Created in COMSOL Multiphysics 5.6

Prestressed Bolts in a Tube Connection

Introduction

A tube connection consisting of a flange with eight prestressed bolts as shown Figure 1 is subjected to a set of loads consisting of an internal pressure, an axial force, and an external bending moment.

In this example, you will apply the preload to the bolts in an initial study step. You will then study how the stress state in the tube and the bolts varies with the applied load.

You will also create a number of stress classification lines, along which a stress linearization is performed.

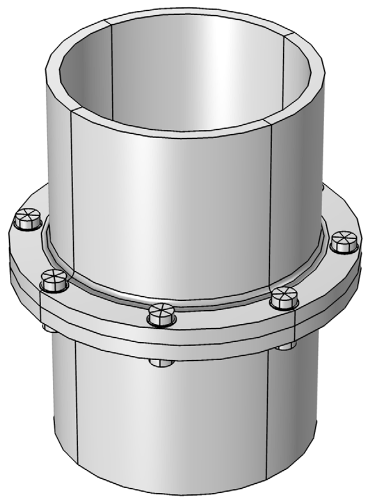


Figure 1: Tube connection.

Model Definition

The tube is made of steel and has an inner diameter of 360 mm and thickness of 20 mm. The flange has an outer diameter of 520 mm and a thickness of 26 mm. At the transition from the pipe to the flange there is a fillet with radius 12 mm.

Eight M24 bolts are used to connect the flanges. Each bolt is prestressed by 150 kN.

The load consists of:

- A constant internal pressure of 30 bar
- A bending moment on the pipe which varies from 0 to 40 kNm
- A constant axial force of 500 kN

Because of the symmetry in both load and geometry, you only need to analyze one half of one of the flanges. As symmetry conditions cannot be applied on the contacting surfaces, the lower half of the model is replaced by a rigid plane.

Two contact regions are modeled. One contact pair acts between the bottom surface of the flange and the rigid plane which supplies the symmetry condition with respect to contact. The other contact pair acts between the washers under the bolt heads and the flange. The possibility to automatically detect potential contact surfaces is used when creating the contact pairs. To improve the accuracy of the contact between the washers and the flange, the augmented Lagrangian contact method is used; the first contact uses the default penalty method.

Results and Discussion

After the pretension step, there is a tensile stress in the bolt, and compressive stress in the flange under the bolt. This is illustrated in Figure 2. The external loads applied after in the second step are visualized in Figure 3.

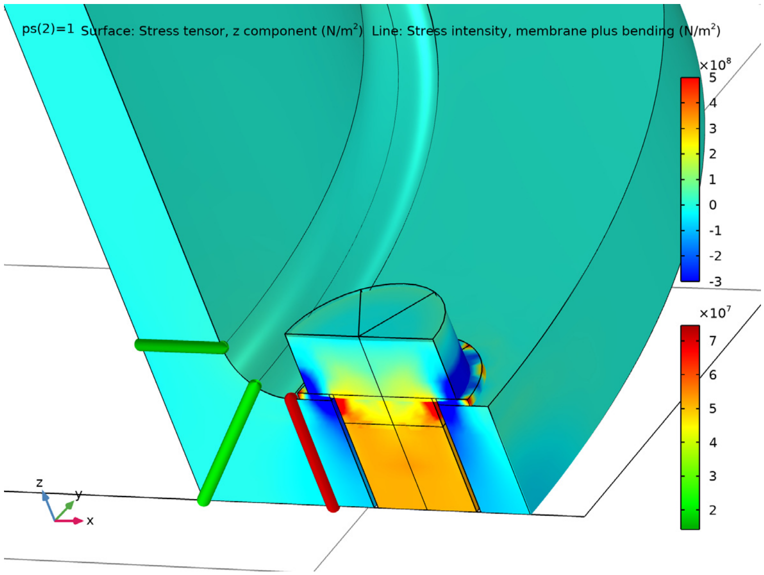


Figure 2: The axial stress after the pretension step.

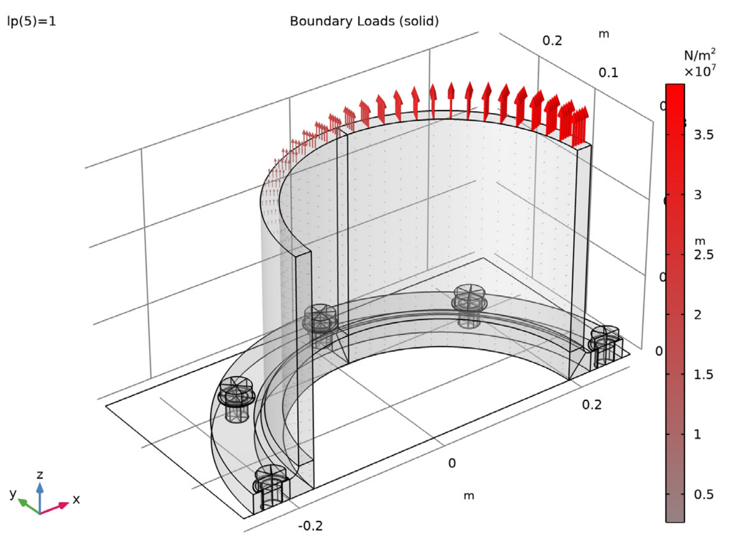


Figure 3: Applied external loads.

The stress state at maximum external load is shown in Figure 4. The stress state on the inner side of the bolt has increased significantly, and the stress is no longer uniform. Furthermore, a stress of the order of 300 MPa has developed in the fillet between the tube and the flange. This stress is caused by local bending, since the flange no longer is in contact with the mating surface at the tensile side.

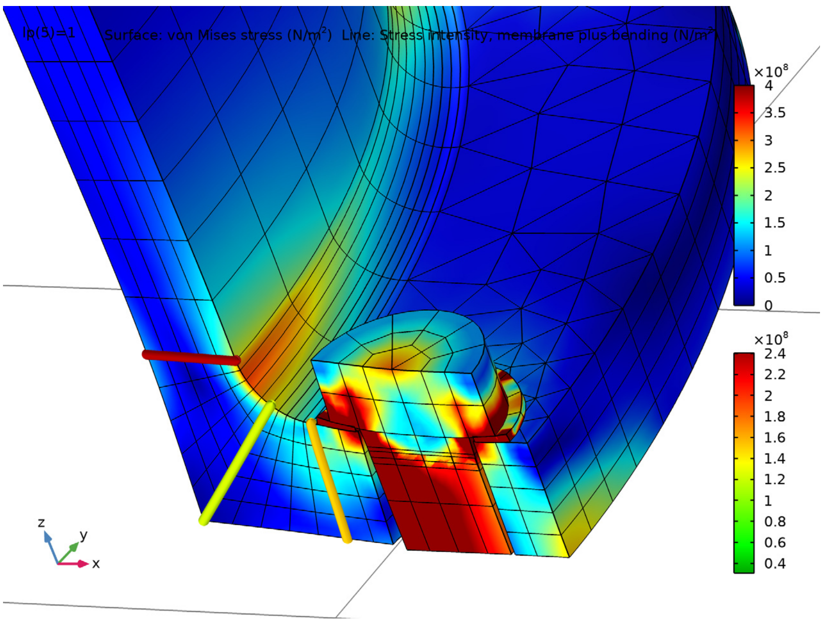


Figure 4: Equivalent stress at the maximum external load.

The applied external load in this example makes the bolted joint excessively loaded. This is displayed in Figure 5, where the bolt forces become significantly uneven. Actually, the conditions are even worse than what the average force indicates. The bolt is subjected to bending with a non-uniform stress distribution over the cross section. The maximum stress has increased from the prestress value and the progression is fast. The development of the axial stress in two points on opposite sides of the bolt is displayed in Figure 6. The points are located in the *xz*-symmetry plane. One point is as close to the tube centerline as possible and the other is as far out as possible.

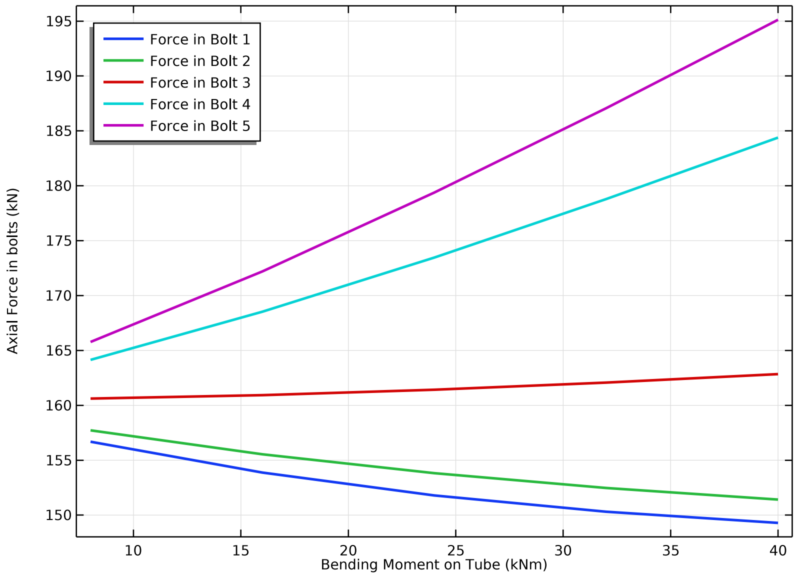


Figure 5: The bolt force as a function of the tensile force.

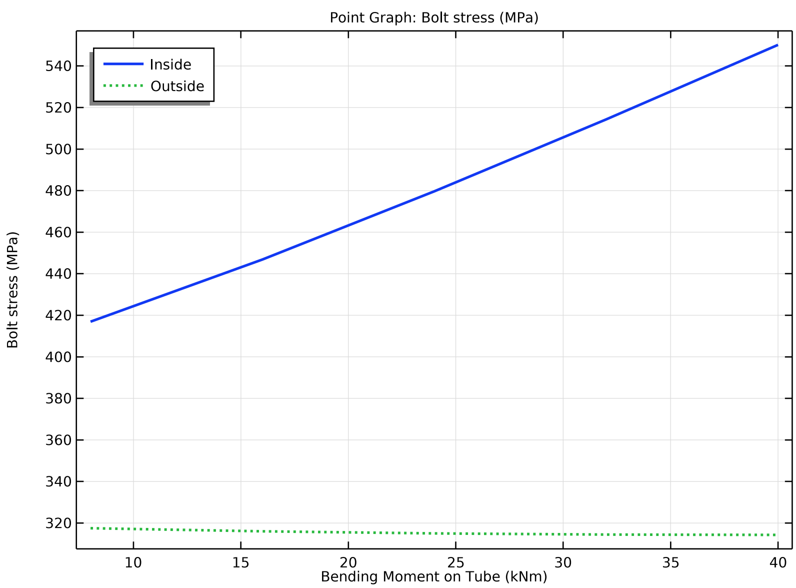


Figure 6: The development of the bolt stress at two different positions in the cross section.

The plots of the contact pressure between the mating flanges are shown in Figure 7 and Figure 8. By comparing these two figures, it is clear that the contact pressure shifts away from the initially prestressed area, which at the peak load becomes almost stress free. This observation indicates that there are too few bolts that connect the two parts with each other.

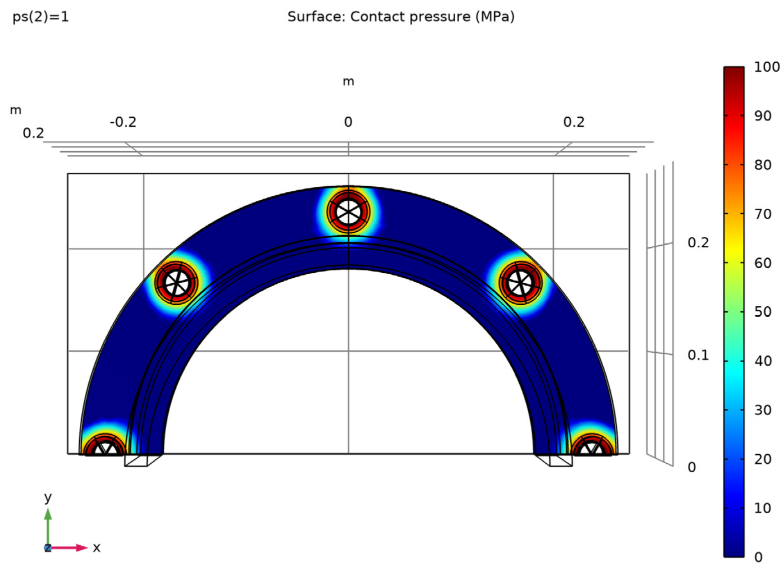


Figure 7: Contact pressure between the flanges after pretensioning the bolts.

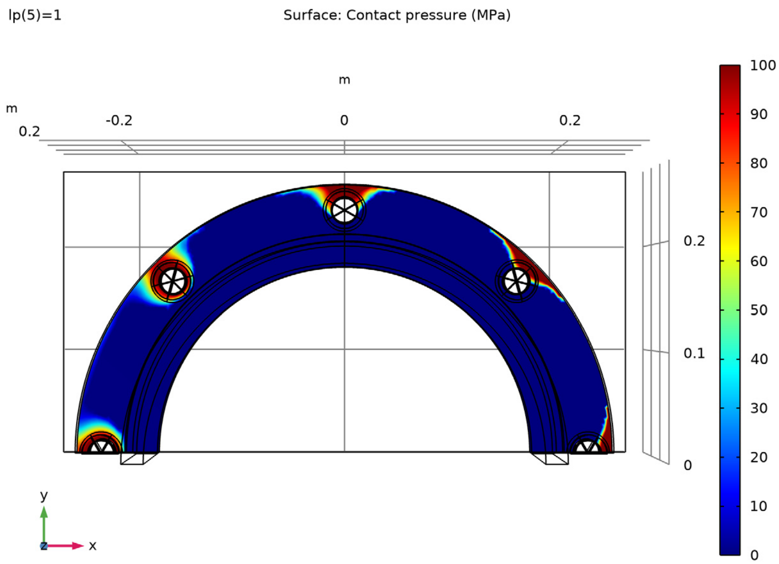


Figure 8: Contact pressure between the flanges at full external load.

The linearized stresses along the six stress classification lines are shown in Figure 9 to Figure 14. The stress component 22 is, with the chosen orientation, the direct stress in the *xz*-plane. The hoop stress would be component 33.

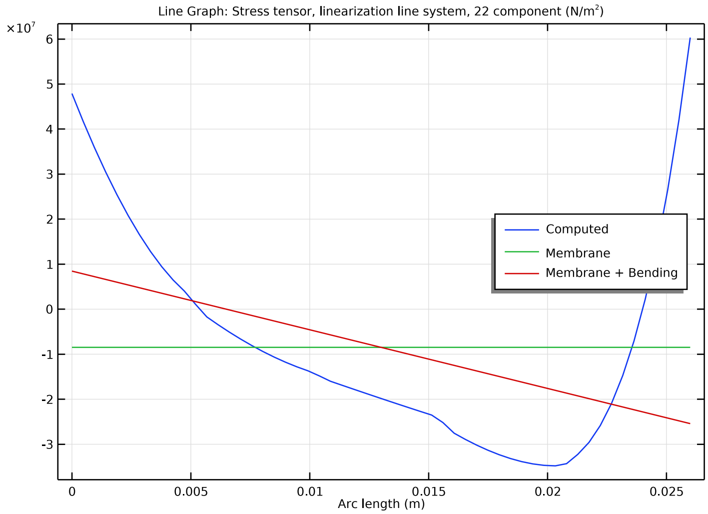


Figure 9: Linearized stresses (hoop direction) along stress classification line 1.

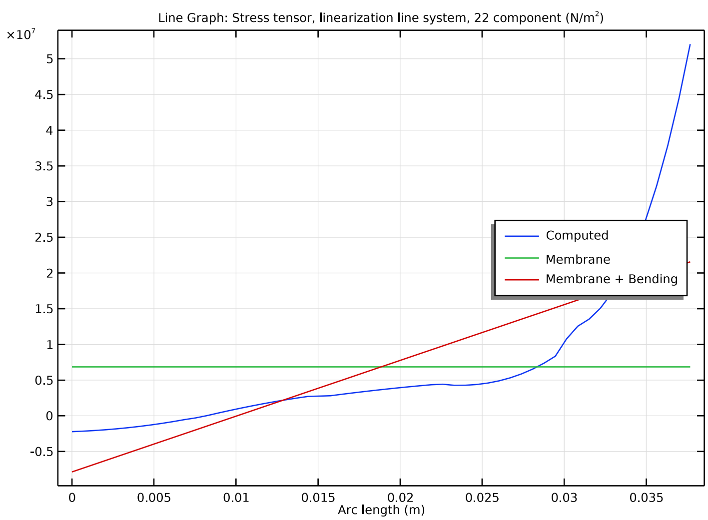


Figure 10: Linearized stresses (hoop direction) along stress classification line 2.

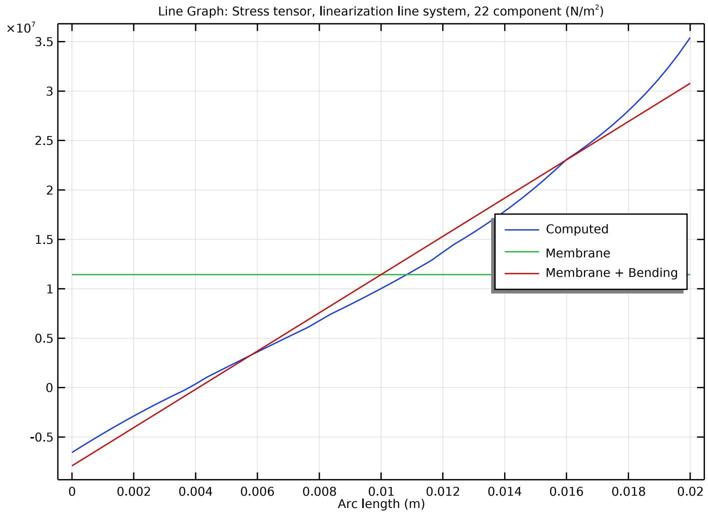


Figure 11: Linearized stresses (hoop direction) along stress classification line 3.

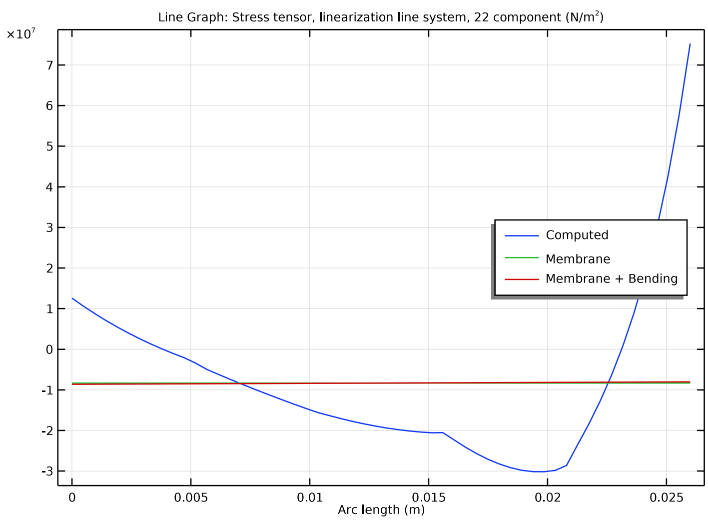


Figure 12: Linearized stresses (hoop direction) along stress classification line 4.

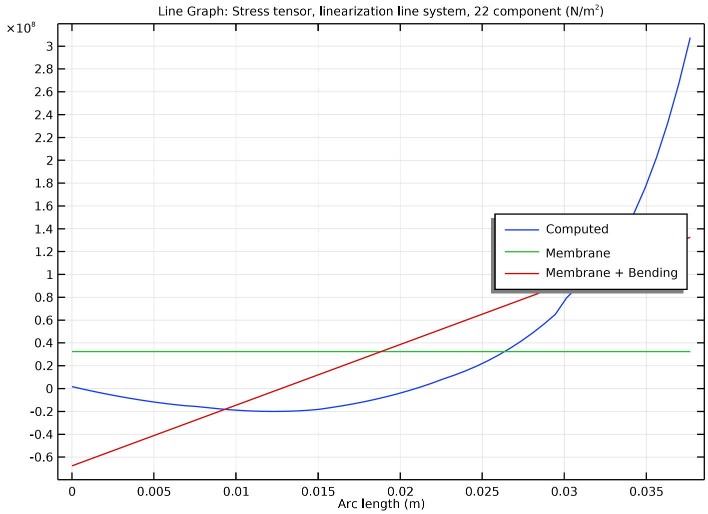


Figure 13: Linearized stresses (hoop direction) along stress classification line 5

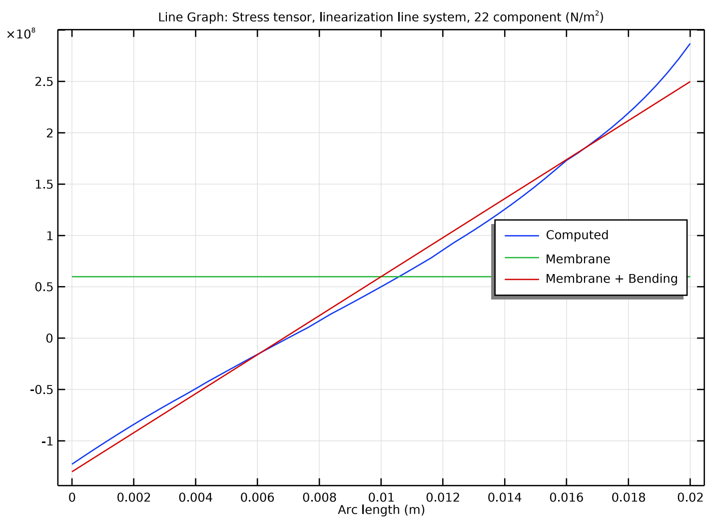


Figure 14: Linearized stresses (hoop direction) along stress classification line 6.

Notes About the COMSOL Implementation

The analysis is performed in two steps. In the first step, the effects of pretensioning the bolts are computed, and in the second step, the external load on the tube is applied as a parametric sweep.

The prestress in the bolts is introduced using the built-in **Bolt Pretension** node. This feature creates one degree of freedom for each bolt, which can be interpreted as the elongation of the bolt caused by the prestress. This degree of freedom is then kept fixed when the service loads are applied.

In order to keep the solution time down, a coarse mesh is used, and it probably needs a refinement if accurate and quantitative stress results are required.

Application Library path: `Structural_Mechanics_Module/Contact_and_Friction/tube_connection`

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)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies>Stationary**.
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 `tube_connection_parameters.txt`.

GEOMETRY 1

Insert the geometry sequence from the `tube_connection_geom_sequence.mph` file.

1 In the **Geometry** toolbar, click **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `tube_connection_geom_sequence.mph`.

Form Union (fin)

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

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

5 Click **Build Selected**.

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

Inspect the contact pair selections and adjust them if necessary. One of the automatically generated contact pairs is not used.

DEFINITIONS

Contact Pair 1 (ap1)

Small sliding is expected so you can speed up the computation by changing the mapping method.

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node, then click **Contact Pair 1 (ap1)**.

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

3 From the **Mapping method** list, choose **Initial configuration**.

Contact Pair 2 (ap2)

This contact pair is only used to collect the bolt boundaries at the symmetry plane, which correspond to the destination boundaries.

1 In the **Model Builder** window, click **Contact Pair 2 (ap2)**.

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

3 Click **Create Selection**.

4 In the **Create Selection** dialog box, type `bolts_symmetry` in the **Selection name** text field.

5 Click **OK**.

Contact Pair 3 (ap3)

1 In the **Model Builder** window, click **Contact Pair 3 (ap3)**.

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

3 From the **Mapping method** list, choose **Initial configuration**.

4 Click the **Swap Source and Destination** button.

Contact Pair 4 (ap4)

Assume a bonded contact between the washers and the bolt heads.

1 In the **Model Builder** window, click **Contact Pair 4 (ap4)**.

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 **Identity pair**.

5 Locate the **Frame** section. From the **Source frame** list, choose **Material (X, Y, Z)**.

6 From the **Destination frame** list, choose **Material (X, Y, Z)**.

COMPONENT 1 (COMP1)

Make sure that a well defined boundary tangents are available for the contact with friction.

DEFINITIONS

Cylindrical System 2 (sys2)

In the **Definitions** toolbar, click **Coordinate Systems** and choose **Cylindrical System**.

Boundary System 1 (sys1)

1 In the **Model Builder** window, click **Boundary System 1 (sys1)**.

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

3 Find the **Coordinate names** subsection. From the **Create first tangent direction from** list, choose **Cylindrical System 2 (sys2)**.

4 From the **Axis** list, choose **r**.

Define selections to use later in the modeling.

Symmetry boundaries (xz-plan)

1 In the **Definitions** toolbar, click **Box**.
2 In the **Settings** window for **Box**, type `Symmetry boundaries (xz-plan)` in the **Label** text field.
3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.
4 Locate the **Box Limits** section. In the **y maximum** text field, type `0`.
5 In the **z minimum** text field, type `0`.
6 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Symmetry

1 In the **Definitions** toolbar, click **Union**.
2 In the **Settings** window for **Union**, locate the **Geometric Entity Level** section.
3 From the **Level** list, choose **Boundary**.
4 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.
5 In the **Add** dialog box, in the **Selections to add** list, choose **bolts_symmetry** and **Symmetry boundaries (xz-plan)**.
6 Click **OK**.
7 In the **Settings** window for **Union**, type `Symmetry` in the **Label** text field.

Now create a selection for each bolt pretension cut.

Bolt_pretension_cut1

1 In the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for **Explicit**, type `Bolt_pretension_cut1` in the **Label** text field.
3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundary 165 only.
5 Select the **Group by continuous tangent** check box.

Bolt_pretension_cut2

1 In the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for **Explicit**, type `Bolt_pretension_cut2` in the **Label** text field.
3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundary 211 only.
5 Select the **Group by continuous tangent** check box.

Bolt_pretension_cut3

1 In the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for **Explicit**, type `Bolt_pretension_cut3` in the **Label** text field.
3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundary 277 only.
5 Select the **Group by continuous tangent** check box.

Bolt_pretension_cut4

1 In the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for **Explicit**, type `Bolt_pretension_cut4` in the **Label** text field.
3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundary 343 only.
5 Select the **Group by continuous tangent** check box.

Bolt_pretension_cut5

1 In the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for **Explicit**, type `Bolt_pretension_cut5` in the **Label** text field.
3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundary 399 only.
5 Select the **Group by continuous tangent** check box.

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

SOLID MECHANICS (SOLID)

Pressure

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Boundary Load**.
2 In the **Settings** window for **Boundary Load**, type `Pressure` in the **Label** text field.
3 Select Boundaries 21, 22, 30, and 33 only.
4 Locate the **Force** section. From the **Load type** list, choose **Pressure**.

5 In the p text field, type `pressure`.

Bending moment and axial force

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

2 In the **Settings** window for **Boundary Load**, type `Bending moment and axial force` in the **Label** text field.

3 Select Boundaries 20 and 35 only.

4 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

0	x
0	y
lp*bendingStress*X/(pipeOuterDiameter/2)+axialStress	z

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_p text field, type `boltPrestressForce`.

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 **Bolt_pretension_cut1**.

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 Selection** section.

3 In the **Bolt label** text field, type `Bolt_2`.

4 Locate the **Boundary Selection** section. From the **Selection** list, choose **Bolt_pretension_cut2**.

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

3 In the **Bolt label** text field, type `Bolt_3`.

4 Locate the **Boundary Selection** section. From the **Selection** list, choose **Bolt_pretension_cut3**.

Bolt Pretension 1

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

Bolt Selection 4

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

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

3 In the **Bolt label** text field, type `Bolt_4`.

4 Locate the **Boundary Selection** section. From the **Selection** list, choose **Bolt_pretension_cut4**.

Bolt Pretension 1

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

Bolt Selection 5

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

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

3 In the **Bolt label** text field, type `Bolt_5`.

4 Locate the **Boundary Selection** section. From the **Selection** list, choose **Bolt_pretension_cut5**.

Spring Foundation 1

1 In the **Physics** toolbar, click **Domains** and choose **Spring Foundation**.

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

3 From the **Selection** list, choose **All domains**.

4 Locate the **Spring** section. From the **Spring type** list, choose **Total spring constant**.

5 In the $\mathbf{k}_{\mathrm{tot}}$ text field, type `1e10*(1-ps)`.

Contact 1

1 In the **Physics** toolbar, click **Pairs** and choose **Contact**.

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

3 Under **Pairs**, click **Add**.

4 In the **Add** dialog box, select **Contact Pair 1 (ap1)** in the **Pairs** list.

5 Click **OK**.

6 In the **Settings** window for **Contact**, locate the **Contact Surface** section.

7 Select the **Source external to current physics** check box.

Contact 2

1 In the **Physics** toolbar, click **Pairs** and choose **Contact**.

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

3 Under **Pairs**, click **Add**.

4 In the **Add** dialog box, select **Contact Pair 3 (ap3)** in the **Pairs** list.

5 Click **OK**.

Use the augmented Lagrangian method to improve the accuracy of the contact forces from the bolts.

6 In the **Settings** window for **Contact**, locate the **Contact Method** section.

7 From the **Formulation** list, choose **Augmented Lagrangian**.

8 Locate the **Contact Pressure Penalty Factor** section. From the **Tuned for** list, choose **Speed**.

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`.

Continuity 1

1 In the **Physics** toolbar, click **Pairs** and choose **Continuity**.

Bonded contact is modeled using a continuity pair condition.

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

3 Under **Pairs**, click **Add**.

4 In the **Add** dialog box, select **Identity Boundary Pair 4 (ap4)** in the **Pairs** list.

5 Click **OK**.

Symmetry 1

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

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

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

Add a constraint to suppress the rigid body motion in the x direction.

Prescribed Displacement 1

1 In the **Physics** toolbar, click **Points** and choose **Prescribed Displacement**.
2 Select Point 35 only.
3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.
4 Select the **Prescribed in x direction** check box.

Add **Stress Linearization** nodes to describe the stress classification lines.

Stress Linearization 1

1 In the **Physics** toolbar, click **Global** and choose **Stress Linearization**.
2 Select Edge 22 only.
3 In the **Settings** window for **Stress Linearization**, locate the **Second Axis Orientation Reference Point** section.
4 Select the **Activate Selection** toggle button.
5 Select Point 1 only.
6 Repeat the previous steps to add **Stress Linearization** node for the edges 33, 36, 107, 110 and 115.

MESH 1

Swept 1

1 In the **Mesh** toolbar, click **Swept**.
2 In the **Settings** window for **Swept**, locate the **Domain Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 From the **Selection** list, choose **Washers**.

   The generation of the swept mesh for the half washer domains is not unique. Specify the source boundaries to control the sweep direction.

5 Click to expand the **Source Faces** section. Select Boundaries 59, 63, 67, 145, 149, and 153 only.

Size 1

1 Right-click **Swept 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**.

Swept 2

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

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

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

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

Size 1

1 Right-click **Swept 2** and choose **Size**.

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

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

Distribution 1

1 In the **Model Builder** window, right-click **Swept 2** and choose **Distribution**.

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

3 In the **Number of elements** text field, type 2.

4 Click **Build Selected**.

Mapped 1

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

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

3 Select Boundaries 10, 14, and 17 only.

Distribution 1

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

2 Select Edges 25, 27, and 33 only.

Distribution 2

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

2 Select Edge 30 only.

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

4 In the **Number of elements** text field, type 20.

5 Click **Build Selected**.

Swept 3

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

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

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

4 Select Domains 2–4 only.

Distribution 1

1 Right-click **Swept 3** and choose **Distribution**.

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

3 In the **Number of elements** text field, type 30.

4 Click **Build Selected**.

Free Triangular 1

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

2 Select Boundaries 5 and 41 only.

Size 1

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

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

3 From the **Predefined** list, choose **Extra fine**.

4 Click **Build Selected**.

Swept 4

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

2 In the **Settings** window for **Swept**, click **Build All**.

Mapped 2

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

2 Select Boundary 1 only.

Distribution 1

1 Right-click **Mapped 2** and choose **Distribution**.

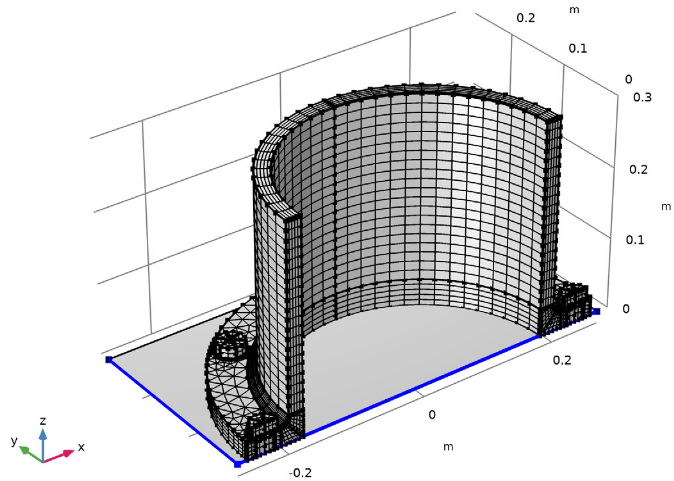
2 Select Edges 1 and 2 only.

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

4 In the **Number of elements** text field, type 1.

5 Click **Build All**.

The mesh should now look as in the figure below.



STUDY 1

The bolt pretension has to be computed first in a separate study step.

Bolt Pretension

1 In the **Study** toolbar, click **Study Steps** and choose **Stationary>Bolt Pretension**.

2 Right-click **Study 1>Step 2: Bolt Pretension** and choose **Move Up**.

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

Disable the external loads during the prestress load step.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

5 In the **Physics and variables selection** tree, select **Component 1 (comp1)> Solid Mechanics (solid), Controls spatial frame>Pressure**.

6 Click **Disable**.

7 In the **Physics and variables selection** tree, select **Component 1 (comp1)> Solid Mechanics (solid), Controls spatial frame>Bending moment and axial force**.

8 Click **Disable**.

9 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

10 Click Add.

11 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
ps (Parameter for spring relaxation)	`0 1`	

Step 2: Stationary

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

2 In the **Settings** window for **Stationary**, 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
lp (Loading parameter)	`range(0.2,0.2,1)`	

Solution 1 (sol1)

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

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Dependent Variables 1** node, then click **Displacement field (comp1.u)**.

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

5 In the **Scale** text field, type `1e-4`.

The default scale for the displacements is 1% of the model size. This is significantly more than can be expected here.

6 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Dependent Variables 2** node, then click **Displacement field (comp1.u)**.

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

8 In the **Scale** text field, type `1e-4`.

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

RESULTS

Stress, Bolt Pretension

Reproduce the plot in Figure 2 with the following steps:

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

2 Locate the **Data** section. From the **Dataset** list, choose **Study 1/Solution Store 1 (sol2)**.

Surface 1

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

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>Stress>Stress tensor (spatial frame) - N/m^2>solid.sz - Stress tensor, z component**.

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

4 In the **Minimum** text field, type `-3e8`.

5 In the **Maximum** text field, type `5e8`.

6 Click to expand the **Quality** section.

Deformation

1 In the **Model Builder** window, expand the **Surface 1** node, then click **Deformation**.

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

3 In the **Scale factor** text field, type `20`.

4 In the **Stress, Bolt Pretension** toolbar, click **Plot**.

Stress Linearization (solid)

The default plot group for stress linearization shows the 22-component of the linearized stress. You can change the line for evaluation by switching between the different datasets.

1 In the **Model Builder** window, click **Stress Linearization (solid)**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Parameter selection (lp)** list, choose **Last**.

The following steps reproduce the plot in Figure 4 :

Stress

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

2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

3 Clear the **Plot dataset edges** check box.

4 In the **Label** text field, type `Stress`.

5 Locate the **Color Legend** section. From the **Position** list, choose **Right double**.

Surface 1

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

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

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

4 Locate the **Range** section. Select the **Manual color range** check box.

5 In the **Maximum** text field, type `4e8`.

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.

4 In the associated text field, type `30`.

Line 1

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

2 In the **Settings** window for **Line**, click to expand the **Inherit Style** section.

3 From the **Plot** list, choose **Surface 1**.

4 Clear the **Color** check box.

5 Clear the **Color and data range** check box.

6 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Stress linearization>solid.SImb - Stress intensity, membrane plus bending - N/m²**.

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

8 Select the **Radius scale factor** check box.

9 In the associated text field, type `0.001`.

10 From the **Color table** list, choose **Traffic**.

Deformation 1

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

Surface 2

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

2 In the **Settings** window for **Surface**, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

5 From the **Color** list, choose **Black**.

6 Select the **Wireframe** check box.

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

8 Clear the **Color** check box.

9 Clear the **Color and data range** check box.

Deformation 1

1 Right-click **Surface 2** and choose **Deformation**.

2 In the **Stress** toolbar, click **Plot**.

Proceed to plot the bolt forces as a function of the applied moment as in Figure 5.

Bolt Force

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

2 In the **Settings** window for **1D Plot Group**, type `Bolt Force` in the **Label** text field.

Global 1

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

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Bolts>Bolt_1>solid.pblt1.sblt1.F_bolt - Bolt force - N**.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	**Unit**	**Description**
`solid.pblt1.sblt1.F_bolt`	`kN`	`Force in Bolt 1`
`solid.pblt1.sblt2.F_bolt`	`kN`	`Force in Bolt 2`
`solid.pblt1.sblt3.F_bolt`	`kN`	`Force in Bolt 3`
`solid.pblt1.sblt4.F_bolt`	`kN`	`Force in Bolt 4`
`solid.pblt1.sblt5.F_bolt`	`kN`	`Force in Bolt 5`

4 Click to expand the **Legends** section. Clear the **Show legends** check box.

5 Click to expand the **Coloring and Style** section. In the **Width** text field, type `2`.

6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

7 In the **Expression** text field, type `bendingMoment*lp/1000`.

Bolt Force

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

2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.

3 Select the **x-axis label** check box.

4 In the associated text field, type `Bending Moment on Tube (kNm)`.

5 Select the **y-axis label** check box.

6 In the associated text field, type `Axial Force in bolts (kN)`.

Global 1

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

2 In the **Settings** window for **Global**, locate the **Legends** section.

3 Select the **Show legends** check box.

4 Click to collapse the **Coloring and Style** section.

Bolt Force

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

2 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

4 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

The following steps create the plot in Figure 6:

Bolt Stress

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

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

Point Graph 1

1 Right-click **Bolt Stress** and choose **Point Graph**.

2 Select Points 311 and 328 only.

3 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Stress>Stress tensor (spatial frame) - N/m^2>solid.sz - Stress tensor, z component**.

4 Locate the **y-Axis Data** section. From the **Unit** list, choose **MPa**.

5 Select the **Description** check box.

6 In the associated text field, type `Bolt stress`.

7 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

8 In the **Expression** text field, type `bendingMoment*lp`.

9 From the **Unit** list, choose **kN*m**.

10 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Cycle**.

11 In the **Width** text field, type 2.

12 Click to expand the **Legends** section. Select the **Show legends** check box.

13 From the **Legends** list, choose **Manual**.

14 In the table, enter the following settings:

Legends
Inside
Outside

Bolt Stress

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

2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.

3 Select the **x-axis label** check box.

4 In the associated text field, type `Bending Moment on Tube (kNm)`.

5 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Finally, reproduce the contact pressure plots shown in Figure 7 and Figure 8 :

Contact Pressure

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/Solution Store 1 (sol2)**.

Surface 1

1 In the **Contact Pressure** toolbar, click **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>Contact 1>solid.cnt1.Tn - Contact pressure - N/m²**.

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

4 Locate the **Range** section. Select the **Manual color range** check box.

5 In the **Maximum** text field, type 100.

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

7 In the **Contact Pressure** toolbar, click **Plot**.

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

Contact Pressure

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

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.

4 From the **Parameter value (lp)** list, choose **1**.

5 In the **Contact Pressure** toolbar, click **Plot**.