

# Electrowetting Lens

## *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,  $\theta_0$ , is given by Young's equation:

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

Here  $\gamma_{s1}$  is the surface energy per unit area between fluid 1 and the solid surface,  $\gamma_{s2}$  is the surface energy per unit area between fluid 2 and the solid surface, and  $\sigma_{12}$  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. For the case when a voltage difference occurs between fluid 1 and the conductor beyond the dielectric Young's equation is modified as follows:

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

Here  $\epsilon$  is the permittivity of the dielectric,  $V$  is the potential difference applied, and  $d_f$  is the dielectric thickness. This equation can be re-written as

$$\cos \theta_{ew} = \cos \theta_0 + \frac{\epsilon V^2}{2\sigma_{12}d_f} \quad (1)$$

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.

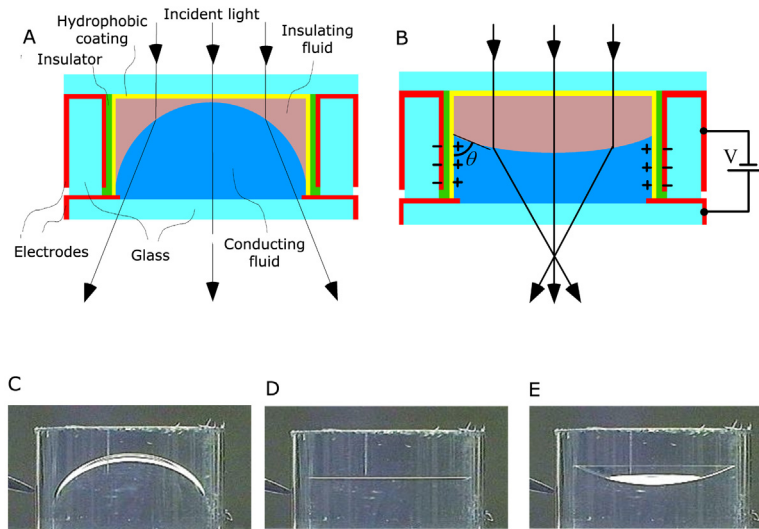


Figure 1: (A) Schematic cross section of the Philips FluidFocus lens. (B) When a voltage is applied, the electrowetting effect alters the contact angle and hence the focal distance of the lens. (C) to (E) Shapes of a 6-mm diameter lens taken at 0 V, 100 V and 120 V respectively. Diagrams and photos: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

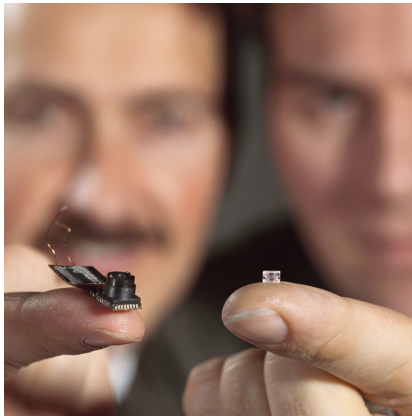


Figure 2: The miniature variable focus lens and the camera that was developed to contain it. Photo: Philips.

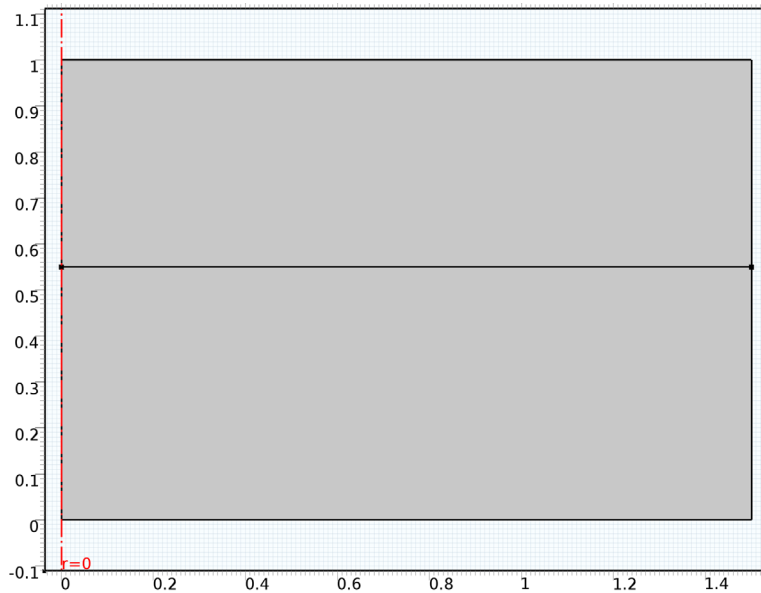
### Model Definition

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  $1120 \text{ kg/m}^3$  and a viscosity of  $1.5 \text{ mPa}\cdot\text{s}$ . 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  $50 \text{ N/m}$ .

The walls of the cylinder are coated with  $3 \text{ }\mu\text{m}$  of paylene N (relative dielectric constant, 2.65). Because this layer is thin it is not modeled explicitly in COMSOL and [Equation 1](#) is used for the contact angle. The contact angle of the fluid in the absence of applied voltage is  $140^\circ$ . In this model the response of the fluid surface is modeled as a function of time after the voltage is switched from  $100 \text{ V}$  to  $120 \text{ V}$ .

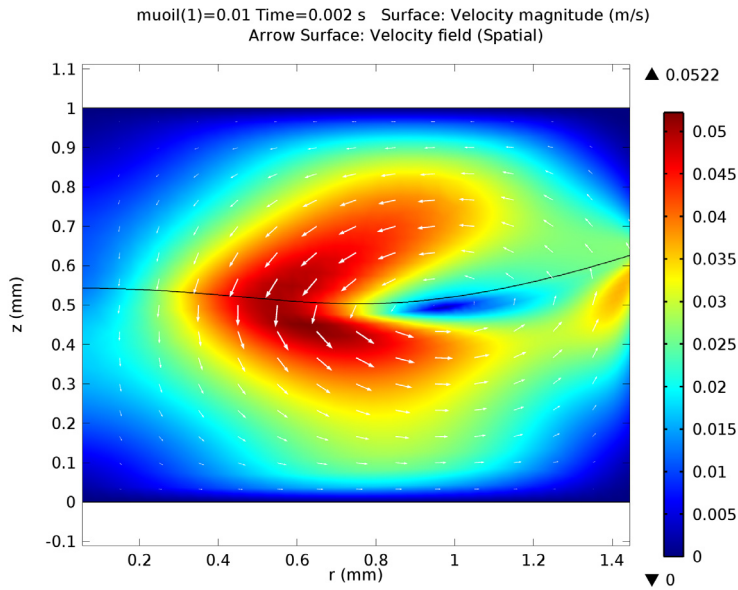
It is desired to optimize the viscosity of the insulating fluid to achieve a fast response time for the switching of the lens, so the time dependent switching of the system is studied. Viscosities of  $10 \text{ mPa}\cdot\text{s}$ ,  $30 \text{ mPa}\cdot\text{s}$ , and  $50 \text{ mPa}\cdot\text{s}$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



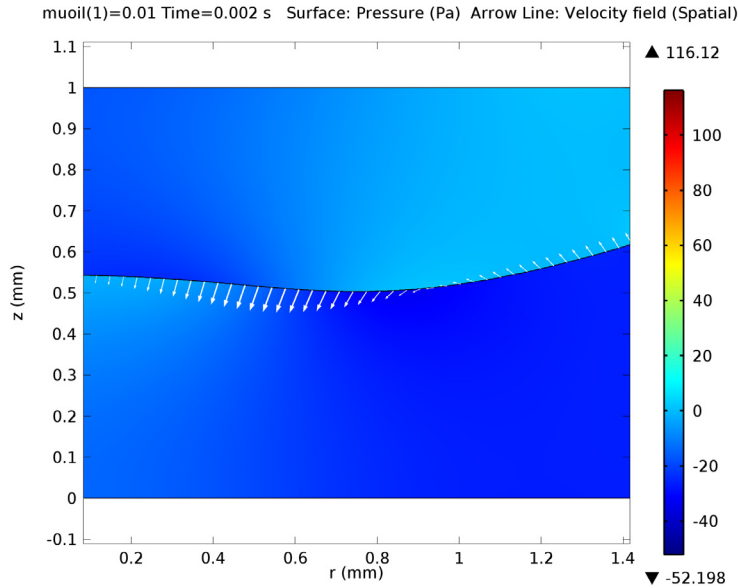
*Figure 3: Axisymmetric model geometry.*

## 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 will produce a system that is close to being critically damped.

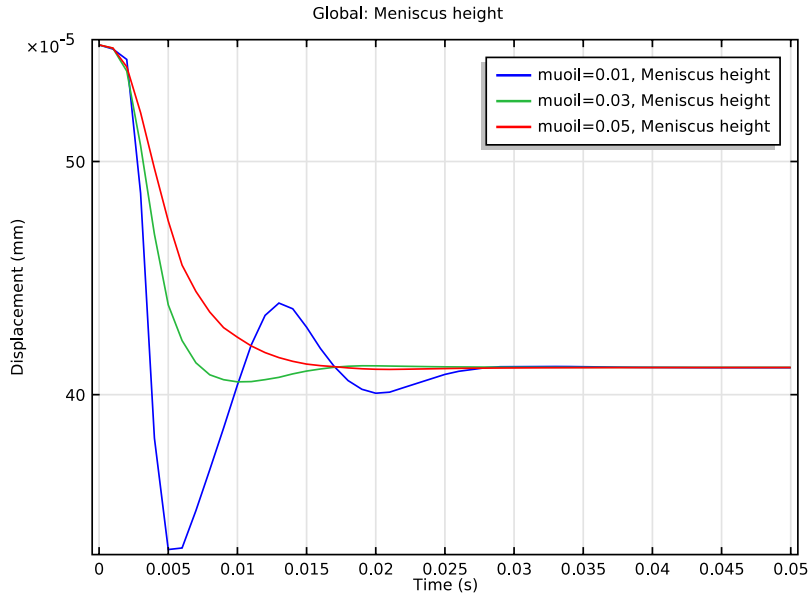


Figure 6: Location of the center of the meniscus as a function of time for different values of the viscosity of the insulating fluid.

### 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>

**Model Library path:** Microfluidics\_Module/Two-Phase\_Flow/  
electrowetting\_lens

## *Modeling Instructions*

---

### **NEW**

- 1 In the **New** window, click the **Model Wizard** button.

### **MODEL WIZARD**

- 1 In the **Model Wizard** window, click the **2D Axisymmetric** button.
- 2 In the **Select physics** tree, select **Fluid Flow>Multiphase Flow>Two-Phase Flow, Moving Mesh>Laminar Two-Phase Flow, Moving Mesh (tpfmm)**.
- 3 Click the **Add** button.
- 4 Click the **Study** button.
- 5 In the tree, select **Preset Studies>Time Dependent**.
- 6 Click the **Done** button.

### **GEOMETRY I**

Define the model geometry.

- 1 In the **Model Builder** window, under **Component I** click **Geometry I**.
- 2 In the **Geometry** settings window, locate the **Units** section.
- 3 From the **Length unit** list, choose **mm**.

#### *Rectangle 1*

- 1 Right-click **Component I>Geometry I** and choose **Rectangle**.
- 2 In the **Rectangle** settings window, locate the **Size** section.
- 3 In the **Width** edit field, type 1.5.

#### *Bézier Polygon 1*

- 1 In the **Model Builder** window, right-click **Geometry I** and choose **Bézier Polygon**.
- 2 In the **Bézier Polygon** settings window, locate the **Polygon Segments** section.
- 3 Find the **Added segments** subsection. Click the **Add Linear** button.
- 4 Find the **Control points** subsection. In row **1**, set **z** to 0.55.
- 5 In row **2**, set **r** to 1.5.
- 6 In row **2**, set **z** to 0.55.
- 7 Click the **Build All Objects** button.
- 8 Click the **Zoom Extents** button on the Graphics toolbar.

Define parameters for the material properties and for the constants in Equation 1.

**GLOBAL DEFINITIONS***Parameters*

- 1 On the **Home** toolbar, click **Parameters**.
- 2 In the **Parameters** settings window, locate the **Parameters** section.
- 3 In the table, enter the following settings:

Name	Expression	Value	Description
theta0	140[deg]	2.443 rad	Zero voltage contact angle
gamma	0.05[N/m]	0.05000 N/m	Surface tension
muoil	8e-3[Pa*s]	0.008000 Pa*s	Insulating fluid viscosity
epsr	2.65	2.650	Relative dielectric constant
d_f	3[um]	3.000E-6 m	Dielectric thickness
Vapp	120[V]	120.0 V	Applied voltage

Define the contact angle according to Equation 1.

**DEFINITIONS***Variables 1*

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.
- 2 In the **Variables** settings window, locate the **Variables** section.
- 3 In the table, enter the following settings:

Name	Expression	Unit	Description
theta	$\text{acos}(\cos(\text{theta0}) + \text{Vapp}^2 * \text{epsr} * \text{epsil} / (2 * \text{gamma} * \text{d\_f}))$	rad	Contact angle

Set up material properties.

**MATERIALS***Material 1*

- 1 In the **Model Builder** window, under **Component 1** right-click **Materials** and choose **New Material**.
- 2 In the **Material** settings window, locate the **Geometric Entity Selection** section.
- 3 Click **Clear Selection**.
- 4 Select Domain 2 only.



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

Property	Name	Value	Unit	Property group
Density	rho	1000	kg/m <sup>3</sup>	Basic
Dynamic viscosity	mu	muo11	Pa·s	Basic

#### Material 2

- 1 In the **Model Builder** window, right-click **Materials** and choose **New Material**.
- 2 Select Domain 1 only.
- 3 In the **Material** settings window, locate the **Material Contents** section.
- 4 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Density	rho	1000	kg/m <sup>3</sup>	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 TWO-PHASE FLOW, MOVING MESH

#### Fluid-Fluid Interface 1

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Fluid-Fluid Interface**.
- 2 Select Boundary 4 only.
- 3 In the **Fluid-Fluid Interface** settings window, locate the **Surface Tension** section.
- 4 In the  $\sigma$  edit field, type gamma.

Define the contact angle settings at the wall fluid interface.

#### Wall-Fluid Interface 1

- 1 Right-click **Component 1 > Laminar Two-Phase Flow, Moving Mesh > Fluid-Fluid Interface 1** and choose **Wall-Fluid Interface**.
- 2 In the **Wall-Fluid Interface** settings window, locate the **Wall-Fluid Interface** section.
- 3 In the  $\theta_w$  edit field, type theta.

The Navier Slip 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.

#### *Navier Slip 1*

1 On the **Physics** toolbar, click **Boundaries** and choose **Navier Slip**.

2 Select Boundaries 6 and 7 only.

Set up the mesh deformation boundary conditions. The mesh is fixed on the upper and lower boundaries.

#### *Prescribed Mesh Displacement 2*

1 On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Mesh Displacement**.

2 Select Boundaries 2 and 5 only.

The mesh is allowed to move vertically on the wall with the three phase contact.

#### *Prescribed Mesh Displacement 1*

1 In the **Model Builder** window, under **Component 1>Laminar Two-Phase Flow, Moving Mesh** click **Prescribed Mesh Displacement 1**.

2 In the **Prescribed Mesh Displacement** settings window, locate the **Prescribed Mesh Displacement** section.

3 Clear the **Prescribed z displacement** check box.

Apply a pressure point constraint so that the pressure is constrained.

#### *Pressure Point Constraint 1*

1 On the **Physics** toolbar, click **Points** and choose **Pressure Point Constraint**.

2 Select Point 6 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 **Model Builder** window, under **Component 1** right-click **Mesh 1** and choose **Scale**.

2 In the **Scale** settings window, 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** edit field, type 0.2.

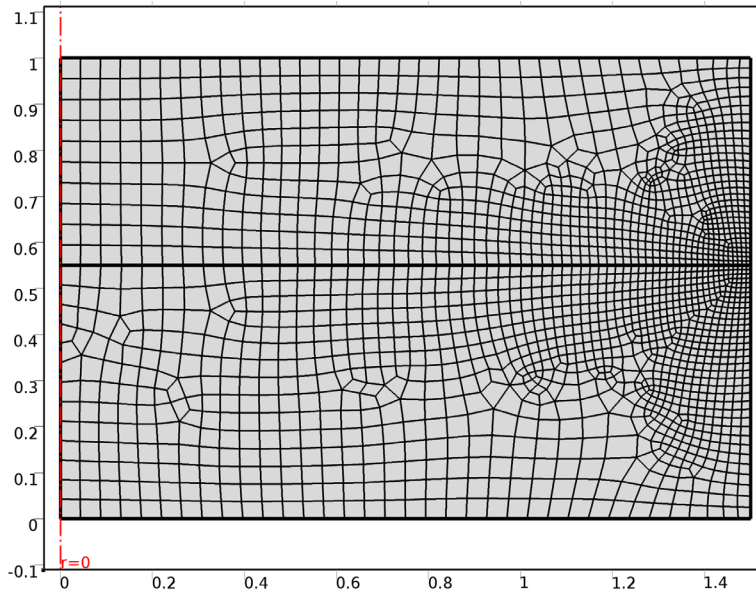
#### *Size*

1 In the **Model Builder** window, under **Component 1>Mesh 1** click **Size**.

- 2 In the **Size** settings window, locate the **Element Size** section.
- 3 From the **Calibrate for** list, choose **Fluid dynamics**.

*Free Quad 1*

- 1 In the **Model Builder** window, right-click **Mesh 1** and choose **Free Quad**.
- 2 Right-click **Free Quad 1** and choose **Build Selected**.



Define an integration operator 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 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.
- 2 In the **Integration** settings window, locate the **Source Selection** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundary 1 only.

Add a parametric sweep on the viscosity of the insulating fluid.

**STUDY 1***Parametric Sweep*

- 1 On the **Study** toolbar, click **Parametric Sweep**.
- 2 In the **Parametric Sweep** settings window, locate the **Study Settings** section.
- 3 Click **Add**.
- 4 In the table, enter the following settings:

Parameter names	Parameter value list
muoil	10e-3 30e-3 50e-3

Solve the problem over an appropriate time interval.

*Step 1: Time Dependent*

- 1 In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.
- 2 In the **Time Dependent** settings window, locate the **Study Settings** section.
- 3 In the **Times** edit field, type range (0, 1e-3, 5e-2).
- 4 On the **Home** toolbar, click **Compute**.

**RESULTS***Velocity (tpfmm) 1*

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

- 1 In the **2D Plot Group** settings window, locate the **Data** section.
- 2 From the **Time (s)** list, choose **0.002**.
- 3 From the **Parameter value (muoil)** 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** edit field, type  $r$  (mm).
- 7 In the **y-axis label** edit field, type  $z$  (mm).
- 8 Right-click **Results>Velocity (tpfmm) 1** and choose **Arrow Surface**.
- 9 In the **Arrow Surface** settings window, locate the **Coloring and Style** section.
- 10 From the **Color** list, choose **White**.

**11** On the **2D Plot Group** toolbar, click **Plot**.

Reproduce the plot shown in [Figure 5](#).

#### *Pressure (tpfmm)*

- 1** In the **Model Builder** window, under **Results** click **Pressure (tpfmm)**.
- 2** In the **2D Plot Group** settings window, locate the **Data** section.
- 3** From the **Time (s)** list, choose **0.002**.
- 4** From the **Parameter value (muoil)** 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** edit field, type  $r$  (mm).
- 8** In the **y-axis label** edit field, type  $z$  (mm).
- 9** In the **Model Builder** window, expand the **Pressure (tpfmm)** node, then click **Surface 1**.
- 10** In the **Surface** settings window, click to expand the **Range** section.
- 11** In the **Model Builder** window, right-click **Pressure (tpfmm)** and choose **Arrow Line**.
- 12** In the **Arrow Line** settings window, locate the **Coloring and Style** section.
- 13** From the **Color** list, choose **White**.
- 14** On the **2D Plot Group** toolbar, click **Plot**.

#### *Velocity (tpfmm)*

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

#### *1D Plot Group 5*

- 1** On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
- 2** In the **1D Plot Group** settings window, locate the **Data** section.
- 3** From the **Data set** list, choose **Solution 2**.
- 4** Locate the **Plot Settings** section. Select the **y-axis label** check box.
- 5** In the associated edit field, type `Displacement` (mm).
- 6** On the **1D Plot Group** toolbar, click **Global**.
- 7** In the **Global** settings window, locate the **y-Axis Data** section.

8 In the table, enter the following settings:

Expression	Unit	Description
intop1(1)		Meniscus height

9 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Inner solutions**.

10 On the **ID Plot Group** toolbar, click **Plot**.