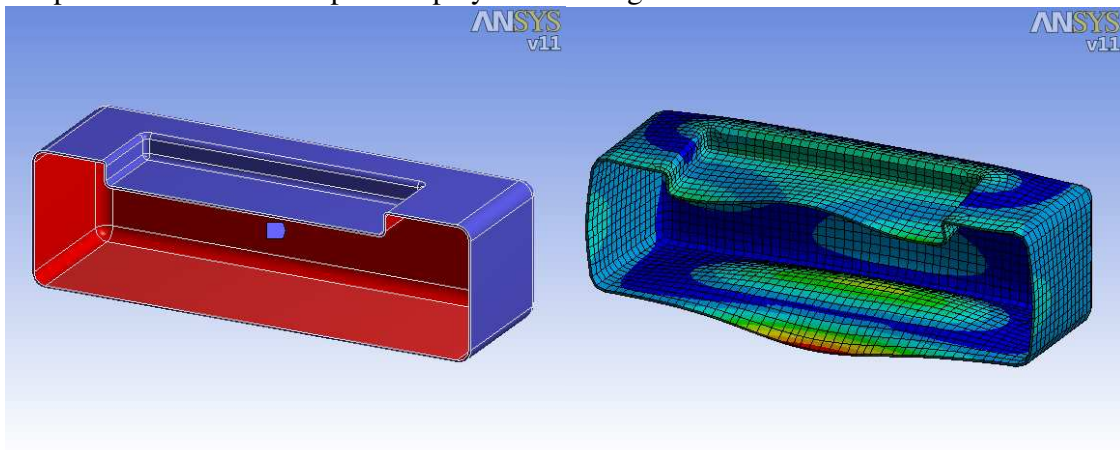
<sup>6</sup> Automatic creation of virtual cells is also available in ANSYS Workbench Simulation, although the author prefers to manually create virtual cells where needed.

## 7. Thin Solid Meshing:

The SOLSH190 solid-shell element is a specially-formulated 8-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 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. In the figure on the left, the interior faces are designated as source surfaces (red), and Workbench Simulation automatically detects the companion target surfaces (purple). The resulting swept mesh and mode shape is displayed on the right.



## 8. Additional Comments and Conclusion:

Although beyond the scope of the present discussion, it is useful to note that ANSYS Workbench offers the ability to create “Inflation layers” (prism elements at surfaces to capture boundary layer effects in CFD and skin effects in Emag) and “Air Gap” meshes (stretched elements in a thin air gap region for Emag), which are helpful for CFD and electromagnetic analyses.

A hex-dominant meshing algorithm is available, which generates brick elements on the exterior surface (with transitional pyramid and tetrahedral elements in the interior). Because of the generation of hex elements on the surface which are then extruded inwards, this method is most useful for ‘bulky’ parts.

As discussed above, ANSYS Workbench provides users with many advanced meshing tools, ranging from physics-based meshing, local mesh control, defeaturing capabilities, and advanced meshing algorithms.