Modeling Bolted Connections

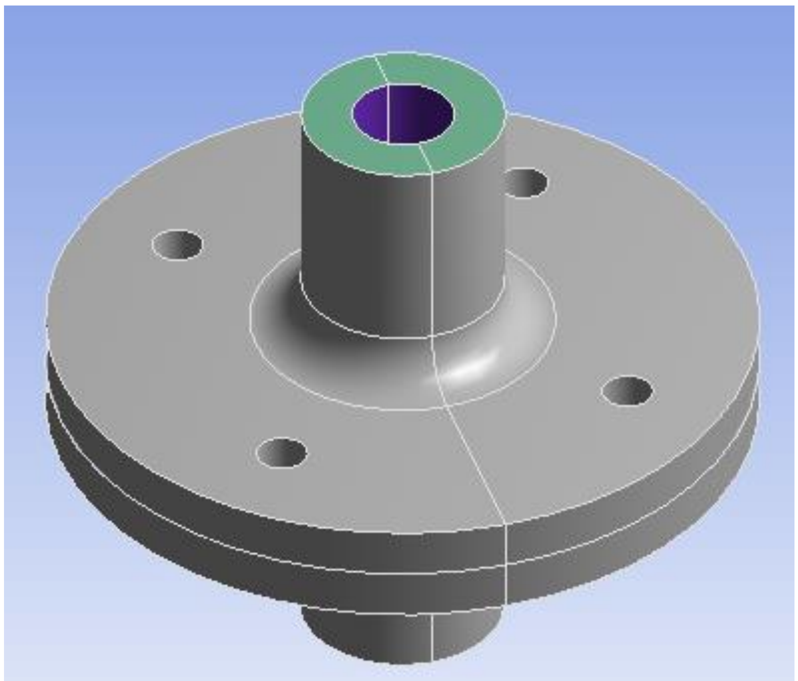
SimuTech Group

Tyler Coffey, FEA Consulting Engineer

Butch Vision, Engineering Manager

Engineers typically use four common methods for modeling bolted connections. In this white paper, SimuTech Group engineers explain these four approaches using ANSYS® Mechanical (Workbench), and they describe the benefits and drawbacks for each method.

Figure 1: Geometry of bolted connections without fasteners modeled.



Method 1: No bolts, bonded contact

The simplest kind of models for a bolted joint do not include the bolts themselves. Instead, bonded contact is defined between the surfaces of the joints. The simplest analysis would simply bond the entire joint surface; this would not allow individual bolt loads to be extracted.

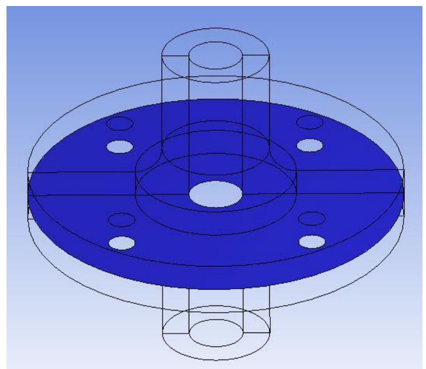


Figure 2: Simple analysis of bonded joint surface.

A more detailed analysis might only bond the area corresponding to the washer diameter or a 30˚ pressure cone. To use this approach, the face needs to be imprinted on the contacting flange surfaces. If separate contacts are used for each area, you can extract a force reaction for each bolt. You can also extract a moment reaction in Workbench using a command object.

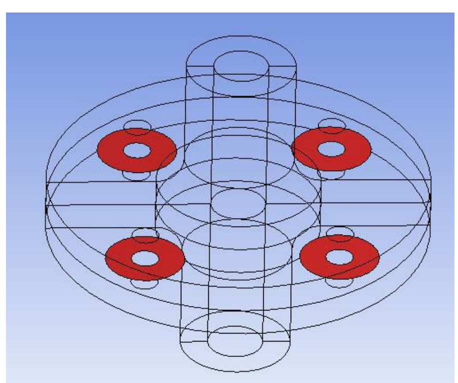


Figure 3: Bonded area corresponds to the washer diameter.

Benefits of using this method include

- minimal (if any) additional geometry prep
- a fast setup
- a linear approach leading to a fast solve
- the extraction bolt loads for individual bolts when bonding a washer diameter or 30 ˚ pressure cone.

The limitations to this method mean

- you can't capture joint separation
- extracting moment loads requires a command object
- it is also less realistic than other methods, and bonding the entire face in particular can be unrealistically stiff.

Method 2: Beam element bolts

Bolts are often modeled using beam elements in two ways:

1. Using a beam connection directly inside of Workbench, or
2. Creating a line body and defining its cross section in ANSYS® Design Modeler or ANSYS® SpaceClaim.

Both methods create beam elements that can be attached to an edge or face; this is included in the definition of a beam connection and is generally done with a fixed joint for line bodies.

Both of these approaches use remote points under the hood. It’s often more realistic to attach the beam elements to a face corresponding to the washer diameter, which distributes the forces from the beam over the washer area. The beam can also be attached to the ID of the bolt hole. The force and moment loads on the beam can be extracted and used in a hand calculation for the bolt margin (see NASA Technical Memo 106943).

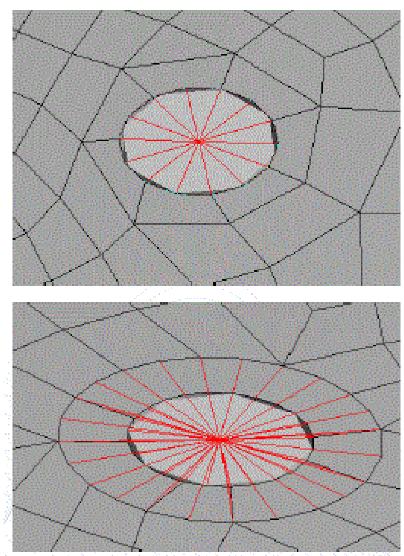


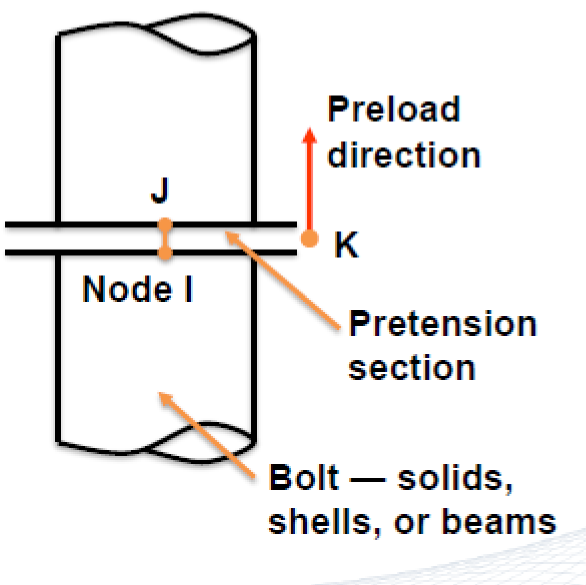
Figure 4: The top image shows the constraint equations attaching a beam to an edge.
The bottom image shows the attachment to the imprinted washer face.

Here are some differences between the using line bodies and beam connections.

Line bodies vs. beam connections

Line bodies	Beam connections
Can be meshed with multiple elements	Meshed with a single beam element
Cross section defined in Design Modeler or SpaceClaim	Cross section defined in Mechanical (must be circular)
Can apply preload without a command object	Applying a preload requires a command object

Preload setup



ANSYS generates preload by cutting the bolt and creating pretension elements between the two halves. This happens automatically when a bolt pretension load is created. A set of pretension elements is identified as a "section." The 2-D or 3-D line elements act like hooks and connect the two halves of a bolt.

Figure 5: Nodes I and J are the end nodes, usually coincident. Node K is the pretension node. The actual line of action is in the pretension load direction.

Node K in Figure 5, the pretension node, has one DOF: UX. Its location is arbitrary. The pretension node is used to define the preload as an FX force or UX displacement. Remember that preload direction is constant: it does not update for rotations.

Workbench splits the bolt into two unconnected groups, which it then ties together with pretension elements.

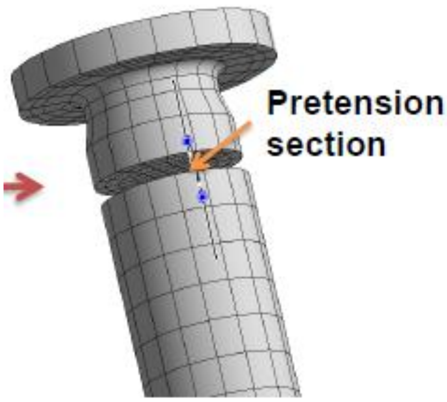


Figure 6: Having a regular (hex) mesh will provide a planar pretension section.

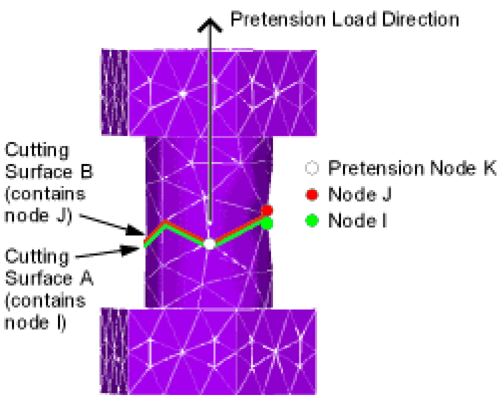


Figure 7: A tetrahedral mesh will create a non-planar pretension section.

.

When preload is applied, the two halves of the bolt will move relative to one another and overlap. Applying the preload in a region that is part of a bonded contact can cause it to function incorrectly because the bond prevents the two halves from moving relative to one another. To avoid this problem, you will want to split the bolt shank face. This can be done using the project or split face tools in ANSYS SpaceClaim or a projection, face split, or imprint faces in ANSYS Design Modeler.

When using solid elements, preload can be scoped to either a cylindrical face or a body. Scoping the load to a body requires a coordinate system to define the load direction and the place the body will be cut.

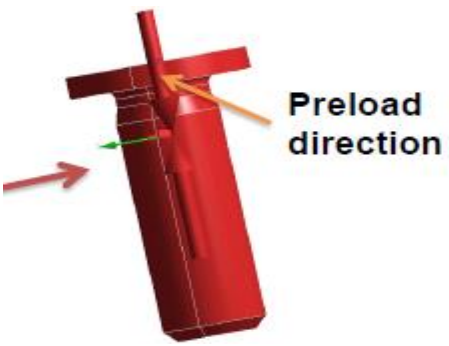


Figure 8: CS origin defines the position of the pretension node. Z-axis defines the pretension direction. XY plane determines approximate pretension section (cut) location, which should be away from any bonded contact.

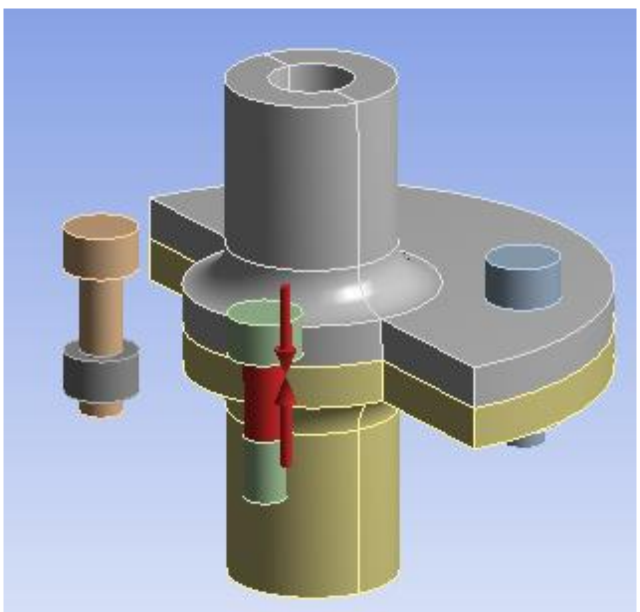


Figure 9: If the XY plane of the selected coordinate system doesn't pass through the bolt shank, an error will occur.

Scoping the load to a cylindrical face will automatically determine the load direction. When you do this, split the face of the bolt to separate any bonded region from the pretension region.

You can apply bolt pretension load using one of these options:

- Load: applies a force as a preload
- Adjustment: applies a length as a pre-adjustment (for example, to model x number of threads)
- Lock: fixes all displacements. Can be set for any step except the first.
- Open: use this option to leave the bolt pretension load open so that the load has no effect on the applied step, effectively suppressing the load for the step. In order to avoid convergence issues under constrained conditions, a small load (.01% of the maximum load across the steps) will be applied. This set can be set for any step.

In analyses with pretension and nonlinear contact, it generally works best to

1. Apply the pretension load in the first load step to establish the contact.
2. Then lock the pretension load and apply the other loads in the subsequent load steps.

Modeling a Joint Using Beam Elements

If you decide to use beam elements, you can use three ways to model the joint: use beam connections without preload or contact, use beam connections with preload and frictional contact between the flanges, or use line bodies with preload and frictional contact between the flanges.

Beam connection bolt, no contact: Commonly used in the aerospace industry, in this linear model, the beam connections are all that holds the two flanges together. The bolts will carry the entire load, which is usually conservative. Hand calculations are required to determine if the bolt would fail. (See NASA Technical memo 106943 for details.)

Beam connection bolt with preload and frictional contact: Beam connections in Workbench use a single beam element between bodies and will not allow pretension to be applied. This is because Mechanical APDL creates a pretension load by attaching pretension elements in between two or more beam or solid elements. In this case, you'll use a command object to apply pretension to the beam connections.

Each command object under the beam connections

- creates a new node at the midpoint of the beam element
- creates two new beam elements between the endpoints of the original and the new node
- deletes the original beam element
- creates a pretension element between the new beam elements

Workbench isn’t aware of the new beam elements generated as part of the command object, so any postprocessing of beam results must happen in another command object.

Line body bolts with preload and frictional contact: In this method, line bodies are used to create beam elements. Line bodies can be created in CAD, but the cross section must be designated in ANSYS Design Modeler or ANSYS SpaceClaim (both require a license.)

Bolt pretension is created with a bolt pretension load. Meanwhile, the line bodies are attached to the flange with fixed joints. The behavior of the flange side of the joint is deformable so that the washer face can deform.

Compare the benefits and drawbacks of these three ways.

Beam element bolts comparison

Beam connection – no contact	**Beam connection – preload + friction**	**Line body – preload + friction**
Pros	Pros	Pros
Linear; easy to use with linear dynamic analyses	Includes preload	Includes preload
Easy setup	Accounts for contact between joint surfaces	Accounts for contact between joint surfaces
Can extract bolt loads	Can extract bolt loads	Can extract bolt loads
Conservative	Doesn’t require any additional geometry prep	Straightforward setup
Fast solve	NLS (or higher) is only license required	
Cons	Cons	Cons
Does not account for preload or contact	More complex setup	More geometry prep
Conservative	Does not account for frictional contact with washer/bolt	Requires Design Modeler or SpaceClaim license
		Does not account for frictional contact with washer/bolt

Method 3: Solid Element Bolts

Simulations that require more detail can use solid elements to model the bolts, which allows the model to account for contact between the washer/bolt and the rest of the joint.

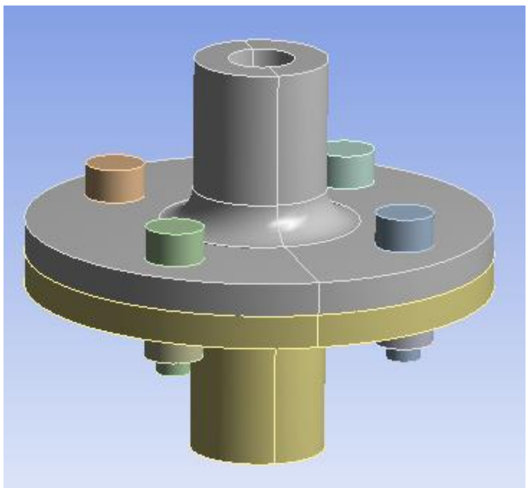


Figure 10: A simpler model might use bonded contact between the bolt head/nut and flanges, while a more complex model might use frictional contact.

These models don't usually model the threads because of the associated computational cost.

Postprocessing bolt results

You can obtain the force and moment loads on the bolts using a construction surface. To create a construction surface

- Right-click on a coordinate system and select Create Construction Surface
- Create a force or moment reaction probe
- Set the Location Method to Surface
- Set the geometry to the bolt body
- Before solving, save the nodal forces to the results file

Placing the construction surface on the cutting surface for the pretension can give you incorrect results. Offsetting the construction surface is usually the easiest way to avoid this problem. You can identify the cutting surface by plotting the directional deformation along the axis of the bolt. (See Figure 11.)

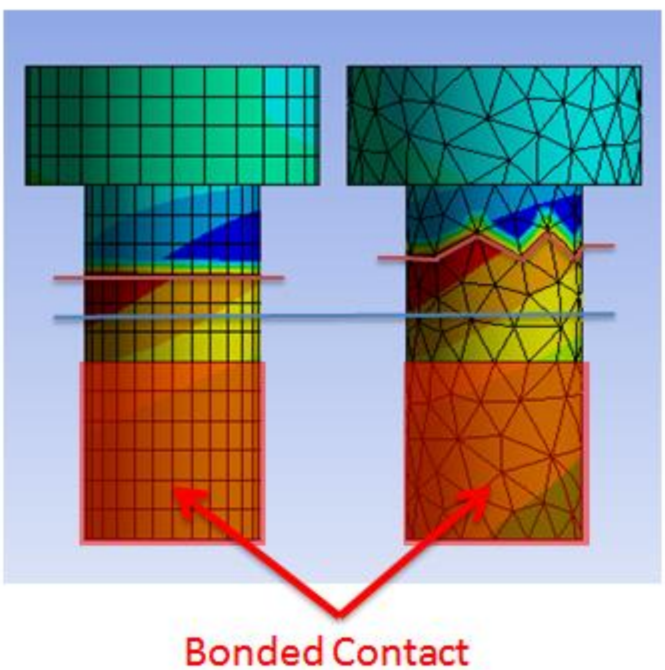


Figure 11: Red lines illustrate how the cutting surface can be identified by plotting the directional deformation along the axis of the bolt.

A good location for the construction surface is shown via the blue line. This should be at least one element away from the cutting surface.

Preload setup using joints

Pretension elements do not move or rotate with the geometry. Preload direction is constant, which can be problematic for large deflection analyses. Joints will rotate with the geometry and can be used instead of pretension elements to preload bolts.

To generate preload with a joint

- Cut the bolt in half in CAD
- Create a translational joint attaching one side of the cut to the other
- Make sure the x-axis of the reference coordinate system is aligned with the bolt axis
- Create a joint load. Like a bolt pretension load, this can be done in terms of displacement or force. Joint can be locked like a pretension load. Sign convention: mobile side of joint will move in specified direction.

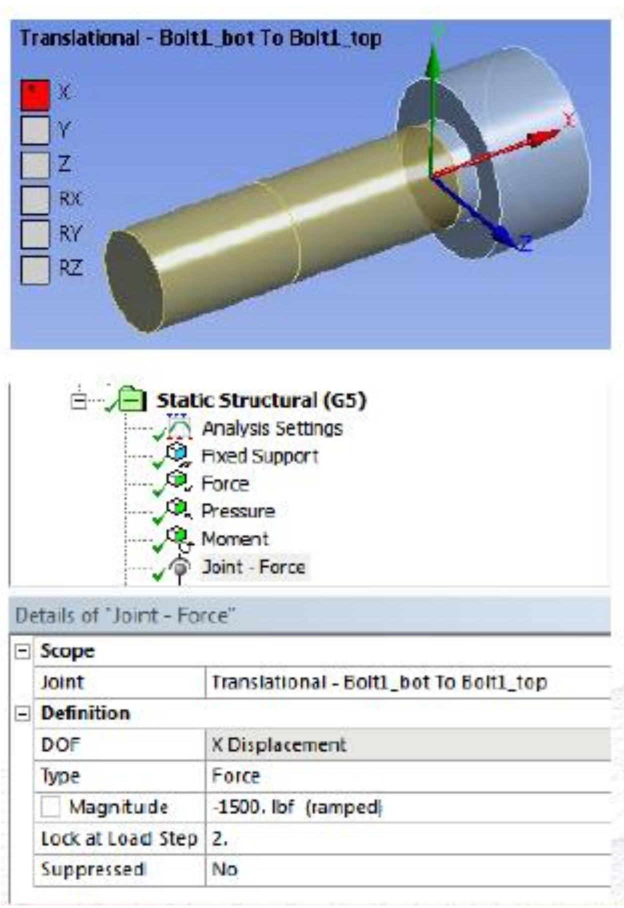


Figure 12: Example input for a joint load

Postprocessing using joints

You can extract bolt reactions by using joint probes. Both forces and moments can be extracted by switching between force probes and moment probes. Joint load boundary conditions are not included in the bolt reaction, so probe results do not include preload.

Bolt thread contact geometry correction

A new feature available in ANSYS R15, a bolt thread contact geometry correction, will distribute the loading over the faces as though the threads were present. To use this feature effectively

- Set Behavior to Asymmetric
- Do not use Nodal-Normal to Target or On Gauss Point detection methods (program controlled defaults to On Gauss Point)
- Do not use with bonded contact
- Make the bolt the contact surface and the bolt hole the target surface
- The mesh size in the threaded region should be smaller than one-fourth the pitch distance

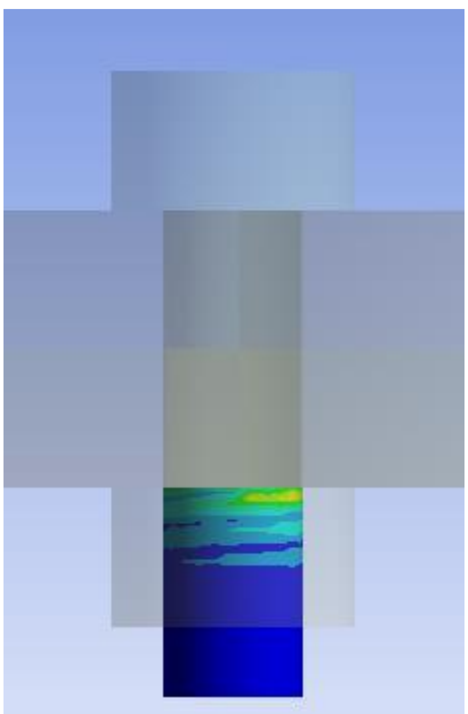


Figure 13.Stress distribution on bolt using bolt thread contact geometry correction

Here's a comparison chart summarizing the pros and cons of solid element bolts.

Bonded contact	**Frictional contact**	**Frictional contact with thread contact correction**
Pros	Pros	Pros
Capture stiffness of bolt, washer, and nut	Capture stiffness of bolt, washer, and nut	Capture stiffness of bolt, washer, and nut
Simpler model than using frictional contact between everything	Frictional contact	Frictional contact
		Closer approximation of stress distribution in the bolt
Cons	Cons	Cons
Harder to get bolt results than beam models	Harder to get bolt results than beam models	Harder to get bolt results than beam models
Less realistic than frictional contact between everything	More complex model	Most complex model that doesn't include threads
		Requires significantly higher mesh density in the threaded region

Note: The contact in the headings above refers to contact between the bolts/nuts and the flanges. The contact between the two flanges is assumed to be frictional.

Method 4: Modeling the Threads

You can explicitly model the thread geometry with frictional contact defined between the threads. However, these models can be *extremely computationally expensive.* A very fine mesh is required in the threads, and it is rarely practical to run a full assembly model with threads included—a submodel is almost always required. Even 3D bolt submodels can be impractically large. Using a 2D axisymmetric model is often more practical, though this is not an exact representation of the real geometry.

Achieving convergence can be difficult, particularly if the geometry isn't clean. Bolted joints are very common and well-researched. Hand calculations based on empirical data are often your best option (see NASA Technical Memo 106943).

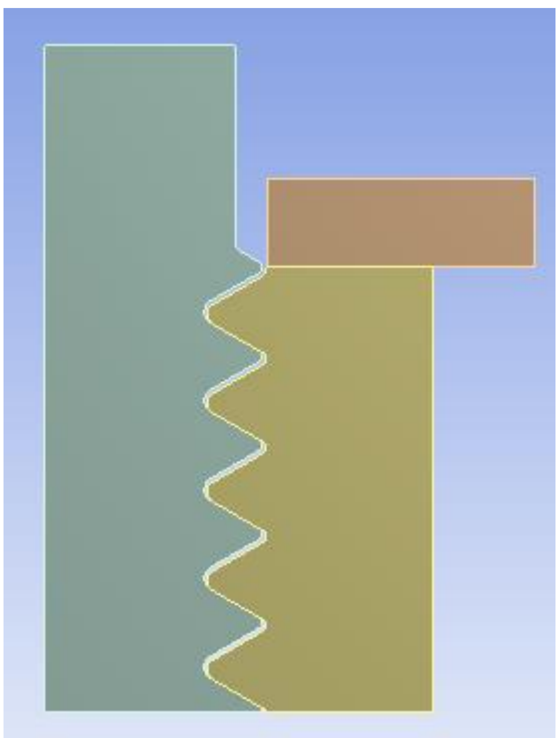


Figure 14. 2D axisymmetric bolt model

<u>Summary Table</u>

No bolts, bonded contact	Beam element bolts	Solid element bolts	Modeling the threads
<u>Pros</u> Easiest setup Linear	<u>Pros</u> Can extract bolt loads easily Accounts for stiffness of bolt Can account for preload Lightweight representation of bolts	<u>Pros</u> Can extract bolt loads Accounts for stiffness of entire fastener Can account for preload Can use frictional contact between parts	<u>Pros</u> Calculate stresses in threads Most closely represents actual situation
<u>Cons</u> Least realistic Preload not included Extracting bolt moments more labor intensive	<u>Cons</u> Doesn't account for stiffness of washer, bolt head, nut, etc.	<u>Cons</u> More difficult to extract bolt loads Larger mesh	<u>Cons</u> Often extremely computationally expensive Difficult to set up Good hand calculations for bolt margins exist and are much easier

Who We Are

SimuTech Group offers a wide range of finite element analysis (FEA) and computational fluid dynamics (CFD) engineering simulation software, support, training, consulting and testing services to engineering and manufacturing companies in the US and Canada.

As the largest full service provider of ANSYS FEA and CFD engineering simulation software in North America, SimuTech Group is well-positioned to provide clients with scalable, cost effective, and timely solutions.

With 11 regional offices and 25+ years of FEA, CFD and testing experience, no project is too small, too large, or too complex for us to handle. We are trusted advisors to our 1,700+ clients that range from one-man consulting shops to large multi-national manufacturers.

Our staff of 50+ FEA, CFD and testing consultants and support engineers have years of industrial experience to assist our clients in delivering cost effective and timely product designs or modifications.

Please visit our website or contact us to discover the FEA and CFD engineering simulation software, support, training, consulting and testing services we provide our clients.

To keep up to date on SimuTech and our simulation software, including ANSYS® Tips & Tricks, join our monthly newsletter. Or for more information, contact:

Eric Underwood
(303) 832-3149 x1751
EUnderwood@SimuTechGroup.com