

Analyzing Bolt Pretension in the ANSYS Workbench Platform

Convenient features enable pretension to be quickly and easily included in analysis of bolted joints.

By Doug Oatis, Senior Mechanical Engineer, Mechanical Simulation, Phoenix Analysis & Design Technologies, Inc., Arizona, U.S.A.

Analyzing a bolted flange used to be a serious undertaking, in part because of difficulties in including pretension loads produced by the installation torque of tightening the bolt. Analysts resorted to a variety of methods to account for pretension, including running a “dummy” thermal analysis to induce thermal expansion loads or creating beams and constraint equations on the flange to add equivalent compressive flange loads.

Pretension elements available in the ANSYS Workbench platform allow the analyst to more readily specify known axial loads or adjustments to groups of elements in accounting for these bolt installation loads. Indeed, bolt pretension is a great example of the user-friendly nature of simulation within the ANSYS Workbench environment.

One point of confusion, however, is in trying to scope a bolt pretension between the two “clamshell” faces of split cylindrical features imported from an external CAD package — for example Pro/ENGINEER® or SolidWorks®. A clamshell face is simply the result of a single cylindrical face being split in half. If both clamshell faces are selected

for a single ‘Bolt Load’ object, a ‘?’ will appear next to the ‘Bolt Load.’

To get around this behavior, a little understanding of how simulation within the ANSYS Workbench environment applies bolt pretension is needed. First, the software divides a meshed body using the PSMESH command. Next, a pretension element effectively writes a constraint equation that relates the displacement of one cut boundary to the other.

Within the simulation environment, the face selected is used as a cutting guide for the PSMESH command. This means that you only need to select one clamshell face to define bolt load on an imported cylinder. The simulation tool will then use the middle of the selected face for the mesh division and pretension element creation. Bolt preload direction is determined automatically, as shown in Figure 1.

If you need more control over the preload application location, the bolt load can be scoped to a single body, as in Figure 2. You are then required to specify a coordinate system that defines the mesh slice plane and bolt load direction. This can be done through the default global or user-defined

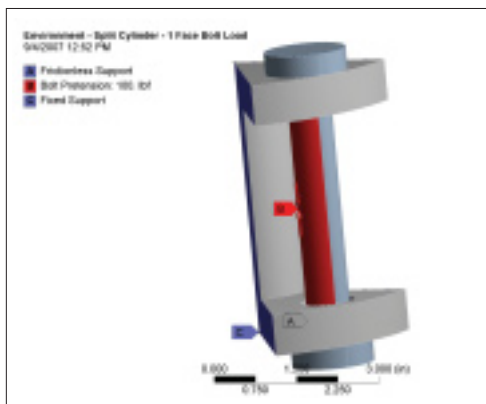


Figure 1. The middle of the bolt face is automatically selected for dividing the mesh and creating pretension elements.

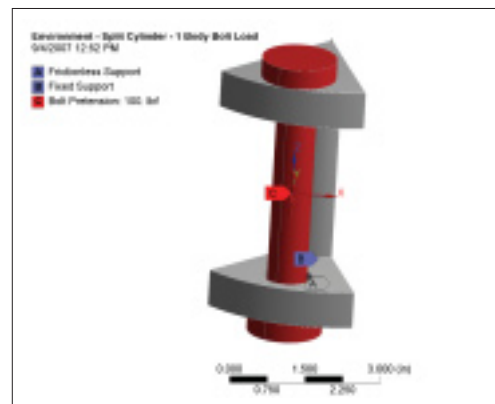


Figure 2. For greater control over preloading conditions, the bolt load can be scoped to a single body.

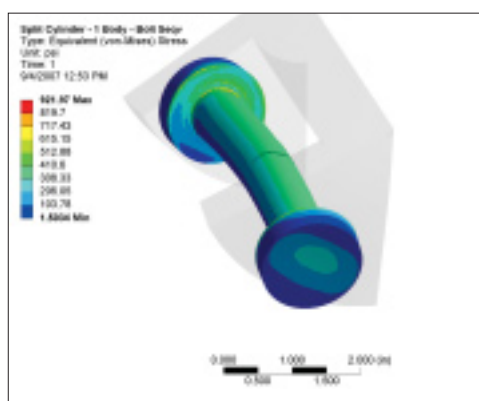
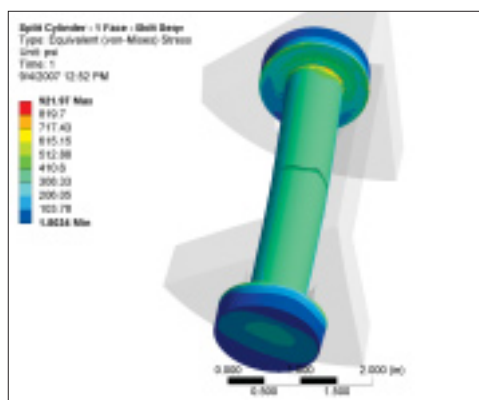


Figure 3. Stress contours of the one-clamshell-face (top) and the body selection methods (bottom) are identical.

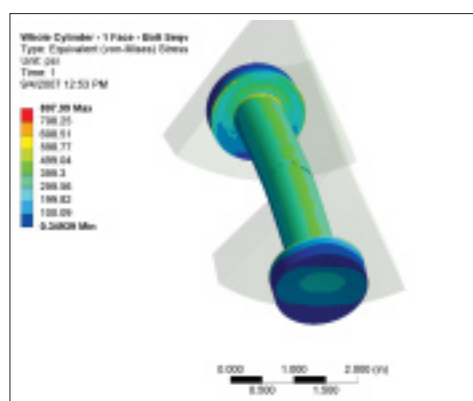
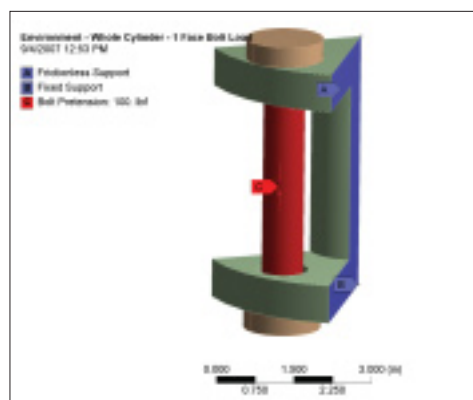


Figure 4. Geometry repaired and treated as a whole cylinder in the ANSYS DesignModeler tool (top) yields similar stress contours to the split cylinder case (bottom).

coordinate system. The XY-plane of the coordinate system defines the PSMESH slice plane, while the Z-axis defines the direction of bolt preload. You don't need to define the coordinate system "inside" the preloaded part.

When you compare the results of the one-clamshell-face with the body selection methods, they are the same, as shown in Figure 3. This is to be expected because the mesh division occurred at the same location for each environment.

Scoping the bolt load to a body and coordinate system not only lets you validate the one-clamshell-face scenario, but also allows the analysis of bolted flanges where the bolt is long relative to the flange thickness (e.g., the bolt midpoint is on the other side of the nut.)

If you create a cylinder using ANSYS DesignModeler software (the geometry modeling tool within ANSYS Workbench), it is defined as a single cylindrical surface. ANSYS DesignModeler capabilities can also be used to clean/repair imported geometry. If you use the 'Face Delete' tool (Create > Face Delete) on one of the two clamshell faces, the cylinder will be "repaired" into a single surface. Although not a required step, this allows you to clean up and simplify the geometry, as in Figure 4. This simplification could be done within a simulation by specifying a virtual cell from the two clamshells, though virtual cells do not support bolt loads.

The minor differences shown between the whole cylinder and split cylinder models occur at the contact interface, which was simplified as bonded. The reaction pretension adjustments were within .05 percent, as shown in Table 1.

When modeling bolted interfaces, the ease of using non-threaded solids to represent the bolt is increasingly attractive. Through a combination of automatic contact detection, multiple meshing controls and an easy-to-use bolt-loading interface, simulation using the ANSYS Workbench platform has made including bolt pretension intuitive and speedy. ■

Table 1. Pretension Differences at the Contact Interface for Variations in Clamshell Models

	Split Model		Whole Cylinder
	1-Face	1-Body	1-Face
Bolt Adjustment [in]	0.00019686	0.00019686	0.00019677

This article is based on a column from the technical newsletter *The Focus* (www.padtinc.com/epubs/focus) from engineering consulting firm PADT (Phoenix Analysis & Design Technologies).