Created in COMSOL Multiphysics 6.2

Disc Brake Pad Wear

Introduction

This example demonstrates how to compute wear of the friction material in a disc brake. Wear is modeled using a generalized form of the well-known Archard equation. The sliding velocity and the friction forces are computed from simple kinematic considerations. In addition, it is assumed that the disc has a much higher wear resistance than the pad, so that only wear of the friction material has to be considered. Thermal effects on the wear rate and thermal expansion are neglected.

Model Definition

The simplified geometry of the disc and brake pad assembly is shown in Figure 1. Assuming a symmetric geometry, only half of the assembly needs to be modeled.

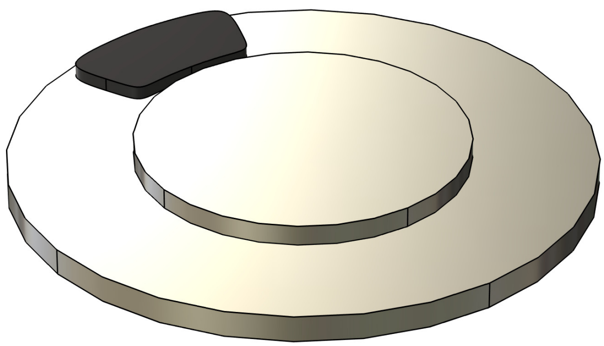

Figure 1: Disc brake geometry.

The entire assembly is considered to be linear elastic, and the brake disc is given material properties of steel. The material of the brake pad is assumed to have a stiffness that is one order of magnitude lower than that of the disc. All material properties are summarized in Table 1.

TABLE 1: MATERIAL PROPERTIES.

PROPERTY	**DISC**	**PAD**
Young's modulus	210 GPa	10 GPa
Poisson's ratio	0.3	0.1
Density	7850 kg/m^3	2000 kg/m^3

The interaction between the disc and the pad is modeled using a contact condition. To ensure that the contact pressure is well resolved, the augmented Lagrangian method is used. The contact pressure is initiated by adding a boundary load to the top surface of the brake pad, and the effect of friction forces is accounted for by prescribing a relative slip velocity between the two parts. The angular velocity, ω, of the brake disc is assumed to be constant during the analysis. Since the origin of the global coordinate system is located in the center of the disc, the relative slip velocity, $\mathbf{v}_{\mathrm{slip}}$, between the pad and the disc is given as

$$\mathbf{v}_{\mathrm{slip}} = \omega \times \mathbf{r}$$

where $\mathbf{r}$ is the position vector to any point on the pad.

Wear involves a gradual removal of material at the surfaces in contact, and the process can be described with a rate equation. In this example, a generalized form of the Archard wear model is used, where the wear rate is given as

$$\dot{h}_{\mathrm{wear}} = k_{\mathrm{wear}}\left(\frac{T_{\mathrm{n}}}{T_{\mathrm{n,ref}}}\right)^{n}\|\mathbf{v}_{\mathrm{slip}}\|$$

Here, h_{wear} is the wear depth, k_{wear} is a dimensionless wear constant, T_{n} is the contact pressure, $T_{\mathrm{n,ref}}$ is a reference pressure, n is a dimensionless exponent. By default, $T_{\mathrm{n,ref}}$ is set to 1 MPa and n to 1, which makes the above equation correspond to the classical form of the Archard wear equation.

Results and Discussion

Figure 2 shows the distribution of the von Mises stress at the end of the simulation. At this point, material has been removed due to wear causing any initial peak in the stress to be smoothened out.

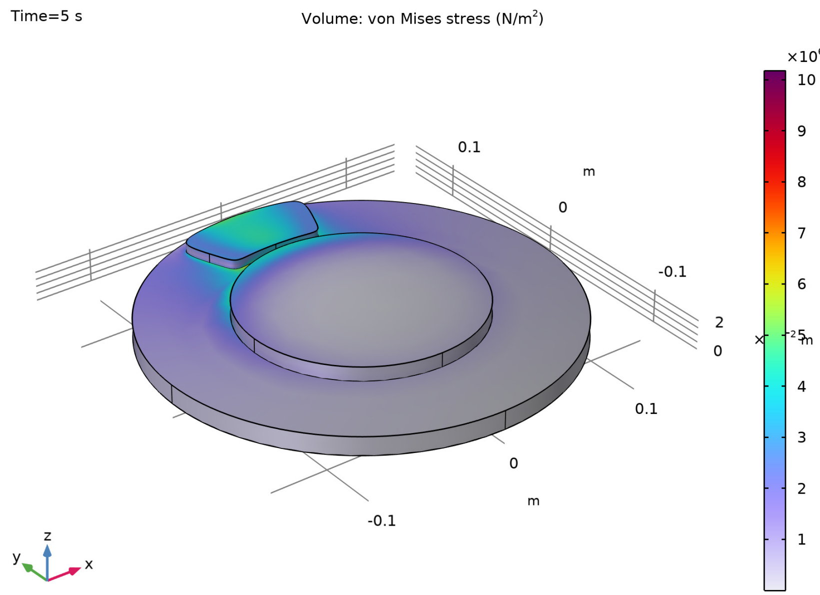

Figure 2: Distribution of von Mises stress in the pad and disc assembly.

If the contact between the pad and the disc was frictionless, the contact pressure would have a symmetric distribution over the pad surface. However, including friction forces results in a higher contact pressure at the leading edge of the brake pad compared to its trailing edge. This is shown in Figure 3 for the worn brake pad. As a result of the locally higher pressure, the wear rate, and subsequently the wear depth, is larger at the leading edge, see Figure 4. Due to the geometric changes resulting from the material removal, the pressure distribution changes and the pressure peak at the leading edge eventually decreases over time.

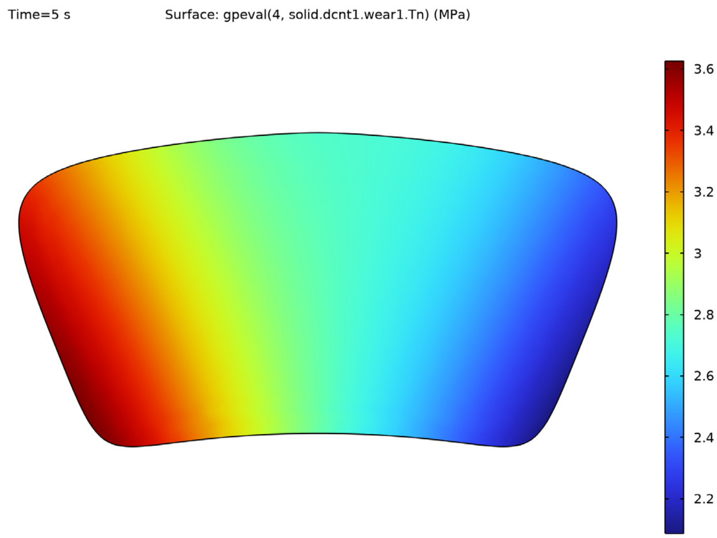

Figure 3: Pressure distribution of the worn pad surface.

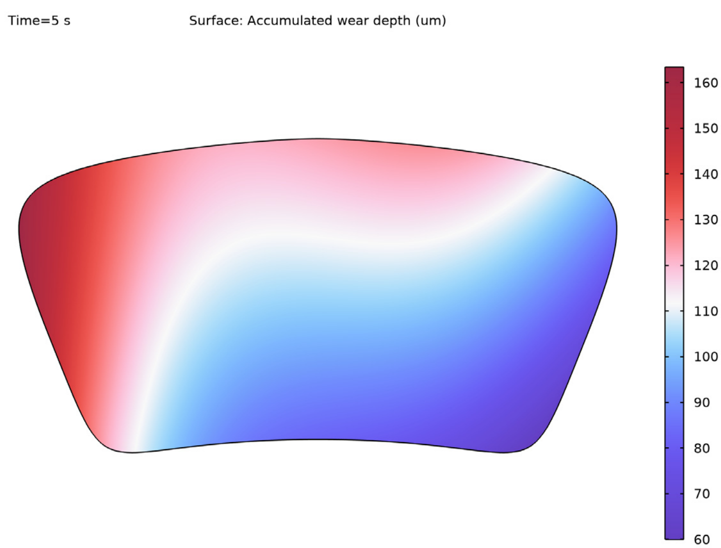

Figure 4: Distribution of the accumulated wear depth.

Notes About the COMSOL Implementation

The initial contact problem is solved in a stationary study. By adding a **Slip Velocity** subnode to the **Contact** node, you can solve for dynamic friction forces even in a stationary study. To solve for the wear depth, add a second time-dependent study, and initialize the study with the values from the first study.

Wear is in this example modeled using the deformed geometry concept. This means that the geometry of the brake pad changes with time by using an adaptive meshing technique. Using this method to model wear thus also updates the contact conditions accordingly during the simulation.

Usually, wear problems require long time periods to generate any significant wear depths. One approach to avoid running long-term studies is to multiply the wear rate with an acceleration factor. The result may be interpreted as running the simulation for a time t times the wear acceleration factor, where t is the time period used in the simulation.

Application Library path: `Structural_Mechanics_Module/Contact_and_Friction/disc_brake_wear`

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 In the table, enter the following settings:

Name	Expression	Value	Description
v	60[km/h]	16.667 m/s	Vehicle speed
r_wheel	0.25[m]	0.25 m	Wheel radius
omega	v/r_wheel	66.667 1/s	Wheel angular velocity
wear_accel	100	100	Wear acceleration factor

GEOMETRY 1

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 `disc_brake_geom_sequence.mph`.

Form Assembly (fin)

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

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

3 Select the **Create pairs** check box.

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

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

DEFINITIONS

Contact Pair 1 (ap1)

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**.

Cylindrical System 2 (sys2)

A boundary system is used to define the orientation of the relative slip velocity between the pad and the disc. In this case it is convenient to define the orientation of the boundary system axes based on cylindrical coordinates.

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 From the **Frame** list, choose **Reference configuration**.

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

5 From the **Axis** list, choose **phi**.

MATERIALS

Disc

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 `Disc` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. In the list, select **3**.

4 Click **Remove from Selection**.

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

Property	Variable	Value	Unit	Property group
Young's modulus	E	`210[GPa]`	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`0.3`	1	Young's modulus and Poisson's ratio
Density	rho	`7850[kg/m^3]`	kg/m³	Basic

6 Click to expand the **Appearance** section. From the **Material type** list, choose **Steel**.

7 In the **Graphics** window toolbar, click next to **Colors**, then choose **Show Material Color and Texture**.

8 In the **Graphics** window toolbar, click next to **Scene Light**, then choose **Indoor Environment**.

Pad

1 Right-click **Materials** and choose **Blank Material**.

2 Select Domain 3 only.

3 In the **Settings** window for **Material**, type `Pad` in the **Label** text field.

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

Property	**Variable**	**Value**	**Unit**	**Property group**
Young's modulus	E	10[GPa]	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.1	1	Young's modulus and Poisson's ratio
Density	rho	2000	kg/m^3	Basic

5 Click to expand the **Appearance** section. From the **Color** list, choose **Black**.

SOLID MECHANICS (SOLID)

Wear is a slow process and inertial effects can be neglected.

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

2 In the **Settings** window for **Solid Mechanics**, locate the **Structural Transient Behavior** section.

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

Fixed Constraint 1

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

2 Select Boundary 8 only.

Symmetry 1

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

2 Select Boundary 3 only.

Prescribed Displacement 1

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

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

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

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

5 Select Boundary 16 only.

Boundary Load 1

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

2 Select Boundary 16 only.

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

4 From the **Load type** list, choose **Total force**.

5 Specify the $\mathbf{F}_{\mathrm{tot}}$ vector as

0	x
0	y
-10[kN]	z

Contact 1

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

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

3 From the list, choose **Augmented Lagrangian**.

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

Slip Velocity 1

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

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

3 In the μ text field, type `0.45`.

4 Locate the **Prescribed Velocity** section. Specify the $\mathbf{v}_{\mathrm{slip}}$ vector as

omega*sys2.r	t1
0	t2

The input is given in the boundary system selected in the **Coordinate System Selection** section.

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

6 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Advanced Physics Options**.

7 Click **OK**.

8 In the **Settings** window for **Slip Velocity**, click to expand the **Advanced** section.

9 Select the **Store accumulated slip** check box.

Contact 1

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

Wear 1

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

2 In the **Settings** window for **Wear**, locate the **Wear Model** section.

3 In the k_{wear} text field, type `1e-14*wear_accel`.

The factor `wear_accel` artificially accelerates the wear rate in order to reduce the computation time. In this case `wear_accel` is set to 100 and the time-dependent solver spans an interval of only 5 s. Alternatively, this model could be run with no acceleration factor but over a time span of 500 s. This would yield practically identical results, though at the cost of a significantly higher computation time.

MESH 1

Free Triangular 1

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

2 Select Boundaries 4 and 8 only.

Size Expression 1

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

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

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

4 Select Boundaries 4 and 8 only.

5 Locate the **Element Size Expression** section. In the **Size expression** text field, type `if((x/60[mm])^2+((y-115[mm])/40[mm])^2<1, 6[mm], 40[mm])`.

6 Click to expand the **Smoothing** section. In the **Maximum size field growth rate** text field, type `1.25`.

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 Domains 1 and 2 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 `2`.

Free Triangular 2

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

2 Select Boundary 16 only.

Size 1

1 Right-click **Free Triangular 2** and choose **Size**.

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 **Maximum element size** check box. In the associated text field, type `3.5[mm]`.

Swept 2

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

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 `3`.

4 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**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Deformed geometry (Component 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 **Contact pressure (comp1.solid.Tn_ap1)**.

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

5 In the **Scale** text field, type `5e6`.

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

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>Contact Forces (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.

6 In the **Home** toolbar, click **Add Predefined Plot** to open the **Add Predefined Plot** window.

RESULTS

Add a predefined contact plot to verify that the initial friction forces on the brake pad, as computed by the **Slip Velocity** node, act in the expected direction.

Contact 1, Pressure

1 In the **Model Builder** window, expand the **Contact Forces (solid)** node.

2 Right-click **Contact 1, Pressure** and choose **Disable**.

Color Expression

1 In the **Model Builder** window, expand the **Contact 1, Friction Force** node, then click **Color Expression**.

2 In the **Settings** window for **Color Expression**, locate the **Coloring and Style** section.

3 From the **Coloring** list, choose **Color table**.

4 Click **Change Color Table**.

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

6 Click **OK**.

ADD STUDY

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

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

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

4 Click **Add Study** in the window toolbar.

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

STUDY 2

Step 1: Time Dependent

1 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

2 In the **Output times** text field, type `range(0, 0.1, 5)`.

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

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

5 From the **Study** list, choose **Study 1, Stationary**.

6 In the **Study** toolbar, click **Get Initial Value**.

Create a **Transformation 3D** dataset to easily visualize the disc's rotation with an arrow plot.

RESULTS

Transformation 3D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Transformation 3D**.

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

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

4 Locate the **Transformation** section. Select the **Rotate** check box.

5 In the **Angle** text field, type `-omega*t/200`.

Surface 1

1 In the **Results** toolbar, click **More Datasets** and choose **Surface**.

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

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

4 Locate the **Parameterization** section. From the **x- and y-axes** list, choose **XY-plane**.

5 Locate the **Selection** section. Click **Paste Selection**.

6 In the **Paste Selection** dialog box, type `15` in the **Selection** text field.

7 Click **OK**.

Wear Depth

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

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

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

4 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

5 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

6 Locate the **Color Legend** section. Select the **Show units** check box.

Volume 1

1. Right-click **Wear Depth** and choose **Volume**.
2. In the **Settings** window for **Volume**, locate the **Expression** section.
3. In the **Expression** text field, type `1`.

Selection 1

1. Right-click **Volume 1** and choose **Selection**.
2. Select Domains 1 and 2 only.

Material Appearance 1

In the **Model Builder** window, right-click **Volume 1** and choose **Material Appearance**.

Arrow Surface 1

1. In the **Model Builder** window, right-click **Wear Depth** and choose **Arrow Surface**.
2. In the **Settings** window for **Arrow Surface**, locate the **Data** section.
3. From the **Dataset** list, choose **Transformation 3D 1**.
4. Locate the **Expression** section. In the **x-component** text field, type `y*omega`.
5. In the **y-component** text field, type `-x*omega`.
6. In the **z-component** text field, type `0`.
7. Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type `100`.
8. Locate the **Coloring and Style** section. From the **Color** list, choose **Black**.

Selection 1

1. Right-click **Arrow Surface 1** and choose **Selection**.
2. Select Boundary 4 only.

Surface 1

1. In the **Model Builder** window, right-click **Wear Depth** and choose **Surface**.
2. In the **Settings** window for **Surface**, locate the **Data** section.
3. From the **Dataset** list, choose **Surface 1**.
4. Locate the **Expression** section. In the **Expression** text field, type `solid.htot`.
5. In the **Unit** field, type `um`.
6. Click to expand the **Quality** section. Locate the **Coloring and Style** section. Click **Change Color Table**.
7. In the **Color Table** dialog box, select **Wave>WaveLight** in the tree.
8. Click **OK**.

Translation 1

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

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

3 In the **z** text field, type `2[mm]`.

STUDY 2

Step 1: Time Dependent

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

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

3 Select the **Plot** check box.

4 From the **Plot group** list, choose **Wear Depth**.

Solver Configurations

In the **Model Builder** window, expand the **Study 2>Solver Configurations** node.

Solution 2 (sol2)

1 In the **Model Builder** window, expand the **Study 2>Solver Configurations> Solution 2 (sol2)>Dependent Variables 1** node, then click **Contact pressure (comp1.solid.Tn_ap1)**.

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

3 In the **Scale** text field, type `5e6`.

4 In the **Model Builder** window, click **Time-Dependent Solver 1**.

5 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

6 From the **Steps taken by solver** list, choose **Strict**.

7 Find the **Algebraic variable settings** subsection. From the **Consistent initialization** list, choose **Off**.

8 In the **Model Builder** window, expand the **Study 2>Solver Configurations> Solution 2 (sol2)>Time-Dependent Solver 1>Segregated 1** node, then click **Solid Mechanics**.

9 In the **Settings** window for **Segregated Step**, locate the **General** section.

10 Under **Variables**, click + **Add**.

11 In the **Add** dialog box, select **Material mesh displacement (comp1.material.disp)** in the **Variables** list.

12 Click **OK**.

13 In the **Settings** window for **Segregated Step**, click to expand the **Method and Termination** section.

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

15 From the **Termination technique** list, choose **Tolerance**.

16 In the **Model Builder** window, right-click **Material Frame Variables** and choose **Delete**.

17 In the **Home** toolbar, click **Compute**.

RESULTS

Arrow Surface 1

1 In the **Model Builder** window, under **Results>Wear Depth** click **Arrow Surface 1**.

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

3 From the **Solution parameters** list, choose **From parent**.

Surface 1

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

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

3 From the **Solution parameters** list, choose **From parent**.

To easily synchronize the output time used in different plot groups, add a **Single-Select Solution** node.

Single-Select Solution 1

1 In the **Results** toolbar, click **Configurations** and choose **Single-Select Solution**.

2 In the **Settings** window for **Single-Select Solution**, locate the **Solution** section.

3 From the **Solution** list, choose **Solution 2 (sol2)**.

Plot the contact pressure. In order to correctly interpolate from discrete Gauss point data to a continuous field, the pressure variable is wrapped in a `gpeval()` operator.

Pressure Distribution

1 In the **Results** toolbar, click **2D Plot Group**.

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

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

4 From the **Solution parameters** list, choose **From configuration**.

Surface 1

1 Right-click **Pressure Distribution** and choose **Surface**.

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

3 In the **Expression** text field, type `gpeval(4, solid.dcnt1.wear1.Tn)`.

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

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

Pressure Distribution

In the **Model Builder** window, right-click **Pressure Distribution** and choose **Duplicate**.

Slip Distance

1 In the **Model Builder** window, under **Results** click **Pressure Distribution 1**.

2 In the **Settings** window for **2D Plot Group**, type `Slip Distance` in the **Label** text field.

Surface 1

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

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

3 In the **Expression** text field, type `gpeval(4, solid.sliptot)*wear_accel`.

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

5 In the **Slip Distance** toolbar, click **Plot**.

Slip Distance

In the **Model Builder** window, right-click **Slip Distance** and choose **Duplicate**.

Wear Rate

1 In the **Model Builder** window, under **Results** click **Slip Distance 1**.

2 In the **Settings** window for **2D Plot Group**, type `Wear Rate` in the **Label** text field.

Surface 1

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

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

3 In the **Expression** text field, type `gpeval(4, solid.h_tEff)`.

4 In the **Unit** field, type `um/s`.

5 In the **Wear Rate** toolbar, click **Plot**.

Evaluation Group 1

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

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

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

4 From the **Time selection** list, choose **Last**.

Surface Average 1

1 Right-click **Evaluation Group 1** and choose **Average>Surface Average**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `15` in the **Selection** text field.

5 Click **OK**.

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

7 In the table, enter the following settings:

Expression	Unit	Description
`solid.htot`	`um`	`Average wear depth`

Surface Minimum 1

1 In the **Model Builder** window, right-click **Evaluation Group 1** and choose **Minimum>Surface Minimum**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `15` in the **Selection** text field.

5 Click **OK**.

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

7 In the table, enter the following settings:

Expression	Unit	Description
`solid.htot`	`um`	`Minimum wear depth`

Surface Maximum 1

1 Right-click **Evaluation Group 1** and choose **Maximum>Surface Maximum**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `15` in the **Selection** text field.

5 Click **OK**.

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

7 In the table, enter the following settings:

Expression	**Unit**	**Description**
`solid.htot`	`um`	`Maximum wear depth`

8 In the **Evaluation Group 1** toolbar, click **=** **Evaluate**.