Model created in COMSOL Multiphysics 6.4

Interference Fit Connection in a Mountain Bike Fork

Introduction

Interference fit is a technique used to join two pipes with each other. The smaller pipe, which is slightly larger than the available space in the larger pipe, is cooled down so that it fits. When shrunk, it is fitted into the larger pipe. When the temperature returns to normal, the expansion of the inner pipe will force the outer pipe to expand and the pipes will be pressed against each other. The contact pressure and friction coefficient between the two surfaces determine the strength of the connection.

In this model of a mountain bike fork, the steerer tube is connected to the crown through a shrink fit.

Model Definition

The geometry of the front fork is shown in Figure 1. The damping elements located in the stanchions have been removed from the model since they do not contribute to the structural response during the interference fit mounting.

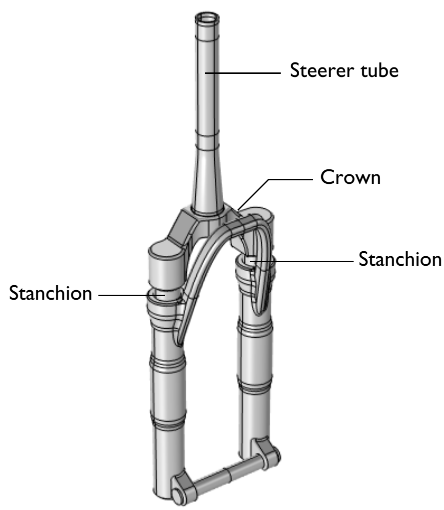

Figure 1: Front fork.

The entire fork is made out of AISI 4130 steel with the following material data:

- Young's modulus E = 210 GPa
- Poisson's ratio ν = 0.29
- Density ρ = 7800 kg/m^3.

The radial overlap between the two parts is 0.04 mm and the static friction coefficient is assumed to be 0.2.

Results and Discussion

The equivalent stress in the assembly is shown in Figure 2. The stresses on the outer surface are about 300 MPa. Much higher stresses are found below the surface close to the interference fit. Isosurfaces of the equivalent stress are shown in Figure 3. The picture clearly shows that the stress gradient is high around the interference fit.

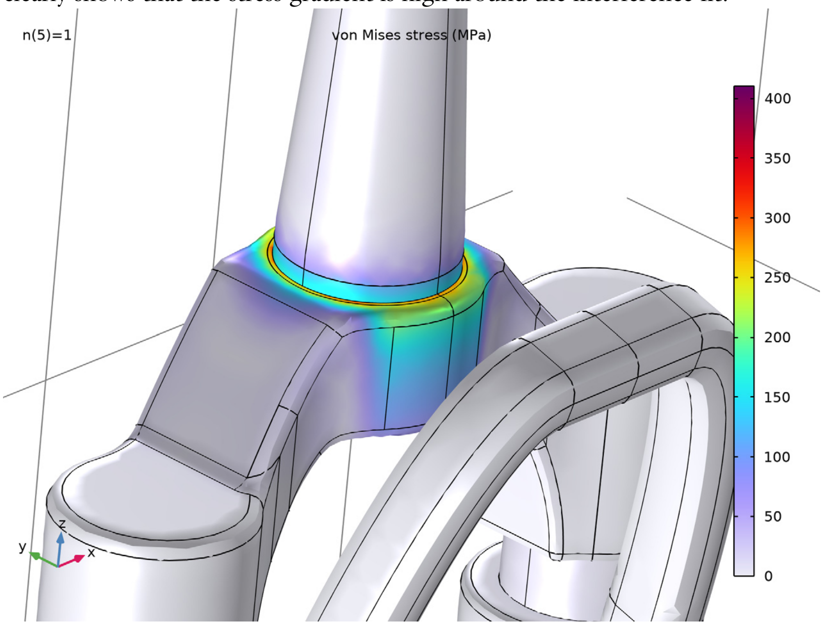

Figure 2: The equivalent stress caused by the interference fit.

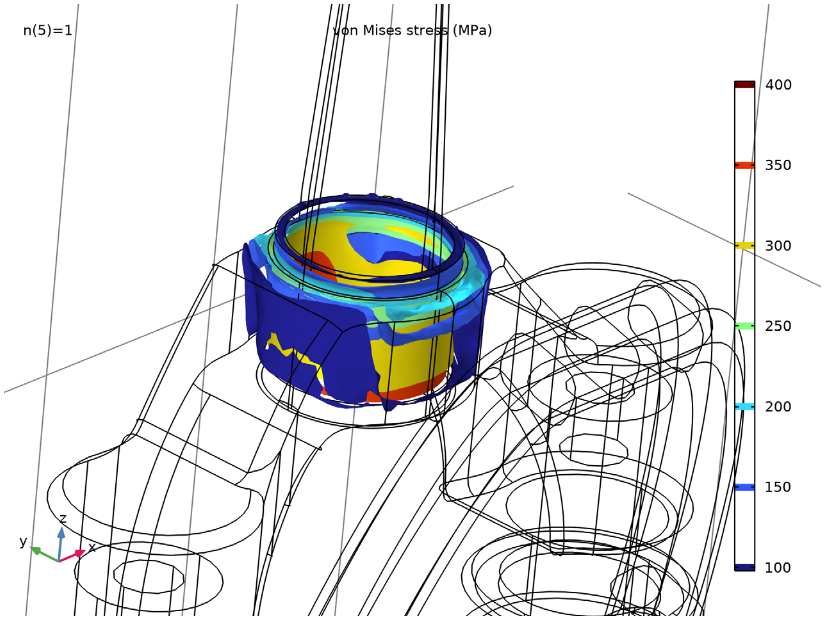

Figure 3: The isosurfaces of the equivalent stress.

The largest tensile and compressive stresses are shown in Figure 4-Figure 5. The largest tensile stress, with a magnitude of about 230 MPa, is located in the crown, while the largest compressive stress, with a magnitude of about -410 MPa, is found in the steerer tube. Since the tube was slightly larger than the available space in the crown, it must shrink, while the crown must expand. The maximum transferable force and moment through the shrink fit is 22 kN and 442 Nm, respectively.

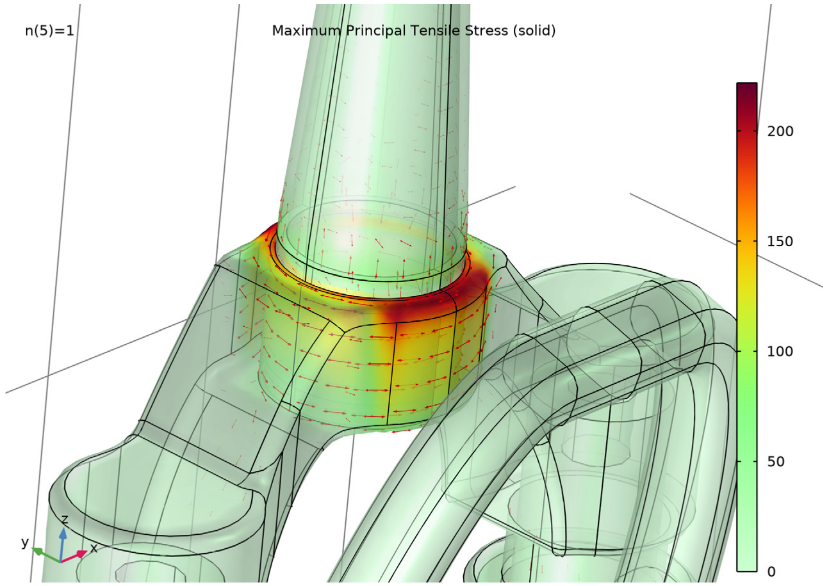

Figure 4: First principal stress.

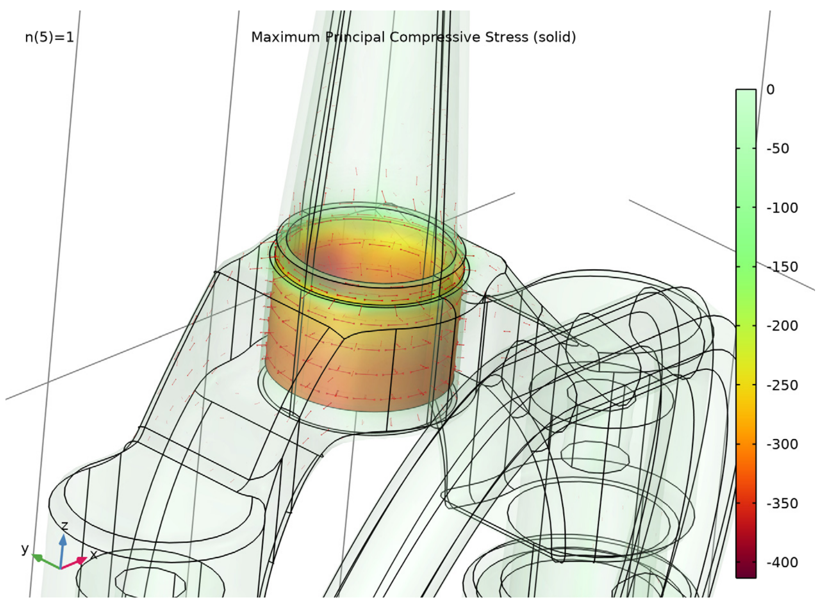

Figure 5: Third principal stress.

The contact pressure in the interference fit is shown in Figure 6.

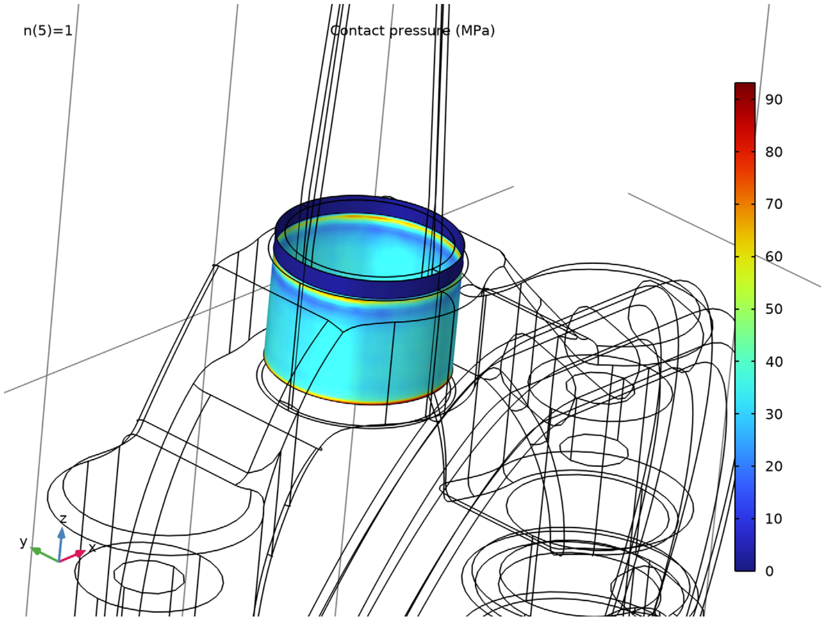

Figure 6: Contact pressure.

Notes About the COMSOL Implementation

When analyzing a mounting process, it is common that the two parts are not in contact in the initial configuration. In order to obtain a well-posed model none of the parts can have possible rigid body motions. In the model this is ensured by using a **Fixed Constraint** on a few boundaries of the crown, and by adding a **Stabilization** node under the **Contact** node.

When modeling contact problems like this, where the contacting boundaries have very small relative movements, the performance can be improved by selecting **Initial configuration** as **Mapping method** in the **Contact pair** node. In order to ensure a smooth contact pressure distribution that is not affected by the mesh discretization on the contact boundaries, **Force zero initial gap** has been selected in the **Contact** node.

Application Library path: `Structural_Mechanics_Module/Contact_and_Friction/mountain_bike_fork`

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**.

GEOMETRY 1

Import 1 (imp1)

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

2 In the **Settings** window for **Import**, locate the **Source** section.

3 From the **Source** list, choose **COMSOL Multiphysics file**.

4 Click **Browse**.

5 Browse to the model's Application Libraries folder and double-click the file `mountain_bike_fork.mphbin`.

6 Click **Import**.

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 **Frame for identity pairs** list, choose **Material (X, Y, Z)**.

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

Disable the analysis of the geometry as the remaining geometric details can be kept.

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

7 In the **Settings** window for **Geometry**, locate the **Cleanup** section.

8 Clear the **Automatic detection of small details** checkbox.

DEFINITIONS

Identity Boundary Pair 2 (ap2)

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Definitions** node, then click **Identity Boundary Pair 2 (ap2)**.

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**.

Identity Boundary Pair 3 (ap3)

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

2 In the **Settings** window for **Pair**, click the **Swap Source and Destination** button.

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 In the table, enter the following settings:

Name	Expression	Value	Description
n	0	0	Interference fit multiplier
mu	0.2	0.2	Friction coefficient

MATERIALS

Steel

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

2 In the **Settings** window for **Material**, type `Steel` in the **Label** text field.

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

Property	Variable	Value	Unit	Property group
Young's modulus	E	`210e9`	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`0.29`	1	Young's modulus and Poisson's ratio
Density	rho	`7800`	kg/m³	Basic

SOLID MECHANICS (SOLID)

Contact 1

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

2 In the **Settings** window for **Contact**, click to expand the **Contact Surface Offset and Adjustment** section.

3 In the $d_{\text{offset,d}}$ text field, type `(0.04[mm])*n`.

4 Select the **Force zero initial gap** checkbox.

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 `mu`.

Contact 1

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

Stabilization 1

In the **Physics** toolbar, click **Attributes** and choose **Stabilization**.

Fixed Constraint 1

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

2 Select Boundary 191 only.

Symmetry 1

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

2 Select Boundaries 1, 10, 13, 18, 163, 168, 171, 176, 179, 183, 185, 187, 189, 196, 198, 200, 202, and 204 only.

MESH 1

Mapped 1

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

2 Select Boundary 166 only.

Distribution 1

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

2 Select Edges 367, 370, 375, and 378 only.

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

4 In the **Number of elements** text field, type 15.

Distribution 2

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

2 Select Edge 366 only.

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

4 From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type 6.

6 In the **Element ratio** text field, type 2.

7 Select the **Reverse direction** checkbox.

Distribution 3

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

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

3 In the list box, select **366**.

4 Click **Remove from Selection**.

5 Select Edge 374 only.

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 Select Domain 2 only.

5 Click to expand the **Source Faces** section. Select Boundary 166 only.

6 Click to expand the **Destination Faces** section. Select Boundary 165 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 10.

Mapped 2

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

2 Select Boundary 174 only.

Distribution 1

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

2 Select Edges 385, 388, 393, and 396 only.

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

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

Distribution 2

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

2 Select Edge 384 only.

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

4 From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type 6.

6 In the **Element ratio** text field, type 2.

Distribution 3

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

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

3 In the list box, select **384**.

4 Click **Remove from Selection**.

5 Select Edge 392 only.

6 Locate the **Distribution** section. Select the **Reverse direction** checkbox.

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 Select Domain 3 only.

Distribution 1

1 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 15.

Free Tetrahedral 1

1 In the **Mesh** toolbar, click **Free Tetrahedral**.

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

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

4 Select Domain 4 only.

Size 1

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

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

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

Free Tetrahedral 2

1 In the **Mesh** toolbar, click **Free Tetrahedral**.

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

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

4 Select Domain 1 only.

Distribution 1

1 Right-click **Free Tetrahedral 2** and choose **Distribution**.

2 Select Edges 27, 28, 31, and 33 only.

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

4 In the **Number of elements** text field, type 10.

Size 1

In the **Model Builder** window, right-click **Free Tetrahedral 2** and choose **Size**.

Size 2

1 Right-click **Free Tetrahedral 2** 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 Boundaries 109 and 110 only.

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

Size 1

1 In the **Model Builder** window, click **Size 1**.
2 In the **Settings** window for **Size**, locate the **Element Size** section.
3 Click the **Custom** button.
4 Locate the **Element Size Parameters** section.
5 Select the **Minimum element size** checkbox. In the associated text field, type `0.011`.
6 Click **Build All**.

STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.
2 In the **Settings** window for **Stationary**, 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
n (Interference fit multiplier)	`0.1, range(0.25, 0.25, 1.0)`

Solution 1 (sol1)

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

  Since the contact surface is assumed to be established, you can use less conservative solver settings to speed up the computations.
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)** > **Stationary Solver 1** node, then click **Fully Coupled 1**.
4 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.
5 From the **Nonlinear method** list, choose **Automatic (Newton)**.
6 In the **Study** toolbar, click **Compute**.

Set default units for result presentation.

RESULTS

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, select **Solid Mechanics** > **Stress tensor (N/m^2)** in the tree.

5 Click **OK**.

6 In the **Settings** window for **Preferred Units**, locate the **Units** section.

7 In the table, enter the following settings:

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

8 Select the **Apply conversions to expressions with the same dimensions** checkbox.

9 Click **Apply**.

Mirror 3D 1

1 In the **Model Builder** window, expand the **Results** > **Datasets** node.

2 Right-click **Results** > **Datasets** and choose **More 3D Datasets** > **Mirror 3D**.

Stress (solid)

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

2 From the **Dataset** list, choose **Mirror 3D 1**.

Volume 1

1 In the **Model Builder** window, expand the **Stress (solid)** node, then click **Volume 1**.

2 In the **Stress (solid)** toolbar, click **Plot**.

Stress (solid)

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

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

3 From the **View** list, choose **View 3D 2**.

4 In the **Stress (solid)** toolbar, click **Plot**.

Use the mouse buttons to zoom in on the region around the interference fit. When done, lock the camera:

5 Click **Go to Source**.

View 3D 2

1 In the **Model Builder** window, under **Results** > **Views** click **View 3D 2**.

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

3 Select the **Lock camera** checkbox.

Equivalent Stress, Isosurface

1 In the **Results** toolbar, click **3D Plot Group**.

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

3 Locate the **Data** section. From the **Dataset** list, choose **Mirror 3D 1**.

Isosurface 1

1 Right-click **Equivalent Stress, Isosurface** and choose **Isosurface**.

2 In the **Settings** window for **Isosurface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics** > **Stress** > **solid.misesGp - von Mises stress - N/m²**.

3 Locate the **Levels** section. From the **Entry method** list, choose **Levels**.

4 In the **Levels** text field, type `range(100,50,400)`.

RESULT TEMPLATES

1 In the **Home** 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** > **Maximum Principal Tensile Stress (solid)**.

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

RESULT TEMPLATES

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

2 In the tree, select **Study 1/Solution 1 (sol1)** > **Solid Mechanics** > **Maximum Principal Compressive Stress (solid)**.

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

RESULTS

Maximum Principal Compressive Stress (solid)

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

Maximum Principal Tensile Stress (solid)

1 In the **Model Builder** window, click **Maximum Principal Tensile Stress (solid)**.
2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3 From the **Dataset** list, choose **Mirror 3D 1**.
4 Locate the **Plot Settings** section. From the **View** list, choose **View 3D 2**.
5 In the **Maximum Principal Tensile Stress (solid)** toolbar, click **Plot**.

Transparency 1

1 In the **Model Builder** window, expand the **Maximum Principal Tensile Stress (solid)** node.
2 Right-click **Surface 1** and choose **Transparency**.
3 In the **Settings** window for **Transparency**, locate the **Transparency** section.
4 Find the **Transparency** subsection. From the **Input** list, choose **Expression**.
5 In the **Expression** text field, type `max(solid.sp1Gp,0)`.
6 In the **Maximum transparency** text field, type `0.5`.
7 Select the **Reverse** checkbox.

Principal Stress Surface 1

1 In the **Model Builder** window, under **Results** > **Maximum Principal Tensile Stress (solid)** click **Principal Stress Surface 1**.
2 In the **Settings** window for **Principal Stress Surface**, locate the **Positioning** section.
3 In the **Number of points** text field, type `10000`.
4 Locate the **Coloring and Style** section.
5 Select the **Scale factor** checkbox. In the associated text field, type `2E-11`.

Maximum Principal Compressive Stress (solid)

1 In the **Model Builder** window, under **Results** click **Maximum Principal Compressive Stress (solid)**.
2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3 From the **Dataset** list, choose **Mirror 3D 1**.
4 Locate the **Plot Settings** section. From the **View** list, choose **View 3D 2**.
5 In the **Maximum Principal Compressive Stress (solid)** toolbar, click **Plot**.

Transparency 1

1 In the **Model Builder** window, expand the **Maximum Principal Compressive Stress (solid)** node.
2 Right-click **Surface 1** and choose **Transparency**.

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

4 Find the **Transparency** subsection. From the **Input** list, choose **Expression**.

5 In the **Expression** text field, type `min(solid.sp3Gp,0)`.

6 In the **Maximum transparency** text field, type `0.9`.

Principal Stress Surface 1

1 In the **Model Builder** window, under **Results** > **Maximum Principal Compressive Stress (solid)** click **Principal Stress Surface 1**.

2 In the **Settings** window for **Principal Stress Surface**, locate the **Coloring and Style** section.

3 Select the **Scale factor** checkbox.

4 Locate the **Positioning** section. In the **Number of points** text field, type `10000`.

5 Locate the **Coloring and Style** section. In the **Scale factor** text field, type `8E-12`.

Contact Pressure

1 In the **Results** toolbar, click **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 **Mirror 3D 1**.

Surface 1

1 Right-click **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**.

Unlock the camera view.

View 3D 2

1 In the **Model Builder** window, under **Results** > **Views** click **View 3D 2**.

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

3 Clear the **Lock camera** checkbox.

Transferable Loads

1 In the **Results** toolbar, click **More Derived Values** and choose **Integration** > **Surface Integration**.

2 In the **Settings** window for **Surface Integration**, type `Transferable Loads` in the **Label** text field.

3 Select Boundaries 172 and 178 only.

4 Locate the **Data** section. From the **Parameter selection (n)** list, choose **Last**.

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

Expression	Unit	Description
2*sqrt(x^2+(y-0.0085)^2)*mu*solid.Tn	N*m	Transferable torque
2*mu*solid.Tn	N	Transferable force

6 Click **Evaluate**.