Created in COMSOL Multiphysics 6.0

Electrowetting Lens

This model is licensed under the COMSOL Software License Agreement 6.0.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

The contact angle of a two-fluid interface with a solid surface is determined by the balance of the forces at the contact point. The equilibrium contact angle, θ_0, is given by Young's equation:

$$\gamma_{s1} + \sigma_{12}\cos\theta_0 = \gamma_{s2} \tag{1}$$

Here γ_{s1} is the surface energy per unit area between fluid 1 and the solid surface, γ_{s2} is the surface energy per unit area between fluid 2 and the solid surface, and σ_{12} is the surface tension at the interface between the two fluids.

In electrowetting the balance of forces at the contact point is modified by the application of a voltage between a conducting fluid and the solid surface. In many applications the solid surface consists of a thin dielectric deposited onto a conducting layer; this is often referred to as "Electrowetting on Dielectric" (EWOD). In this case the capacitance of the dielectric layer dominates over the double layer capacitance at the solid-liquid interface (Ref. 1). The energy stored in the capacitor formed between the conducting liquid and the conducting layer in the solid reduces the effective surface energy of the liquid to which the voltage is applied. If there is a voltage difference between fluid 1 and the conductor beyond the dielectric, Young's equation is modified as follows:

$$\gamma_{s1} - \frac{\varepsilon V^2}{2d_f} + \sigma_{12}\cos\theta_{\mathrm{ew}} = \gamma_{s2} \tag{2}$$

Here ε is the permittivity of the dielectric, V is the potential difference applied, and d_f is the dielectric thickness. Combining Equation 1 and Equation 2 yields

$$\cos\theta_{\mathrm{ew}} = \cos\theta_0 + \frac{\varepsilon V^2}{2\sigma_{12}d_f} \tag{3}$$

Electrowetting can therefore be used to modify the contact angle dynamically by changing the voltage applied to the conducting liquid.

In this example, the meniscus between two immiscible liquids is used as an optical lens. A change in curvature of the meniscus caused by the electrowetting effect is used to change the focal length of the lens over a large range. This model is based on the work of the Philips FluidFocus team (Ref. 2). The principle of the device is illustrated in Figure 1 and the miniature, variable focus camera developed around the technology is shown in Figure 2.

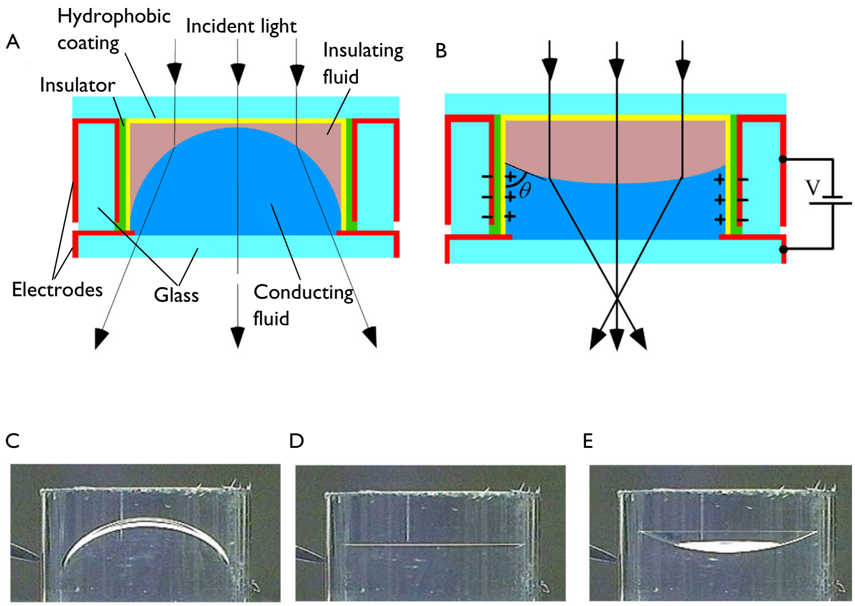


Figure 1: (A) Schematic cross section of the Philips FluidFocus lens. (B) When a voltage is applied, the electrowetting effect alters the contact angle and hence the focal distance of the lens. (C) to (E) Shapes of a 6-mm diameter lens taken at 0 V, 100 V and 120 V respectively. Diagrams and photos: Philips.

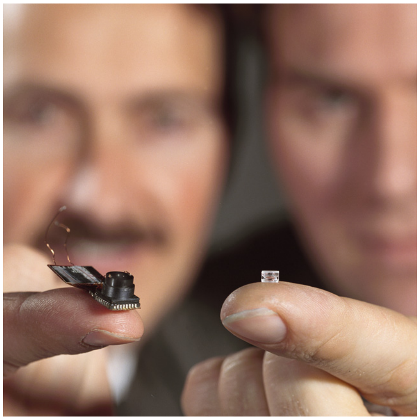


Figure 2: The miniature variable focus lens and the camera that was developed to contain it. Photo: Philips.

Model Definition

The model consists of a sealed chamber with radius 1.5 mm filled with two immiscible liquids. Because the geometry is cylindrical, the axisymmetric geometry shown in Figure 3 can be used.

The lower fluid in Figure 3 is a conducting solution of lithium chloride, with a density of 1000 kg/m^3 and a viscosity of 1.5 mPa·s. The upper fluid is insulating, with a matching density and a viscosity that is altered by varying its composition to optimize the camera performance. The surface tension at the interface between the two fluids is 50 mN/m.

The walls of the cylinder are coated with 3 µm of paylene N (relative dielectric constant, 2.65). Because this layer is thin it is not modeled explicitly in COMSOL and Equation 3 is used for the contact angle. The contact angle of the fluid in the absence of applied voltage is 140°. In this model the response of the fluid surface is modeled as a function of time after the voltage is switched from 100 V to 120 V.

It is desired to optimize the viscosity of the insulating fluid to achieve a fast response time for the switching of the lens, so the time dependent switching of the system is studied. Viscosities of 10 mPa·s, 30 mPa·s, and 50 mPa·s are investigated.

Figure 3: Axisymmetric model geometry.

Results and Discussion

When the voltage is switched the contact angle of the fluid changes abruptly but the system takes some time to respond to the change in the force at the contact point. The resonant modes of the interface are excited by this disturbance and, depending on the system damping, the oscillations of the interface take some time to decay. The higher order modes are damped out more rapidly than the fundamental mode, but are apparent in the plots shown in Figure 4 and Figure 5, which show the fluid velocity and pressure respectively 2 ms after the voltage is switched.

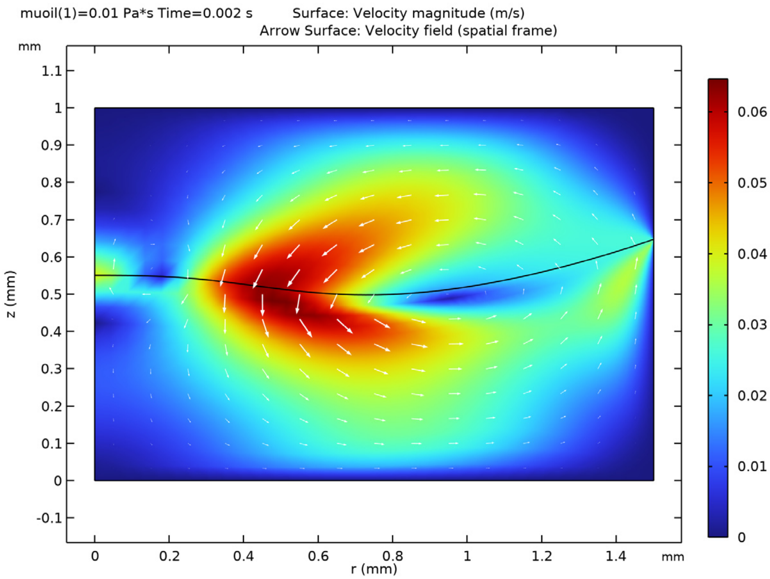


Figure 4: Fluid velocity magnitude (color) and direction (arrows) for a lens 2ms after the voltage is switched from 100 V to 120 V. The viscosity of the insulating fluid is 10 mPa·s.

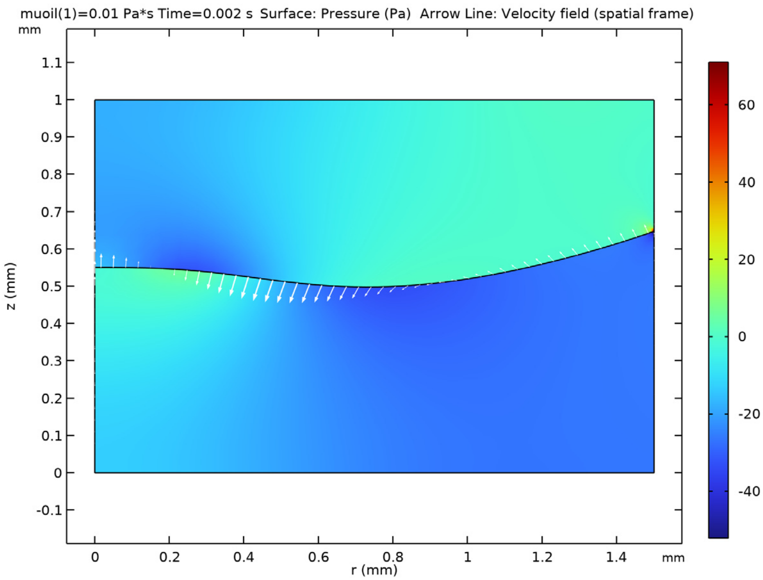


Figure 5: Pressure in the fluid (color) and velocity of the boundary (arrows) for a lens 2 ms after the voltage is switched from 100 V to 120 V. The viscosity of the insulating fluid is 10 mPa·s.

Clearly for optimum performance of the lens, the oscillation of the meniscus should be damped out as rapidly as possible; the system should therefore be critically damped. Because the viscosity of the insulating fluid can be altered by changing its composition, it is possible to adjust the damping and hence to produce a lens with the fastest possible response time. Figure 6 shows the response of the system for three different values of the viscosity of the insulating fluid. From this plot it is clear that a viscosity of 50 mPa·s produces a system that is close to being critically damped.

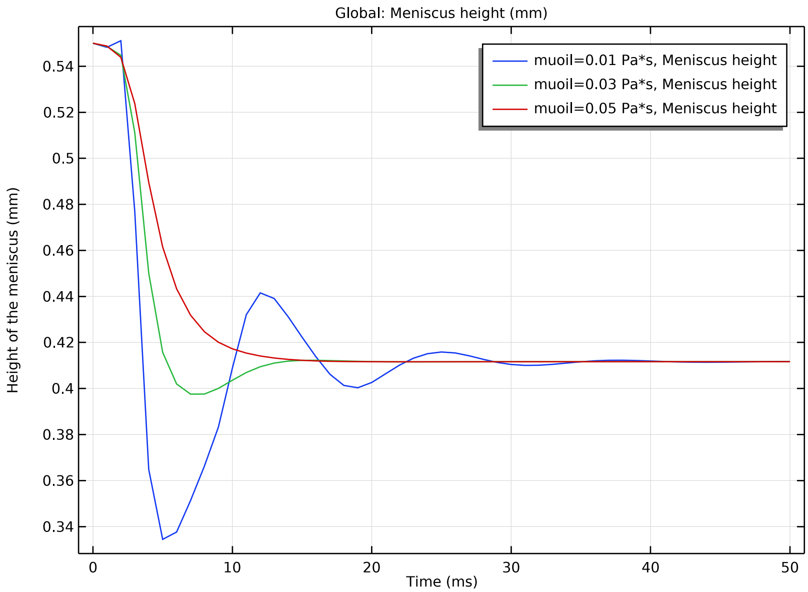


Figure 6: Location of the center of the meniscus as a function of time for different values of the viscosity of the insulating fluid.

References

1. F. Mugele and J.-C. Baret, “Electrowetting: from basics to applications,” *J. Phys. Condens. Matter*, vol. 17, pp. R705–R774, 2005.

2. S. Kuiper and B.W. Hendriks, “Variable focus lens for miniature cameras,” *Appl. Phys. Lett.*, vol. 85, no. 7, pp. 1128–1130, 2004. See also: http://www.research.philips.com/technologies/fluidfocus.html

Application Library path: `Microfluidics_Module/Two-Phase_Flow/electrowetting_lens`

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 Axisymmetric**.

2 In the **Select Physics** tree, select **Fluid Flow>Multiphase Flow>Two-Phase Flow, Moving Mesh>Laminar Two-Phase Flow, Moving Mesh**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GEOMETRY 1

Define the model geometry.

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 `1.5`.

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

Layer name	Thickness (mm)
Layer 1	0.55

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
theta0	140[deg]	2.4435 rad	Zero voltage contact angle
gamma	0.05[N/m]	0.05 N/m	Surface tension
muoil	8e-3[Pa*s]	0.008 Pa·s	Insulating fluid viscosity
epsr	2.65	2.65	Relative dielectric constant
d_f	3[um]	3E-6 m	Dielectric thickness
Vapp	120[V]	120 V	Applied voltage

Define the contact angle according to Equation 3.

DEFINITIONS

Variables 1

1 In the **Home** toolbar, click a= **Variables** and choose **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
theta	acos(cos(theta0)+Vapp^2*epsr*epsilon0_const/(2*gamma*d_f))	rad	Contact angle

Set up material properties.

MATERIALS

Insulating fluid

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 `Insulating fluid` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. Click **Clear Selection**.

4 Select Domain 2 only.

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

Property	Variable	Value	Unit	Property group
Density	rho	1000	kg/m³	Basic
Dynamic viscosity	mu	muoil	Pa·s	Basic

Lithium chloride solution

1 Right-click **Materials** and choose **Blank Material**.

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

3 Select Domain 1 only.

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

Property	Variable	Value	Unit	Property group
Density	rho	1000	kg/m³	Basic
Dynamic viscosity	mu	1.5e-3	Pa·s	Basic

Define the physics settings for the problem. Boundary conditions must be applied for both the moving mesh and the fluid flow.

Select the **Fluid-Fluid Interface** boundary condition for the two phase boundary.

LAMINAR FLOW (SPF)

Fluid-Fluid Interface 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Laminar Flow (spf)** and choose **Fluid Interface Features>Fluid-Fluid Interface**.

2 Select Boundary 4 only.

3 In the **Settings** window for **Fluid-Fluid Interface**, locate the **Surface Tension** section.

4 From the **Surface tension coefficient** list, choose **User defined**. In the σ text field, type `gamma`.

Define the contact angle settings at the wall fluid interface.

Contact Angle 1

1 In the **Model Builder** window, expand the **Fluid-Fluid Interface 1** node, then click **Contact Angle 1**.

2 In the **Settings** window for **Contact Angle**, locate the **Contact Angle** section.

3 In the θ_w text field, type `theta`.

4 Locate the **Normal Wall Velocity** section. Select the **Constrain wall-normal velocity** check box.

The **Navier Slip** option for the **Wall** boundary condition must be used in the moving mesh interface for a boundary on which a contact point moves. Use this condition for the wall on which the electrowetting effect occurs.

Wall 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Wall**.
2 Select Boundaries 6 and 7 only.
3 In the **Settings** window for **Wall**, locate the **Boundary Condition** section.
4 From the **Wall condition** list, choose **Navier slip**.

Apply a **Pressure Point Constraint** so that the pressure is constrained.

Pressure Point Constraint 1

1 In the **Physics** toolbar, click **Points** and choose **Pressure Point Constraint**.
2 Select Point 6 only.

Set up the mesh deformation boundary conditions.

DEFINITIONS

Symmetry/Roller 1

1 In the **Definitions** toolbar, click **Moving Mesh** and choose **Symmetry/Roller**.
2 Select Boundaries 1, 3, 6, and 7 only.

Quadrilateral elements are used for the mesh as they are typically stiffer and hence less susceptible to inverted elements than triangular elements. The mesh is also scaled at the contact point to improve the accuracy of the simulation.

MESH 1

Scale 1

1 In the **Mesh** toolbar, click **Modify** and choose **Mesh>Scale**.
2 In the **Settings** window for **Scale**, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Point**.
4 Select Point 5 only.
5 Locate the **Scale** section. In the **Element size scale** text field, type `0.2`.

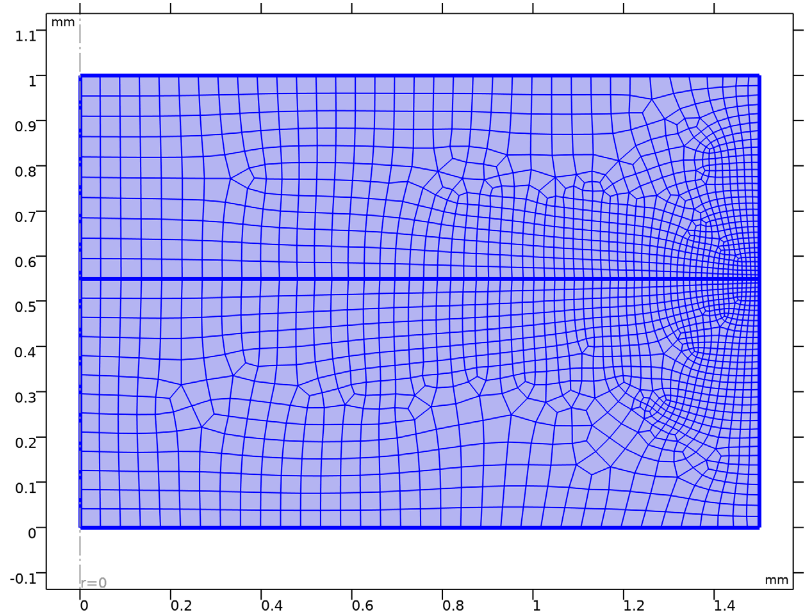
Size

1 In the **Model Builder** window, click **Size**.
2 In the **Settings** window for **Size**, locate the **Element Size** section.
3 From the **Calibrate for** list, choose **Fluid dynamics**.

Free Quad 1

1 In the **Mesh** toolbar, click **Free Quad**.

2 In the **Settings** window for **Free Quad**, click **Build All**.



Define a nonlocal integration coupling that can be used to compute the height of the center of the meniscus above the base of the lens.

DEFINITIONS

Integration 1 (intop1)

1. In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.
2. In the **Settings** window for **Integration**, locate the **Source Selection** section.
3. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundary 1 only.
5. Locate the **Advanced** section. Clear the **Compute integral in revolved geometry** check box.

Add a **Parametric Sweep** on the viscosity of the insulating fluid.

STUDY 1

Parametric Sweep

1. In the **Study** toolbar, click **Parametric Sweep**.
2. In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.
3. Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
muoil (Insulating fluid viscosity)	`10e-3 30e-3 50e-3`	`Pa*s`

Solve the problem over an appropriate time interval.

Step 1: Time Dependent

1 In the **Model Builder** window, 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(0,1e-3,5e-2)`.

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

RESULTS

Velocity (spf)

Reproduce the plot shown in Figure 4. Note that to display the deformed geometry the plot should be viewed in the default spatial reference frame rather than in the material frame.

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

2 From the **Time (s)** list, choose **0.002**.

3 From the **Parameter value (muoil (Pa*s))** list, choose **0.01**.

4 Locate the **Plot Settings** section. Select the **x-axis label** check box.

5 Select the **y-axis label** check box.

6 In the **x-axis label** text field, type `r (mm)`.

7 In the **y-axis label** text field, type `z (mm)`.

Arrow Surface 1

1 Right-click **Velocity (spf)** and choose **Arrow Surface**.

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

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

4 In the **Velocity (spf)** toolbar, click **Plot**.

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

Reproduce the plot shown in Figure 5.

Pressure (spf)

1 In the **Model Builder** window, click **Pressure (spf)**.

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

3 From the **Time (s)** list, choose **0.002**.

4 From the **Parameter value (muoil (Pa*s))** list, choose **0.01**.

5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 Select the **y-axis label** check box.

7 In the **x-axis label** text field, type `r (mm)`.

8 In the **y-axis label** text field, type `z (mm)`.

Contour

1 In the **Model Builder** window, expand the **Pressure (spf)** node.

2 Right-click **Contour** and choose **Delete**. Click **Yes** to confirm.

Surface 1

1 In the **Model Builder** window, right-click **Pressure (spf)** and choose **Surface**.

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

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

Arrow Line 1

1 Right-click **Pressure (spf)** and choose **Arrow Line**.

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

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

4 In the **Pressure (spf)** toolbar, click **Plot**.

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

2D Plot Group 4

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

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

3 From the **Frame** list, choose **Spatial (r, phi, z)**.

Surface 1

1 Right-click **2D Plot Group 4** and choose **Surface**.

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

3 In the **Expression** text field, type `spf.rho`.

4 In the **2D Plot Group 4** toolbar, click **Plot**.

Velocity, 3D (spf)

Reproduce the plot shown in Figure 6. The integration operator `intop1(1)` is used to integrate unity along the centerline of the lens, to compute the height of the meniscus as a function of time.

1D Plot Group 5

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

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

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

5 In the associated text field, type `Height of the meniscus (mm)`.

Global 1

1 Right-click **1D Plot Group 5** 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
`intop1(1)`	`mm`	`Meniscus height`

4 Locate the **x-Axis Data** section. From the **Unit** list, choose **ms**.

5 From the **Axis source data** list, choose **Inner solutions**.

6 In the **1D Plot Group 5** toolbar, click **Plot**.