Model created in COMSOL Multiphysics 6.4

Needle Penetration

Introduction

Modeling needle penetration into soft materials such as gelatin involves several mechanical challenges, including large deformations, high friction, contact forces, and material failure. In this example, the insertion process into tissue-like material is simulated using a cohesive zone model to represent fracture behavior. The setup is inspired by the study presented in Ref. 1.

Model Definition

A fully resolved 3D model of crack propagation would require excessive computational resources. Therefore, a plane-strain approximation is used in this example. In this approach, the needle (see Figure 1) is represented by a rigid body represented by a moving boundary mesh, prescribed to penetrate a gelatin block at a constant velocity of 10 mm/s.

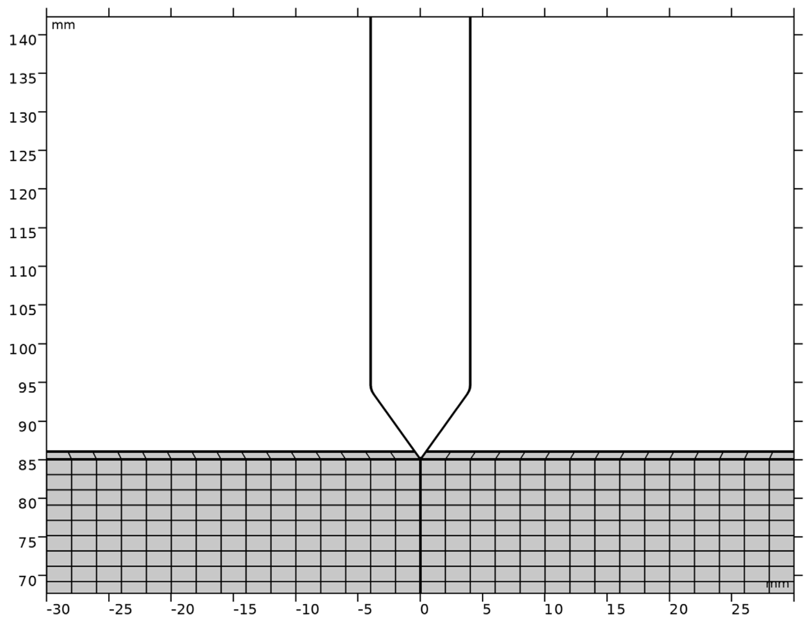

Figure 1: Model with a cohesive zone formulation along the center line of the gelatin block.

To further simplify the analysis, the crack path is predefined along the centerline. Fracture behavior along this path is modeled using a displacement-based linear traction–separation law. The gelatin is described as a Neo-Hookean hyperelastic material with a high Poisson's ratio to approximate incompressibility. The material properties used in the simulation are summarized in the following table.

Material properties	Value	Description
G_c	17.55[J/m^2]	Energy release rate
K	1.55[N/m^3]	Initial stiffness of the traction law
σ	4648[Pa]	Peak stress of the traction law
E	5[kPa]	Young's modulus of gelatin
υ	0.485	Poisson's ratio of gelatin

The sides of the gelatin block are fixed, while the bottom boundary is subject to a roller constraint. Two contact pairs are defined, one between the needle and the gelatin, and another along the predefined crack path in the gelatin block.

Results and Discussion

Figure 2 shows the reaction force as a function of needle insertion depth. The noisy curve arises from the undamped dynamic response and the stick–slip behavior between the needle and the gelatin.

Before puncture (i.e., prior to point A in Figure 2 and illustrated in Figure 3), the external work is stored as strain energy until the onset of crack propagation, at which point a slight relaxation occurs due to damage softening.

Figure 2: Reaction force versus insertion depth. Crack propagation begins at point A. At point B, deeper penetration increases the contact surface, raising frictional forces. At point C, the needle tip emerges from the bottom, and the reaction force reaches equilibrium.

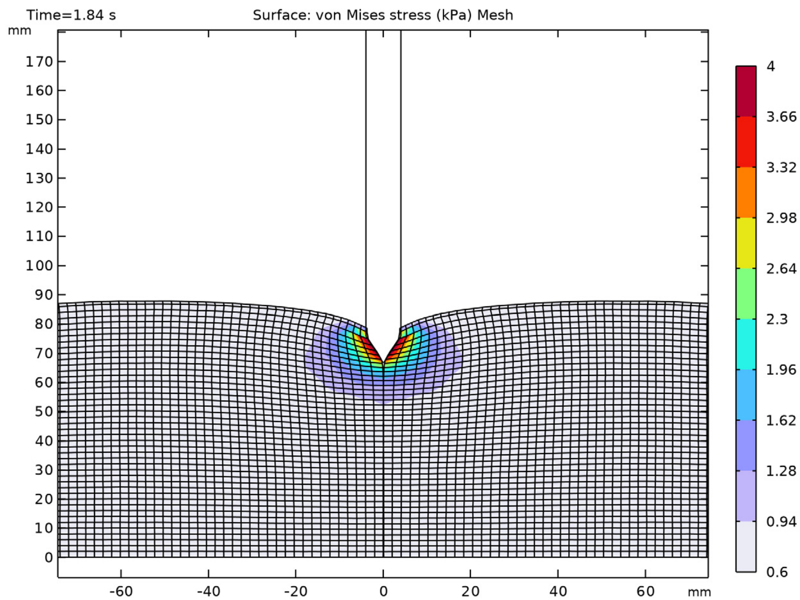

Figure 3: von Mises stress right before the needle tip penetrates the gelatin block.

Between point A and point B, the external work is balanced between further crack propagation, deformation of the material to open the crack, and overcoming friction. As the contact area between the needle and the gelatin increases, the reaction force gradually increases, see Figure 4.

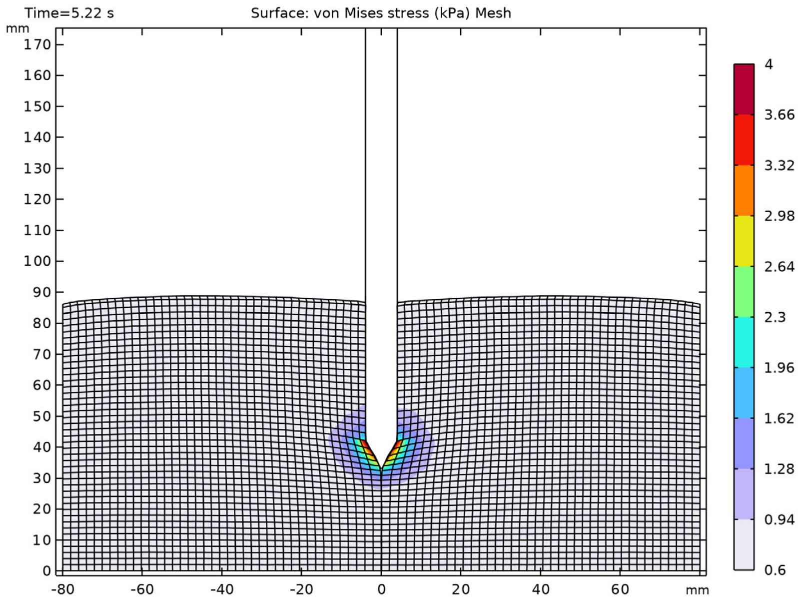

Figure 4: During crack propagation.

Around point C, when the needle tip exits the bottom surface (Figure 5), no further crack opening nor damage occurs, leading to a drop in force. The reaction force stabilizes at a steady-state value dominated by frictional sliding.

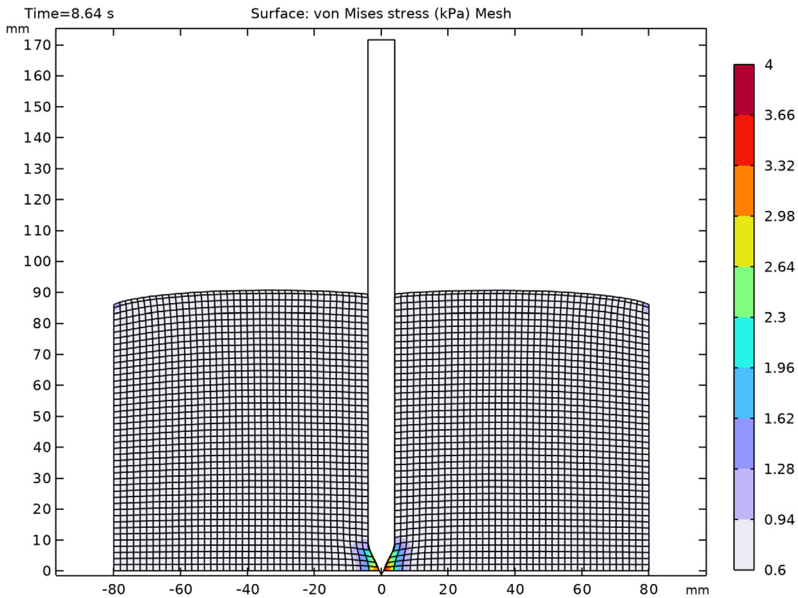

Figure 5: Needle tip starts to emerge from the bottom of the gelatin block.

Reference

1. M. Oldfield, D. Dini, G. Giordano, and F. Rodriguez y Baena, "Detailed finite element modelling of deep needle insertions into a soft tissue phantom using a cohesive approach," *Comput. Methods Biomech. Biomed. Eng.*, vol. 16, pp. 530–543, 2013; dx.doi.org/10.1080/10255842.2011.628448.

Application Library path: `Nonlinear_Structural_Materials_Module/Hyperelasticity/needle_penetration`

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** > **Explicit Dynamics** > **Solid Mechanics, Explicit Dynamics (solid)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces** > **Explicit Dynamics**.

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 `needle_penetration_parameters.txt`.

GEOMETRY 1

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

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Rectangle 1 (r1)

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

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

3 In the **Width** text field, type `W`.

4 In the **Height** text field, type `H`.

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

Layer name	Thickness (mm)
Layer 1	1

6 Select the **Layers on top** checkbox.

7 Clear the **Layers on bottom** checkbox.

Polygon 1 (pol1)

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

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:

x (mm)	y (mm)
0	85
4	94
4	3*H
0	3*H
0	85

4 Click **Build Selected**.

Difference 1 (dif1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **r1** only.

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

4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

5 Select the object **pol1** only.

6 Select the **Keep objects to subtract** checkbox.

7 Click **Build Selected**.

Mirror 1 (mir1)

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

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

3 In the **Settings** window for **Mirror**, locate the **Input** section.

4 Select the **Keep input objects** checkbox.

5 Click **Build Selected**.

Union 1 (uni1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Select the objects **mir1(2)** and **pol1** only.

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

4 Clear the **Keep interior boundaries** checkbox.

5 Click **Build Selected**.

Fillet 1 (fil1)

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

2 On the object **uni1**, select Point 3 only.

3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type `0.1`.

5 Click **Build Selected**.

Fillet 2 (fil2)

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

2 On the object **fil1**, select Points 1 and 6 only.

3 In the **Settings** window for **Fillet**, locate the **Radius** section.

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

5 Click **Build Selected**.

Convert to Curve 1 (ccur1)

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

2 Select the object **fil2** only.

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

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**.

DEFINITIONS

Define a Contact Pair for both needle and gelatin block, and an additional pair to describe the interaction between the two gelatin segments. Ensure that the needle is the source as it is a rigid body in the context of Contact.

Contact Pair 1 (p1)

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

2 Select Boundary 17 only.

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

4 Click to select the **Activate Selection** toggle button.

5 Select Boundary 7 only.

6 Locate the **Advanced** section. From the **Mapping method** list, choose **Initial configuration**.

Contact Pair 2 (p2)

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

2 Select Boundaries 8–16 only.

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

4 Click to select the **Activate Selection** toggle button.

5 Select Boundaries 6, 7, 17, and 19 only.

MATERIALS

Gelatin

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

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Manual**.

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

Property	Variable	Value	Unit	Property group
Young's modulus	E	6[kPa]	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.485	1	Young's modulus and Poisson's ratio
Density	rho	720	kg/m³	Basic

COMPONENT 1 (COMP1)

Prescribed Deformation 1

1 In the **Physics** toolbar, click **Moving Mesh** and choose **Prescribed Deformation**.

2 In the **Settings** window for **Prescribed Deformation**, locate the **Geometric Entity Selection** section.

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

4 Select Boundaries 8–16 only.

5 Locate the **Prescribed Deformation** section. Specify the dx vector as

vel*t	Y

The Moving Mesh feature enables the creation of a pseudo rigid body by selecting a geometric entity such as a line and prescribing a deformation. Since it lacks a material model, this body cannot deform, but it can still participate in a Contact Pair within the physics interface.

SOLID MECHANICS, EXPLICIT DYNAMICS (SOLID)

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

2 In the **Settings** window for **Solid Mechanics, Explicit Dynamics**, locate the **Thickness** section.

3 In the d text field, type `thickness`.

Hyperelastic Material 1

1 In the **Physics** toolbar, click **Domains** and choose **Hyperelastic Material**.

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

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

4 Locate the **Quadrature Settings** section. Clear the **Reduced integration** checkbox.

Contact - Gelatin

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

2 In the **Settings** window for **Contact**, type `Contact - Gelatin` in the **Label** text field.

3 Locate the **Contact Pressure Penalty Factor** section. From the **Penalty factor control** list, choose **Manual tuning**.

4 In the f_p text field, type `0.5`.

Adhesion 1

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

2 In the **Settings** window for **Adhesion**, locate the **Adhesive Activation** section.

3 From the **Activation criterion** list, choose **Always active**.

4 Locate the **Adhesive Stiffness** section. From the **Adhesive stiffness** list, choose **User defined**.

5 Specify the **k** vector as

K	tl
K	n

Contact - Gelatin

In the **Model Builder** window, click **Contact - Gelatin**.

Decohesion 1

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

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

3 In the σ_{ts} text field, type `sigma`.

4 In the σ_{ss} text field, type `sigma`.

5 In the G_{ct} text field, type `Gc`.

6 In the G_{cs} text field, type `Gc`.

Contact - Needle-Gelatin

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

2 In the **Settings** window for **Contact**, type `Contact - Needle-Gelatin` in the **Label** text field.

3 Locate the **Pair Selection** section. Click **Add**.

4 In the **Add** dialog, select **Contact Pair 2 (p2)** in the **Pairs** list.

5 Click **OK**.

6 In the **Settings** window for **Contact**, locate the **Contact Pressure Penalty Factor** section.

7 From the **Penalty factor control** list, choose **Manual tuning**.

8 In the f_p text field, type `5`.

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`.

4 Locate the **Friction Force Penalty Factor** section. From the **Penalty factor control** list, choose **From parent**.

Fixed Constraint 1

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

2 Select Boundaries 1, 3, 22, and 23 only.

Roller 1

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

2 Select Boundaries 2 and 18 only.

MESH 1

Mapped 1

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

Size

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

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

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

4 Click **Build Selected**.

Mapped 1

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

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

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

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

5 Click **Build Selected**.

Size 1

1 Right-click **Mapped 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 2.

Edge 1

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

2 Select Boundaries 8–16 only.

3 In the **Settings** window for **Edge**, click **Build Selected**.

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 0.5.

6 Click **Build Selected**.

STUDY 1

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

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

3 Clear the **Generate default plots** checkbox.

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

RESULT TEMPLATES

1 In the **Results** 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, Explicit Dynamics** > **Stress (solid)**.

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

RESULTS

Surface 1

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

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

3 From the **Unit** list, choose **kPa**.

4 Locate the **Coloring and Style** section. From the **Color table type** list, choose **Discrete**.

5 Click to expand the **Range** section. Select the **Manual color range** checkbox.

6 In the **Minimum** text field, type .6.

7 In the **Maximum** text field, type 4.

Mesh 1

1 In the **Model Builder** window, right-click **Stress (solid)** and choose **Mesh**.

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

3 From the **Element color** list, choose **None**.

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

STUDY 1

Solution 1 (sol1)

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

2 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Dependent Variables 1** node.

Step 1: Explicit Dynamics

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

2 In the **Settings** window for **Explicit Dynamics**, locate the **Study Settings** section.

3 In the **Output times** text field, type `range(0,0.02,10)`.

4 Click to expand the **Results While Solving** section. Select the **Plot** checkbox.

5 From the **Update at** list, choose **Time steps taken by solver**.

6 Click to expand the **Explicit Dynamics Settings** section.

7 Select the **Time step safety factor** checkbox. In the associated text field, type `1/2`.

Solution 1 (sol1)

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

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

3 From the **Method** list, choose **Manual**.

4 Locate the **Residual Scaling** section. From the **Method** list, choose **Manual**.

5 In the **Model Builder** window, under **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Dependent Variables 1** click **Displacement Field (comp1.u)**.

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

7 In the **Scale** text field, type `1`.

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

9 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

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

11 Click **Run**.

RESULTS

Force-Displacement Curve

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

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

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Force-Displacement Curve`

5 Locate the **Plot Settings** section.

6 Select the **x-axis label** checkbox. In the associated text field, type `Insertion depth (mm)`.

7 Select the **y-axis label** checkbox. In the associated text field, type `Reaction force (N)`.

Global 1

1 Right-click **Force-Displacement Curve** 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.RFtotaly	N	Reaction force

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

5 In the **Expression** text field, type `-vel*t`.

6 Click to expand the **Coloring and Style** section. From the **Color** list, choose **Red**.

7 From the **Width** list, choose **2**.

8 Click to expand the **Legends** section. Clear the **Show legends** checkbox.

Annotation 1

1 In the **Model Builder** window, right-click **Force-Displacement Curve** and choose **Annotation**.

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

3 In the **Text** text field, type `\textbf{A}`.

4 Select the **LaTeX markup** checkbox.

5 Locate the **Position** section. In the **x** text field, type `13`.

6 In the **y** text field, type `1.1`.

7 Locate the **Coloring and Style** section. Clear the **Show point** checkbox.

Annotation 2

1 Right-click **Annotation 1** and choose **Duplicate**.

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

3 In the **Text** text field, type `\textbf{B}`.

4 Locate the **Position** section. In the **x** text field, type `86`.

5 In the **y** text field, type `1.2`.

Annotation 3

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

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

3 In the **Text** text field, type `\textbf{C}`.

4 Locate the **Position** section. In the **x** text field, type `95`.

5 In the **y** text field, type `0.66`.

Animation 1

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

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