Created in COMSOL Multiphysics 5.6

A Piezoelectric Micropump

Introduction

This tutorial model is kindly provided by Riccardo Vietri, James Ransley, and Andrew Spann at Veryst Engineering, LLC. Piezoelectric micropumps are frequently used in medical applications because of their ability to precisely control the metering of very small volumes of fluids or gases. This example shows how to simulate a simple, non-resonant micropump, suitable for low flow rate applications. The model demonstrates the combination of piezoelectric materials with a fluid-structure interaction, and also illustrates the use of a simple velocity-dependent formula to account for the presence of valves on the inlet and outlet boundaries.

Model Definition

The model geometry consists of an annular piezoelectric actuator on top of the fluid domain which is connected to a flexible membrane (highlighted in Figure 1). Due to the symmetry of the physics, only half of the geometry needs to be included. The fluid flows out when a voltage is applied, and the actuator expands. The circumference of the membrane is fixed as the actuator moves, and it imposes a force on the fluid beneath it, pulling in fluid from the left and pushing fluid out of the channel on the right.

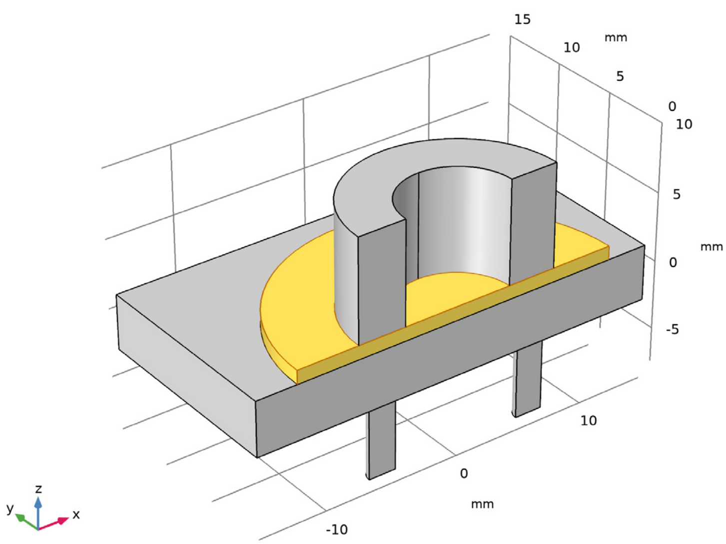


Figure 1: The geometry of the piezoelectric micropump. The membrane is highlighted.

In a real micropump, a stacked piezoelectric actuator would be employed with many separate piezoelectric layers and electrical contacts. For the purposes of this model, neglect the stiffness of the thin metal layers and model the actuator as a monolithic block of piezoelectric. Consequently, apply a high potential difference of 1500 V across the entire piezoelectric. This corresponds to an electric field of 0.2 V/μm. The voltage required in a real device would depend on the thickness of each layer in the stacked actuator.

The inlet and outlet work by employing check valves to ensure a one-way flow. In the model the valve is represented by a simple boundary condition based on K-factor piping losses, where the losses are high when flowing against the valve and low when flowing in the direction of operation. The back pressure resulting from the valve is represented by the following equation:

$$p = A\rho u_{\mathrm{av}}^2 \tag{1}$$

where u_{av} is the average velocity of the fluid normal to the boundary, ρ is the fluid density and A is a dimensionless constant that changes magnitude depending on the sign of u_{av}. The back pressure is applied as a normal stress at the end of a short length of pipe - which ensures that the fluid flow in the domain is realistic, despite this approximate boundary condition. This boundary condition can be employed to represent a simple fluid valve or diode. The constants used for the outlet boundary are reversed with respect to those used in the inlet, representing a different orientation of a similar valve. This encourages flow through the pump in the desired direction. To represent a low resistance valve (such as a simple flapper valve), set A to 5,000 for closed condition and to 0.1 for the open condition. Careful tuning of these values and potentially refinement of this model would be desirable in a real application.

To save computation time for the purposes of this example, a relatively coarse mesh is used, as shown in the figure below.

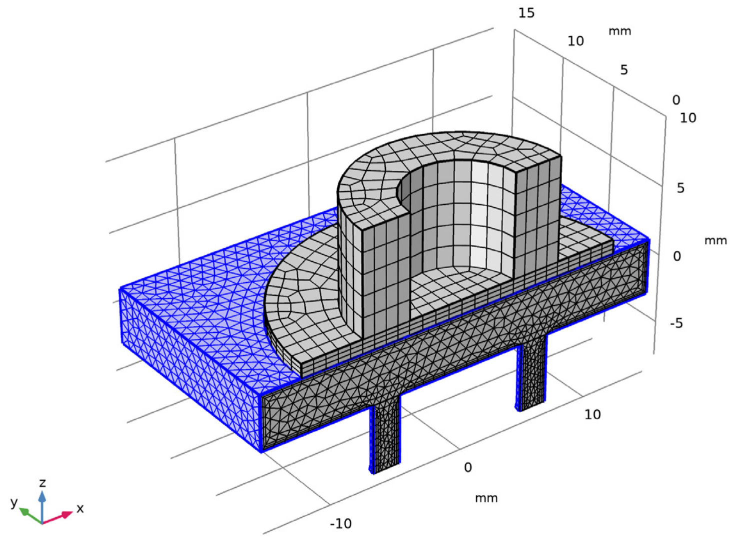


Figure 2: The mesh used in the model. Note the fluid boundary layers next to the highlighted fluid walls.

Results and Discussion

Figure 3 shows the inlet and outlet flow rates and confirms the conservation of fluid volume within the device. The drive voltage is ramped up during the first 3/4 of the actuation period. Afterwards, a consistent time periodic flow is quickly established. The difference in inlet and outlet flow matches the volume of the fluid displaced by the membrane due to the piezoelectric stroke, confirming the volume conservation.

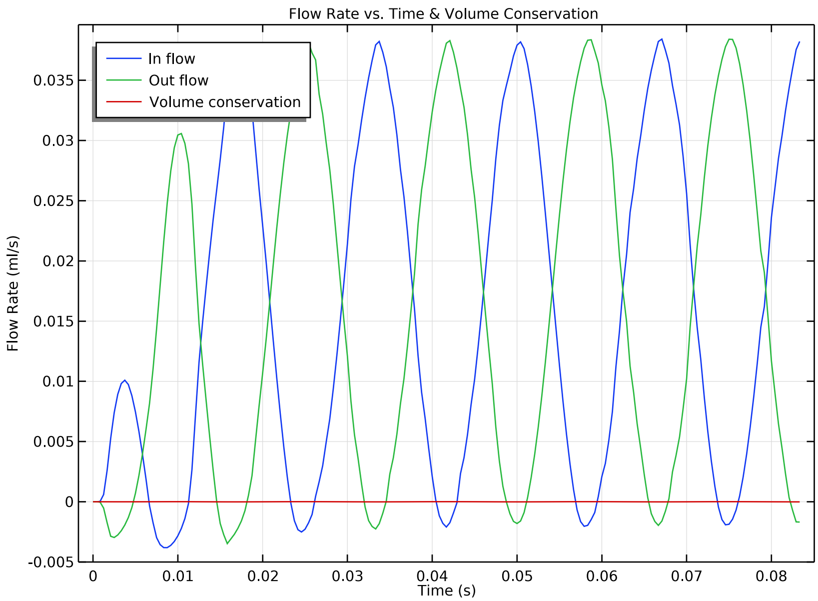


Figure 3: Flow rates and volume conservation.

The time integrated flows into the inlet and out of the outlet are calculated using a global ODE and are shown in Figure 4. The net flow through the pump in 0.05 s is 1.8 µL, corresponding to 36 µL/s or 2.16 ml/min. This is typical for such a nonresonant design.

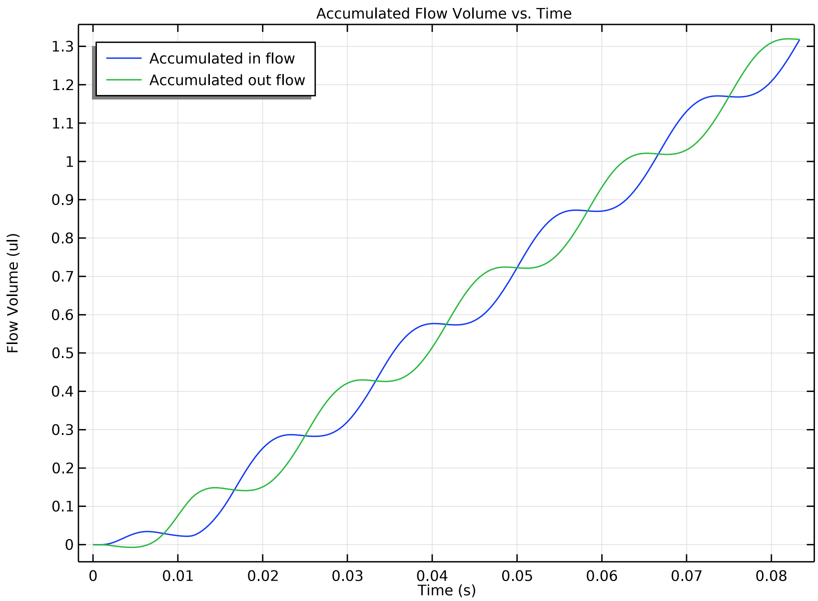


Figure 4: Net fluid flow through the inlet and the outlet over time, computed using a global ordinary differential equation.

[Figure 5](#) shows the flow through the inlet (left) and outlet (right) when fluid is being drawn into the chamber, whilst [Figure 6](#) shows the case when fluid is being forced out of the chamber.

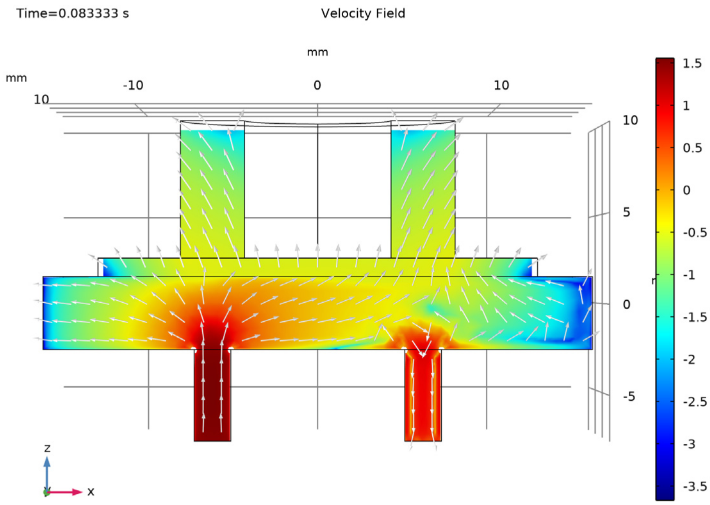


Figure 5: Velocity (base 10 logarithm in mm/s) when fluid is being drawn through the inlet.

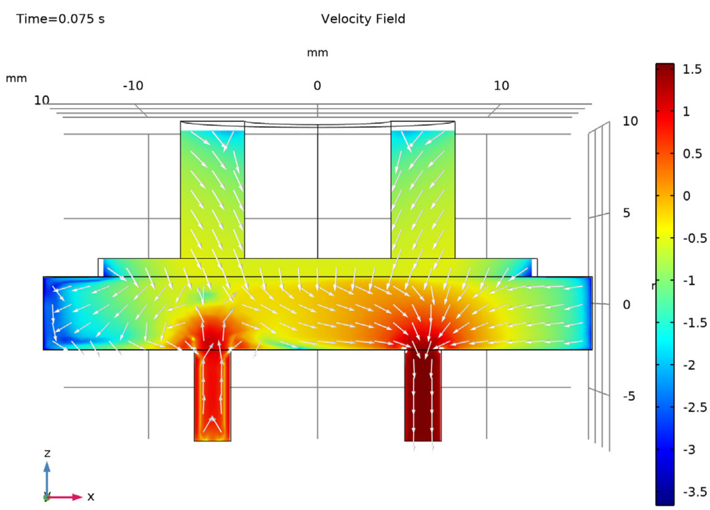


Figure 6: Velocity (base 10 logarithm in mm/s) when fluid is being pushed through the outlet.

Figure 7 and Figure 8 show the fluid streamlines in the corresponding cases.

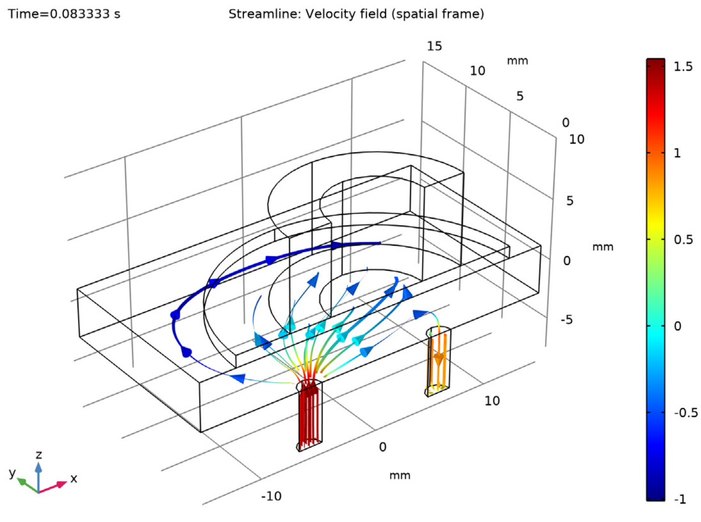


Figure 7: Flow streamlines when fluid is being drawn through the inlet. The color shows the base 10 logarithm of the velocity in mm/s.

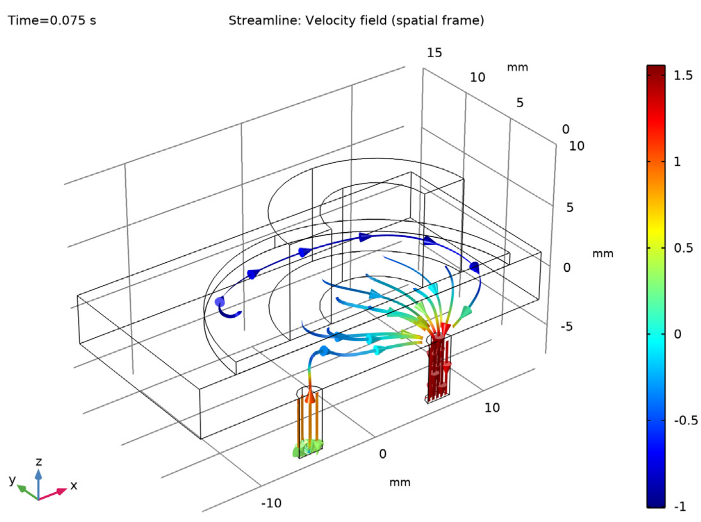


Figure 8: Flow streamlines when fluid is being pushed through the outlet. The color shows the base 10 logarithm of the velocity in mm/s.

Application Library path: `MEMS_Module/Fluid-Structure_Interaction/piezoelectric_micropump`

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**.

In addition to piezoelectricity and laminar flow, a global equation will be used to keep track of the total flow through the inlet and the outlet.

2 In the **Select Physics** tree, select **Structural Mechanics>Electromagnetics-Structure Interaction>Piezoelectricity>Piezoelectricity, Solid**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Fluid Flow>Single-Phase Flow>Laminar Flow (spf)**.

5 Click **Add**.

6 In the **Select Physics** tree, select **Mathematics>ODE and DAE Interfaces>Global ODEs and DAEs (ge)**.

7 Click **Add**.

8 Click **Study**.

9 In the **Select Study** tree, select **General Studies>Time Dependent**.

10 Click **Done**.

GEOMETRY 1

The Model Wizard starts the COMSOL Desktop at the **Geometry** node. Take the opportunity to set the length unit to mm for convenience.

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**.

Enter some global parameters. The expression containing the parameter r_inlet appears red initially and turns to black once r_inlet is defined.

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
w_block	30 [mm]	0.03 m	Width of base
depth_block	30 [mm]	0.03 m	Depth of base
h_block	5 [mm]	0.005 m	Base thickness
h_exit	5*r_inlet	0.005 m	Height of Inlet/Outlet
h_memb	1 [mm]	0.001 m	Height of the Membrane
ID	8[mm]	0.008 m	Disc actuator inner diameter
OD	15[mm]	0.015 m	Disc actuator outer diameter
r_inlet	1 [mm]	0.001 m	Fluid Inlet Radius
r_memb	12 [mm]	0.012 m	Radius of the Membrane
r_outlet	1 [mm]	0.001 m	Fluid Outlet Radius
t0	0.1[mm]	1E-4 m	Piezoelectric layer thickness
n	75	75	Number of layers in actuator
E0	0.2[V/um]	2E5 V/m	Electric field strength
V0	E0*t0*n	1500 V	Applied voltage
frequency	60[Hz]	60 Hz	Frequency of piston actuation
high_stress	5e3	5000	Boundary Stress (High)
low_stress	1e-1	0.1	Boundary Stress (Low)

Build the geometry. We will model only half of the device using the symmetry of the problem. Use **Form Assembly** with **Create imprints** to allow different meshes in the solid and the fluid domains, while still creating a matching pair of surfaces for the solid-fluid interface.

The fluid chamber is a simple block.

GEOMETRY 1

Block 1 - Fluid Chamber

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

2 In the **Settings** window for **Block**, type `Block 1 - Fluid Chamber` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `w_block`.

4 In the **Depth** text field, type `depth_block`.

5 In the **Height** text field, type `h_block-h_memb`.

6 Locate the **Position** section. From the **Base** list, choose **Center**.

7 In the **z** text field, type `-h_memb/2`.

The piezoelectric stack is a hollow cylinder, which can be constructed by taking the difference between two cylinders.

Cylinder 1 - Piezo OD

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

2 In the **Settings** window for **Cylinder**, type `Cylinder 1 - Piezo OD` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Radius** text field, type `OD/2`.

4 In the **Height** text field, type `t0*n`.

5 Locate the **Position** section. In the **z** text field, type `h_block/2`.

Cylinder 2 - Piezo ID

1 Right-click **Cylinder 1 - Piezo OD** and choose **Duplicate**.

2 In the **Settings** window for **Cylinder**, type `Cylinder 2 - Piezo ID` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Radius** text field, type `ID/2`.

Difference 1 - Piezo

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

2 In the **Settings** window for **Difference**, type `Difference 1 - Piezo` in the **Label** text field.

3 Select the object **cyl1** only.

4 Locate the **Difference** section. Find the **Objects to subtract** subsection. Select the **Activate Selection** toggle button.

5 Select the object **cyl2** only.

The actuated membrane is a simple cylinder, so are the inlet and outlet pipes.

Cylinder 3 - Membrane

1 In the **Geometry** toolbar, click **Cylinder**.
2 In the **Settings** window for **Cylinder**, type `Cylinder 3 - Membrane` in the **Label** text field.
3 Locate the **Size and Shape** section. In the **Radius** text field, type `r_memb`.
4 In the **Height** text field, type `h_memb`.
5 Locate the **Position** section. In the **z** text field, type `h_block/2-h_memb`.

Cylinder 4 - Inlet

1 In the **Geometry** toolbar, click **Cylinder**.
2 In the **Settings** window for **Cylinder**, type `Cylinder 4 - Inlet` in the **Label** text field.
3 Locate the **Size and Shape** section. In the **Radius** text field, type `r_inlet`.
4 In the **Height** text field, type `h_exit`.
5 Locate the **Position** section. In the **x** text field, type `-(OD+ID)/4`.
6 In the **z** text field, type `-h_block/2-h_exit`.

Cylinder 5 - Outlet

1 Right-click **Cylinder 4 - Inlet** and choose **Duplicate**.
2 In the **Settings** window for **Cylinder**, type `Cylinder 5 - Outlet` in the **Label** text field.
3 Locate the **Size and Shape** section. In the **Radius** text field, type `r_outlet`.
4 Locate the **Position** section. In the **x** text field, type `(OD+ID)/4`.

Use a workplane to divide the geometry to two symmetric parts and remove one of them.

Work Plane 1 - Symmetry Plane

1 In the **Geometry** toolbar, click **Work Plane**.
2 In the **Settings** window for **Work Plane**, type `Work Plane 1 - Symmetry Plane` in the **Label** text field.
3 Locate the **Plane Definition** section. From the **Plane** list, choose **xz-plane**.

Partition Objects 1 (par1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Objects**.
2 Select the object only.
3 Click in the **Graphics** window and then press Ctrl+A to select all objects.
4 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.
5 From the **Partition with** list, choose **Work plane**.

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 **par1(1)**, select Domain 1 only.

5 On the object **par1(2)**, select Domain 1 only.

6 On the object **par1(3)**, select Domain 1 only.

7 On the object **par1(4)**, select Domain 1 only.

8 On the object **par1(5)**, select Domain 1 only.

Unite the solid and fluid parts of the geometry separately with a selection defined for each, and then finalize the geometry using **Form Assembly**.

Union 1 - Solid

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

2 In the **Settings** window for **Union**, type `Union 1 - Solid` in the **Label** text field.

3 Select the objects **del1(2)** and **del1(5)** only.

4 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

Union 2 - Fluid

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

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

3 Select the objects **del1(1)**, **del1(3)**, and **del1(4)** only.

4 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

Form Union (fin)

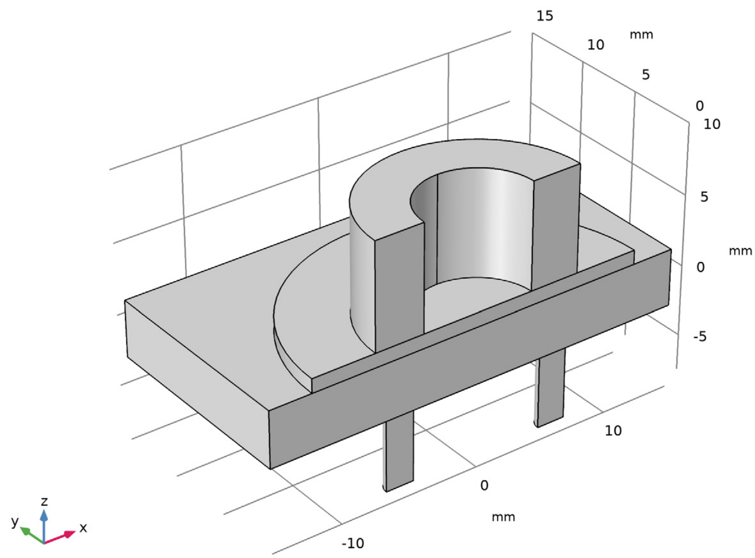
1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Form Union (fin)**.

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 Select the **Create imprints** check box.

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



Create selections for the subsequent settings. First define the piezoelectric stack domain using a box selection.

DEFINITIONS

Box 1 - Piezo

1 In the **Definitions** toolbar, click **Box**.

2 In the **Settings** window for **Box**, type `Box 1 - Piezo` in the **Label** text field.

3 Locate the **Box Limits** section. In the **x minimum** text field, type `h_block/2-h_memb/2`.

4 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

5 Locate the **Box Limits** section. In the **x minimum** text field, type `-inf`.

6 In the **z minimum** text field, type `h_block/2-h_memb/2`.

The membrane domain is given by the difference between the solid domain and the piezoelectric stack.

Difference 1 - Membrane

1 In the **Definitions** toolbar, click **Difference**.

2 In the **Settings** window for **Difference**, type `Difference 1 - Membrane` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to add**, click + **Add**.

4 In the **Add** dialog box, select **Union 1 - Solid** in the **Selections to add** list.

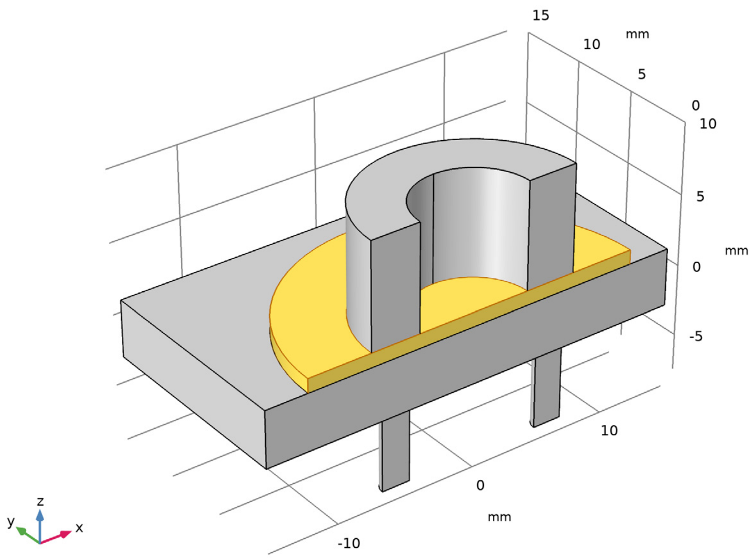
5 Click **OK**.

6 In the **Settings** window for **Difference**, locate the **Input Entities** section.

7 Under **Selections to subtract**, click + **Add**.

8 In the **Add** dialog box, select **Box 1 - Piezo** in the **Selections to subtract** list.

9 Click **OK**.



Define the inlet and outlet boundaries using box selections.

Box 2 - Inlet

1 In the **Definitions** toolbar, click **Box**.

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

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Box Limits** section. In the **x maximum** text field, type `0`.

5 In the **z maximum** text field, type `-h_block/2-h_exit/2`.

6 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Use wireframe rendering to see the inlet boundary selection better.

7 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

Box 3 - Outlet

1 Right-click **Box 2 - Inlet** and choose **Duplicate**.

2 In the **Settings** window for **Box**, type `Box 3 - Outlet` in the **Label** text field.

3 Locate the **Box Limits** section. In the **x minimum** text field, type `0`.

4 In the **x maximum** text field, type `Inf`.

Use a box selection to define the symmetry plane.

Box 4 - Symmetry Plane

1 In the **Definitions** toolbar, click **Box**.

2 In the **Settings** window for **Box**, type `Box 4 - Symmetry Plane` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Box Limits** section. In the **y maximum** text field, type `min(min(r_inlet, r_outlet),ID)/2`.

5 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Create a selection for the fluid walls by removing the symmetry plane and the in/outlets from the boundaries adjacent to the fluid. This selection can be used to create boundary layer mesh for the flow problem.

Adjacent 1 - All Fluid Boundaries

1 In the **Definitions** toolbar, click **Adjacent**.

2 In the **Settings** window for **Adjacent**, type `Adjacent 1 - All Fluid Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Input selections**, click **Add**.

4 In the **Add** dialog box, select **Union 2 - Fluid** in the **Input selections** list.

5 Click **OK**.

Difference 2 - Fluid Walls

1 In the **Definitions** toolbar, click **Difference**.

2 In the **Settings** window for **Difference**, type `Difference 2 - Fluid Walls` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

5 In the **Add** dialog box, select **Adjacent 1 - All Fluid Boundaries** in the **Selections to add** list.

6 Click **OK**.

7 In the **Settings** window for **Difference**, locate the **Input Entities** section.

8 Under **Selections to subtract**, click **Add**.

9 In the **Add** dialog box, select **Box 2 - Inlet** in the **Selections to subtract** list.

10 Click **OK**.

11 In the **Settings** window for **Difference**, locate the **Input Entities** section.

12 Under **Selections to subtract**, click **Add**.

13 In the **Add** dialog box, select **Box 3 - Outlet** in the **Selections to subtract** list.

14 Click **OK**.

15 In the **Settings** window for **Difference**, locate the **Input Entities** section.

16 Under **Selections to subtract**, click **Add**.

17 In the **Add** dialog box, select **Box 4 - Symmetry Plane** in the **Selections to subtract** list.

18 Click **OK**.

Create a selection for the membrane boundary to be used to keep track of the volume change of the fluid chamber.

Box 5

1 In the **Definitions** toolbar, click **Box**.

2 In the **Settings** window for **Box**, locate the **Geometric Entity Level** section.

3 From the **Level** list, choose **Boundary**.

4 Locate the **Box Limits** section. In the **x minimum** text field, type `-eps`.

5 In the **x maximum** text field, type `eps`.

6 In the **y minimum** text field, type `-eps`.

7 In the **y maximum** text field, type `eps`.

8 In the **z minimum** text field, type `h_block/2-h_memb-eps`.

Intersection 1 - Fluid Membrane

1 In the **Definitions** toolbar, click **Intersection**.

2 In the **Settings** window for **Intersection**, type `Intersection 1 - Fluid Membrane` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to intersect**, click **Add**.

5 In the **Add** dialog box, in the **Selections to intersect** list, choose **Difference 2 - Fluid Walls** and **Box 5**.

6 Click **OK**.

Finally select the top surface of the piezoelectric stack and the circumference of the membrane for the fixed constraint in the solid problem.

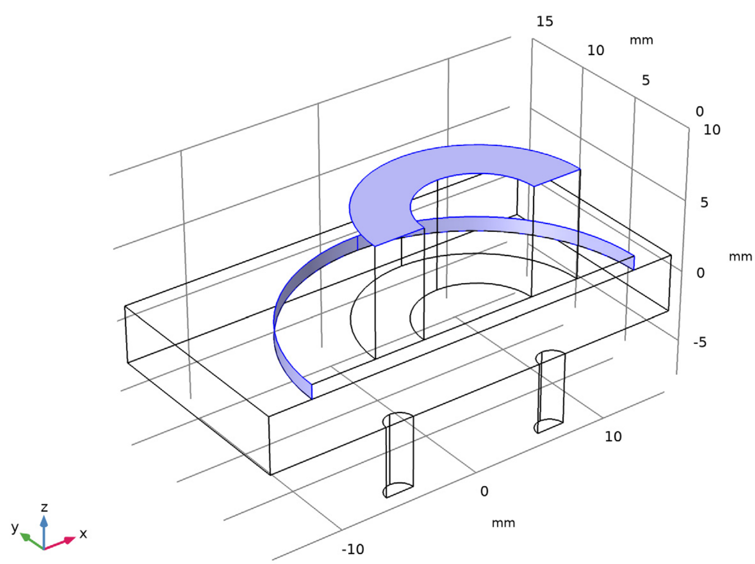
Explicit 1 - Fixed Boundaries

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Explicit 1 - Fixed Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 18, 25, and 30 only.



Add material data to the corresponding selections.

ADD MATERIAL

1 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>Water, liquid**.

4 Click **Add to Component** in the window toolbar.

5 In the tree, select **Piezoelectric>Lead Zirconate Titanate (PZT-5H)**.

6 Click **Add to Component** in the window toolbar.

7 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Water, liquid (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Water, liquid (mat1)**.

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

3 From the **Selection** list, choose **Union 2 - Fluid**.

Lead Zirconate Titanate (PZT-5H) (mat2)

1 In the **Model Builder** window, click **Lead Zirconate Titanate (PZT-5H) (mat2)**.

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

3 From the **Selection** list, choose **Box 1 - Piezo**.

Membrane

1 In the **Model Builder** window, right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Membrane` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Difference 1 - Membrane**.

At this point the set of properties shown for the user-defined membrane material is not as intended, because the physics selections haven't been configured yet.

Set up the physics selections relevant for the material data.

SOLID MECHANICS (SOLID)

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

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

3 From the **Selection** list, choose **Union 1 - Solid**.

Piezoelectric Material 1

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

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

3 From the **Selection** list, choose **Box 1 - Piezo**.

ELECTROSTATICS (ES)

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

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

3 From the **Selection** list, choose **Box 1 - Piezo**.

LAMINAR FLOW (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

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

3 From the **Selection** list, choose **Union 2 - Fluid**.

Now we are ready to enter the material properties for the membrane.

MATERIALS

Membrane (mat3)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Membrane (mat3)**.

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	`200[MPa]`	Pa	Basic
Poisson's ratio	nu	`0.45`	1	Basic
Density	rho	`2320[kg/m^3]`	kg/m³	Basic

Continue setting up the physics. Add fixed constraint and symmetry boundary condition to solid mechanics, using the boundary selections defined earlier.

SOLID MECHANICS (SOLID)

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

Fixed Constraint 1

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

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

3 From the **Selection** list, choose **Explicit 1 - Fixed Boundaries**.

Symmetry 1

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

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

3 From the **Selection** list, choose **Box 4 - Symmetry Plane**.

Add electrical terminals to the top and bottom surfaces of the piezoelectric stack, using a terminal boundary condition on the top and a ground boundary condition at the bottom.

ELECTROSTATICS (ES)

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

Terminal 1

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

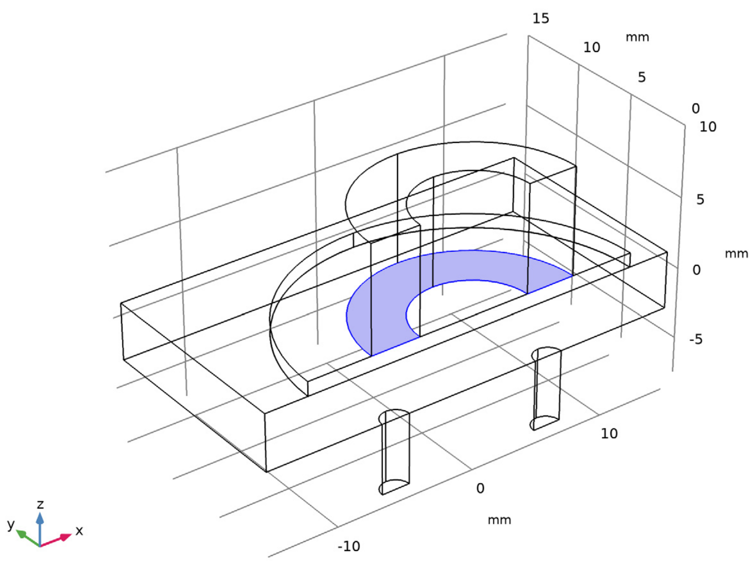
2 In the **Settings** window for **Terminal**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Explicit 1 - Fixed Boundaries**.

Ground 1

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

2 Select Boundary 24 only.



The micropump is driven by a sinusoidal voltage at the terminal. Define a ramp function to smoothly ramp up the sinusoidal drive. Then enter the expression for the terminal voltage.

DEFINITIONS

Ramp 1 (rm1)

1 In the **Home** toolbar, click f(X) **Functions** and choose **Local>Ramp**.

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

3 In the **Location** text field, type `0.1`.

4 In the **Slope** text field, type `1.2`.

5 Select the **Cutoff** check box.

6 Click to expand the **Smoothing** section. Select the **Size of transition zone at start** check box.

7 Select the **Size of transition zone at cutoff** check box.

ELECTROSTATICS (ES)

Terminal 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Electrostatics (es)** click **Terminal 1**.

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

3 From the **Terminal type** list, choose **Voltage**.

4 In the V_0 text field, type `V0*sin(t*frequency*2*pi)*rm1(t*frequency*4/3)`.

Use the selections defined earlier to set up the symmetry, inlet, and outlet boundary conditions for the fluid flow.

LAMINAR FLOW (SPF)

In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

Inlet 1

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

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

3 From the **Selection** list, choose **Box 2 - Inlet**.

Outlet 1

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

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

3 From the **Selection** list, choose **Box 3 - Outlet**.

Symmetry 1

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

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

3 From the **Selection** list, choose **Box 4 - Symmetry Plane**.

Define average operators to compute the average fluid velocity at the in/outlet and then use it in the analytic formula for the pressure to account for the effect of the valve at the in/outlet.

DEFINITIONS

Average 1 - Inlet

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

2 In the **Settings** window for **Average**, type `Average 1 - Inlet` in the **Label** text field.

3 In the **Operator name** text field, type `av_in`.

4 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

5 From the **Selection** list, choose **Box 2 - Inlet**.

Average 2 - Outlet

1 Right-click **Average 1 - Inlet** and choose **Duplicate**.

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

3 In the **Operator name** text field, type `av_out`.

4 Locate the **Source Selection** section. From the **Selection** list, choose **Box 3 - Outlet**.

LAMINAR FLOW (SPF)

Inlet 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Laminar Flow (spf)** click **Inlet 1**.

2 In the **Settings** window for **Inlet**, locate the **Boundary Condition** section.

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

4 Locate the **Pressure Conditions** section. In the p_0 text field, type `if(av_in(w2)>0,-low_stress, high_stress)*(av_in(w2)^2)*av_in(spf.rho)`.

5 From the **Flow direction** list, choose **User defined**. Specify the $\mathbf{d_u}$ vector as

0	x
0	y
1	z

Outlet 1

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

2 In the **Settings** window for **Outlet**, locate the **Pressure Conditions** section.

3 In the p_0 text field, type `if(av_out(w2)<0,low_stress, -high_stress)*(av_out(w2)^2)*av_out(spf.rho)`.

Define integration operators to compute the flow rate at the in/outlet and then use it in the global equation to keep track of the accumulated flow volume at the in/outlet.

DEFINITIONS

Integration 1 - Inlet

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

2 In the **Settings** window for **Integration**, type `Integration 1 - Inlet` in the **Label** text field.

3 In the **Operator name** text field, type `int_in`.

4 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

5 From the **Selection** list, choose **Box 2 - Inlet**.

Integration 2 - Outlet

1 Right-click **Integration 1 - Inlet** and choose **Duplicate**.

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

3 In the **Operator name** text field, type `int_out`.

4 Locate the **Source Selection** section. From the **Selection** list, choose **Box 3 - Outlet**.

GLOBAL ODES AND DAES (GE)

Global Equations 1 - Accumulated Flow Volume

1 In the **Model Builder** window, under **Component 1 (comp1)>Global ODEs and DAEs (ge)** click **Global Equations 1**.

2 In the **Settings** window for **Global Equations**, type `Global Equations 1 - Accumulated Flow Volume` in the **Label** text field.

3 Locate the **Global Equations** section. In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)	Description
`Q_in`	`Q_int-int_in(w2)`	`0`	`0`	`Accumulated in flow`
`Q_out`	`Q_outt-int_out(-w2)`	`0`	`0`	`Accumulated out flow`

The in flow is in the +z direction and the out flow is in the -z direction, thus the opposite signs in the formulas above.

4 Locate the **Units** section. Click **Define Dependent Variable Unit**.

5 In the **Dependent variable quantity** table, enter the following settings:

Dependent variable quantity	Unit
Custom unit	`m^3`

6 Click **Define Source Term Unit**.

7 In the **Source term quantity** table, enter the following settings:

Source term quantity	Unit
Custom unit	`m^3/s`

Add fluid-structure interaction multiphysics coupling.

MULTIPHYSICS

Fluid-Structure Interaction, Pair 1 (fsip1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Multiphysics** and choose **Fluid-Structure Interaction, Pair**.

2 In the **Settings** window for **Fluid-Structure Interaction, Pair**, locate the **Pair Selection** section.

3 Under **Pairs**, click **Add**.

4 In the **Add** dialog box, select **Identity Boundary Pair 1 (ap1)** in the **Pairs** list.

5 Click **OK**.

Create a box selection for the quad mesh surfaces. Then create a swept mesh for the solid domains and boundary layer mesh for the fluid domains.

DEFINITIONS

Box 6 - Quad Mesh

1 In the **Definitions** toolbar, click **Box**.
2 In the **Settings** window for **Box**, type `Box 6 - Quad Mesh` in the **Label** text field.
3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.
4 Locate the **Box Limits** section. In the **z minimum** text field, type `h_block/2-eps`.
5 In the **z maximum** text field, type `h_block/2+eps`.
6 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

MESH 1

Free Quad 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Free Quad**.
2 In the **Settings** window for **Free Quad**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Box 6 - Quad Mesh**.

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 **Extra fine**.

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 From the **Selection** list, choose **Union 1 - Solid**.

Distribution 1

1 Right-click **Swept 1** and choose **Distribution**.
2 In the **Settings** window for **Distribution**, locate the **Domain Selection** section.
3 From the **Selection** list, choose **Box 1 - Piezo**.

Distribution 2

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

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

3 From the **Selection** list, choose **Difference 1 - Membrane**.

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

Size 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Size**.

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

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

4 From the **Selection** list, choose **Union 2 - Fluid**.

5 Locate the **Element Size** section. From the **Calibrate for** list, choose **Fluid dynamics**.

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

Boundary Layers 1

1 In the **Mesh** toolbar, click **Boundary Layers**.

2 In the **Settings** window for **Boundary Layers**, locate the **Geometric Entity Selection** section.

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

4 From the **Selection** list, choose **Union 2 - Fluid**.

Boundary Layer Properties

1 In the **Model Builder** window, click **Boundary Layer Properties**.

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

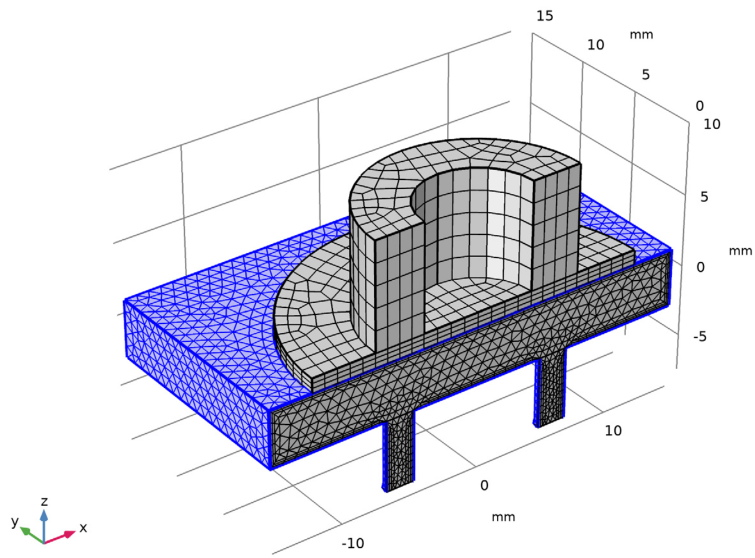
3 From the **Selection** list, choose **Difference 2 - Fluid Walls**.

4 Locate the **Boundary Layer Properties** section. In the **Number of boundary layers** text field, type `3`.

5 From the **Thickness of first layer** list, choose **Manual**.

6 In the **Thickness** text field, type `0.1`.

7 Click **Build All**.



Before computing the model, define an integration operator to calculate the volume change of the fluid chamber due to the membrane displacement. Also define two global probes to monitor the in/out flow rate during the solution process. Finally define the union of the inlet and outlet for the streamline plot after solving.

DEFINITIONS

Integration 3 - Fluid Membrane

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

2 In the **Settings** window for **Integration**, type `Integration 3 - Fluid Membrane` in the **Label** text field.

3 In the **Operator name** text field, type `int_mem`.

4 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

5 From the **Selection** list, choose **Intersection 1 - Fluid Membrane**.

Global Variable Probe 1 - In flow rate

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

2 In the **Settings** window for **Global Variable Probe**, type `Global Variable Probe 1 - In flow rate` in the **Label** text field.

3 In the **Variable name** text field, type `flowrate_in`.

4 Locate the **Expression** section. In the **Expression** text field, type `int_in(w2)`.

5 In the **Table and plot unit** field, type `ml/s`.

Global Variable Probe 2 - Out flow rate

1 Right-click **Global Variable Probe 1 - In flow rate** and choose **Duplicate**.

2 In the **Settings** window for **Global Variable Probe**, type `Global Variable Probe 2 - Out flow rate` in the **Label** text field.

3 In the **Variable name** text field, type `flowrate_out`.

4 Locate the **Expression** section. In the **Expression** text field, type `int_out(-w2)`.

Union 1 - Inlet and Outlet

1 In the **Definitions** toolbar, click **Union**.

2 In the **Settings** window for **Union**, type `Union 1 - Inlet and Outlet` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

5 In the **Add** dialog box, in the **Selections to add** list, choose **Box 2 - Inlet** and **Box 3 - Outlet**.

6 Click **OK**.

Set up the study to compute the model. Follow the time evolution for 5 drive cycles. Use **Strict** time stepping to minimize interpolation. Use fully coupled solver to save computation time.

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(0,0.025,5)/frequency`.

4 Select the **Include geometric nonlinearity** check box.

Solution 1 (sol1)

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

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

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

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

5 Right-click **Time-Dependent Solver 1** and choose **Fully Coupled**.

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

7 In the **Settings** window for **Fully Coupled**, click **= Compute**.

The default plot for the global equation shows the accumulated flow volume as a function of time.

RESULTS

Accumulated Flow Volume vs. Time

1 In the **Settings** window for **1D Plot Group**, type `Accumulated Flow Volume vs. Time` in the **Label** text field.

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

3 Locate the **Plot Settings** section. Select the **y-axis label** check box.

4 In the associated text field, type `Flow Volume (ul)`.

5 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Global 1

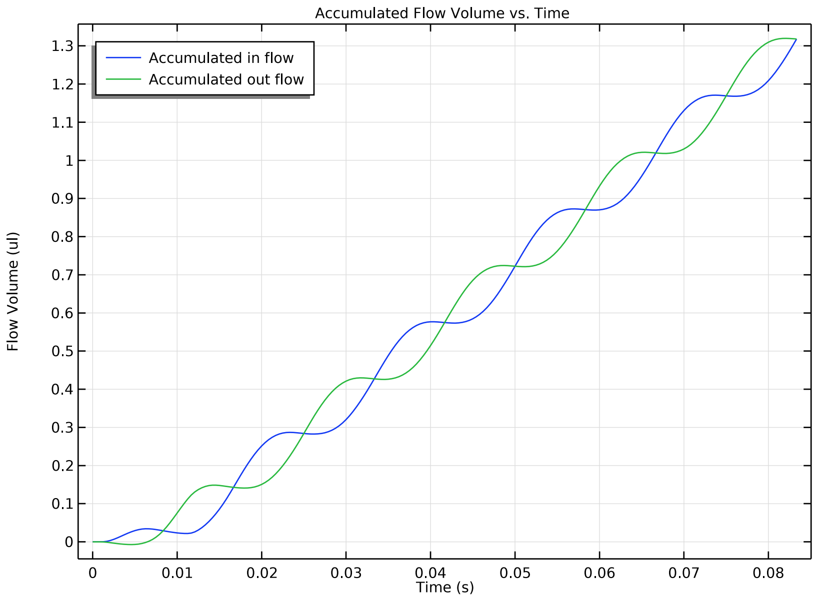
1 In the **Model Builder** window, expand the **Accumulated Flow Volume vs. Time** node, then click **Global 1**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
Q_in	ul	Accumulated in flow
Q_out	ul	Accumulated out flow

4 In the **Accumulated Flow Volume vs. Time** toolbar, click **Plot**.



We see that the in flow and out flow are staggered as one would expect from how the pump works. This can also be seen in the flow rate as a function of time. The volume conservation is shown by plotting the sum of the flow rates and the rate of volume change by the deflection of the membrane.

Flow Rate vs. Time & Volume Conservation

1 In the **Model Builder** window, right-click **Accumulated Flow Volume vs. Time** and choose **Duplicate**.

2 In the **Model Builder** window, click **Accumulated Flow Volume vs. Time 1**.

3 In the **Settings** window for **1D Plot Group**, type `Flow Rate vs. Time & Volume Conservation` in the **Label** text field.

4 Locate the **Plot Settings** section. In the **y-axis label** text field, type `Flow Rate (ml/s)`.

Global 1

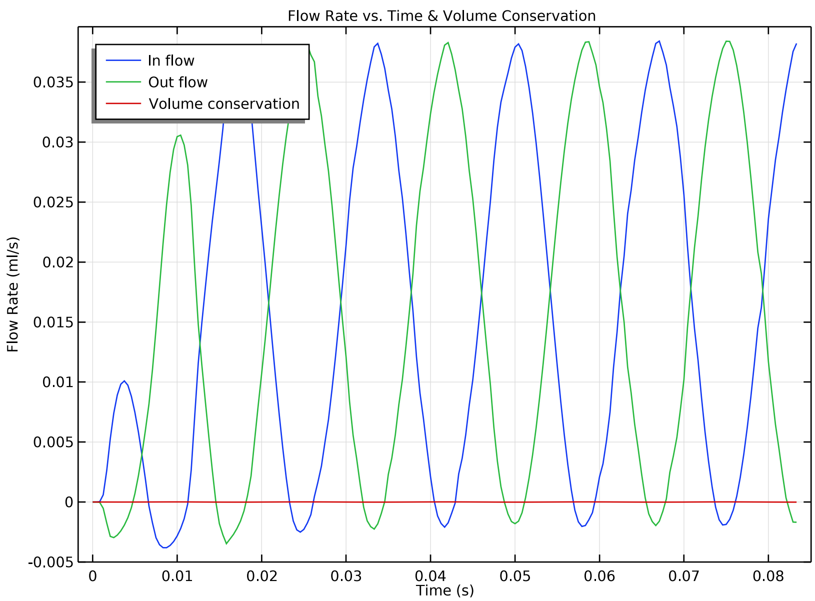
1 In the **Model Builder** window, click **Global 1**.

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

3 In the table, enter the following settings:

Expression	**Unit**	**Description**
`int_in(w2)`	`ml/s`	`In flow`
`int_out(-w2)`	`ml/s`	`Out flow`
`int_in(w2)+int_out(w2)+int_mem(-w2)`	`ml/s`	`Volume conservation`

4 In the **Flow Rate vs. Time & Volume Conservation** toolbar, click **Plot**.



The sum of the rates is zero, confirming the volume conservation. Now create a plot of the velocity field showing the fluid and solid movement on the symmetry plane.

Velocity Field

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

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

Surface 1

1 Right-click **Velocity Field** and choose **Surface**.

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

3 In the **Expression** text field, type `log10(spf.U/1[mm/s])`.

Selection 1

1 Right-click **Surface 1** and choose **Selection**.

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

3 From the **Selection** list, choose **Box 4 - Symmetry Plane**.

Surface 2

1 In the **Model Builder** window, under **Results>Velocity Field** right-click **Surface 1** and choose **Duplicate**.

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

3 In the **Expression** text field, type `log10(solid.vel/1[mm/s])`.

4 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Arrow Surface 1

1 In the **Model Builder** window, right-click **Velocity Field** and choose **Arrow Surface**.

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

3 In the **x component** text field, type `u2`.

4 In the **y component** text field, type `v2`.

5 In the **z component** text field, type `w2`.

6 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Normalized**.

7 From the **Color** list, choose **White**.

Selection 1

1 Right-click **Arrow Surface 1** and choose **Selection**.

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

3 From the **Selection** list, choose **Box 4 - Symmetry Plane**.

Arrow Surface 2

1 In the **Model Builder** window, under **Results>Velocity Field** right-click **Arrow Surface 1** and choose **Duplicate**.

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

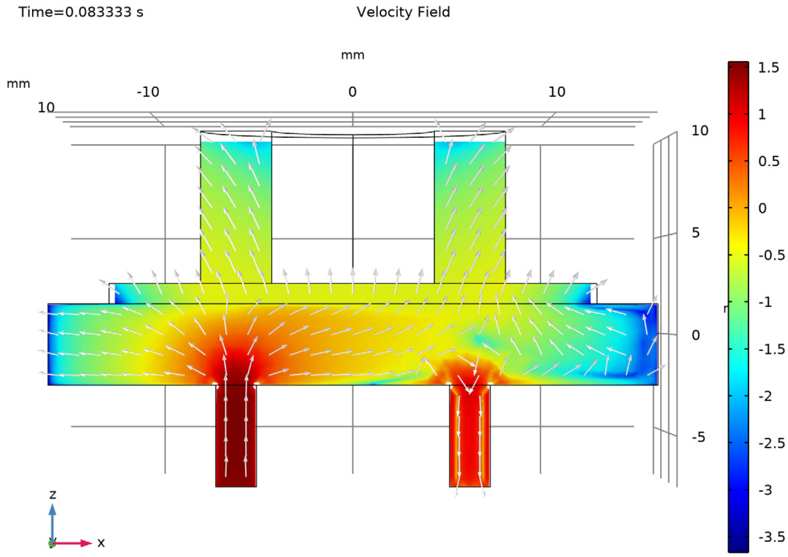
3 In the **x component** text field, type `ut`.

4 In the **y component** text field, type `vt`.

5 In the **z component** text field, type `wt`.

6 In the **Velocity Field** toolbar, click **Plot**.

7 Click the **Go to XZ View** button in the **Graphics** toolbar.



This shows a time point when the fluid is being drawn in through the inlet. Take a look at a different time point when the fluid is being pushed out through the outlet.

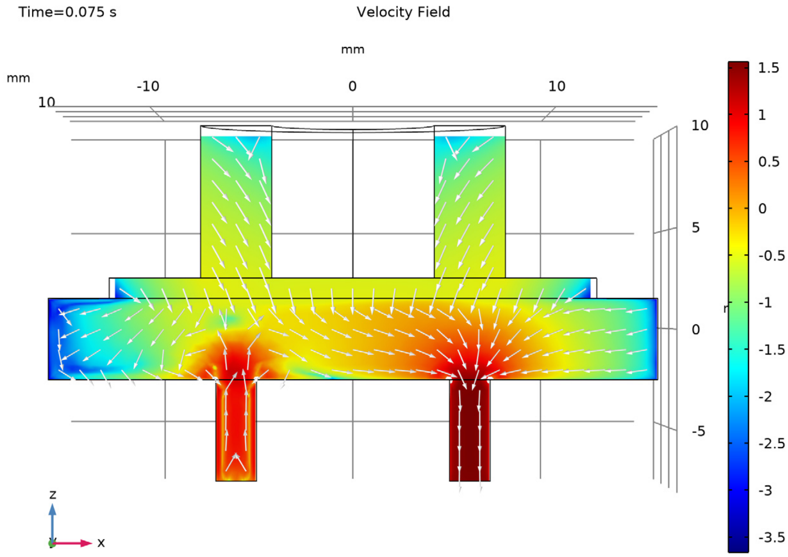
Velocity Field

1 In the **Model Builder** window, click **Velocity Field**.

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

3 From the **Time (s)** list, choose **0.075**.

4 In the **Velocity Field** toolbar, click **Plot**.



Create streamline plots for the same two time points.

Fluid Streamlines

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

Streamline 1

1 Right-click **Fluid Streamlines** and choose **Streamline**.

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

3 In the **x component** text field, type `u2`.

4 In the **y component** text field, type `v2`.

5 In the **z component** text field, type `w2`.

Color Expression 1

1 Right-click **Streamline 1** and choose **Color Expression**.

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

3 In the **Expression** text field, type `log10(spf.U/1[mm/s])`.

Streamline 1

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

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

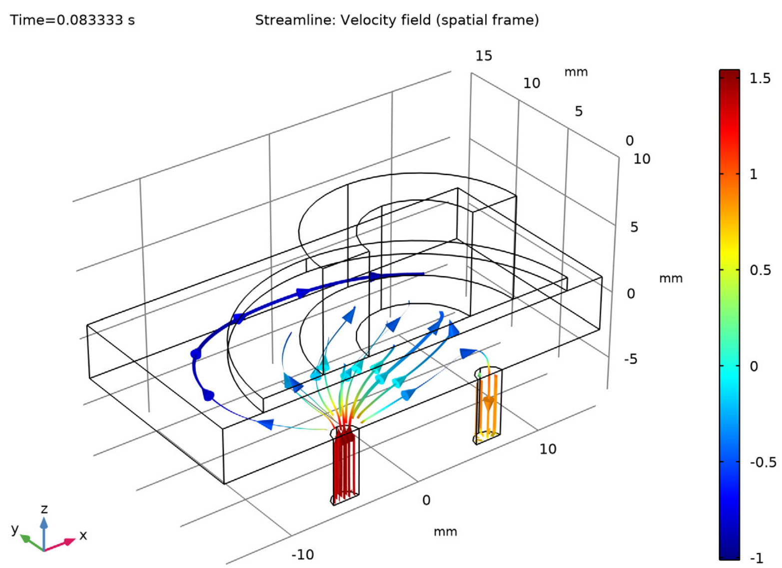
3 From the **Selection** list, choose **Union 1 - Inlet and Outlet**.

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Type** list, choose **Ribbon**.

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

6 In the **Fluid Streamlines** toolbar, click **Plot**.

7 Click the **Go to Default View** button in the **Graphics** toolbar.



This shows a time point when the fluid is being drawn in through the inlet. Take a look at a different time point when the fluid is being pushed out through the outlet.

Fluid Streamlines

1 In the **Model Builder** window, click **Fluid Streamlines**.

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

3 From the **Time (s)** list, choose **0.075**.

4 In the **Fluid Streamlines** toolbar, click **Plot**.