Model created in COMSOL Multiphysics 6.4

Transient Rolling Contact

Introduction

This conceptual example shows how to handle a transient contact problem with stick–slip friction transition. A soft hollow pipe subjected to a gravity load is released at the top of a half-pipe. The motion varies between sliding and rolling, depending on the position in the half-pipe and the velocity of the pipe. The pipe deforms such that its cross-section becomes oval due to the contact and the inertial forces. An analysis of the energy balance validates the accuracy of the solution.

Model Definition

As illustrated in Figure 1, the geometry consists of a section cut from a hollow pipe and a half-pipe. The pipe radius is 15 cm and the thickness is 2 cm. The half-pipe radius is about 1 m and has a transition length of 50 cm.

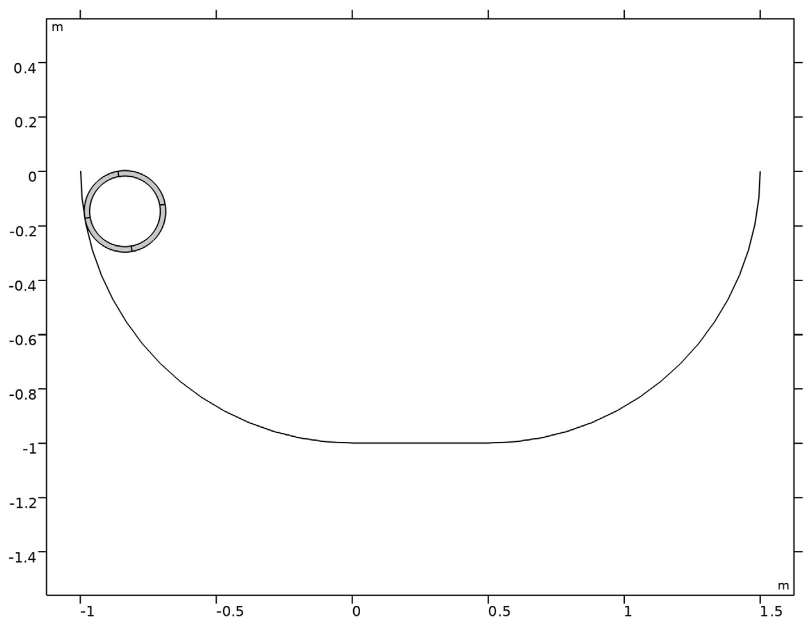


Figure 1: Model geometry.

The half-pipe is rigid, so it is modeled only by a meshed edge without any physics defined. The hollow pipe is subjected to gravity and dropped in the half-pipe with its centroid at a height of 75 cm above the horizontal plane. The pipe is always in contact with the half-pipe.

The friction coefficient is defined as a function of the slip velocity through the exponential dynamic Coulomb friction model:

$$\mu = \mu_{\mathrm{dyn}} + (\mu_{\mathrm{stat}} - \mu_{\mathrm{dyn}})e^{-\alpha\|\mathbf{v}\|} \tag{1}$$

Here, $\mu_{\mathrm{dyn}} = 0.3$ is the dynamic friction coefficient, and $\mu_{\mathrm{stat}} = 0.5$ is the static friction coefficient. The friction decay coefficient $\alpha = 1$, and $\mathbf{v}$ is the slip velocity.

The solution is computed for 4 s. The pipe displacement and the energy balance are the quantities of interest.

Results and Discussion

Figure 2 shows the von Mises stress distribution in the pipe at the final step together with the trajectory of a point located on the outer surface of the pipe. You can notice the deformation of the pipe due to gravity, and that the trajectory path clearly shows a transition between the stick and slip friction stages. In the former, the trajectory is smoothly following the rotation of the pipe, while in the latter stage the trajectory has a slightly more elongated path. Figure 3 below shows the position of the pipe at every 0.4 s.

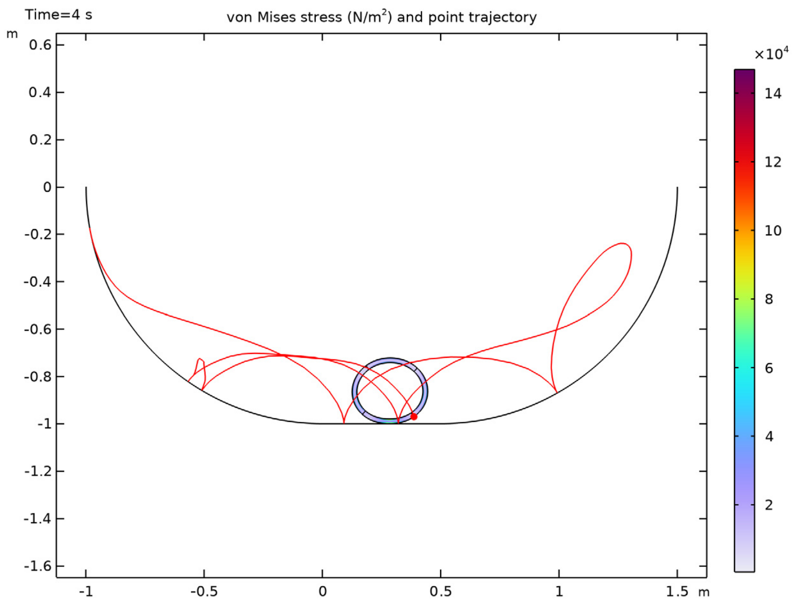


Figure 2: Stress distribution and point trajectory of the pipe.

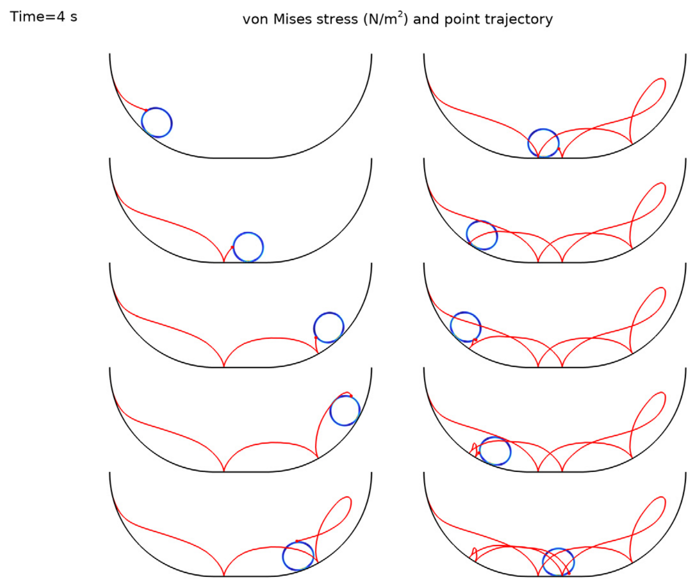


Figure 3: Position of the pipe at every 0.4 s.

In Figure 4, you can see the energy balance. As expected, the potential energy decreases as the kinetic energy increases. Because of frictional dissipation, the pipe never reaches its original height in the half-pipe. Most of the energy is lost due to friction when the pipe reaches the steeper slope region of the half-pipe. After 2 s, the pipe remains in the region with lower slope and will roll rather than slide. Note also that part of the total energy is stored as elastic strain energy, due to the deformation of the pipe. Lastly, Figure 5 shows the friction coefficient as function of time.

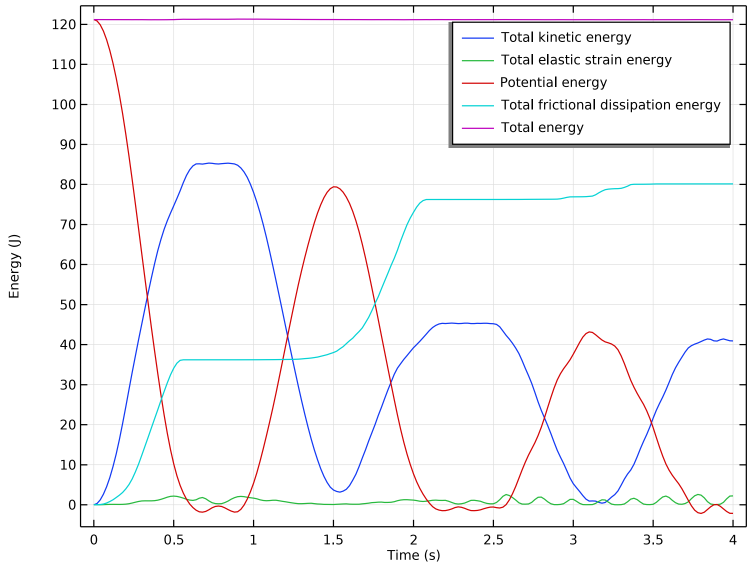


Figure 4: Energy balance versus time.

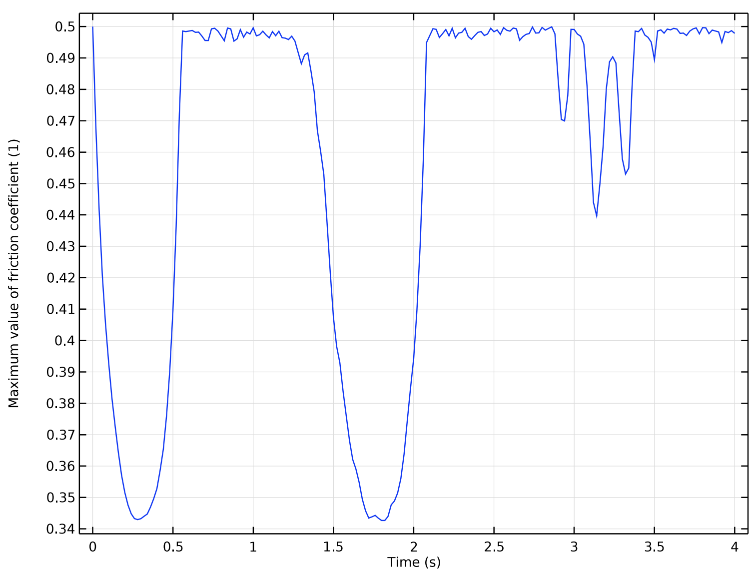


Figure 5: Friction coefficient versus time.

Notes About the COMSOL Implementation

To speed up the computation, manual time stepping is used to solve the model. Note, however, that for manual time stepping there is no error control, so it is good practice to inspect the total energy in a probe plot to verify the energy balance. In Figure 6 you can see that the conservation of the total energy is fulfilled for all time steps taken by the solver.

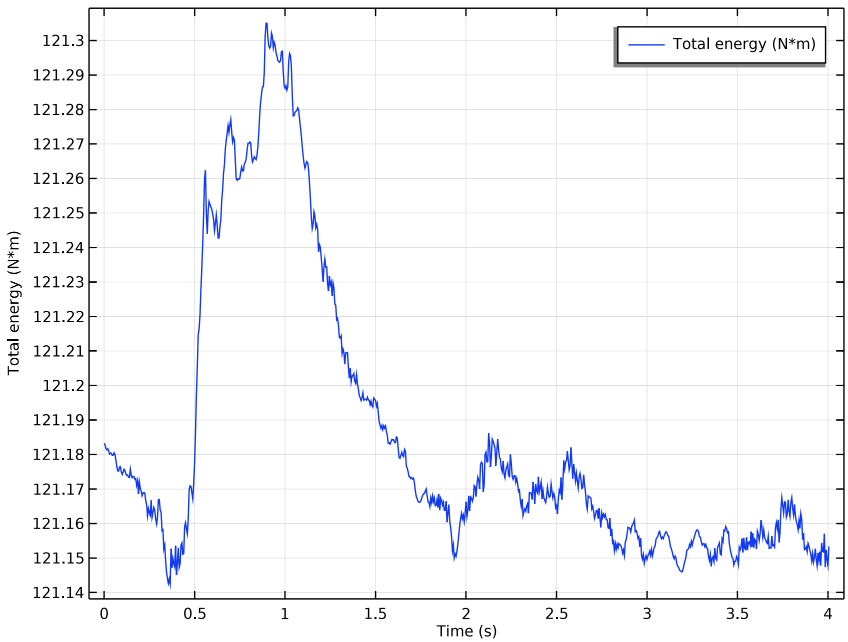


Figure 6: Total energy versus time probe plot. Note the scale on the y-axis.

Application Library path: `Structural_Mechanics_Module/Contact_and_Friction/transient_rolling_contact`

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 **2D**.
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 > Time Dependent**.
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
rp	15[cm]	0.15 m	Pipe radius
th	2[cm]	0.02 m	Pipe thickness
theta	10[deg]	0.17453 rad	Auxiliary geometric parameter

GEOMETRY 1

Circular Arc 1 (ca1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Circular Arc**.
2 In the **Settings** window for **Circular Arc**, locate the **Properties** section.
3 From the **Specify** list, choose **Endpoints and start angle**.
4 Locate the **Starting Point** section. In the **x** text field, type -1.
5 Locate the **Endpoint** section. In the **y** text field, type -1.
6 Locate the **Angles** section. In the **Start angle** text field, type 180.

Circular Arc 2 (ca2)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Circular Arc**.
2 In the **Settings** window for **Circular Arc**, locate the **Properties** section.

3 From the **Specify** list, choose **Endpoints and start angle**.

4 Locate the **Starting Point** section. In the **x** text field, type `0.5`.

5 In the **y** text field, type `-1`.

6 Locate the **Endpoint** section. In the **x** text field, type `1.5`.

7 In the **y** text field, type `0`.

8 Locate the **Angles** section. In the **Start angle** text field, type `270`.

9 Click **Build All Objects**.

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 On the object **ca1**, select Point 2 only.

3 In the **Settings** window for **Line Segment**, locate the **Endpoint** section.

4 Click to select the **Activate Selection** toggle button for **End vertex**.

5 On the object **ca2**, select Point 1 only.

Convert to Curve 1 (ccur1)

1 In the **Geometry** toolbar, click **Conversions** and choose **Convert to Curve**.

2 Click in the **Graphics** window and then press Ctrl+A to select all objects.

3 In the **Settings** window for **Convert to Curve**, locate the **Selections of Resulting Entities** section.

4 Find the **Cumulative selection** subsection. Click **New**.

5 In the **New Cumulative Selection** dialog, type `Contact Source` in the **Name** text field.

6 Click **OK**.

7 In the **Settings** window for **Convert to Curve**, click **Build Selected**.

Circle 1 (c1)

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

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `rp`.

4 Locate the **Position** section. In the **x** text field, type `-1+rp`.

5 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	th

Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

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

4 On the object **c1**, select Domain 5 only.

Rotate 1 (rot1)

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

2 Select the object **del1** only.

3 In the **Settings** window for **Rotate**, locate the **Rotation** section.

4 In the **Angle** text field, type `theta`.

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 Clear the **Create pairs** checkbox.

5 Click **Build Selected**.

MATERIALS

Material 1 (mat1)

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

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

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	`5[MPa]`	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`0.3`	1	Young's modulus and Poisson's ratio
Density	rho	`1e3[kg/m^3]`	kg/m³	Basic

SOLID MECHANICS (SOLID)

Gravity 1

In the **Physics** toolbar, click **Global** and choose **Gravity**.

DEFINITIONS

Contact Destination

1 In the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for **Explicit**, type `Contact Destination` in the **Label** text field.
3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4 Select the **Group by continuous tangent** checkbox.
5 Select Boundaries 8, 9, 12, and 15 only.

Contact Pair 1 (p1)

1 In the **Definitions** toolbar, click **Pairs** and choose **Contact Pair**.
2 In the **Settings** window for **Pair**, locate the **Source Boundaries** section.
3 From the **Selection** list, choose **Contact Source**.
4 Locate the **Destination Boundaries** section. Click to select the **Activate Selection** toggle button.
5 From the **Selection** list, choose **Contact Destination**.

SOLID MECHANICS (SOLID)

Contact 1

The stick-slip friction is better resolved using the **Augmented Lagrangian** method.

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics (solid)** 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 **Penalty factor control** list, choose **Manual tuning**.
5 From the **Use relaxation** list, choose **Never**.

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 From the **Friction model** list, choose **Exponential dynamic Coulomb**.

4 In the μ_{stat} text field, type `0.5`.

5 In the μ_{dyn} text field, type `0.3`.

6 In the α_{dcf} text field, type `1`.

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

8 In the **Show More Options** dialog, in the tree, select the checkbox for the node **Physics** > **Advanced Physics Options**.

9 Click **OK**.

10 In the **Settings** window for **Friction**, click to expand the **Advanced** section.

11 Select the **Compute frictional dissipation** checkbox.

DEFINITIONS

Integration 1 (intop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

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

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

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	**Expression**	**Unit**	**Description**
V	intop1(g_const*solid.rho*(y+0.85[m]))*1[m]	N·m	Potential energy

Maximum 1 (maxop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Maximum**.

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

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

4 From the **Selection** list, choose **Contact Destination**.

5 Locate the **Advanced** section. From the **Point type** list, choose **Integration points**.

Variables 1

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

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
mu_max	maxop1(if(solid.dcnt1.isContact_p1,solid.dcnt1.mu_fric,0))		Maximum value of friction coefficient

MESH 1

Edge 1

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

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

3 From the **Selection** list, choose **Contact Source**.

Size 1

1 Right-click **Edge 1** 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** checkbox. In the associated text field, type `0.005`.

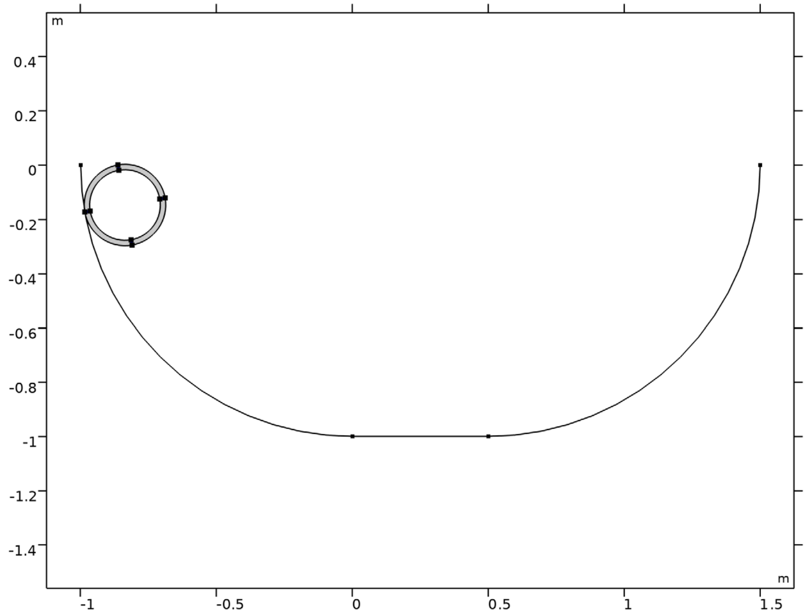
Mapped 1

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

Distribution 1

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

2 Select Boundaries 4–7 only.



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

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

Distribution 2

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

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

3 From the **Selection** list, choose **Contact Destination**.

4 Locate the **Distribution** section. In the **Number of elements** text field, type `20`.

STUDY 1

Step 1: Time Dependent

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

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

3 In the **Output times** text field, type `range(0,2e-2,4)`.

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_p1)**.

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

5 In the **Scale** text field, type `5e4`.

6 Continue with setting manual scaling for the following variables:

Variable name	Scale
comp1.solid.Tn_p1	5e4
comp1.solid.Tt_p1	5e4
comp1.solid.Wfric_p1	5e-3*5e4
comp1.u	5e-3

7 In the **Model Builder** window, under **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** click **Time-Dependent Solver 1**.

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

9 From the **Steps taken by solver** list, choose **Manual**.

10 In the **Time step** text field, type `5e-3`.

11 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Time-Dependent Solver 1** node, then click **Segregated 1**.

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

13 In the **Tolerance factor** text field, type `0.1`.

14 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Time-Dependent Solver 1** > **Segregated 1** node, then click **Solid Mechanics**.

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

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

17 From the **Jacobian update** list, choose **On every iteration**.

18 From the **Termination technique** list, choose **Iterations or tolerance**.

DEFINITIONS

Variables 1

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

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
`W_tot`	`solid.Wk_tot+solid.Ws_tot+V+ solid.Wfric_tot`	N·m	`Total energy`

Global Variable Probe 1 (var1)

1 In the **Definitions** toolbar, click **Probes** and choose **Global Variable Probe**.

2 In the **Settings** window for **Global Variable Probe**, locate the **Expression** section.

3 In the **Expression** text field, type `W_tot`.

STUDY 1

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

RESULTS

Stress (solid)

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

2 From the **Title type** list, choose **Manual**.

3 In the **Title** text area, type `von Mises stress (N/m<sup>2</sup>) and point trajectory`.

4 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (x, y, z)**.

Point Trajectories 1

1 In the **Stress (solid)** toolbar, click **More Plots** and choose **Point Trajectories**.

2 In the **Settings** window for **Point Trajectories**, locate the **Trajectory Data** section.

3 From the **Plot data** list, choose **Points**.

4 In the **X-expression** text field, type `u-cos(theta)`.

5 In the **Y-expression** text field, type `v-sin(theta)`.

6 Select Point 5 only.

7 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Color** list, choose **Red**.

8 Find the **Point style** subsection. From the **Type** list, choose **Point**.

9 Select the **Radius scale factor** checkbox. In the associated text field, type `15`.

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

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

Animation 1

1 In the **Results** toolbar, click **Animation** and choose **Player**.

2 In the **Settings** window for **Animation**, locate the **Scene** section.

3 From the **Subject** list, choose **Stress (solid)**.

Surface 1

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

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

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

Animation 1

1 In the **Model Builder** window, under **Results > Export** click **Animation 1**.

2 In the **Settings** window for **Animation**, locate the **Frames** section.

3 From the **Frame selection** list, choose **All**.

4 Locate the **Playing** section. In the **Display each frame for** text field, type `1e-1`.

5 Click the **Play** button in the **Graphics** toolbar.

Energy

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

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

3 Click to expand the **Title** section. From the **Title type** list, choose **None**.

4 Locate the **Plot Settings** section.

5 Select the **y-axis label** checkbox. In the associated text field, type `Energy (J)`.

Global 1

1 Right-click **Energy** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
solid.Wk_tot	J	Total kinetic energy
solid.Ws_tot	J	Total elastic strain energy
V	N*m	Potential energy
solid.Wfric_tot	J	Total frictional dissipation energy
W_tot	J	Total energy

4 In the **Energy** toolbar, click **Plot**.

Friction Coefficient

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

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

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

4 Locate the **Legend** section. Clear the **Show legends** checkbox.

Global 1

1 Right-click **Friction Coefficient** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
mu_max	1	Maximum value of friction coefficient

4 In the **Friction Coefficient** toolbar, click **Plot**.