Created in COMSOL Multiphysics 6.1

# Modeling of Pretensioned Bolts

## *Introduction*

In many structures, joined parts are clamped by bolts. In order to obtain a good clamping effect, the bolts are tightened so that the axial stresses are high. In an analysis, a correct state of the prestressed structure is often essential when evaluating the effect of service loads. For example, friction forces between joined parts may be crucial for the load carrying capacity. Also, if the effect of the pretension is ignored, the change in bolt forces due to service loads may be overestimated by one order of magnitude.

In most cases, the tensioning order of the bolts has little effect. However, if there are significant nonlinear phenomena, such as plasticity or frictional sliding, the sequence may have to be taken into consideration in the analysis.

In this example, different approaches for modeling pretensioned bolts are explored. The geometry does not show any realistic structure, as the focus is entirely on bolt modeling.

## *Model Definition*

Two steel plates, 150 mm-by-100 mm, are joined using five M10 bolts. The upper plate has a thickness of 10 mm and the lower plate has a thickness of 20 mm. There is an internal cavity formed by matching imprints in the two plates. The geometry is shown in Figure 1.

Two of the bolts are modeled with solid elements, and three by using beam elements. The connections to the plates are created by using different approximations.

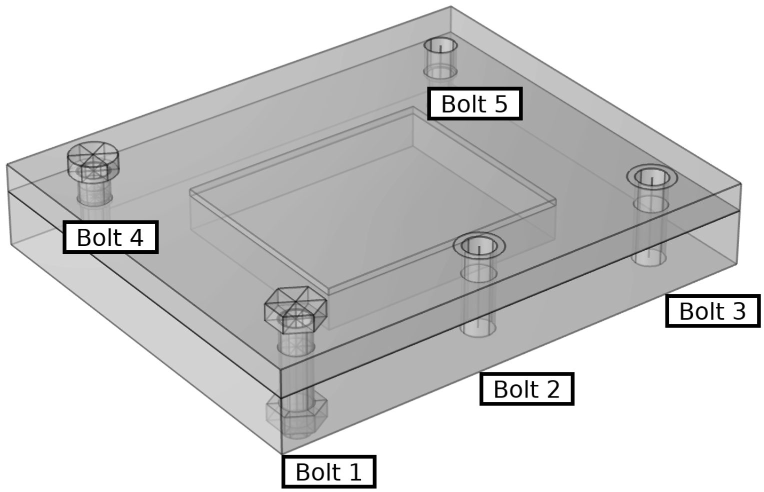



*Figure 1: The geometry of the component, and the bolt numbering.*

Details about how the bolts are modeled are given in Table 1.

TABLE 1: MODELING OF THE VARIOUS BOLTS.

| BOLT | ELEMENT TYPE | HEAD END | OTHER END |
|---|---|---|---|
| Bolt 1 | Solid | Continuity between head and upper plate | Continuity between nut and upper plate |
| Bolt 2 | Beam | Rigid connector from beam end to representative surface on upper plate | Nut, modeled using rigid connector from beam end to bolt hole edge on lower plate |
| Bolt 3 | Beam | Solid-Beam Connection from beam end to representative surface on upper plate | Nut, modeled using Solid-Beam Connection from beam end to bolt hole edge on lower plate |

TABLE 1: MODELING OF THE VARIOUS BOLTS.

| BOLT | ELEMENT TYPE | HEAD END | OTHER END |
|---|---|---|---|
| Bolt 4 | Solid | Contact between head and upper plate | Internal thread in lower plate, modeled using Continuity between bolt and plate |
| Bolt 5 | Beam | Solid-Beam Connection between beam end geometrical region on top surface of upper plate. | Internal thread in lower plate, modeled using Solid-Beam Connection from beam end to part of thread boundary in solid |

The final prestress force in the bolts is set to $P$ = 50 kN. However, not all bolts are tightened to the full prestress force simultaneously. Rather, the bolts are tightened one by one, with the three first bolts tightened only to 70% of the full value during the first cycle. In all, there are eight steps in the tightening cycle, as summarized in Table 2.

TABLE 2: FORCES IN THE BOLTS.

| STEP | BOLT 1 | BOLT 2 | BOLT 3 | BOLT 4 | BOLT 5 |
|---|---|---|---|---|---|
| 1 | 70% of P | Inactive | Inactive | Inactive | Inactive |
| 2 | From solution | 70% of P | Inactive | Inactive | Inactive |
| 3 | From solution | From solution | 70% of P | Inactive | Inactive |
| 4 | From solution | From solution | From solution | 100% of P | Inactive |
| 5 | From solution | From solution | From solution | From solution | 100% of P |
| 6 | 100% of P | From solution | From solution | From solution | From solution |
| 7 | From solution | 100% of P | From solution | From solution | From solution |
| 8 | From solution | From solution | 100% of P | From solution | From solution |

Between the two plates, as well as under the head of Bolt 4, there are contact conditions. The coefficient of friction is assumed to be 0.15 everywhere.

The service load is an internal pressure with a maximum value of 4 MPa.

## *Results and Discussion*

Table 3 summarizes the bolt forces in the different steps of the pretensioning sequence. As can be seen, the variation of the bolt forces from their prescribed values is very small in this case. Typically, the force in the already tightened bolts drops somewhat due to the compression from the neighboring bolts.

TABLE 3: COMPUTED FORCES IN THE BOLTS.

| STEP | BOLT 1 | BOLT 2 | BOLT 3 | BOLT 4 | BOLT 5 |
|---|---|---|---|---|---|
| 1 | 35000 | 500 | 500 | 500 | 500 |
| 2 | 35001 | 35000 | 503 | 499 | 501 |
| 3 | 34998 | 34992 | 35000 | 502 | 499 |
| 4 | 34995 | 34992 | 34999 | 50000 | 508 |
| 5 | 34986 | 34993 | 34997 | 49992 | 50000 |
| 6 | 50000 | 34991 | 34996 | 49975 | 49971 |
| 7 | 50000 | 50000 | 34994 | 49973 | 49970 |
| 8 | 50000 | 49997 | 50000 | 49973 | 49967 |

Note that the inactive bolts actually have been assigned a very small force; 1% of the final force. This will make the analysis run much faster than if the bolts were without force. The reason is that the contact problem has a very slow convergence rate when two boundaries are barely touching. There are other possible approaches, for example including gravity in the analysis, so that the upper plate rests slightly on the lower plate.

[Figure 2](#) shows stresses in the bolts as well as the contact pressure between the upper and the lower block.

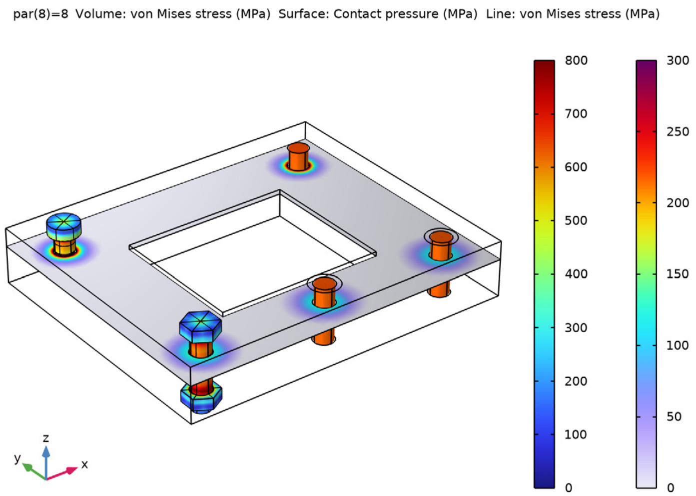



*Figure 2: Stresses in the bolts and contact pressure between the plates after the tightening sequence.*

All bolts have the same stress at the cross section where the bolt prestress is applied. In the bolts modeled with beam elements, the stress is exact and uniform all through the length, whereas when the bolts are modeled as solids, this is only true in an average sense. The detailed stress field is affected by stress concentrations.

The distribution of the contact pressure differs between the through bolts and the bolts that end in an internal thread. In the latter case, the pressure is higher close to the holes, since the path of the force is shorter. Note also that there is a significant contact pressure only in a circle within a diameter of two to three times the hole size. This is why, in practice, gaskets are needed to avoid leakage.

In Figure 3 shows the transverse ($Z$ direction) stress in the top and bottom blocks. As can be seen, the general picture is the same, irrespective of whether the bolt is modeled using beams or solid elements. The details of the stress field at the threads is more sensitive.

When modeling with beams, it is important to use a suitable effective bolt length, as well as a suitable coupling length inside the thread.

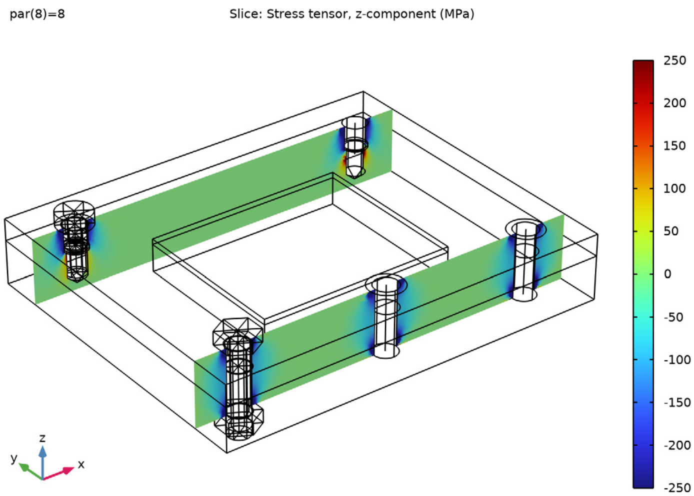



*Figure 3: Stress in the Z direction at the bolt holes after the tightening sequence.*

In Figure 4 to Figure 6, the bolt stress and the contact pressures are shown for three different levels of the internal pressure in the cavity. There is a significant redistribution of the contact pressure at higher load levels, which indicates that the joints are no longer operating as intended.

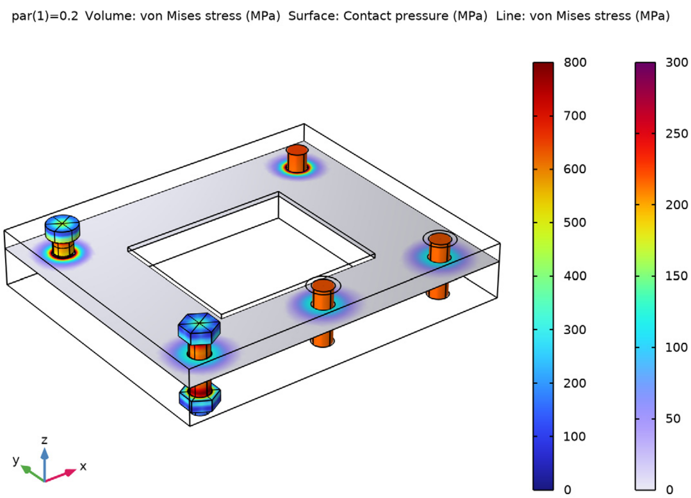



*Figure 4: Stresses in the bolts and contact pressure between the plates after applying 20% of the service load.*

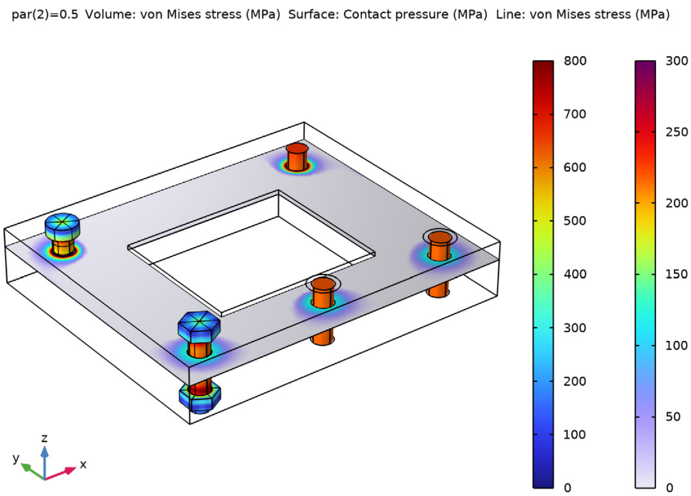



*Figure 5: Stresses in the bolts and contact pressure between the plates after applying 50% of the service load.*

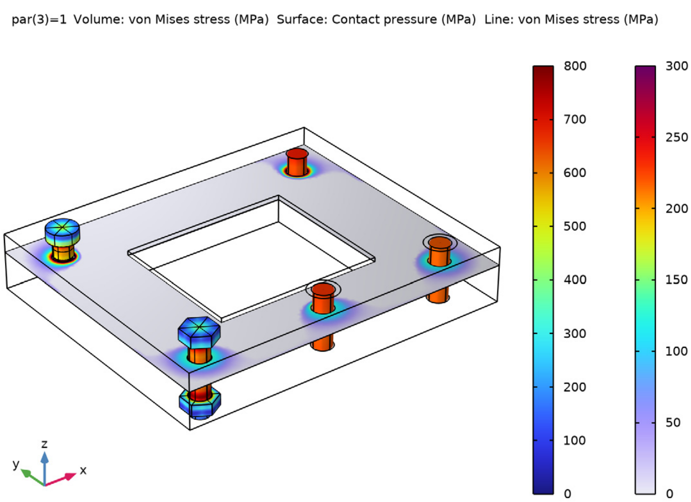



*Figure 6: Stresses in the bolts and contact pressure between the plates after applying 100% of the service load.*

**Application Library path:** `Structural_Mechanics_Module/Tutorials/bolt_pretension_tutorial`

## *Modeling Instructions*

From the **File** menu, choose **New**.

### NEW

In the **New** window, click **Model Wizard**.

### MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)** and **Structural Mechanics>Beam (beam)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Bolt Pretension**.

6 Click **Done**.

### GLOBAL DEFINITIONS

#### *Parameters 1*

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `bolt_pretension_tutorial_parameters.txt`.

### GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

### *Form Union (fin)*

1 In the **Model Builder** window, expand the **Geometry 1** node, then click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.

3 From the **Action** list, choose **Form an assembly**.

### *Block: Bottom*

1 In the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, type `Block: Bottom` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `plateLen`.

4 In the **Depth** text field, type `plateWidth`.

5 In the **Height** text field, type `thicLow`.

6 Click **Build Selected**.

### *Block: Top*

1 Right-click **Block: Bottom** and choose **Duplicate**.

2 In the **Settings** window for **Block**, type `Block: Top` in the **Label** text field.

3 Locate the **Position** section. In the **z** text field, type `thicLow`.

4 Locate the **Size and Shape** section. In the **Height** text field, type `thicUp`.

5 Click **Build Selected**.

### *Block: Cavity*

1 In the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, type `Block: Cavity` in the **Label** text field.

3 Click the **Transparency** button in the **Graphics** toolbar.

4 Locate the **Size and Shape** section. In the **Width** text field, type `plateLen/2`.

5 In the **Depth** text field, type `plateWidth/2`.

6 In the **Height** text field, type `(thicUp+thicLow)/2`.

7 Locate the **Position** section. In the **x** text field, type `plateLen/4`.

8 In the **y** text field, type `plateWidth/4`.

9 In the **z** text field, type `(thicUp+thicLow)/4`.

### *Cylinder: Bolt Hole*

1 In the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for **Cylinder**, type `Cylinder: Bolt Hole` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Radius** text field, type `holeDia/2`.

4 In the **Height** text field, type `thicLow+thicUp+2[mm]`.

5 Locate the **Position** section. In the **x** text field, type `20`.

6 In the **y** text field, type `20`.

7 In the **z** text field, type `-1`.

8 Click **Build Selected**.

*Array: Bolt Holes*

1 In the **Geometry** toolbar, click **Transforms** and choose **Array**.

2 In the **Settings** window for **Array**, type `Array: Bolt Holes` in the **Label** text field.

3 Select the object **cyl1** only.

4 Locate the **Size** section. In the **x size** text field, type `3`.

5 Locate the **Displacement** section. In the **x** text field, type `boltSpacing`.

6 Click **Build Selected**.

## PART LIBRARIES

1 In the **Geometry** toolbar, click **Parts** and choose **Part Libraries**.

2 In the **Part Libraries** window, select **Structural Mechanics Module>Bolts> hex_bolt_no_thread** in the tree.

3 Click **Add to Geometry**.

## GEOMETRY 1

*Hex Bolt, No Thread 1 (pi1)*

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Hex Bolt, No Thread 1 (pi1)**.

2 In the **Settings** window for **Part Instance**, locate the **Input Parameters** section.

3 In the table, enter the following settings:

| Name | Expression | Value | Description |
|---|---|---|---|
| hgrip | `headDia` | 16 mm | Head grip |
| hthic | 6 | 6 mm | Head thickness |
| ndia | `boltDia` | 10 mm | Nominal diameter |
| blen | `thicUp+thicLow+10[mm]` | 40 mm | Bolt length |

4 Locate the **Position and Orientation of Output** section. Find the **Displacement** subsection. In the **xw** text field, type `20`.

5 In the **yw** text field, type `20`.

6 In the **zw** text field, type `-10`.

7 Click **Build Selected**.

8 Click to expand the **Boundary Selections** section. In the table, enter the following settings:

| Name | Keep | Physics | Contribute to |
|---|---|---|---|
| Pretension cut | √ | √ | None |

9 Click to expand the **Domain Selections** section. In the table, enter the following settings:

| Name | Keep | Physics | Contribute to |
|---|---|---|---|
| All | √ | √ | None |

10 Click to select row number 1 in the table.

11 Click **New Cumulative Selection**.

12 In the **New Cumulative Selection** dialog box, type `Bolts and Nuts` in the **Name** text field.

13 Click **OK**.

### PART LIBRARIES

1 In the **Geometry** toolbar, click **Parts** and choose **Part Libraries**.

2 In the **Model Builder** window, click **Geometry 1**.

3 In the **Part Libraries** window, select **Structural Mechanics Module>Bolts>hex_nut** in the tree.

4 Click **Add to Geometry**.

### GEOMETRY 1

*Hexagonal Nut 1 (pi2)*

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Hexagonal Nut 1 (pi2)**.

2 In the **Settings** window for **Part Instance**, locate the **Input Parameters** section.

3 In the table, enter the following settings:

| Name | Expression | Value | Description |
|---|---|---|---|
| hgrip | `headDia` | 16 mm | Head grip |

| Name | Expression | Value | Description |
|---|---|---|---|
| hdia | boltDia | 10 mm | Nominal hole diameter |
| thickness | 6 | 6 mm | Thickness |

4 Locate the **Position and Orientation of Output** section. Find the **Displacement** subsection. In the **xw** text field, type 20.

5 In the **yw** text field, type 20.

6 In the **zw** text field, type -6.

7 Click **Build Selected**.

8 Locate the **Domain Selections** section. In the table, enter the following settings:

| Name | Keep | Physics | Contribute to |
|---|---|---|---|
| All | √ | √ | Bolts and Nuts |

### PART LIBRARIES

1 In the **Geometry** toolbar, click **Parts** and choose **Part Libraries**.

2 In the **Model Builder** window, click **Geometry 1**.

3 In the **Part Libraries** window, select **Structural Mechanics Module>Bolts>simple_bolt_drill** in the tree.

4 Click **Add to Geometry**.

### GEOMETRY 1

*Simple Bolt, With Drill 1 (pi3)*

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Simple Bolt, With Drill 1 (pi3)**.

2 In the **Settings** window for **Part Instance**, locate the **Input Parameters** section.

3 In the table, enter the following settings:

| Name | Expression | Value | Description |
|---|---|---|---|
| hdia | headDia | 16 mm | Head diameter |
| hthic | 6 | 6 mm | Head thickness |
| ndia | boltDia | 10 mm | Nominal diameter |
| sdia | boltDia-1 | 9 mm | Stress diameter |

| Name | Expression | Value | Description |
|---|---|---|---|
| blen | thicUp+threadDepth-(boltDia-1)/(2*tan(50[deg])) | 22.224 mm | Bolt length |
| tlen | threadDepth-(boltDia-1)/(2*tan(50[deg])) | 12.224 mm | Thread length |

4 Locate the **Position and Orientation of Output** section. Find the **Displacement** subsection. In the **xw** text field, type 20.

5 In the **yw** text field, type plateWidth-20.

6 In the **zw** text field, type thicLow-threadDepth+(boltDia-1)/(2*tan(50[deg])).

7 Locate the **Boundary Selections** section. In the table, enter the following settings:

| Name | Keep | Physics | Contribute to |
|---|---|---|---|
| Pretension cut | √ | √ | None |

8 Locate the **Domain Selections** section. In the table, enter the following settings:

| Name | Keep | Physics | Contribute to |
|---|---|---|---|
| All | | √ | None |
| Bolt | √ | √ | Bolts and Nuts |

9 Click **Build All Objects**.

*Copy: Drill*

1 In the **Geometry** toolbar, click **Transforms** and choose **Copy**.

2 In the **Settings** window for **Copy**, type Copy: Drill in the **Label** text field.

3 Select the object **pi3(2)** only.

4 Locate the **Displacement** section. In the **x** text field, type 2*boltSpacing.

5 Click **Build Selected**.

*Difference: Bolt Holes and Cavity, Upper*

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 In the **Settings** window for **Difference**, type Difference: Bolt Holes and Cavity, Upper in the **Label** text field.

3 Select the object **blk2** only.

4 Locate the **Difference** section. Select the **Keep objects to subtract** check box.

5 Find the **Objects to subtract** subsection. Click to select the **Activate Selection** toggle button.

6 Select the objects **arr1(1,1,1)**, **arr1(2,1,1)**, **arr1(3,1,1)**, **blk3**, **copy1**, and **pi3(2)** only.

7 Click **Build Selected**.

*Difference: Bolt Holes and Cavity, Lower*

1 Right-click **Difference: Bolt Holes and Cavity, Upper** and choose **Duplicate**.

2 In the **Settings** window for **Difference**, type `Difference: Bolt Holes and Cavity, Lower` in the **Label** text field.

3 Locate the **Difference** section. Find the **Objects to add** subsection. Click to select the **Activate Selection** toggle button.

4 Select the object **blk1** only.

5 Clear the **Keep objects to subtract** check box.

6 Click **Build Selected**.

7 In the **Geometry** toolbar, click **Build All**.

*Work Plane 1 (wp1)*

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Offset type** list, choose **Through vertex**.

4 Find the **Offset vertex** subsection. Click to select the **Activate Selection** toggle button.

5 On the object **dif1**, select Point 2 only.

*Work Plane 1 (wp1)>Plane Geometry*

In the **Model Builder** window, click **Plane Geometry**.

*Circle: Imprint for Bolt Head*

1 In the **Work Plane** toolbar, click **Circle**.

2 In the **Settings** window for **Circle**, type `Circle: Imprint for Bolt Head` in the **Label** text field.

3 Locate the **Object Type** section. From the **Type** list, choose **Curve**.

4 Locate the **Size and Shape** section. In the **Radius** text field, type `headDia/2`.

5 Locate the **Position** section. In the **xw** text field, type `20+boltSpacing`.

6 In the **yw** text field, type `20`.

7 Click **Build Selected**.

*Copy: Imprints for Bolt Heads*

1 In the **Work Plane** toolbar, click **Transforms** and choose **Copy**.

2 In the **Settings** window for **Copy**, type `Copy: Imprints for Bolt Heads` in the **Label** text field.

3 Select the object **c1** only.

4 Locate the **Displacement** section. In the **xw** text field, type `boltSpacing`.

5 Click **Build Selected**.

*Union: Imprints for Bolt Heads*

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Booleans and Partitions>Union**.

2 In the **Settings** window for **Union**, type `Union: Imprints for Bolt Heads` in the **Label** text field.

3 Select the objects **dif1** and **wp1** only.

4 Click **Build All Objects**.

*Polygon: Bolt 2 Beam*

1 In the **Geometry** toolbar, click **More Primitives** and choose **Polygon**.

2 In the **Settings** window for **Polygon**, type `Polygon: Bolt 2 Beam` in the **Label** text field.

3 Locate the **Coordinates** section. In the table, enter the following settings:

| x (mm) | y (mm) | z (mm) |
|---|---|---|
| 20+boltSpacing | 20 | 0 |
| 20+boltSpacing | 20 | 15 |
| 20+boltSpacing | 20 | thicLow+thicUp |

4 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

5 In the **New Cumulative Selection** dialog box, type `Beams` in the **Name** text field.

6 Click **OK**.

*Copy: Bolt 3 Beam*

1 In the **Geometry** toolbar, click **Transforms** and choose **Copy**.

2 In the **Settings** window for **Copy**, type `Copy: Bolt 3 Beam` in the **Label** text field.

3 Select the object **pol1** only.

4 Locate the **Displacement** section. In the **x** text field, type `boltSpacing`.

5 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Bolts and Nuts**.

6 Click **Build Selected**.

### *Polygon: Bolt 5 Beam*

1 In the **Geometry** toolbar, click **More Primitives** and choose **Polygon**.

2 In the **Settings** window for **Polygon**, type `Polygon: Bolt 5 Beam` in the **Label** text field.

3 Locate the **Coordinates** section. In the table, enter the following settings:

| x (mm) | y (mm) | z (mm) |
|---|---|---|
| `20+2*boltSpacing` | `plateWidth-20` | `thicLow-boltDia/2` |
| `20+2*boltSpacing` | `plateWidth-20` | `thicLow+5` |
| `20+2*boltSpacing` | `plateWidth-20` | `thicLow+thicUp` |

4 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Beams**.

5 Click **Build Selected**.

Take a look at the detailed bolt geometries.

6 In the **Graphics** window toolbar, click next to **Clipping**, then choose **Add Clip Plane**.

## DEFINITIONS

In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

### *Clip Plane 1*

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions>View 1** node, then click **Clip Plane 1**.

2 In the **Settings** window for **Clip Plane**, locate the **Position** section.

3 Find the **Definition** subsection. In the **x** text field, type `20.01`.

4 Select the objects **pi1**, **pi2**, and **pi3(1)** only.

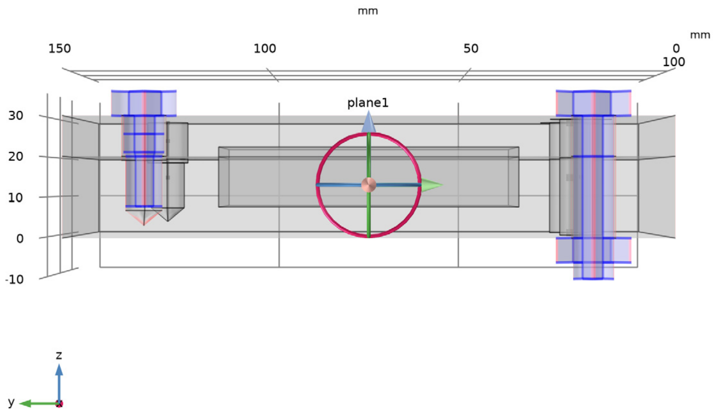
5 Click the **Go to YZ View** button in the **Graphics** toolbar.

6 Click the **Go to YZ View** button in the **Graphics** toolbar.

7 Click the **Go to YZ View** button in the **Graphics** toolbar.

8 In the **Graphics** window toolbar, click next to **Clipping Active**, then choose **Show Frames**.

**9** Click the **Zoom Extents** button in the **Graphics** toolbar.




**10** In the **Graphics** window toolbar, click ▾ next to **Clipping**, then choose **Delete Plane1**.

## ROOT

Click the **Go to Default View** button in the **Graphics** toolbar.

## BEAM (BEAM)

**1** In the **Model Builder** window, under **Component 1 (comp1)** click **Beam (beam)**.

**2** In the **Settings** window for **Beam**, locate the **Edge Selection** section.

**3** From the **Selection** list, choose **Beams**.

### *Cross Section M10*

**1** In the **Model Builder** window, under **Component 1 (comp1)>Beam (beam)** click **Cross-Section Data 1**.

**2** In the **Settings** window for **Cross-Section Data**, type `Cross Section M10` in the **Label** text field.

**3** Locate the **Cross-Section Definition** section. From the list, choose **Common sections**.

**4** From the **Section type** list, choose **Circular**.

**5** In the $d_o$ text field, type `boltDia`.

### *Section Orientation 1*

1 In the **Model Builder** window, click **Section Orientation 1**.

2 In the **Settings** window for **Section Orientation**, locate the **Section Orientation** section.

3 From the **Orientation method** list, choose **Orientation vector**.

4 Specify the *V* vector as

| | |
|---|---|
| 1 | X |
| 0 | Y |
| 0 | Z |

## ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Built-in>Structural steel**.

4 Right-click and choose **Add to Global Materials**.

5 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

## MATERIALS

### *Material Link 1 (matlnk1)*

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **More Materials>Material Link**.

### *Material Link 2 (matlnk2)*

1 Right-click **Materials** and choose **More Materials>Material Link**.

2 In the **Settings** window for **Material Link**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Edge**.

4 From the **Selection** list, choose **Beams**.

Bolt 2 is connected using rigid connectors. Create the Solid Mechanics part.

## SOLID MECHANICS (SOLID)

In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

### *RC Bolt 2, Head*

1 In the **Physics** toolbar, click **Boundaries** and choose **Rigid Connector**.

2 In the **Settings** window for **Rigid Connector**, type `RC Bolt 2, Head` in the **Label** text field.

3 Select Boundary 67 only.

*RC Bolt 2, Nut*

1 In the **Physics** toolbar, click **Edges** and choose **Rigid Connector**.

2 In the **Settings** window for **Rigid Connector**, type `RC Bolt 2, Nut` in the **Label** text field.

3 Select Edges 58, 59, 63, and 66 only.

In the continuity conditions, connections will be established between mismatching meshes. It is often more efficient to use a Nitsche formulation than a large number of coupled pointwise constraints.

*Continuity 1*

1 In the **Model Builder** window, click **Continuity 1**.

2 In the **Settings** window for **Continuity**, locate the **Method** section.

3 From the list, choose **Nitsche**.

### BEAM (BEAM)

Add the Beam side of the rigid connectors. In order to couple the rigid connectors between the physics interfaces, **Advanced Physics Options** must be enabled.

In the **Model Builder** window, under **Component 1 (comp1)** click **Beam (beam)**.

*RC Bolt 2, Head*

1 In the **Physics** toolbar, click **Points** and choose **Rigid Connector**.

2 Click the **Show More Options** button in the **Model Builder** toolbar.

3 In the **Show More Options** dialog box, select **Physics>Advanced Physics Options** in the tree.

4 In the tree, select the check box for the node **Physics>Advanced Physics Options**.

5 Click **OK**.

6 In the **Settings** window for **Rigid Connector**, type `RC Bolt 2, Head` in the **Label** text field.

7 Select Point 247 only.

8 Click to expand the **Advanced** section. From the **Connect to** list, choose **RC Bolt 2, Head (solid)**.

*RC Bolt 2, Nut*

1 In the **Physics** toolbar, click **Points** and choose **Rigid Connector**.

2 In the **Settings** window for **Rigid Connector**, type `RC Bolt 2, Nut` in the **Label** text field.

3 Select Point 245 only.

4 Locate the **Advanced** section. From the **Connect to** list, choose **RC Bolt 2, Nut (solid)**.

## MULTIPHYSICS

*SBC, Bolt 3, Head*

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Global>Solid-Beam Connection**.
2 In the **Settings** window for **Solid-Beam Connection**, type `SBC, Bolt 3, Head` in the **Label** text field.
3 Locate the **Connection Settings** section. Select the **Manual control of selections** check box.
4 Select Boundary 73 only.
5 Locate the **Point Selection, Beam** section. Click to select the **Activate Selection** toggle button.
6 Select Point 250 only.

*SBC, Bolt 3, Nut*

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Global>Solid-Beam Connection**.
2 In the **Settings** window for **Solid-Beam Connection**, type `SBC, Bolt 3, Nut` in the **Label** text field.
3 Locate the **Connection Settings** section. From the **Connection type** list, choose **Solid edges to beam points**.
4 Locate the **Edge Selection, Solid** section. Click to select the **Activate Selection** toggle button.
5 Select Edges 74, 75, 90, and 93 only.
6 Locate the **Point Selection, Beam** section. Click to select the **Activate Selection** toggle button.
7 Select Point 248 only.

*SBC, Bolt 5, Head*

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Global>Solid-Beam Connection**.
2 In the **Settings** window for **Solid-Beam Connection**, type `SBC, Bolt 5, Head` in the **Label** text field.
3 Locate the **Connection Settings** section. Select the **Manual control of selections** check box.
4 Select Boundary 53 only.
5 Locate the **Point Selection, Beam** section. Click to select the **Activate Selection** toggle button.

6 Select Point 253 only.

7 Locate the **Connection Settings** section. From the **Connected region** list, choose **Distance (manual)**.

8 In the $r_c$ text field, type `headDia/2`.

*SBC, Bolt 5, Thread*

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Global>Solid-Beam Connection**.

2 In the **Settings** window for **Solid-Beam Connection**, type `SBC, Bolt 5, Thread` in the **Label** text field.

3 Locate the **Connection Settings** section. Select the **Manual control of selections** check box.

4 Select Boundaries 37, 38, 44, and 47 only.

5 Locate the **Point Selection, Beam** section. Click to select the **Activate Selection** toggle button.

6 Select Point 251 only.

7 Locate the **Connection Settings** section. From the **Connected region** list, choose **Connection criterion**.

8 In the text field, type `Z>thicLow-boltDia/2`.

## GLOBAL DEFINITIONS

Create functions returning the prestress values and the times when they are changed. Using such functions makes the input in each **Bolt Selection** node more readable.

*Analytic 1 (an1)*

1 In the **Home** toolbar, click f(x) **Functions** and choose **Global>Analytic**.

2 In the **Settings** window for **Analytic**, type `forceValue` in the **Function name** text field.

3 Locate the **Definition** section. In the **Arguments** text field, type `active,full,time`.

4 In the **Expression** text field, type `if(time<active,0.01,if(time<full,0.7,1))`.

5 Locate the **Units** section. In the **Function** text field, type `1`.

6 In the table, enter the following settings:

| Argument | Unit |
|---|---|
| active | 1 |
| full | 1 |
| time | 1 |

*Analytic 2 (forceValue2)*

1 Right-click **Analytic 1 (forceValue)** and choose **Duplicate**.

2 In the **Settings** window for **Analytic**, type `setPre` in the **Function name** text field.

3 Locate the **Definition** section. In the **Expression** text field, type `time==1 || abs(time-active)<0.001 || abs(time-full)<0.001`.

## SOLID MECHANICS (SOLID)

In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

*Bolt Pretension 1*

1 In the **Physics** toolbar, click **Global** and choose **Bolt Pretension**.

2 In the **Settings** window for **Bolt Pretension**, locate the **Bolt Pretension** section.

3 In the $F_\mathrm{p}$ text field, type `boltForce`.

4 Select the **Compute tightening torque** check box.

5 In the $l$ text field, type `1.5[mm]`.

6 From the **Bolt head type** list, choose **Hexagonal**.

*Bolt Selection 1*

1 In the **Model Builder** window, expand the **Bolt Pretension 1** node, then click **Bolt Selection 1**.

2 In the **Settings** window for **Bolt Selection**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Pretension cut (Hex Bolt, No Thread 1)**.

4 Locate the **Bolt Pretension** section. Select the **Sequential tightening** check box.

5 From the **Pretension type** list, choose **Pretension force**.

6 In the $F_\mathrm{p}$ text field, type `boltForce*forceValue(1,6,par)`.

7 In the **Pretensioning expression** text field, type `setPre(1,6,par)`.

*Bolt Pretension 1*

In the **Model Builder** window, click **Bolt Pretension 1**.

*Bolt Selection 2*

1 In the **Physics** toolbar, click **Attributes** and choose **Bolt Selection**.

2 In the **Settings** window for **Bolt Selection**, locate the **Bolt Label** section.

3 In the text field, type `Bolt_4`.

4 Locate the **Boundary Selection** section. From the **Selection** list, choose **Pretension cut (Simple Bolt, With Drill 1)**.

5 Locate the **Bolt Pretension** section. Select the **Sequential tightening** check box.

6 From the **Pretension type** list, choose **Pretension force**.

7 In the $F_\mathrm{p}$ text field, type `boltForce*forceValue(4,4,par)`.

8 In the **Pretensioning expression** text field, type `setPre(4,4,par)`.

## BEAM (BEAM)

In the **Model Builder** window, under **Component 1 (comp1)** click **Beam (beam)**.

### *Bolt Pretension 1*

1 In the **Physics** toolbar, click **Global** and choose **Bolt Pretension**.

2 In the **Settings** window for **Bolt Pretension**, locate the **Bolt Pretension** section.

3 In the $F_\mathrm{p}$ text field, type `boltForce`.

### *Bolt Selection 1*

1 In the **Model Builder** window, expand the **Bolt Pretension 1** node, then click **Bolt Selection 1**.

2 In the **Settings** window for **Bolt Selection**, locate the **Bolt Label** section.

3 In the text field, type `Bolt_2`.

4 Select Point 246 only.

5 Locate the **Bolt Pretension** section. Select the **Sequential tightening** check box.

6 From the **Pretension type** list, choose **Pretension force**.

7 In the $F_\mathrm{p}$ text field, type `boltForce*forceValue(2,7,par)`.

8 In the **Pretensioning expression** text field, type `setPre(2,7,par)`.

### *Bolt Selection 2*

1 Right-click **Component 1 (comp1)>Beam (beam)>Bolt Pretension 1>Bolt Selection 1** and choose **Duplicate**.

2 In the **Settings** window for **Bolt Selection**, locate the **Bolt Label** section.

3 In the text field, type `Bolt_3`.

4 Locate the **Point Selection** section. Click **Clear Selection**.

5 Select Point 249 only.

6 Locate the **Bolt Pretension** section. In the $F_\mathrm{p}$ text field, type `boltForce*forceValue(3,8,par)`.

7 In the **Pretensioning expression** text field, type `setPre(3,8,par)`.

### *Bolt Pretension 1*

In the **Model Builder** window, click **Bolt Pretension 1**.

### *Bolt Selection 3*

1 In the **Physics** toolbar, click **Attributes** and choose **Bolt Selection**.
2 Select Point 252 only.
3 In the **Settings** window for **Bolt Selection**, locate the **Bolt Label** section.
4 In the text field, type `Bolt_5`.
5 Locate the **Bolt Pretension** section. Select the **Sequential tightening** check box.
6 From the **Pretension type** list, choose **Pretension force**.
7 In the $F_{\mathrm{p}}$ text field, type `boltForce*forceValue(5,5,par)`.
8 In the **Pretensioning expression** text field, type `setPre(5,5,par)`.

## DEFINITIONS

### *Identity Boundary Pair 1a (ap1)*

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **Identity Boundary Pair 1a (ap1)**.
2 In the **Settings** window for **Pair**, locate the **Pair Type** section.
3 Select the **Manual control of selections and pair type** check box.
4 From the **Pair type** list, choose **Contact pair**.
5 Locate the **Advanced** section. In the **Extrapolation tolerance** text field, type `1e-2`.
6 From the **Mapping method** list, choose **Initial configuration**.

### *Identity Boundary Pair 5a (ap5)*

1 In the **Model Builder** window, click **Identity Boundary Pair 5a (ap5)**.
2 In the **Settings** window for **Pair**, locate the **Pair Type** section.
3 Select the **Manual control of selections and pair type** check box.
4 From the **Pair type** list, choose **Contact pair**.
5 Locate the **Advanced** section. From the **Mapping method** list, choose **Initial configuration**.

   Now that there are contact pairs in the model, a default **Contact** node appears. Add friction to it.

## SOLID MECHANICS (SOLID)

### *Contact 1*

In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Contact 1**.

### *Friction 1*

1 In the **Physics** toolbar, click **Attributes** and choose **Friction**.

2 In the **Settings** window for **Friction**, locate the **Friction Parameters** section.

3 In the μ text field, type `0.15`.

### *Rigid Motion Suppression 1*

Since all loads in this model are self-equilibrating, the only constraints needed are for suppressing possible rigid body motions.

1 In the **Physics** toolbar, click **Domains** and choose **Rigid Motion Suppression**.

2 Select Domain 1 only.

A contact analysis implies a geometrically nonlinear analysis. In a case like this, the deformations are however small, and it may be more efficient to use a linear formulation for the material models.

### *Linear Elastic Material 1*

1 In the **Model Builder** window, click **Linear Elastic Material 1**.

2 In the **Settings** window for **Linear Elastic Material**, locate the **Geometric Nonlinearity** section.

3 Select the **Geometrically linear formulation** check box.

## BEAM (BEAM)

### *Linear Elastic Material 1*

1 In the **Model Builder** window, under **Component 1 (comp1)>Beam (beam)** click **Linear Elastic Material 1**.

2 In the **Settings** window for **Linear Elastic Material**, locate the **Geometric Nonlinearity** section.

3 Select the **Geometrically linear formulation** check box.

## MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Sequence Type** section.

**3** From the list, choose **User-controlled mesh**.

### *Free Tetrahedral 1*

**1** In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Free Tetrahedral 1**.

**2** In the **Settings** window for **Free Tetrahedral**, locate the **Domain Selection** section.

**3** From the **Geometric entity level** list, choose **Domain**.

**4** Select Domains 1 and 2 only.

### *Size 1*

**1** Right-click **Free Tetrahedral 1** and choose **Size**.

**2** In the **Settings** window for **Size**, locate the **Element Size** section.

**3** From the **Predefined** list, choose **Finer**.

**4** Click **Build Selected**.

### *Size 2*

**1** Right-click **Free Tetrahedral 1** and choose **Size**.

**2** In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

**3** From the **Geometric entity level** list, choose **Boundary**.

**4** Select Boundary 52 only.

**5** Locate the **Element Size** section. From the **Predefined** list, choose **Finer**.

**6** Click the **Custom** button.

**7** Locate the **Element Size Parameters** section.

**8** Select the **Maximum element size** check box. In the associated text field, type 5.

**9** Click **Build All**.

### *Swept 1*

**1** In the **Mesh** toolbar, click **Swept**.

**2** In the **Settings** window for **Swept**, click to expand the **Source Faces** section.

**3** Select Boundaries 86, 87, 92, 99, 104, and 109 only.

### *Distribution 1*

**1** Right-click **Swept 1** and choose **Distribution**.

**2** In the **Settings** window for **Distribution**, locate the **Distribution** section.

**3** In the **Number of elements** text field, type 3.

*Distribution 2*

1 In the **Model Builder** window, right-click **Swept 1** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domains 12, 13, 16, 19, 21, and 22 only.

5 Click **Build All**.

*Edge 1*

1 In the **Mesh** toolbar, click **Boundary** and choose **Edge**.

2 In the **Settings** window for **Edge**, locate the **Edge Selection** section.

3 From the **Selection** list, choose **Beams**.

## STUDY 1

*Step 1: Bolt Pretension*

1 In the **Model Builder** window, under **Study 1** click **Step 1: Bolt Pretension**.

2 In the **Settings** window for **Bolt Pretension**, click to expand the **Study Extensions** section.

3 Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

| Parameter name | Parameter value list | Parameter unit |
|---|---|---|
| par (Solution parameter) | `range(1,1,8)` | |

*Solution 1 (sol1)*

In the **Study** toolbar, click **Show Default Solver**.

*Step 1: Bolt Pretension*

1 In the **Model Builder** window, under **Study 1** click **Step 1: Bolt Pretension**.

2 In the **Settings** window for **Bolt Pretension**, click to expand the **Results While Solving** section.

3 Select the **Plot** check box.

*Solution 1 (sol1)*

During initial pretensioning, the displacements in the beams are close to zero, so the automatic scaling of variables may be problematic.

1. In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Dependent Variables 1** node, then click **Displacement field (material and geometry frames) (comp1.beam.uLin)**.
2. In the **Settings** window for **Field**, locate the **Scaling** section.
3. From the **Method** list, choose **Manual**.
4. In the **Scale** text field, type `1e-3`.
5. In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Dependent Variables 1** click **Rotation field (material and geometry frames) (comp1.beam.thLin)**.
6. In the **Settings** window for **Field**, locate the **Scaling** section.
7. From the **Method** list, choose **Manual**.
8. In the **Scale** text field, type `0.01`.

   Using a more aggressive iteration scheme is often faster for this class of problems.
9. In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** node, then click **Fully Coupled 1**.
10. In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.
11. From the **Nonlinear method** list, choose **Constant (Newton)**.

    Since the prestress values are changed at discrete parameter values, it is not meaningful to let the solver automatically choose parameter steps.
12. In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** click **Parametric 1**.
13. In the **Settings** window for **Parametric**, click to expand the **Continuation** section.
14. Select the **Tuning of step size** check box.
15. In the **Initial step size** text field, type `1`.
16. In the **Minimum step size** text field, type `1`.
17. In the **Maximum step size** text field, type `1`.
18. In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** click **Advanced**.
19. In the **Settings** window for **Advanced**, click to expand the **Assembly Settings** section.
20. Clear the **Reuse sparsity pattern** check box.

    Update the plot for every iteration.

21 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** click **Fully Coupled 1**.

22 In the **Settings** window for **Fully Coupled**, click to expand the **Results While Solving** section.

23 Select the **Plot** check box.

24 In the **Study** toolbar, click **Show Default Plots**.

## RESULTS

Set up a suitable plot for monitoring the solution process.

### *Bolt Stress and Contact Pressure*

1 In the **Settings** window for **3D Plot Group**, type `Bolt Stress and Contact Pressure` in the **Label** text field.

2 Click to collapse the **Data** section.

### *Volume 1*

1 In the **Model Builder** window, expand the **Bolt Stress and Contact Pressure** node, then click **Volume 1**.

2 In the **Settings** window for **Volume**, locate the **Expression** section.

3 From the **Unit** list, choose **MPa**.

4 Click to expand the **Range** section. Select the **Manual color range** check box.

5 In the **Maximum** text field, type `800`.

6 Locate the **Coloring and Style** section. Click **Change Color Table**.

7 In the **Color Table** dialog box, select **Rainbow>Rainbow** in the tree.

8 Click **OK**.

### *Selection 1*

1 Right-click **Volume 1** and choose **Selection**.

2 Clear all domains.

3 In the **Settings** window for **Selection**, locate the **Selection** section.

4 From the **Selection** list, choose **Bolts and Nuts**.

### *Surface 1*

1 In the **Model Builder** window, right-click **Bolt Stress and Contact Pressure** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Contact>solid.Tn - Contact pressure - N/m²**.

3 Locate the **Expression** section. From the **Unit** list, choose **MPa**.

4 Locate the **Coloring and Style** section. Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Rainbow>Prism** in the tree.

6 Click **OK**.

7 In the **Settings** window for **Surface**, click to expand the **Range** section.

8 Select the **Manual color range** check box.

9 In the **Maximum** text field, type 300.

*Deformation 1*

1 Right-click **Surface 1** and choose **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Scale** section.

3 Select the **Scale factor** check box.

*Line 1*

1 In the **Model Builder** window, expand the **Results>Stress (beam)** node.

2 Right-click **Line 1** and choose **Copy**.

*Line 1*

1 In the **Model Builder** window, right-click **Bolt Stress and Contact Pressure** and choose **Paste Line**.

2 In the **Settings** window for **Line**, locate the **Expression** section.

3 From the **Unit** list, choose **MPa**.

4 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Volume 1**.

5 Click the **Transparency** button in the **Graphics** toolbar.

## STUDY 1

In the **Study** toolbar, click **Compute**.

## RESULTS

*Bolt Stress and Contact Pressure*

1 Click the **Show Grid** button in the **Graphics** toolbar.

2 In the **Bolt Stress and Contact Pressure** toolbar, click **Plot**.

*Transverse Stress in the Bolt Planes*

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Transverse Stress in the Bolt Planes` in the **Label** text field.

*Slice 1*

1 Right-click **Transverse Stress in the Bolt Planes** and choose **Slice**.

2 In the **Settings** window for **Slice**, locate the **Expression** section.

3 In the **Expression** text field, type `solid.sz`.

4 From the **Unit** list, choose **MPa**.

5 Locate the **Plane Data** section. From the **Plane** list, choose **ZX-planes**.

6 From the **Entry method** list, choose **Coordinates**.

7 In the **Y-coordinates** text field, type `20 plateWidth-20`.

8 Click to expand the **Range** section. Select the **Manual color range** check box.

9 In the **Minimum** text field, type `-250`.

10 In the **Maximum** text field, type `250`.

*Selection 1*

1 Right-click **Slice 1** and choose **Selection**.

2 Select Domains 1 and 2 only.

3 In the **Transverse Stress in the Bolt Planes** toolbar, click **Plot**.

*Bolt Forces: Bolt Pretension 1 (Study 1) (solid)*

1 In the **Model Builder** window, under **Results** click **Bolt Forces: Bolt Pretension 1 (Study 1) (solid)**.

2 In the **Bolt Forces: Bolt Pretension 1 (Study 1) (solid)** toolbar, click **Evaluate**.

*Bolt Forces: Bolt Pretension 1 (Study 1) (beam)*

1 In the **Model Builder** window, click **Bolt Forces: Bolt Pretension 1 (Study 1) (beam)**.

2 In the **Bolt Forces: Bolt Pretension 1 (Study 1) (beam)** toolbar, click **Evaluate**.

## SOLID MECHANICS (SOLID)

Add the service load, a pressure inside the cavity.

In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

*Boundary Load 1*

1 In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

2 In the **Settings** window for **Boundary Load**, locate the **Force** section.

3 From the **Load type** list, choose **Pressure**.

4 In the *p* text field, type `4[MPa]*par`.

5 Select Boundaries 23–26, 31, 63–66, and 72 only.

## STUDY 1

In case you need to recompute the pretensioning step, the pressure load should not be active there.

### *Step 1: Bolt Pretension*

1 In the **Model Builder** window, under **Study 1** click **Step 1: Bolt Pretension**.

2 In the **Settings** window for **Bolt Pretension**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Boundary Load 1**.

5 Right-click and choose **Disable**.

## ROOT

Add a new study for the service load.

## ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

4 Right-click and choose **Add Study**.

## STUDY 2

### *Step 1: Stationary*

1 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

Study the effect of 20%, 50%, and 100% of the service load.

2 In the **Model Builder** window, under **Study 2** click **Step 1: Stationary**.

3 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

4 Select the **Auxiliary sweep** check box.

5 Click **Add**.

6 In the table, enter the following settings:

| Parameter name | Parameter value list | Parameter unit |
|---|---|---|
| par (Solution parameter) | `0.2 0.5 1` | |

Pick up the prestress solution from the previous study.

7 Click to expand the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.

8 From the **Method** list, choose **Solution**.

9 From the **Study** list, choose **Study 1, Bolt Pretension**.

10 Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

11 From the **Method** list, choose **Solution**.

12 From the **Study** list, choose **Study 1, Bolt Pretension**.

*Solution 2 (sol2)*

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

3 In the **Model Builder** window, expand the **Study 2>Solver Configurations> Solution 2 (sol2)>Dependent Variables 1** node, then click **Rotation field (material and geometry frames) (comp1.beam.thLin)**.

4 In the **Settings** window for **Field**, locate the **Scaling** section.

5 From the **Method** list, choose **Manual**.

6 In the **Scale** text field, type `0.01`.

7 In the **Model Builder** window, under **Study 2>Solver Configurations>Solution 2 (sol2)> Dependent Variables 1** click **Displacement field (material and geometry frames) (comp1.beam.uLin)**.

8 In the **Settings** window for **Field**, locate the **Scaling** section.

9 From the **Method** list, choose **Manual**.

10 In the **Scale** text field, type `0.001`.

11 In the **Model Builder** window, expand the **Study 2>Solver Configurations> Solution 2 (sol2)>Stationary Solver 1** node, then click **Advanced**.

12 In the **Settings** window for **Advanced**, locate the **Assembly Settings** section.

13 Clear the **Reuse sparsity pattern** check box.

**14** In the **Study** toolbar, click **Compute**.

## RESULTS

### *Bolt Stress and Contact Pressure, Service Load*

1 In the **Model Builder** window, right-click **Bolt Stress and Contact Pressure** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Bolt Stress and Contact Pressure, Service Load` in the **Label** text field.

   Examine the results for different load levels.

3 Click to expand the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 From the **Parameter value (par)** list, choose **0.2**.

5 In the **Bolt Stress and Contact Pressure, Service Load** toolbar, click **Plot**.

6 From the **Parameter value (par)** list, choose **0.5**.

7 In the **Bolt Stress and Contact Pressure, Service Load** toolbar, click **Plot**.

8 From the **Parameter value (par)** list, choose **1**.

9 In the **Bolt Stress and Contact Pressure, Service Load** toolbar, click **Plot**.

Evaluate the bolt forces.

### *Bolt Forces: Bolt Pretension 1 (Study 2) (solid)*

1 In the **Model Builder** window, click **Bolt Forces: Bolt Pretension 1 (Study 2) (solid)**.

2 In the **Bolt Forces: Bolt Pretension 1 (Study 2) (solid)** toolbar, click **Evaluate**.

### *Bolt Forces: Bolt Pretension 1 (Study 2) (beam)*

1 In the **Model Builder** window, click **Bolt Forces: Bolt Pretension 1 (Study 2) (beam)**.

2 In the **Bolt Forces: Bolt Pretension 1 (Study 2) (beam)** toolbar, click **Evaluate**.

### *Tightening Torque*

Evaluate the required tightening torque for full prestress.

1 In the **Results** toolbar, click **Evaluation Group**.

2 In the **Settings** window for **Evaluation Group**, type `Tightening Torque` in the **Label** text field.

### *Global Evaluation 1*

1 Right-click **Tightening Torque** and choose **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)> Solid Mechanics>Bolts>Bolt_1>solid.pblt1.sblt1.M_pre - Tightening torque - N·m**.

*Tightening Torque*

1 In the **Model Builder** window, click **Tightening Torque**.

2 In the **Settings** window for **Evaluation Group**, locate the **Data** section.

3 From the **Parameter selection (par)** list, choose **First**.

4 In the **Tightening Torque** toolbar, click **=** **Evaluate**.