



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.

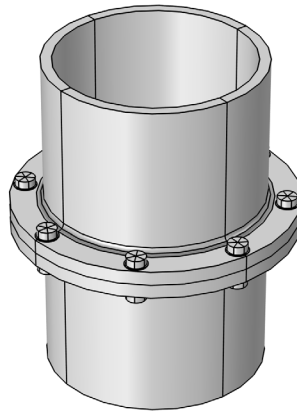


Figure 1: Tube connection.

Model Definition

The tube is made of steel and has an inner diameter of 360 mm and a 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 in the second step are shown in [Figure 3](#).

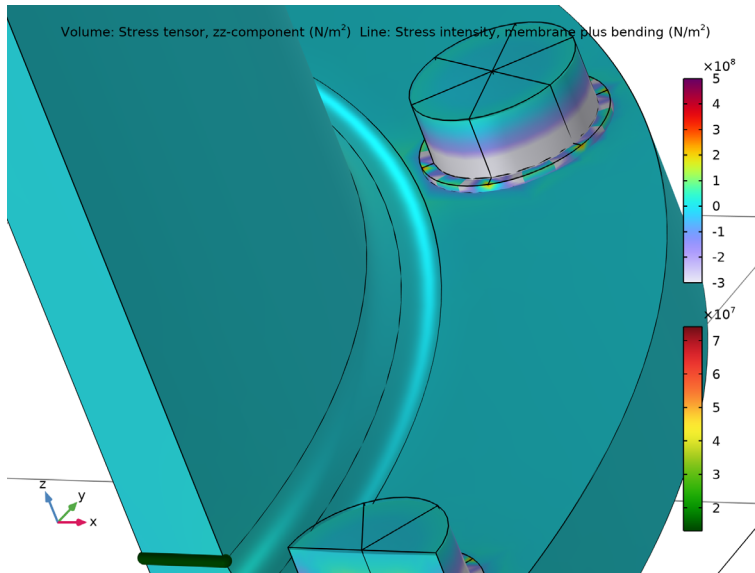


Figure 2: The axial stress after the pretension step.

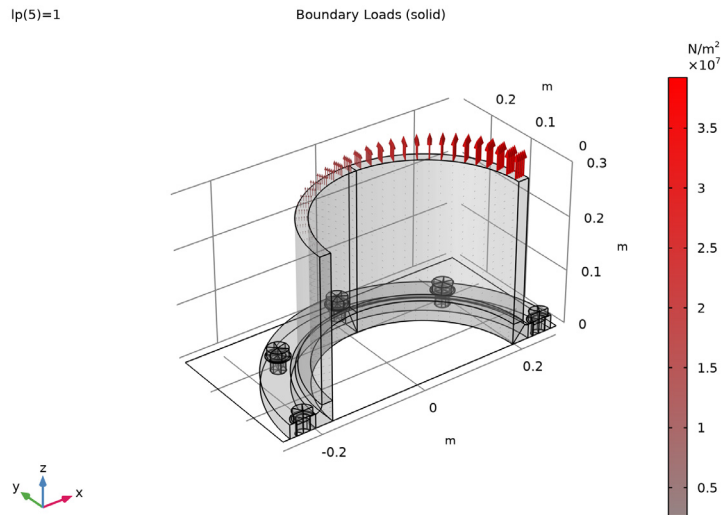


Figure 3: Applied external loads.

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 on 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 on the tensile side.

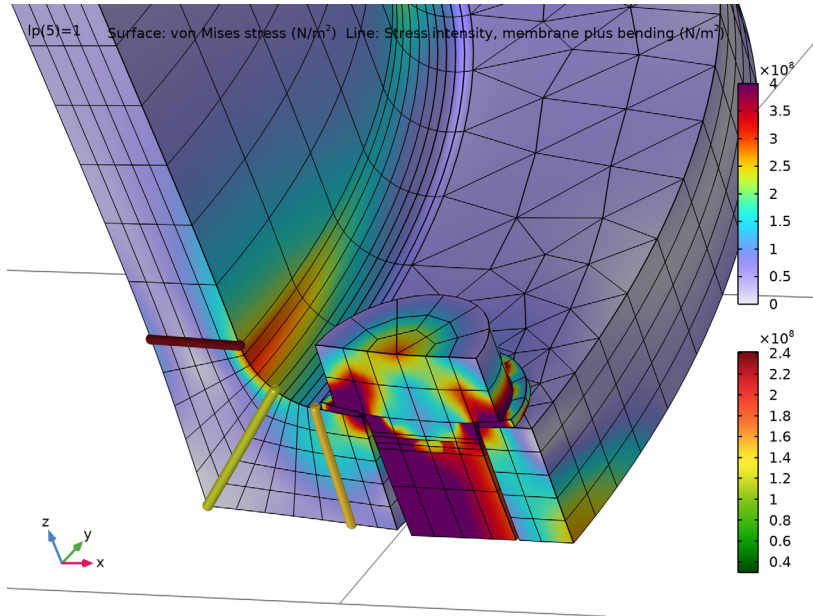


Figure 4: Equivalent stress at the maximum external load.

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 nonuniform 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.

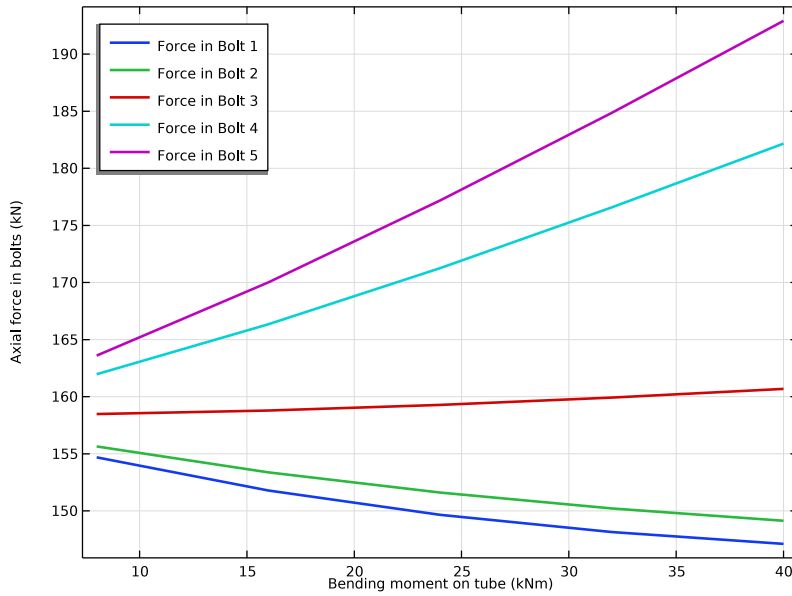


Figure 5: The bolt force as a function of the tensile force.

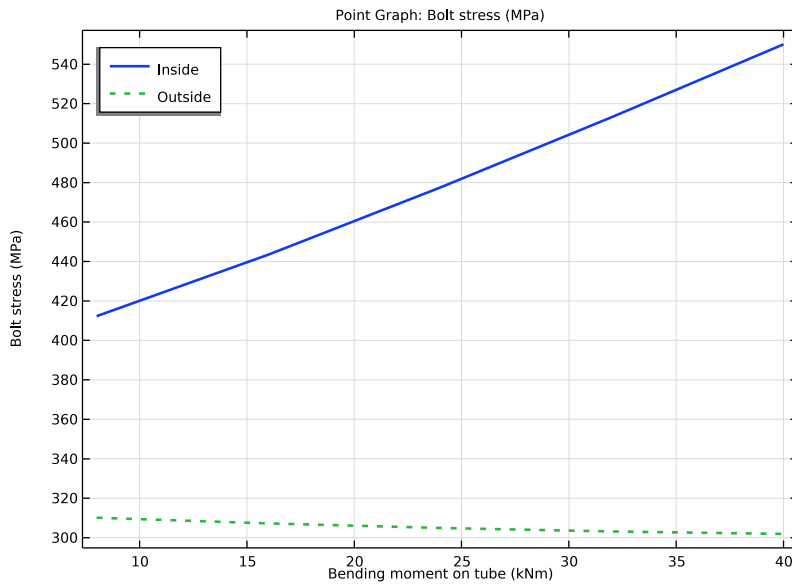


Figure 6: The development of the bolt stress at two different positions in the cross section.

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.

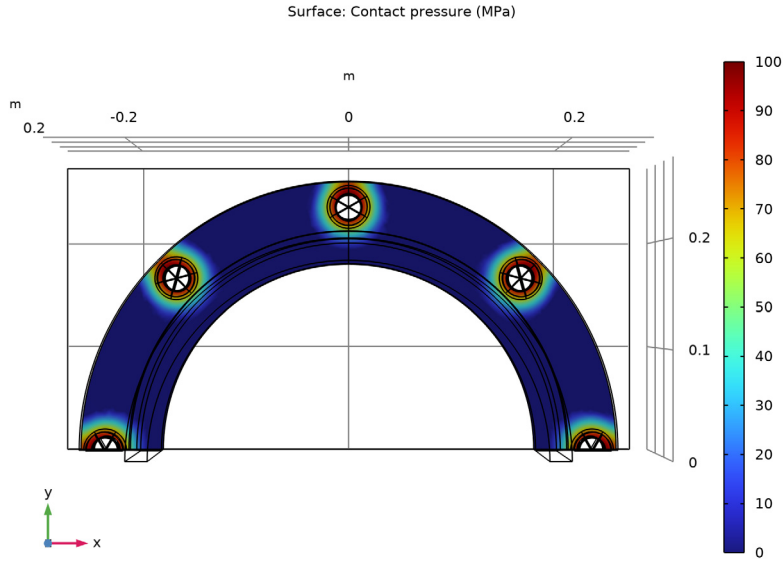


Figure 7: Contact pressure between the flanges after pretensioning the bolts.

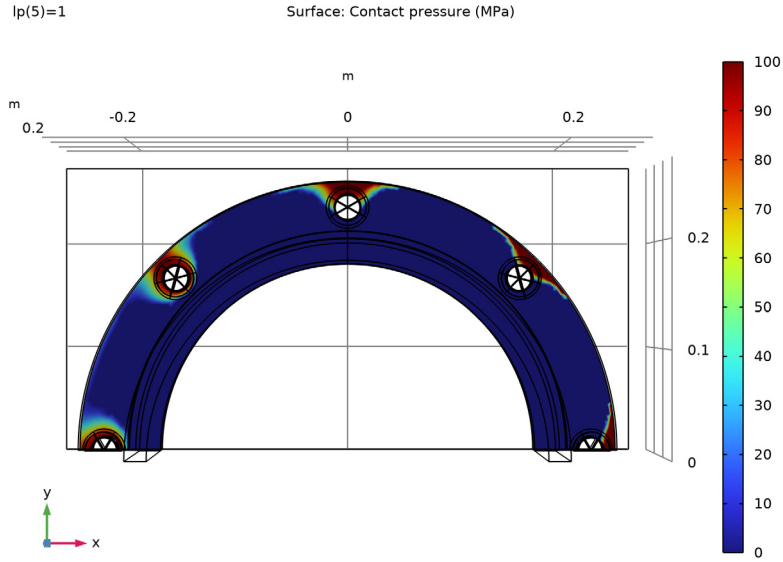


Figure 8: Contact pressure between the flanges at maximum external load.

The linearized stresses along the six stress classification lines are shown in [Figure 9](#) to [Figure 14](#). In the local coordinate system chosen for the stress linearization lines, the 22 component is the direct stress in the x_2 direction, which lies in the xz -plane and is perpendicular to the stress linearization line. The hoop stress is component 33.

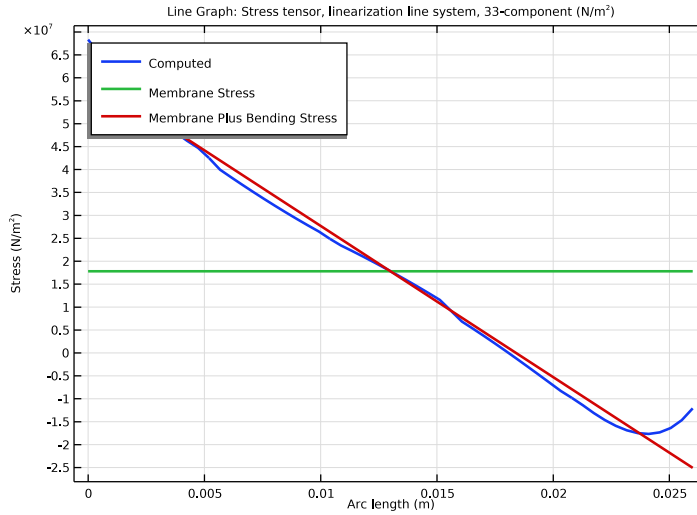


Figure 9: Linearized stresses (hoop direction) along stress classification line 1.

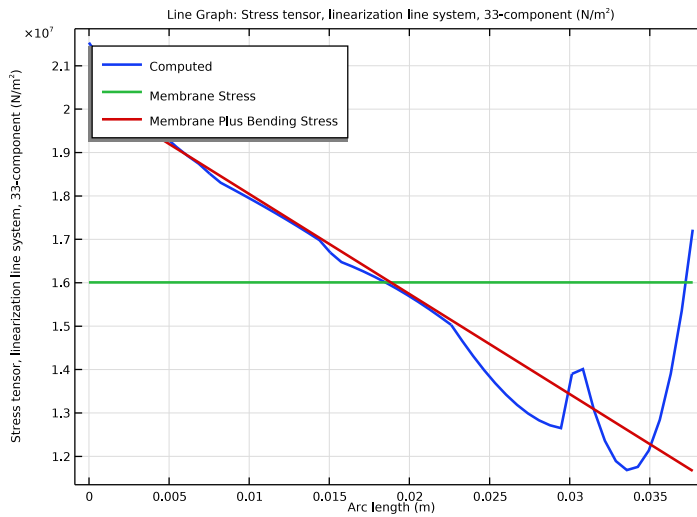


Figure 10: Linearized stresses (hoop direction) along stress classification line 2.

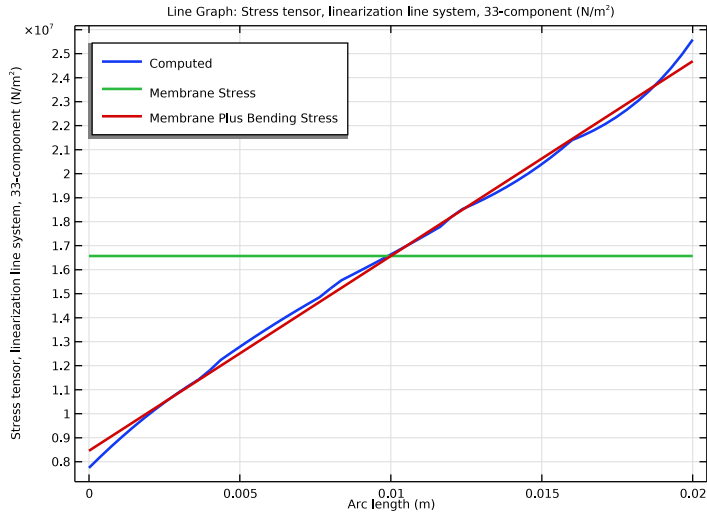


Figure 11: Linearized stresses (hoop direction) along stress classification line 3.

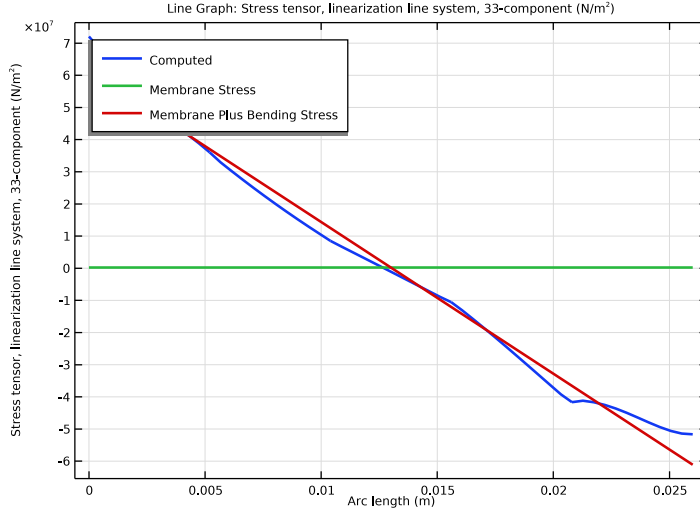


Figure 12: Linearized stresses (hoop direction) along stress classification line 4.

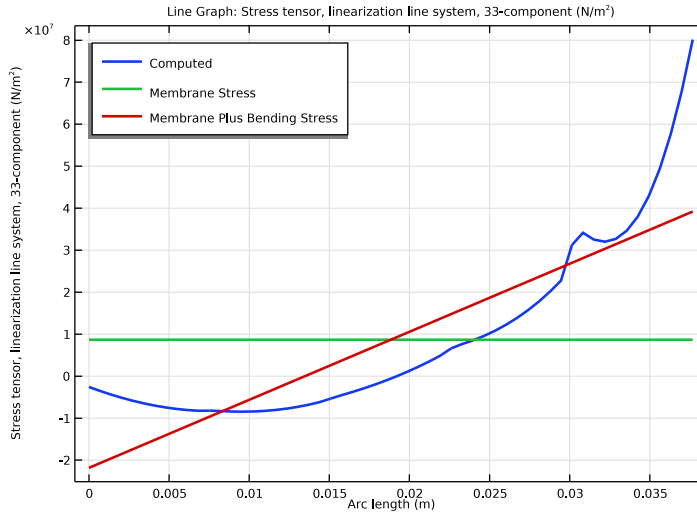


Figure 13: Linearized stresses (hoop direction) along stress classification line 5.

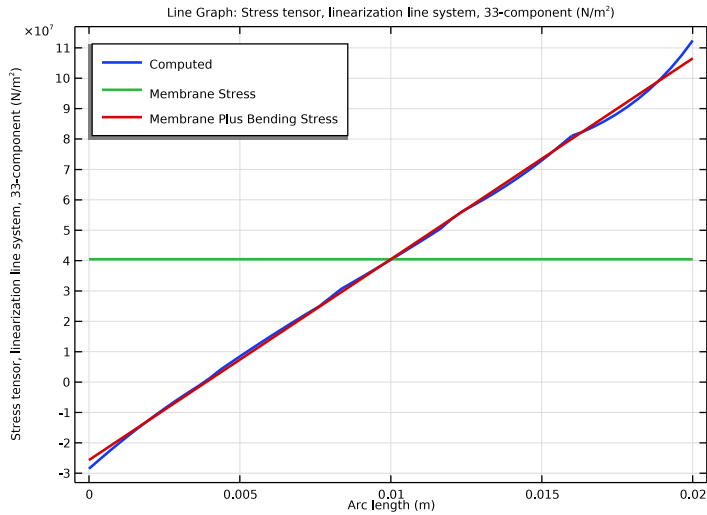


Figure 14: Linearized stresses (hoop direction) along stress classification line 6.

Notes About the COMSOL Implementation

The analysis is performed in two steps. In the first step, the effects of pretensioning the bolts are computed; 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; a mesh refinement study is required if quantitative stress results are of interest.

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 I

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

- 1 In the **Geometry** toolbar, click **Insert Sequence** and choose **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 From the **Frame for identity pairs** list, choose **Material (X, Y, Z)**.
- 6 Click  **Build Selected**.
- 7 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. Create a new selection, then delete the pair.

- 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**.
- 6 Right-click **Contact Pair 2 (ap2)** and choose **Delete**.

Contact Pair 3 (ap3)

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

COMPONENT 1 (COMP1)

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


DEFINITIONS

Boundary System 1 (sys1)

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** 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 **Axis** list, choose **x**.



Define selections to use later in the modeling.

Symmetry Boundaries (xz-plane)

- 1 In the **Definitions** toolbar, click  **Box**.
- 2 In the **Settings** window for **Box**, type Symmetry Boundaries (xz-plane) 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-plane)**.
- 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, Cut 1

- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type Bolt Pretension, Cut 1 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, Cut 2


- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type Bolt Pretension, Cut 2 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, Cut 3


- 1 In the **Definitions** toolbar, click  **Explicit**.

- 2 In the **Settings** window for **Explicit**, type Bolt Pretension, Cut 3 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, Cut 4

- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type Bolt Pretension, Cut 4 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, Cut 5

- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type Bolt Pretension, Cut 5 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. From the **Load type** list, choose **Resultant**.
- 5 Specify the **F** vector as

0	x
0	y
axialForce	z


- 6 Specify the **M** vector as

0	x
-1p*bendingMoment	y
0	z

- 7 Click to expand the **Symmetry** section. From the list, choose **Symmetry**.
- 8 Specify the **n_{sym}** vector as

1	Y
---	---

Bolt Pretension I

- 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 I

- 1 In the **Model Builder** window, click **Bolt Selection I**.
- 2 In the **Settings** window for **Bolt Selection**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Bolt Pretension, Cut I**.

Bolt Pretension I

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


Bolt Selection 2

- 1 In the **Physics** toolbar, click  **Attributes** and choose **Bolt Selection**.
- 2 In the **Settings** window for **Bolt Selection**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Bolt Pretension, Cut 2**.

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 **Boundary Selection** section.
- 3 From the **Selection** list, choose **Bolt Pretension, Cut 3**.

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 **Boundary Selection** section.
- 3 From the **Selection** list, choose **Bolt Pretension, Cut 4**.



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 **Boundary Selection** section.
- 3 From the **Selection** list, choose **Bolt Pretension, Cut 5**.

Contact 1a


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


Friction I

- 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.


Contact Ia

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

Stabilization I


- 1 In the **Physics** toolbar, click  **Attributes** and choose **Stabilization**.
The bolts are initially in contact. It is therefore enough to add stabilization in the first iteration.
- 2 In the **Settings** window for **Stabilization**, locate the **Quasistatic Stabilization** section.
- 3 From the **Method** list, choose **Manual**.

Symmetry I

- 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 I

- 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 From the **Displacement in x direction** list, choose **Prescribed**.

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





Stress Linearization I

- 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 Click to 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.


Add a **Section Forces** node to compute the integrated section forces for the cross section near the fillet.

Section Forces I

- 1 In the **Physics** toolbar, click  **Global** and choose **Section Forces**.
- 2 Click the  **Wireframe Rendering** button in the **Graphics** toolbar.
- 3 Select Boundaries 19 and 34 only.
- 4 Click the  **Wireframe Rendering** button in the **Graphics** toolbar.
- 5 In the **Settings** window for **Section Forces**, locate the **Cut Plane** section.
- 6 Select the **Symmetry plane** check box.
- 7 Locate the **Symmetry Plane Selection** section. Click to select the  **Activate Selection** toggle button.
- 8 Select Boundary 17 only.

MESH I

Swept I

- 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 I

- 1 Right-click **Swept I** 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  **More Generators** 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  **More Generators** 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

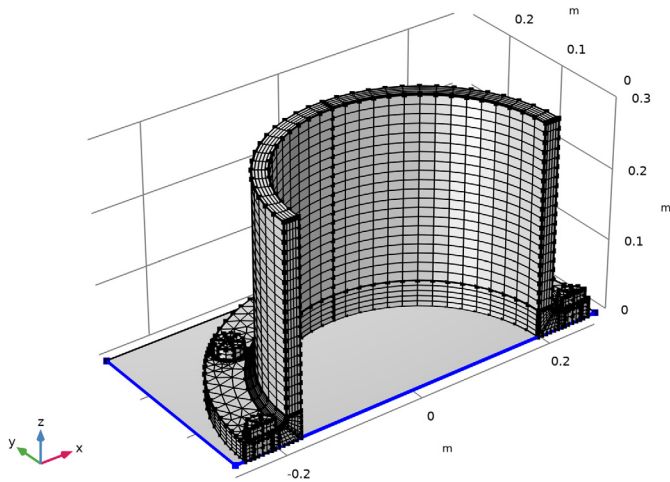
- 1 In the **Mesh** toolbar, click  **More Generators** and choose **Mapped**.
- 2 Select Boundary 1 only.

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 I

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

Step 2: Bolt Pretension


- 1 In the **Study** toolbar, click  **Study Steps** and choose **Stationary>Bolt Pretension**.
- 2 Right-click **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 tree, select **Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Pressure**.
- 6 Click  **Disable**.
- 7 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Bending Moment and Axial Force**.
- 8 Click  **Disable**.


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)**.

Volume 1


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

- 2 In the **Settings** window for **Volume**, 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/m²>solid.sGpzz - Stress tensor, zz-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 **Volume 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**.

Graph Plot Style 1

- 1 In the **Results** toolbar, click  **Configurations** and choose **Graph Plot Style**.
- 2 In the **Settings** window for **Graph Plot Style**, locate the **Coloring and Style** section.
- 3 Find the **Line style** subsection. From the **Width** list, choose **2**.
- 4 Locate the **Legends** section. Find the **Include in automatic mode** subsection. Clear the **Solution** check box.
- 5 Select the **Label** check box.
- 6 Clear the **Point** check box.






Stress Linearization 1 (solid)

The default plot group for stress linearization shows the 22-component of the linearized stress. Here, the local x2 direction is in the xz-plane and perpendicular to the SCL chosen. The x3 direction will be the hoop direction.

- 1 In the **Model Builder** window, under **Results** click **Stress Linearization 1 (solid)**.
- 2 In the **Settings** window for **ID Plot Group**, click to expand the **Title** section.
- 3 From the **Title type** list, choose **Manual**.
- 4 In the **Title** text area, type Line Graph: Stress tensor, linearization line system, 33-component (N/m²).
- 5 Locate the **Plot Settings** section. Select the **x-axis label** check box.
- 6 Select the **y-axis label** check box. In the associated text field, type Stress (N/m²).

- 7 Click to expand the **Style Configuration** section. From the **Configuration** list, choose **Graph Plot Style 1**.
- 8 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

FIND AND REPLACE


- 1 Press Ctrl+F.
- 2 Go to the **Find and Replace** window.
- 3 Click  next to  **Add Filter**, then choose **Node Filter**.
- 4 In the **Node Filter** dialog box, type Results in the **Find** text field.
- 5 Click **OK**.
- 6 Go to the **Find and Replace** window.
- 7 In the **Contains** text field, type S11s22.
- 8 In the **Replace with** text field, type S11s33.
- 9 Click  **Search**.
- 10 Click  next to  **Replace**, then choose **Replace All**.
- 11 Repeat steps 7-10 with the following expressions:

Contains	Replace with
Sm22	Sm33
SmbS22	SmbS33
Smbe22	Smbe33


RESULTS


Section Forces (secf1)

The Section Forces node generates plots to visualize the resultant moment and force but also an evaluation group to evaluate the resultants at each load parameter.

- 1 In the **Model Builder** window, click **Section Forces (secf1)**.
- 2 In the **Section Forces (secf1)** toolbar, click  **Evaluate**.

ADD PREDEFINED PLOT


- 1 In the **Home** toolbar, click  **Add Predefined Plot** to open the **Add Predefined Plot** window.
- 2 Go to the **Add Predefined Plot** window.

- 3 In the tree, select **Study 1/Solution 1 (sol1)>Solid Mechanics>Section Forces (secf1)** and **Study 1/Solution 1 (sol1)>Solid Mechanics>Section Moments (secm1)**.
- 4 Click **Add Plot** in the window toolbar.
- 5 In the **Home** toolbar, click  **Add Predefined Plot** to close the **Add Predefined Plot** window.


The following steps reproduce the plot in [Figure 4](#) :

RESULTS

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.misesGp`.
- 4 Click to expand the **Range** section. Select the **Manual color range** check box.
- 5 In the **Maximum** text field, type `4e8`.
- 6 Locate the **Coloring and Style** section. Click  **Change Color Table**.
- 7 In the **Color Table** dialog box, select **Rainbow>Prism** in the tree.
- 8 Click **OK**.

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. 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.Slmb - 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. In the associated text field, type 0.001.
- 9 Click  **Change Color Table**.
- 10 In the **Color Table** dialog box, select **Traffic>Traffic** in the tree.
- 11 Click **OK**.


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 **ID Plot Group**.
- 2 In the **Settings** window for **ID 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 I (compI)>Solid Mechanics>Bolts>Bolt_I>solid.pb1t1.sblt1.F_bolt - Bolt force - N**.

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

Expression	Unit	Description
solid.pb1t1.sblt1.F_bolt	kN	Force in Bolt 1
solid.pb1t1.sblt2.F_bolt	kN	Force in Bolt 2
solid.pb1t1.sblt3.F_bolt	kN	Force in Bolt 3
solid.pb1t1.sblt4.F_bolt	kN	Force in Bolt 4
solid.pb1t1.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. From the **Width** list, choose **2**.

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

7 In the **Expression** text field, type $\text{bendingMoment} \cdot l_p / 1000$.

Bolt Force

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

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

3 Select the **x-axis label** check box. In the associated text field, type **Bending moment on tube (kNm)**.

4 Select the **y-axis label** check box. In the associated text field, type **Axial force in bolts (kN)**.

Global I

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

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 **ID Plot Group**, locate 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 **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type **Bolt Stress** in the **Label** text field.

Point Graph

- 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/m²>solid.sGpzz - Stress tensor, zz-component**.
- 4 Locate the **y-Axis Data** section. From the **Unit** list, choose **MPa**.
- 5 Select the **Description** check box. In the associated text field, type **Bolt stress**.
- 6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
- 7 In the **Expression** text field, type **bendingMoment*1p**.
- 8 From the **Unit** list, choose **kN*m**.
- 9 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Cycle**.
- 10 From the **Width** list, choose **2**.
- 11 Click to expand the **Legends** section. Select the **Show legends** check box.
- 12 From the **Legends** list, choose **Manual**.
- 13 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 **ID Plot Group**, locate the **Plot Settings** section.
- 3 Select the **x-axis label** check box. In the associated text field, type **Bending moment on tube (kNm)**.
- 4 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Next, 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.dcnt1.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 In the **Contact Pressure** toolbar, click  **Plot**.

Finally, add **Boundary Loads (solid)** from the predefined plots to visualize the applied loads.

ADD PREDEFINED PLOT

- 1 In the **Home** toolbar, click  **Add Predefined Plot** to open the **Add Predefined Plot** window.
- 2 Go to the **Add Predefined Plot** window.
- 3 In the tree, select **Study 1/Solution 1 (sol1)>Solid Mechanics>Applied Loads (solid)>Boundary Loads (solid)**.
- 4 Click **Add Plot** in the window toolbar.
- 5 In the **Home** toolbar, click  **Add Predefined Plot** to close the **Add Predefined Plot** window.

RESULTS

Boundary Loads (solid)

In the **Boundary Loads (solid)** toolbar, click  **Plot**.