

# Prestressed Micromirror

# Introduction

One method of creating spring-like structures or inducing curvature in plated structures is to plate materials onto a substrate such that the layer has a residual stress after the plating process. The plating process can control this stress, which can be either compressive or tensile, even for similar materials. The automotive industry has studied this phenomenon at length because highly stressed chrome is appreciably shinier than nonstressed chrome. MEMS device manufacturers sometimes use this effect to create curved cantilevers or spring-loaded micromechanical structures that lift off the substrate when deliberately undercut by an etchant.

One such device is the electrostatically controlled micromirror. It is typically quite small, and arrays of such devices can implement a projection system. They serve as optical redirectors and similar reflection devices. This section shows the fundamentals of how to set up and solve lift-off of a prestressed plated device.

# Model Definition

This single-physics model uses 3D structural analysis. The micromirror has a stiff, flat, reflective center portion, which is supported by four prestressed plated cantilever springs. To keep the mesh size small and the solution time reasonable, this exercise studies the plated structure with two layers. It also assumes that the plating process creates equal and opposite (compressive and tensile) initial stresses in the top and bottom layers. This convenience makes the model straightforward to set up. You can make the initial-stress distribution as complex as desired and set it up as shown in this example. Depending on the magnitude of the deformations, you are likely best advised to solve such simulations with a large-deformation analysis using a nonlinear or parametric nonlinear solver, noting that the latter is more likely to converge. Thus this illustrative model uses the large-deformation analysis type with both the linear and parametric linear solvers.

Note in particular that a 3D structure with thin layers such as the one in this model leads to a very large unstructured tetrahedral mesh. To avoid this case, this example first generates a 2D quadrilateral mesh by mesh mapping and then extrudes it into 3D to produce a mesh with hexahedral (brick) elements. This way you can have the mesher create structured elements with a high aspect ratio.

One of the key process parameters you wish to determine in this class of problem is generally what prestress level is necessary to result in a desired lift-off. Another common concern is how much effect variations in the prestress might have on displacement. A parametric study answers this question.

# Results and Discussion

The following two images compare lift-off for aluminum and steel plates. The steel, being stiffer than aluminum, deforms less.



Volume: Displacement magnitude (mm)

Figure 1: Lift-off for aluminum.



Figure 2: Lift-off for steel.

The following two figures show the mirror response to the different levels of applied prestress. According to Figure 3, the centerpoint deflection is almost linearly dependent



on the prestress. Figure 4 shows the mirror's curvature along its centerline for different values of prestress. In line with expectations, increasing the stress bends the mirror more.

Figure 3: Total deflection of the center of the mirror for different prestress levels.



Figure 4: Mirror curvature along the centerline for different prestress levels.

# Reference

1. G. Kovacs, Micromachined Transducers Sourcebook, WCM McGraw-Hill, 1998.

Application Library path: MEMS\_Module/Actuators/micromirror

# Modeling Instructions

From the File menu, choose New.

#### NEW

In the New window, click 🙆 Model Wizard.

# MODEL WIZARD

I In the Model Wizard window, click 间 3D.

#### 6 | PRESTRESSED MICROMIRROR

- 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 M Done.

The Model Wizard starts the COMSOL Desktop with the **Geometry** node selected. We can take the chance to set the length unit to a more convenient one.

# GEOMETRY I

- I In the Model Builder window, under Component I (compl) click Geometry I.
- 2 In the Settings window for Geometry, locate the Units section.
- 3 From the Length unit list, choose mm.

Now set up some global parameters.

## **GLOBAL DEFINITIONS**

Parameters 1

I In the Model Builder window, under Global Definitions click Parameters I.

2 In the Settings window for Parameters, locate the Parameters section.

**3** In the table, enter the following settings:

Name	Expression	Value	Description
sigma_pre	5[GPa]	5E9 Pa	Initial normal stress

#### GEOMETRY I

Work Plane I (wp1)

I In the Geometry toolbar, click 📥 Work Plane.

2 In the Settings window for Work Plane, click 📥 Show Work Plane.

Work Plane I (wpI)>Plane Geometry

In the Model Builder window, click Plane Geometry.

Work Plane I (wp1)>Square I (sq1)

- I In the Work Plane toolbar, click Square.
- 2 In the Settings window for Square, click 🔚 Build Selected.

Work Plane 1 (wp1)>Square 2 (sq2)

- I In the Work Plane toolbar, click Square.
- 2 In the Settings window for Square, locate the Size section.
- **3** In the **Side length** text field, type **0.2**.
- 4 Locate the Position section. In the yw text field, type 1.
- 5 Click 틤 Build Selected.
- 6 Click the 🕂 Zoom Extents button in the Graphics toolbar.

# Work Plane I (wp1)>Chamfer I (cha1)

- I In the Work Plane toolbar, click // Chamfer.
- 2 On the object sq2, select Point 4 only.
- 3 In the Settings window for Chamfer, locate the Distance section.
- 4 In the Distance from vertex text field, type 0.1.
- 5 Click 📄 Build Selected.

Work Plane I (wp1)>Rectangle I (r1)

- I In the Work Plane toolbar, click 📃 Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- **3** In the **Width** text field, type **0.9**.
- 4 In the **Height** text field, type 0.1.
- **5** Locate the **Position** section. In the **xw** text field, type **0.2**.
- 6 In the **yw** text field, type 1.1.
- 7 Click 틤 Build Selected.

Work Plane I (wp1)>Rotate I (rot1)

- I In the Work Plane toolbar, click 💭 Transforms and choose Rotate.
- 2 Select the objects chal and rl only.
- 3 In the Settings window for Rotate, locate the Rotation section.
- 4 In the Angle text field, type range (90,90,360).
- 5 Locate the Center of Rotation section. In the xw text field, type 0.5.
- 6 In the **yw** text field, type 0.5.
- 7 Click 틤 Build Selected.

#### Extrude I (extI)

- In the Model Builder window, under Component I (compl)>Geometry I right-click
  Work Plane I (wpl) and choose Extrude.
- 2 In the Settings window for Extrude, locate the Distances section.
- **3** In the table, enter the following settings:

#### Distances (mm)

- 0.02
- 0.04

# 4 Click 틤 Build Selected.

This creates two layers with thickness of 0.02 mm each (not one of 0.04 mm on top of one of 0.02 mm).

## Form Union (fin)

- I In the Model Builder window, click Form Union (fin).
- 2 In the Settings window for Form Union/Assembly, click 🖺 Build Selected.
- **3** Click the **Graphics** toolbar.

#### ADD MATERIAL

- I In the Home toolbar, click 👯 Add Material to open the Add Material window.
- 2 Go to the Add Material window.
- 3 In the tree, select Built-in>Aluminum.
- 4 Click Add to Component in the window toolbar.
- 5 In the Home toolbar, click 🙀 Add Material to close the Add Material window.

## MATERIALS

#### Aluminum (mat1)

By default the first material you add applies to all domains, so you do not need to change the settings in the **Geometric Entity Selection**.

You can inspect which material properties are used and what values they take in the **Material Contents** table.

#### SOLID MECHANICS (SOLID)

#### Linear Elastic Material I

In the Model Builder window, under Component I (comp1)>Solid Mechanics (solid) click Linear Elastic Material I.

Initial Stress and Strain I

- I In the Physics toolbar, click 戻 Attributes and choose Initial Stress and Strain.
- 2 In the Settings window for Initial Stress and Strain, locate the Domain Selection section.
- 3 Click Clear Selection.
- **4** Select Domains 4, 6, 12, and 18 only.



5 Locate the Initial Stress and Strain section. In the  $S_0$  table, enter the following settings:

sigma_pre	0	0
0	sigma_pre	0
0	0	0

Linear Elastic Material I

In the Model Builder window, click Linear Elastic Material I.

Initial Stress and Strain 2

- I In the Physics toolbar, click 层 Attributes and choose Initial Stress and Strain.
- 2 In the Settings window for Initial Stress and Strain, locate the Domain Selection section.

- 3 Click Clear Selection.
- **4** Select Domains 3, 5, 11, and 17 only.

5 Locate the Initial Stress and Strain section. In the  $S_0$  table, enter the following settings:

-sigma_pre	0	0
0	-sigma_pre	0
0	0	0

# Fixed Constraint I

- I In the Physics toolbar, click 🔚 Boundaries and choose Fixed Constraint.
- **2** Select Boundaries 15–17, 20, 92, 95, 102, and 103 only.

It might be easier to select the correct boundaries by using the **Selection List** window. To open this window, in the **Home** toolbar click **Windows** and choose **Selection List**. (If you are running the cross-platform desktop, you find **Windows** in the main menu.)

**3** Click the **Graphics** toolbar.

# MESH I

Mapped I

- I In the Mesh toolbar, click  $\bigwedge$  Boundary and choose Mapped.
- 2 Select Boundaries 7, 14, 23, 38, 47, 62, 71, 88, and 97 only.

#### Distribution I

I Right-click Mapped I and choose Distribution.

# **2** Select Edges 15, 29, 57, 62, 88, 101, 134, and 177 only.



- 3 In the Settings window for Distribution, locate the Distribution section.
- 4 In the Number of elements text field, type 12.

#### Distribution 2

- I In the Model Builder window, right-click Mapped I and choose Distribution.
- 2 Select Edges 21, 28, 47, 87, 114, 142, 155, and 170 only.
- 3 In the Settings window for Distribution, locate the Distribution section.
- 4 In the Number of elements text field, type 3.

#### Distribution 3

- I Right-click Mapped I and choose Distribution.
- 2 Select Edges 7, 80, 109, and 182 only.
- 3 In the Settings window for Distribution, locate the Distribution section.
- 4 In the Number of elements text field, type 6.

# Mapped I

- I Right-click Mapped I and choose Build Selected.
- **2** Click the **Come Extents** button in the **Graphics** toolbar.

#### Swept I

I In the Mesh toolbar, click A Swept.

#### 12 | PRESTRESSED MICROMIRROR

2 In the Settings window for Swept, click 📗 Build Selected.

## STUDY I

Step 1: Stationary

- I In the Model Builder window, under Study I click Step I: Stationary.
- 2 In the Settings window for Stationary, locate the Study Settings section.
- **3** Select the **Include geometric nonlinearity** check box.

Solution 1 (soll)

- I In the Study toolbar, click **here** Show Default Solver.
- 2 In the Model Builder window, expand the Solution I (soll) node.
- 3 In the Model Builder window, expand the Study I>Solver Configurations> Solution I (soll)>Stationary Solver I node, then click Fully Coupled I.
- **4** In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.
- 5 From the Nonlinear method list, choose Constant (Newton).

Sometimes the automatic Newton nonlinear solver is too conservative, so here we choose the solver with constant damping.

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

# RESULTS

The following steps reproduce the displacement plot shown in Figure 1.

Volume 1

- I In the Model Builder window, expand the Results>Stress (solid) node, then click Volume I.
- 2 In the Settings window for Volume, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I (comp1)>Solid Mechanics> Displacement>solid.disp - Displacement magnitude - m.
- 3 In the Stress (solid) toolbar, click **I** Plot.
- **4** Click the 4 **Zoom Extents** button in the **Graphics** toolbar.

# MATERIALS

Change the plate material to steel.

# ADD MATERIAL

I In the Home toolbar, click 🙀 Add Material to open the Add Material window.

- 2 Go to the Add Material window.
- 3 In the tree, select Built-in>Structural steel.
- 4 Click Add to Component in the window toolbar.
- 5 In the Home toolbar, click 🙀 Add Material to close the Add Material window.

#### MATERIALS

Structural steel (mat2)

I Select Domains 1, 2, 7–10, and 13–16 only.

The remaining domains keep their previous material, that is aluminum.

# STUDY I

Step 1: Stationary

- I In the Model Builder window, under Study I click Step I: Stationary.
- 2 In the Settings window for Stationary, click to expand the Study Extensions section.
- 3 Select the Auxiliary sweep check box.
- 4 Click + Add.
- 5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
sigma_pre (Initial normal stress)	range(0,1,10)	GPa

To keep the sequence **Stationary Solver I** intact, disable it and generate a new solver sequence.

Solution 1 (soll)

In the Model Builder window, under Study I>Solver Configurations right-click Solution I (soll) and choose Disable.

Solution 2 (sol2)

- I In the Study toolbar, click **here** Show Default Solver.
- 2 In the Model Builder window, expand the Solution 2 (sol2) node.

Repeat the settings you previously made for Stationary Solver 1.

- 3 In the Model Builder window, expand the Study I>Solver Configurations> Solution 2 (sol2)>Stationary Solver I node, then click Fully Coupled I.
- 4 In the Settings window for Fully Coupled, locate the Method and Termination section.
- 5 From the Nonlinear method list, choose Constant (Newton).

6 In the Model Builder window, right-click Solution 2 (sol2) and choose Compute.

## RESULTS

Follow the steps below to reproduce the plot in Figure 2.

Volume 1

- I In the Model Builder window, expand the Results>Stress (solid) I node, then click Volume I.
- 2 In the Settings window for Volume, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose solid.disp Displacement magnitude m.
- 3 In the Stress (solid) I toolbar, click 💿 Plot.
- 4 Click the 🕂 Zoom Extents button in the Graphics toolbar.

Next, reproduce the plot in Figure 3, showing the total deflection at the center of the mirror as a function of the prestress level.

Cut Point 3D I

- I In the Results toolbar, click 💽 Cut Point 3D.
- 2 In the Settings window for Cut Point 3D, locate the Data section.
- 3 From the Dataset list, choose Study I/Solution 2 (sol2).
- 4 Locate the **Point Data** section. In the **X** text field, type 0.5.
- **5** In the **Y** text field, type **0.5**.
- 6 In the Z text field, type 0.04.

## ID Plot Group 3

- I In the Results toolbar, click  $\sim$  ID Plot Group.
- 2 In the Settings window for ID Plot Group, locate the Data section.
- 3 From the Dataset list, choose Cut Point 3D I.
- 4 Locate the Plot Settings section.
- 5 Select the x-axis label check box. In the associated text field, type Prestress (GPa).

## Point Graph 1

- I Right-click ID Plot Group 3 and choose Point Graph.
- In the Settings window for Point Graph, click Replace Expression in the upper-right corner of the y-Axis Data section. From the menu, choose Component I (compl)> Solid Mechanics>Displacement>solid.disp Displacement magnitude m.

3 In the ID Plot Group 3 toolbar, click 💿 Plot.

Finally, reproduce the plot in Figure 4 of the total deflection along the centerline for the prestress levels included in the parametric sweep.

# Cut Line 3D I

- I In the **Results** toolbar, click Cut Line 3D.
- 2 In the Settings window for Cut Line 3D, locate the Data section.
- 3 From the Dataset list, choose Study I/Solution 2 (sol2).
- 4 Locate the Line Data section. In row Point I, set Y to 0.5 and Z to 0.04.

#### ID Plot Group 4

- I In the Results toolbar, click  $\sim$  ID Plot Group.
- 2 In the Settings window for ID Plot Group, locate the Data section.
- 3 From the Dataset list, choose Cut Line 3D I.

#### Line Graph 1

I Right-click ID Plot Group 4 and choose Line Graph.

Keep the default plot expression.

- 2 In the Settings window for Line Graph, click Replace Expression in the upper-right corner of the x-Axis Data section. From the menu, choose Component I (compl)>Geometry> Coordinate (spatial frame)>x x-coordinate.
- 3 Click to expand the Legends section. Select the Show legends check box.
- 4 In the ID Plot Group 4 toolbar, click 💿 Plot.