Model created in COMSOL Multiphysics 6.4

# 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, 205 mm-by-40 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. Between the two plates, there is a 0.5 mm Nylon gasket. There is an internal cavity formed by matching imprints in the two plates. Due to symmetry, only half the geometry is modeled as 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.

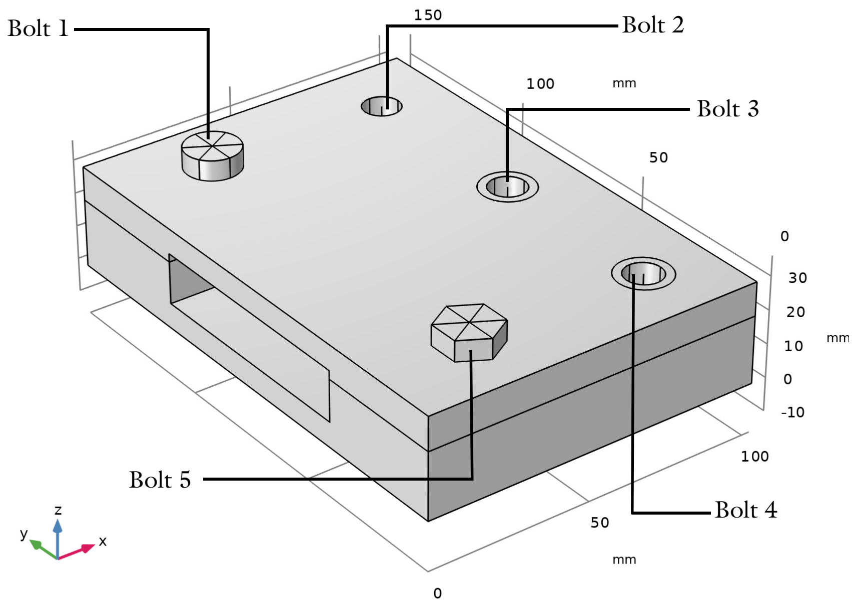



*Figure 1: The geometry of the component, and the bolt numbering.*

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 | Contact between head and upper plate | Internal thread in lower plate, modeled using Continuity between bolt and plate |
| Bolt 2 | 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 |
| 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 | 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 5 | Solid | Continuity between head and upper plate | Continuity between nut and upper plate |

The final prestress force in the bolts is set to $P$ = 30 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 ten 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 | 70% of P | Inactive |
| 5 | From solution | From solution | From solution | From solution | 70% 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 |
| 9 | From solution | From solution | From solution | 100% of P | From solution |
| 10 | From solution | From solution | From solution | From solution | 100% of P |

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.

A 0.5 mm nylon gasket is included between the two plates.

The service load is an internal pressure with a maximum value of 4 MPa.The pressure is applied between the two plates when they are no longer in contact.

## *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 | 21000 | 300 | 300 | 300 | 300 |
| 2 | 21185 | 21000 | 500 | 300 | 305 |
| 3 | 21215 | 21170 | 21000 | 545 | 325 |
| 4 | 21215 | 21165 | 21110 | 21000 | 620 |
| 5 | 21220 | 21170 | 21135 | 21115 | 21000 |
| 6 | 30000 | 21240 | 21145 | 21115 | 21000 |
| 7 | 30060 | 30000 | 21190 | 21115 | 21000 |
| 8 | 30075 | 30070 | 30000 | 21160 | 21010 |
| 9 | 30075 | 30065 | 30045 | 30000 | 21055 |
| 10 | 30075 | 30070 | 30055 | 30045 | 30000 |

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.

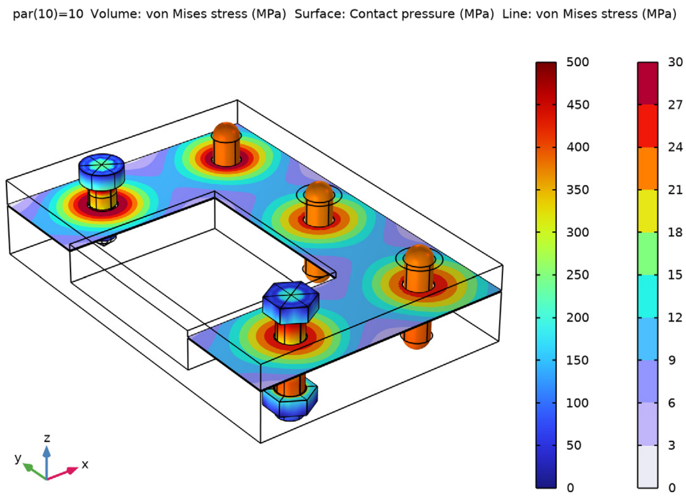



*Figure 2: Stresses in the bolts and contact pressure between the plates after the tightening sequence.*

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 the contact pressure is continuous between the bolts, this is because of the gasket. Without a gasket the contact pressure would only be applied in a circle within a diameter of two to three times the hole size causing risk of 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.

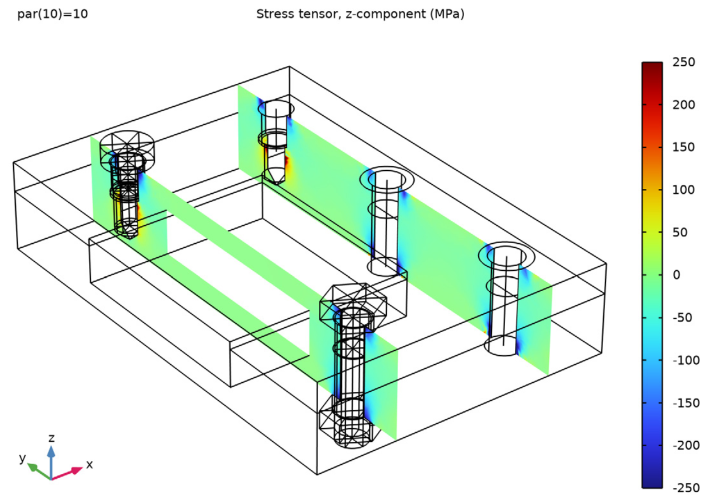



*Figure 3: Stress in the Z direction at the bolt holes after the tightening sequence.*

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.

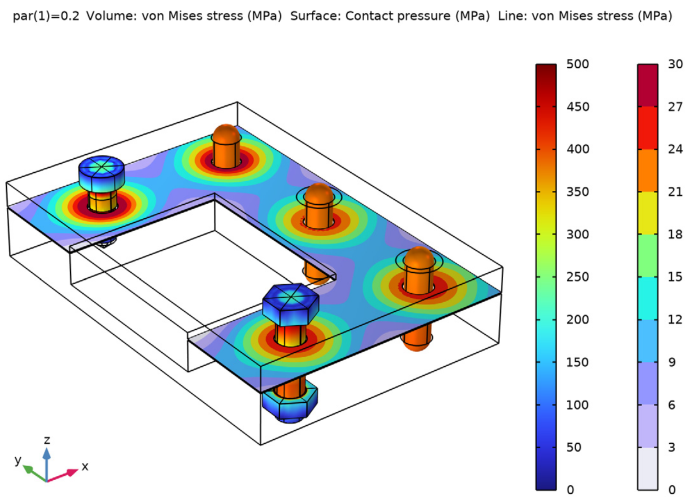



*Figure 4: Stresses in the bolts and contact pressure between the plates after applying 20% of the service load.*

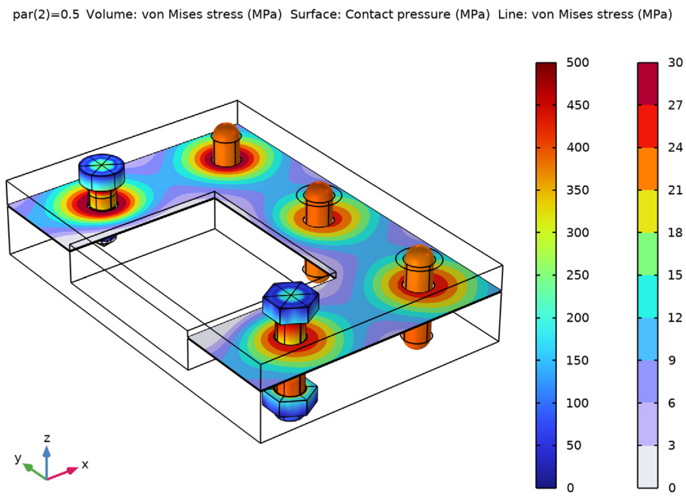



*Figure 5: Stresses in the bolts and contact pressure between the plates after applying 50% of the service load.*

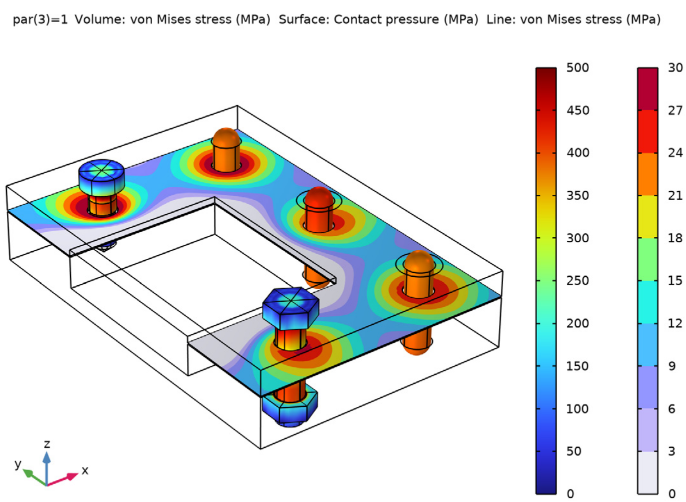



*Figure 6: Stresses in the bolts and contact pressure between the plates after applying 100% of the service load.*

In Figure 7 the applied pressure is shown. It is noticeable that there is for a 4 Mpa pressure there no leakage is observed.

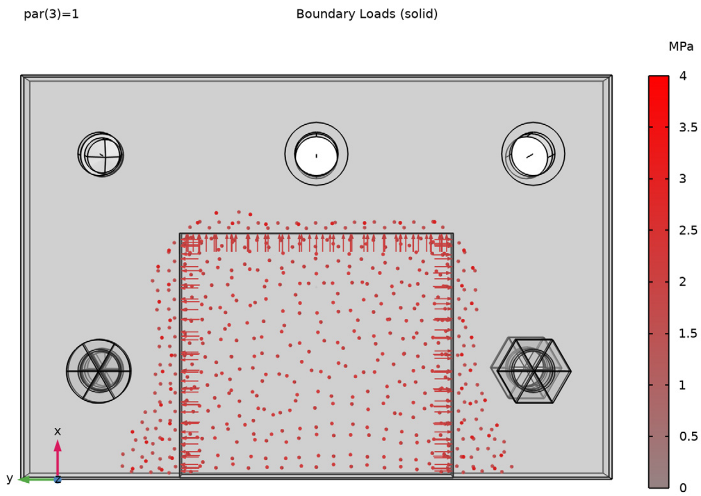



*Figure 7: Applied pressure at full service load.*

## *Note About the COMSOL Implementation*

When the service load is introduced in the second study, the problem becomes strongly nonlinear. As the plates lose contact because of the pressure, the loaded area increases and thus the plate deformation increases even more, until an equilibrium is reached. This causes high residual values in the intermediate nonlinear iterations that are difficult to handle for the automatic Newton solver. A more robust approach in this specific case is to use the constant Newton solver.

**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/2`.

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

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

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-contactWidth`.

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

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

7 Locate the **Position** section. In the **x** text field, type `0`.

8 In the **y** text field, type `contactWidth`.

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 `boltSpacing/2`.

6 In the **y** text field, type `contactWidth/2`.

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

8 Click **Build Selected**.

*Move 1 (mov1)*

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

2 Select the object **cyl1** only.

3 In the **Settings** window for **Move**, locate the **Input** section.

4 Select the **Keep input objects** checkbox.

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

6 Click **Build Selected**.

*Move 2 (mov2)*

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

2 Select the object **mov1** only.

3 In the **Settings** window for **Move**, locate the **Input** section.

4 Select the **Keep input objects** checkbox.

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

6 Click **Build Selected**.

## PART LIBRARIES

1 In the **Geometry** toolbar, click **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 **xwi** text field, type `boltSpacing/2`.

5 In the **ywi** text field, type `contactWidth/2`.

6 In the **zwi** 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 the first row in the table.

11 Click **New Cumulative Selection**.

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

13 Click **OK**.

### PART LIBRARIES

1 In the **Geometry** toolbar, click **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 |
| 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 **xwi** text field, type `boltSpacing/2`.

5 In the **ywi** text field, type `contactWidth/2`.

6 In the **zwi** 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 **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 |
| 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 **xwi** text field, type `boltSpacing/2`.

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

6 In the **zwi** 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 `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** checkbox.

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

6 Select the objects **blk3**, **copy1**, **cyl1**, **mov1**, **mov2**, 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. Click to select the **Activate Selection** toggle button for **Objects to add**.

4 Select the object **blk1** only.

5 Clear the **Keep objects to subtract** checkbox.

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 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 `3/2*boltSpacing`.

6 In the **yw** text field, type `contactWidth/2`.

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 **yw** 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) |
|---|---|---|
| 3/2*boltSpacing | contactWidth/2 | 0 |
| 3/2*boltSpacing | contactWidth/2 | 15 |
| 3/2*boltSpacing | contactWidth/2 | thicLow+thicUp |

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

5 In the **New Cumulative Selection** dialog, 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 **y** 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) |
|---|---|---|
| 3/2*boltSpacing | plateWidth-contactWidth/2 | thicLow-boltDia/2 |

| x (mm) | y (mm) | z (mm) |
|---|---|---|
| 3/2*boltSpacing | plateWidth-contactWidth/2 | thicLow+5 |
| 3/2*boltSpacing | plateWidth-contactWidth/2 | 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.

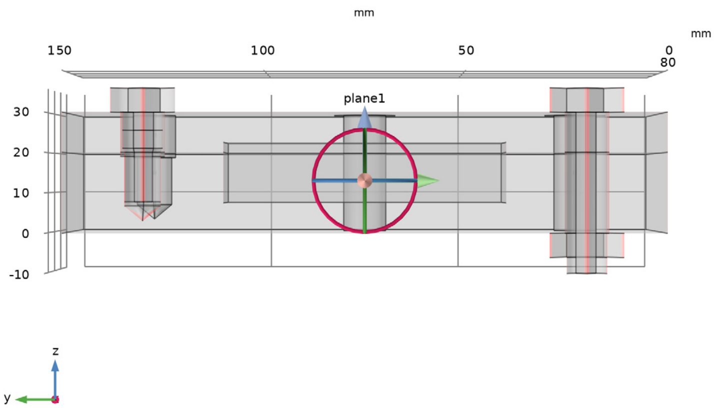
3 Find the **Definition** subsection. In the **x** text field, type `boltSpacing/2`.

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

5 Click the **Go to YZ View** button in the **Graphics** toolbar, then repeat the operation twice.

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

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




**8** 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 **Section type** list, choose **Circular**.

**4** 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 **Materials** 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 **Materials** 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**.

### SOLID MECHANICS (SOLID)

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

*RC Bolt 4, Head*

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

**2** In the **Settings** window for **Rigid Connector**, type `RC Bolt 4, Head` in the **Label** text field.

**3** Select Boundary 66 only.

*RC Bolt 4, Nut*

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

2 In the **Settings** window for **Rigid Connector**, type `RC Bolt 4, Nut` in the **Label** text field.

3 Select Edges 62, 63, 83, and 86 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 **Constraint Settings** section.

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

### 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 4, 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, select **Physics** > **Advanced Physics Options** in the tree.

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

5 Click **OK**.

6 In the **Settings** window for **Rigid Connector**, type `RC Bolt 4, 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 4, Head (solid)**.

*RC Bolt 4, Nut*

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

2 In the **Settings** window for **Rigid Connector**, type `RC Bolt 4, Nut` in the **Label** text field.

3 Select Point 245 only.

4 Locate the **Advanced** section. From the **Connect to** list, choose **RC Bolt 4, 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** checkbox.

4 Select Boundary 67 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 67, 68, 89, and 92 only.

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

7 Select Point 248 only.

*SBC, Bolt 2, 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 2, Head` in the **Label** text field.

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

4 Select Boundary 52 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 2, 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 2, Thread` in the **Label** text field.

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

**4** Select Boundaries 34, 35, 43, and 46 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 `forceScaling` in the **Function name** text field.

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

**4** In the **Expression** text field, type `if(time<active,0.01,if(time<(active+numBolt),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 |
| time | 1 |

*Analytic 2 (forceScaling2)*

1 Right-click **Analytic 1 (forceScaling)** 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-(active+numBolt))<0.001`.

## 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 Select the **Compute tightening torque** checkbox.

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

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

*Bolt Selection 1*

1 In the **Model Builder** window, 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 (Simple Bolt, With Drill 1)**.

4 Locate the **Bolt Pretension** section. From the **Pretension type** list, choose **Pretension force**.

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

6 Select the **Sequential tightening** checkbox.

7 In the **Pretensioning expression** text field, type `setPre(1,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_5`.

4 Locate the **Boundary Selection** section. From the **Selection** list, choose **Pretension cut (Hex Bolt, No Thread 1)**.

5 Locate the **Bolt Pretension** section. From the **Pretension type** list, choose **Pretension force**.

6 In the $F_p$ text field, type `forceScaling(5,par)*boltForce`.

7 Select the **Sequential tightening** checkbox.

8 In the **Pretensioning expression** text field, type `setPre(5,par)`.

### BEAM (BEAM)

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

*Bolt Pretension 1*

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

*Bolt Selection 1*

1 In the **Model Builder** window, 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 252 only.

5 Locate the **Bolt Pretension** section. From the **Pretension type** list, choose **Pretension force**.

6 In the $F_p$ text field, type `forceScaling(2,par)*boltForce`.

7 Select the **Sequential tightening** checkbox.

8 In the **Pretensioning expression** text field, type `setPre(2,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 $F_p$ text field, type `forceScaling(3,par)*boltForce`.

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

*Bolt Selection 3*

1 Right-click **Bolt Selection 2** and choose **Duplicate**.

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

3 In the text field, type `Bolt_4`.

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

5 Select Point 246 only.

6 Locate the **Bolt Pretension** section. In the $F_p$ text field, type `forceScaling(4,par)*boltForce`.

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

### DEFINITIONS

*Identity Boundary Pair 1 (ap1)*

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Definitions** click **Identity Boundary Pair 1 (ap1)**.

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

3 Select the **Manual control of selections and pair type** checkbox.

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 5 (ap5)*

1 In the **Model Builder** window, click **Identity Boundary Pair 5 (ap5)**.

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

3 Select the **Manual control of selections and pair type** checkbox.

4 From the **Pair type** list, choose **Contact pair**.

5 Locate the **Advanced** section. From the **Mapping method** list, choose **Initial configuration**.

### SOLID MECHANICS (SOLID)

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

*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`.

*Fixed Constraint 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 **Points** and choose **Fixed Constraint**.

2 Select Point 1 only.

*Prescribed Displacement 1*

1 In the **Physics** toolbar, click **Points** and choose **Prescribed Displacement**.

2 Select Point 7 only.

3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

4 From the **Displacement in z direction** list, choose **Prescribed**.

*Symmetry 1*

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

2 Select Boundaries 1 and 49 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 From the **Formulation** list, choose **Geometrically linear**.

## BEAM (BEAM)

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** checkbox.

## GLOBAL DEFINITIONS

Add a gasket to enforce a continuous contact pressure between the bolt, and so reduce the risk of leakage.

#### ADD MATERIAL

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

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

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

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

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

#### MATERIALS

*Material Link 3 (matlnk3)*

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

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

3 From the **Material** list, choose **Nylon (mat2)**.

4 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

5 Select Boundary 51 only.

#### SOLID MECHANICS (SOLID)

*Gasket*

1 In the **Physics** toolbar, click **Boundaries** and choose **Thin Layer**.

2 In the **Settings** window for **Thin Layer**, type `Gasket` in the **Label** text field.

3 Select Boundary 51 only.

4 Locate the **Boundary Properties** section. In the $L_{th}$ text field, type `5e-4[m]`.

#### 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 **Geometric Entity Selection** section.

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

4 Select Boundary 51 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** checkbox. 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 84, 85, 90, 97, 102, and 107 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 1.

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

6 Click **Build All**.

*Edge 1*

1 In the **Mesh** toolbar, click **More Generators** 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** checkbox.

4 Click **Add**.

5 In the table, enter the following settings:

| Parameter name | Parameter value list | Parameter unit |
|---|---|---|
| par (Solution parameter) | range(1,1,10) | |

6 In the **Model Builder** window, click **Study 1**.

7 In the **Settings** window for **Study**, locate the **Study Settings** section.

8 Clear the **Generate default plots** checkbox.

*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** checkbox.

*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** checkbox.

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** checkbox.

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** checkbox.

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

## RESULTS

Set up a suitable plot for monitoring the solution process.

*Preferred Units 1*

1 In the **Results** toolbar, click **Configurations** and choose **Preferred Units**.
2 In the **Settings** window for **Preferred Units**, locate the **Units** section.
3 Click **Add Physical Quantity**.
4 In the **Physical Quantity** dialog, type `stress` in the text field.
5 In the tree, select **Solid Mechanics** > **Stress tensor (N/m^2)**.
6 Click **OK**.
7 In the **Settings** window for **Preferred Units**, locate the **Units** section.
8 In the table, enter the following settings:

| Quantity | Unit | Preferred unit |
|---|---|---|
| Stress tensor | N/m^2 | MPa |

9 Select the **Apply conversions to expressions with the same dimensions** checkbox.
10 Click **Apply**.

## RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.
2 Go to the **Result Templates** window.
3 In the tree, select **Study 1/Solution 1 (sol1)** > **Solid Mechanics (solid)** > **Stress (solid)**.
4 Click the **Add Result Template** button in the window toolbar.
5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

## RESULTS

*Bolt Stress and Contact Pressure*

1 In the **Model Builder** window, expand the **Results** > **Stress (solid)** node, then click **Stress (solid)**.
2 In the **Settings** window for **3D Plot Group**, type `Bolt Stress and Contact Pressure` in the **Label** text field.

*Volume 1*

1 In the **Model Builder** window, expand the **Results** > **Bolt Stress and Contact Pressure** > **Volume 1** node, then click **Volume 1**.
2 In the **Settings** window for **Volume**, click to expand the **Range** section.
3 Select the **Manual color range** checkbox.

4 In the **Maximum** text field, type 500.

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **Rainbow**.

### *Selection 1*

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

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

3 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 **Coloring and Style** section. From the **Color table** list, choose **Prism**.

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

5 In the **Maximum** text field, type 30.

6 Locate the **Coloring and Style** section. From the **Color table type** list, choose **Discrete**.

### *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** checkbox.

### *Line 1*

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

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

3 In the **Expression** text field, type beam.mises.

4 Locate the **Coloring and Style** section. From the **Line type** list, choose **Tube**.

5 In the **Tube radius expression** text field, type beam.re.

6 Select the **Radius scale factor** checkbox.

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

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

### *Deformation 1*

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

2 In the **Settings** window for **Deformation**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Beam** > **Displacement** > **u2,v2,w2 - Displacement field**.

### 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 **Results** toolbar, click **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 Locate the **Plane Data** section. From the **Entry method** list, choose **Coordinates**.

5 In the **X-coordinates** text field, type `boltSpacing/2 3*boltSpacing/2`.

6 Click to expand the **Range** section. Select the **Manual color range** checkbox.

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

8 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 Force*

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

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

*Global Evaluation 1*

1 Right-click **Bolt Force** and choose **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, click **Add 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.F_bolt - Bolt force - N**.

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

| Expression | Unit | Description |
|---|---|---|
| solid.pblt1.sblt1.F_bolt | N | Bolt 1 |

4 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Beam** > **Bolts** > **Bolt_2** > **beam.pblt1.sblt1.F_bolt - Bolt force - N**.

5 Locate the **Expressions** section. In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| beam.pblt1.sblt1.F_bolt | N | Bolt 2 |

6 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Beam** > **Bolts** > **Bolt_3** > **beam.pblt1.sblt2.F_bolt - Bolt force - N**.

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

| Expression | Unit | Description |
|---|---|---|
| beam.pblt1.sblt2.F_bolt | N | Bolt 3 |

8 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Beam** > **Bolts** > **Bolt_4** > **beam.pblt1.sblt3.F_bolt - Bolt force - N**.

9 Locate the **Expressions** section. In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| beam.pblt1.sblt3.F_bolt | N | Bolt 4 |

10 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics** > **Bolts** > **Bolt_5** > **solid.pblt1.sblt2.F_bolt - Bolt force - N**.

11 Locate the **Expressions** section. In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| solid.pblt1.sblt2.F_bolt | N | Bolt 5 |

12 In the **Bolt Force** 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.

3 Locate the **Data** section. From the **Parameter selection (par)** list, choose **Last**.

*Global Evaluation 1*

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

2 In the **Settings** window for **Global Evaluation**, click **Add 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**.

3 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics** > **Bolts** > **Bolt_5** > **solid.pblt1.sblt2.M_pre - Tightening torque - N·m**.

*Tightening Torque*

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

2 In the **Tightening Torque** toolbar, click **Evaluate**.

## SOLID MECHANICS (SOLID)

Add the service load, a pressure inside the cavity.

*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 4–7, 26, 51, 53–55, and 65 only.

*Contact 1*

Now adjust the contact detection tolerance to apply fallback condition when the gap distance is lower than 1 um.

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

2 In the **Settings** window for **Contact**, click to expand the **Advanced** section.

3 From the **Multiphysics contact detection** list, choose **Manual**.

4 In the $\Delta_{contact}$ text field, type `1[um]`.

### 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** checkbox.

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**.

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

### STUDY 2

#### *Step 1: Stationary*

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

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

2 Select the **Auxiliary sweep** checkbox.

3 Click **Add**.

**4** 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.

**5** 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**.

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

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

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

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

**10** 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** checkbox.

14 In the **Model Builder** window, under **Study 2** > **Solver Configurations** > **Solution 2 (sol2)** > **Stationary Solver 1** click **Fully Coupled 1**.

15 In the **Settings** window for **Fully Coupled**, locate the **Method and Termination** section.

16 From the **Nonlinear method** list, choose **Constant (Newton)**.

17 In the **Model Builder** window, click **Study 2**.

18 In the **Settings** window for **Study**, locate the **Study Settings** section.

19 Clear the **Generate default plots** checkbox.

20 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 Locate 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**.

## RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 2/Solution 2 (sol2)** > **Solid Mechanics** > **Applied Loads (solid)** > **Boundary Loads (solid)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

## RESULTS

*Boundary Loads (solid)*

1 Click the **Go to XY View** button in the **Graphics** toolbar.

2 Click the **Rotate Left 90°** button in the **Graphics** toolbar.

Evaluate the bolt shear forces with respect to the service load.

*Bolt Shear Force (service load)*

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

2 In the **Settings** window for **Evaluation Group**, type `Bolt Shear Force (service load)` in the **Label** text field.

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

*Global Evaluation 1*

1 Right-click **Bolt Shear Force (service load)** and choose **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, click **Add 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.F_shear - Bolt shear force - N**.

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

| **Expression** | **Unit** | **Description** |
|---|---|---|
| `solid.pblt1.sblt1.F_shear` | N | Bolt 1 |

4 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Beam** > **Bolts** > **Bolt_2** > **beam.pblt1.sblt1.F_shear - Bolt shear force - N**.

5 Locate the **Expressions** section. In the table, enter the following settings:

| **Expression** | **Unit** | **Description** |
|---|---|---|
| `beam.pblt1.sblt1.F_shear` | N | Bolt 2 |

6 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Beam** > **Bolts** > **Bolt_3** > **beam.pblt1.sblt2.F_shear - Bolt shear force - N**.

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

| **Expression** | **Unit** | **Description** |
|---|---|---|
| `beam.pblt1.sblt2.F_shear` | N | Bolt 3 |

8 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Beam** > **Bolts** > **Bolt_4** > **beam.pblt1.sblt3.F_shear - Bolt shear force - N**.

9 Locate the **Expressions** section. In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| beam.pblt1.sblt3.F_shear | N | Bolt 4 |

10 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics** > **Bolts** > **Bolt_5** > **solid.pblt1.sblt2.F_shear - Bolt shear force - N**.

11 Locate the **Expressions** section. In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| solid.pblt1.sblt2.F_shear | N | Bolt 5 |

12 In the **Bolt Shear Force (service load)** toolbar, click **Evaluate**.