

Piezoelectric Valve

This model is licensed under the COMSOL Software License Agreement 6.1. All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

Piezoelectric valves are frequently employed in medical and laboratory applications. They have a number of advantages over competing technologies, including minimal heat dissipation, quiet operation, energy efficiency, durability, low weight and fast response times. These valves typically consist of a seal that is pushed up against an opening to close the valve, or moved away from the opening to open the valve, by a piezoelectric actuator. The actuator itself often has a complex internal structure, with stacked layers of piezoelectric separated by thin conducting layers that are connected together in such a way that the applied field leads to a large deformation.

This model shows how to model a piezoelectric valve in COMSOL. The valve is actuated by a stacked piezoelectric bimorph disc actuator, which compresses a hyperelastic seal against the valve opening to shut off the flow. The detailed construction and operation of the stacked actuator is considered in the model.

Note: This application requires the Nonlinear Structural Materials Module.

Model Definition

Figure 1 shows both an axisymmetric slice through the geometry and a 3D rendering of the geometry. In this simple valve design a disc actuator compresses a hyperelastic seal directly onto an annular opening in a stainless steal support structure. The construction of the actuator itself is illustrated in Figure 2. The outer edge of the disc annulus is clamped to a stainless steel base and supporting structure. When a voltage is applied to the actuator the disc bends causing a vertical motion of the central opening of the annular actuator. With an appropriate polarity the opening moves downward, toward an annular opening in the base (supported at regular intervals by struts not included in the model). As the actuator moves toward the opening a hyperelastic seal is compressed against a mating structure, sealing up the opening. Within the model, the contact between the seal and the mating structure is modeled in detail, as is the operation of the actuator.



Figure 1: Axisymmetric model geometry (top) and full 3D geometry (bottom). Key components of the geometry are labeled.



Figure 2: Detail of the actuator and seal region, showing the construction of the actuator itself. The actuator consists of layers of PZT with metal layers between the various layers. Alternate metal layers are connected to ground whilst a voltage is applied to the other layers. Alternate layers of PZT are polarized in opposite directions. Two such actuators are stacked in such a manner that the applied potential causes contraction of one half of the beam and expansion of the other half. This results in a net bending moment acting on the beam.

Results and Discussion

Figure 3 shows the strain in the hyperelastic seal when the applied voltage is 60 V. The strain is localized in the vicinity of the contact region. Figure 4 shows the von Mises strain in the piezoelectric and its supporting structures at the same applied voltage. The stress is maximal in the PZT close to the contact. The potential within the actuator is shown in Figure 5. It is clear that the applied potentials match those shown in Figure 2. Finally the contact pressure is shown in Figure 6. The maximum pressure is 6×10^5 Pa on the surface of the seal that separates the inlet of the valve from the outlet.



Figure 3: ZZ component of the strain in the vicinity of the contact.



Figure 4: Three-dimensional visualization of the von Mises stress in the valve at an applied voltage of 50 V.



Figure 5: Electric field inside the actuator in the vicinity of the contact.



Figure 6: Contact pressure as a function of position along the surface of the seal.

6 | PIEZOELECTRIC VALVE

Application Library path: MEMS_Module/Piezoelectric_Devices/ piezoelectric_valve

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 🚈 2D Axisymmetric.
- 2 In the Select Physics tree, select Structural Mechanics>Electromagnetics-Structure Interaction>Piezoelectricity>Piezoelectricity, Solid.
- 3 Click Add.
- 4 Click 🔿 Study.
- 5 In the Select Study tree, select General Studies>Stationary.
- 6 Click 🗹 Done.

GEOMETRY I

For convenience, the device geometry is inserted from an existing file. You can read the instructions for creating the geometry in the Appendix — Geometry Instructions.

- I 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 piezoelectric valve geom sequence.mph.
- **3** In the **Geometry** toolbar, click 🛄 **Build All**.

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
V0	50[V]	50 V	Applied voltage

DEFINITIONS

- + Polarized
- I In the Definitions toolbar, click 🐚 Explicit.
- **2** Select Domains 5, 7, 9, 10, 12, 14, 18, 20, 22, 23, 25, 27, 32, 34, 36, 37, 39, 41, 47, 49, 51, 52, 54, and 56 only.
- 3 In the Settings window for Explicit, type + Polarized in the Label text field.

- Polarized

- I In the **Definitions** toolbar, click **herefore Explicit**.
- 2 In the Settings window for Explicit, type Polarized in the Label text field.
- **3** Select Domains 4, 6, 8, 11, 13, 15, 17, 19, 21, 24, 26, 28, 31, 33, 35, 38, 40, 42, 46, 48, 50, 53, 55, and 57 only.

Piezoelectric

- I In the **Definitions** toolbar, click **H Union**.
- 2 In the Settings window for Union, type Piezoelectric in the Label text field.
- 3 Locate the Input Entities section. Under Selections to add, click + Add.
- 4 In the Add dialog box, in the Selections to add list, choose + Polarized and Polarized.
- 5 Click OK.

Ground

- I In the **Definitions** toolbar, click **herefore Explicit**.
- 2 In the Settings window for Explicit, locate the Input Entities section.
- 3 From the Geometric entity level list, choose Boundary.
- 4 In the Label text field, type Ground.
- **5** Select Boundaries 10, 14, 18, 22, 26, 30, 33, 36, 40, 44, 48, 52, 56, 59, 76, 80, 84, 88, 92, 96, 99, 109, 113, 117, 121, 125, 129, and 133 only.

Voltage

- I In the **Definitions** toolbar, click **here Explicit**.
- 2 In the Settings window for Explicit, type Voltage in the Label text field.

- 3 Locate the Input Entities section. From the Geometric entity level list, choose Boundary.
- **4** Select Boundaries 12, 16, 20, 24, 28, 32, 38, 42, 46, 50, 54, 58, 78, 82, 86, 90, 94, 98, 111, 115, 119, 123, 127, and 131 only.

Mapped Mesh Steel

- I In the **Definitions** toolbar, click **here Explicit**.
- 2 In the Settings window for Explicit, type Mapped Mesh Steel in the Label text field.
- **3** Select Domains 1, 2, 29, 43, and 44 only.

Mapped Mesh

- I In the **Definitions** toolbar, click 💾 **Union**.
- 2 In the Settings window for Union, locate the Input Entities section.
- 3 Under Selections to add, click + Add.
- 4 In the Add dialog box, in the Selections to add list, choose Piezoelectric and Mapped Mesh Steel.
- 5 Click OK.
- 6 In the Settings window for Union, type Mapped Mesh in the Label text field.

Contact Pair I (p1)

- I In the Definitions toolbar, click **Pairs** and choose Contact Pair.
- 2 Select Boundaries 8, 62, 71, 72, and 150–153 only.
- 3 In the Settings window for Pair, locate the Destination Boundaries section.
- **4** Click to select the **EXAMPLA** Activate Selection toggle button.
- **5** Select Boundaries 61 and 66 only.

+Z Polarized

- I In the Model Builder window, under Component I (compl)>Definitions click Material XZplane System (compl_xz_sys).
- 2 In the Settings window for Base Vector System, type +Z Polarized in the Label text field.

-Z Polarized

- I Right-click +Z Polarized and choose Duplicate.
- 2 In the Settings window for Base Vector System, locate the Base Vectors section.

3 In the table, enter the following settings:

	r	Z
xl	1	0
x3	0	-1

4 In the Label text field, type -Z Polarized.

SOLID MECHANICS (SOLID)

Piezoelectric Material 2

- I In the Model Builder window, under Component I (compl) right-click Solid Mechanics (solid) and choose Material Models>Piezoelectric Material.
- **2** In the Settings window for Piezoelectric Material, locate the Coordinate System Selection section.
- **3** From the Coordinate system list, choose -Z Polarized (compl_xz_sysl).
- 4 Locate the Domain Selection section. From the Selection list, choose Polarized.

Piezoelectric Material I

- I In the Model Builder window, click Piezoelectric Material I.
- 2 In the Settings window for Piezoelectric Material, locate the Domain Selection section.
- 3 From the Selection list, choose + Polarized.

Hyperelastic Material I

- I In the Physics toolbar, click 🔵 Domains and choose Hyperelastic Material.
- **2** Select Domain 16 only.
- 3 In the Settings window for Hyperelastic Material, locate the Hyperelastic Material section.
- 4 From the Material model list, choose Mooney-Rivlin, two parameters.
- **5** In the κ text field, type 1e4[MPa].

Fixed Constraint 1

- I In the Physics toolbar, click 🔵 Domains and choose Fixed Constraint.
- 2 Select Domains 1, 2, 29, 43, and 44 only.

Contact I a

- I In the Physics toolbar, click Pairs and choose Contact.
- 2 In the Settings window for Contact, locate the Pair Selection section.
- 3 Under Pairs, click + Add.

- 4 In the Add dialog box, select Contact Pair I (pl) in the Pairs list.
- 5 Click OK.
- 6 In the Settings window for Contact, locate the Contact Method section.
- 7 From the list, choose Augmented Lagrangian.

ELECTROSTATICS (ES)

- I In the Model Builder window, under Component I (compl) click Electrostatics (es).
- 2 In the Settings window for Electrostatics, locate the Domain Selection section.
- 3 From the Selection list, choose Piezoelectric.

Terminal I

- I In the Physics toolbar, click Boundaries and choose Terminal.
- 2 In the Settings window for Terminal, locate the Boundary Selection section.
- 3 From the Selection list, choose Voltage.
- 4 Locate the Terminal section. From the Terminal type list, choose Voltage.
- **5** In the V_0 text field, type V0.

Ground I

- I In the Physics toolbar, click Boundaries and choose Ground.
- 2 In the Settings window for Ground, locate the Boundary Selection section.
- 3 From the Selection list, choose Ground.

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 Piezoelectric>Lead Zirconate Titanate (PZT-5H).
- 4 Right-click and choose Add to Component I (compl).
- 5 In the tree, select Built-in>Steel AISI 4340.
- 6 Right-click and choose Add to Component I (compl).

MATERIALS

Lead Zirconate Titanate (PZT-5H) (mat1)

- I In the Model Builder window, under Component I (compl)>Materials click Lead Zirconate Titanate (PZT-5H) (matl).
- 2 In the Settings window for Material, locate the Geometric Entity Selection section.

3 From the Selection list, choose Piezoelectric.

Steel AISI 4340 (mat2)

- I In the Model Builder window, click Steel AISI 4340 (mat2).
- **2** Select Domains 1–3, 29, 30, and 43–45 only.

Seal

- I In the Model Builder window, right-click Materials and choose Blank Material.
- 2 In the Settings window for Material, type Seal in the Label text field.
- **3** Select Domain 16 only.
- 4 Locate the Material Contents section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Model parameters	C10	0.37[MPa]	Pa	Mooney-Rivlin
Model parameters	C01	0.11[MPa]	Pa	Mooney-Rivlin
Density	rho	1800[kg/m^3]	kg/m³	Basic

5 In the Home toolbar, click 🙀 Add Material to close the Add Material window.

MESH I

Edge 1

- I In the Mesh toolbar, click A Edge.
- **2** Select Boundaries 61, 62, 66, 71, and 150–153 only.

Size I

- I Right-click Edge I and choose Size.
- 2 In the Settings window for Size, locate the Geometric Entity Selection section.
- 3 Click Clear Selection.
- 4 Select Boundaries 61 and 66 only.
- 5 Locate the Element Size section. From the Predefined list, choose Extremely fine.
- 6 Click the **Custom** button.
- 7 Locate the Element Size Parameters section.
- 8 Select the Maximum element size check box. In the associated text field, type w0/100.

Size 2

- I In the Model Builder window, right-click Edge I and choose Size.
- 2 In the Settings window for Size, locate the Geometric Entity Selection section.

- 3 Click Clear Selection.
- 4 Select Boundaries 62, 71, and 150–153 only.
- 5 Locate the Element Size section. From the Predefined list, choose Extremely fine.
- 6 Click the **Custom** button.
- 7 Locate the Element Size Parameters section.
- 8 Select the Maximum element size check box. In the associated text field, type w0/50.

Size

- I In the Model Builder window, under Component I (compl)>Mesh I click Size.
- 2 In the Settings window for Size, locate the Element Size section.
- **3** From the **Predefined** list, choose **Extremely fine**.
- 4 Click the **Custom** button.
- **5** Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type w0.

Mapped I

- I In the Mesh toolbar, click Mapped.
- 2 In the Settings window for Mapped, locate the Domain Selection section.
- 3 From the Geometric entity level list, choose Domain.
- 4 From the Selection list, choose Mapped Mesh.

Distribution I

- I Right-click Mapped I and choose Distribution.
- **2** Select Boundaries 75, 77, 79, 81, 83, 85, 87, 89, 91, 93, 95, and 97 only.
- 3 In the Settings window for Distribution, locate the Distribution section.
- 4 In the Number of elements text field, type 2.

Distribution 2

- I In the Model Builder window, right-click Mapped I and choose Distribution.
- 2 Select Boundaries 36 and 109 only.
- 3 In the Settings window for Distribution, locate the Distribution section.
- 4 In the Number of elements text field, type 40.

Distribution 3

- I Right-click Mapped I and choose Distribution.
- 2 In the Settings window for Distribution, locate the Distribution section.

- 3 In the Number of elements text field, type 1.
- 4 Select Boundaries 2, 63, 67, and 101 only.

Free Triangular 1

- I In the Mesh toolbar, click Kree Triangular.
- 2 In the Settings window for Free Triangular, click 📗 Build All.

Compare the resulting mesh with that shown below. Note that there are purposefully very few elements in the rigid domains.



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 From the list in the **Parameter name** column, choose **V0 (Applied voltage)**, then specify values and unit as follows:

Parameter name	Parameter value list	Parameter unit
V0 (Applied voltage)	range(0,5,60)	٧

For assistance in entering ranges of different kinds in the **Parameter value list** column, click the **Range** button to launch the **Range** dialog.

Solution 1 (soll)

When modeling contact, both the contact pressure and the auxiliary pressure need to be manually scaled. It is good practice to modify the manual scaling of these variables to an appropriate value.

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

The contact pressure is expected to be of the order of MPa, so set the scales accordingly.

- 3 In the Model Builder window, expand the Study I>Solver Configurations>
 Solution I (sol1)>Dependent Variables I node, then click
 Auxiliary pressure (compl.solid.hmml.pw).
- 4 In the Settings window for Field, locate the Scaling section.
- 5 In the Scale text field, type 1e6.
- 6 In the Model Builder window, under Study I>Solver Configurations>Solution I (soll)> Dependent Variables I click Contact pressure (compl.solid.Tn_pl).
- 7 In the Settings window for Field, locate the Scaling section.
- 8 In the Scale text field, type 1e6.
- 9 In the Model Builder window, expand the Study I>Solver Configurations> Solution I (soll)>Stationary Solver I node.

Use the Automatic Newton solver for faster and more reliable convergence.

- I0 In the Model Builder window, expand the Study I>Solver Configurations>
 Solution I (solI)>Stationary Solver I>Segregated I node, then click Merged variables.
- **II** In the **Settings** window for **Segregated Step**, click to expand the **Method and Termination** section.
- 12 From the Nonlinear method list, choose Automatic (Newton).
- **I3** In the **Study** toolbar, click **= Compute**.

RESULTS

Strain (ZZ component)

- I In the Home toolbar, click 🚛 Add Plot Group and choose 2D Plot Group.
- 2 In the Settings window for 2D Plot Group, type Strain (ZZ component) in the Label text field.
- 3 Locate the Plot Settings section. From the Frame list, choose Spatial (r, phi, z).

Surface 1

- I Right-click Strain (ZZ component) 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 I (compl)>Solid Mechanics> Strain>Strain tensor (material and geometry frames)>solid.eZZ - Strain tensor, ZZcomponent.
- **3** Locate the **Coloring and Style** section. From the **Color table transformation** list, choose **Reverse**.
- 4 In the Strain (ZZ component) toolbar, click 💽 Plot.

Zoom in on the contact region and compare the plot with Figure 3.

Electric Field (Z component)

- I In the Model Builder window, right-click Strain (ZZ component) and choose Duplicate.
- 2 In the Model Builder window, click Strain (ZZ component) I.
- **3** In the **Settings** window for **2D Plot Group**, type Electric Field (Z component) in the **Label** text field.

Surface 1

- I In the Model Builder window, click Surface I.
- 2 In the Settings window for Surface, click to collapse the Expression section.
- 3 Click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I (compl)>Electrostatics>Electric> Electric field (material and geometry frames) - V/m>es.EZ - Electric field, Z-component.
- **4** In the **Electric Field (Z component)** toolbar, click **O Plot**.

Contact Pressure

- I In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for ID Plot Group, type Contact Pressure in the Label text field.
- 3 Locate the Data section. From the Parameter selection (V0) list, choose Last.

Line Graph I

- I Right-click Contact Pressure and choose Line Graph.
- **2** Select Boundaries 60, 61, 65, 66, and 73 only.
- In the Settings window for Line Graph, click Replace Expression in the upper-right corner of the y-Axis Data section. From the menu, choose Component I (compl)>
 Solid Mechanics>Contact>solid.Tn Contact pressure N/m².
- **4** In the **Contact Pressure** toolbar, click **I Plot**.

Appendix — Geometry Instructions

From the File menu, choose New.

NEW

In the New window, click Solution Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click 📥 2D Axisymmetric.
- 2 Click M Done.

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
t0	0.05[mm]	5E-5 m	Piezoelectric layer thickness
ID	7 [mm]	0.007 m	Disc actuator inner diameter
OD	50[mm]	0.05 m	Disc actuator outer diameter
n	12	12	Number of layers in actuator
ts	0.2[mm]	2E-4 m	Thickness of seal
wO	0.5[mm]	5E-4 m	Through hole dimension
w1	ID/2	0.0035 m	Clamp region dimension
w2	ID/4	0.00175 m	Overall clamp dimension

Name	Expression	Value	Description
h0	5*t0*n	0.003 m	Base thickness
deltaz	16[um]	1.6E-5 m	Contact offset at O[V]

GEOMETRY I

Rectangle 1 (r1)

- I 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 (OD-ID)/2.
- 4 In the **Height** text field, type t0.
- 5 Locate the **Position** section. In the **r** text field, type ID/2.
- 6 In the z text field, type ts.
- 7 Click to expand the Layers section. Select the Layers to the left check box.
- 8 Clear the Layers on bottom check box.
- **9** In the table, enter the following settings:

Layer name	Thickness (m)	
Layer 1	0.5*w0	
Layer 2	3*w0	

Array I (arr I)

- I In the Geometry toolbar, click 💭 Transforms and choose Array.
- 2 In the Settings window for Array, locate the Size section.
- **3** In the **z** size text field, type n.
- 4 Locate the **Displacement** section. In the **z** text field, type t0.
- **5** Select the object **r1** only.

Rectangle 2 (r2)

- I 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 3*w0.
- **4** In the **Height** text field, type ts.
- 5 Locate the Position section. In the r text field, type ID/2+0.5*w0.

Fillet I (fill)

- I In the **Geometry** toolbar, click *Fillet*.
- 2 On the object r2, select Points 1 and 2 only.
- 3 In the Settings window for Fillet, locate the Radius section.
- 4 In the Radius text field, type ts/3.

Rectangle 3 (r3)

- I 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 2*w0.
- 4 In the **Height** text field, type 2*w0+h0.
- 5 Locate the Position section. In the r text field, type ID/2-0.5*w0.
- 6 In the z text field, type -2*w0-deltaz-h0.

Chamfer 1 (cha1)

- I In the **Geometry** toolbar, click 🧹 **Chamfer**.
- 2 On the object **r3**, select Point 4 only.
- 3 In the Settings window for Chamfer, locate the Distance section.
- 4 In the Distance from vertex text field, type 1.8*w0.

Fillet 2 (fil2)

- I In the **Geometry** toolbar, click **Fillet**.
- 2 On the object chal, select Points 3 and 5 only.
- 3 In the Settings window for Fillet, locate the Radius section.
- 4 In the Radius text field, type 0.1*w0.

Polygon I (poll)

- I 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:

r (m)	z (m)
ID/2+1.2*w0	0
ID/2+1.6*w0	0

Mirror I (mirI)

I In the Geometry toolbar, click 💭 Transforms and choose Mirror.

- 2 Select the objects fil2 and poll only.
- 3 In the Settings window for Mirror, locate the Input section.
- **4** Select the **Keep input objects** check box.
- 5 Locate the Point on Line of Reflection section. In the r text field, type ID/2+2*w0.

Rectangle 4 (r4)

- I 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 ID/2+1.5*w0.
- **4** In the **Height** text field, type h0.
- **5** Locate the **Position** section. In the **z** text field, type -2*w0-deltaz-h0.

Rectangle 5 (r5)

- I 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 0D/2+w1-ID/2-2.5*w0.
- **4** In the **Height** text field, type h0.
- 5 Locate the **Position** section. In the **r** text field, type ID/2+2.5*w0.
- 6 In the z text field, type -2*w0-deltaz-h0.

Rectangle 6 (r6)

- I 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 w2+w1.
- **4** In the **Height** text field, type h0+4*w0+n*t0+deltaz.
- 5 Locate the Position section. In the r text field, type 0D/2-w2.
- 6 In the z text field, type -2*w0-deltaz-h0.
- 7 Click 틤 Build Selected.