Created in COMSOL Multiphysics 5.2a

Dielectrophoretic Separation of Platelets from Red Blood Cells

Introduction

By exploiting the fact that platelets are the smallest cells in blood, it is possible to perform size-based fractionation of blood (that is, separate platelets from red blood cells) using dielectrophoresis. This model demonstrates the continuous separation of platelets from red blood cells (RBCs) using the Dielectrophoretic Force feature available in the Particle Tracing for Fluid Flow interface.

This model also requires one of the following: CFD Module, Microfluidics Module, or Subsurface Flow Module.

Model Definition

Dielectrophoresis is the movement of particles in a non-uniform electric field due to the interaction of the particles' induced dipoles with the spatial gradient of the electric field norm.

When the electric field is computed in the frequency domain, the **Dielectrophoretic Force** feature adds the following contribution to the total force applied on the particles:

$$\mathbf{F}_{\mathrm{ext}} = 2\pi r_p^3\varepsilon_0\mathrm{real}(\varepsilon_f^*)\mathrm{real}\left(\frac{\varepsilon_p^*-\varepsilon_f^*}{\varepsilon_p^*+2\varepsilon_f^*}\right)\nabla|\mathbf{E}_{\mathrm{rms}}|^2$$

where ε_f^* (dimensionless) is the complex relative permittivity of the fluid, ε_p^* (dimensionless) is the complex relative permittivity of the particle, and $\mathbf{E}_{\mathrm{rms}}$ is the root mean square electric field. For fields that are computed in the frequency domain, the complex permittivity can be expressed as

$$\varepsilon^* = \varepsilon - \frac{i\sigma}{\omega}$$

where ε is the permittivity, σ is the electrical conductivity, and ω is the angular frequency of the electric field.

The **Shell** subnode can be added to the **Dielectrophoretic Force** node to model the dielectrophoretic force on particles with thin dielectric shells. The complex permittivity of the shell can differ from the complex permittivity of the rest of the particle. When computing the dielectrophoretic force, the complex permittivity of the particle is replaced by the equivalent complex relative permittivity $\varepsilon_{\mathrm{eq}}^*$ of a homogeneous particle comprising both the shell and the interior of the particle:

$$\varepsilon_{eq}^* = \varepsilon_s^* \frac{\left(\frac{r_0}{r_i}\right)^3 + 2\left(\frac{\varepsilon_p^* - \varepsilon_s^*}{\varepsilon_p^* + 2\varepsilon_s^*}\right)}{\left(\frac{r_0}{r_i}\right)^3 - \left(\frac{\varepsilon_p^* - \varepsilon_s^*}{\varepsilon_p^* + 2\varepsilon_s^*}\right)}$$

where r_0 and r_i (SI unit: m) are the outer and inner radii of the shell, respectively; ε_p^* (dimensionless) is the complex relative permittivity of the particle, and ε_s^* (dimensionless) is the complex relative permittivity of the outer shell. For this model, the shell parameters for platelets and RBCs are respectively obtained from Ref. 2 and Ref. 3.

The present model is based on a lab-on-a-chip device described in detail in Ref. 1. It consists of two inlets, two outlets and a separation region in which a non-uniform electric field created by an arrangement of electrodes of alternating polarity alter the particle trajectories. Figure 1 shows the schematic of the modeled geometry. As seen on the figure, the inlet velocity for the lower inlet is significantly higher (853 μm/s) than the upper inlet (154 μm/s) in order to focus all the injected particles toward the upper outlet.

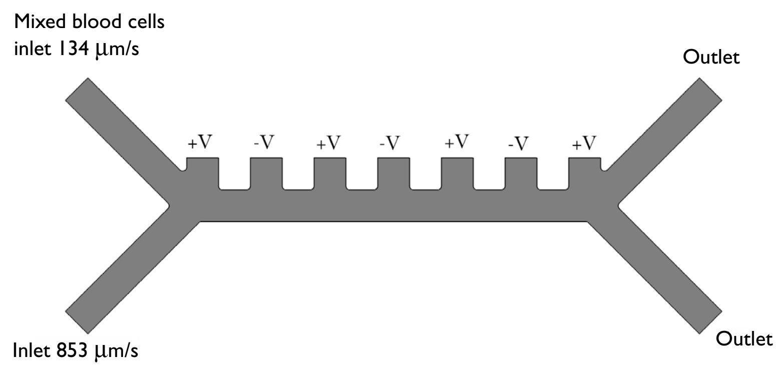

Figure 1: Two dimensional geometry of the modeled device. Details are presented in Ref. 1. The inlet velocity for the bottom inlet is significantly higher than the upper inlet to focus all the injected particles toward the upper outlet (Flow Focusing).

The model uses the following physics interfaces:

1 Creeping Flow (Microfluidics Module) to model the fluid flow,

2 Electric Currents to model the electric field in the microchannel, and

3 Particle Tracing for Fluid Flow (Particle Tracing Module) to compute the trajectories of RBCs and platelets under the influence of drag and dielectrophoretic forces

Three studies are also used:

1 Study 1 solves for the steady state fluid dynamics and frequency domain (AC) electric potential.

2 Study 2 uses a Time Dependent study step which utilizes the solution from Study 1 and estimates the particle trajectories without the dielectrophoretic force, as a result of which all particles (platelets and RBC) are focused to the same outlet.

3 Study 3 computes the particle trajectories while including the effect of the dielectrophoretic force.

Results and Discussion

Figure 2 shows the electric potential in the microfluidic device. When no dielectrophoretic force is applied, the red blood cells and platelets follow the same path and exit through the same outlet, as shown in Figure 3. When the dielectrophoretic force is applied, the two species are separated due to the differences in their dielectric properties, as shown in Figure 4.

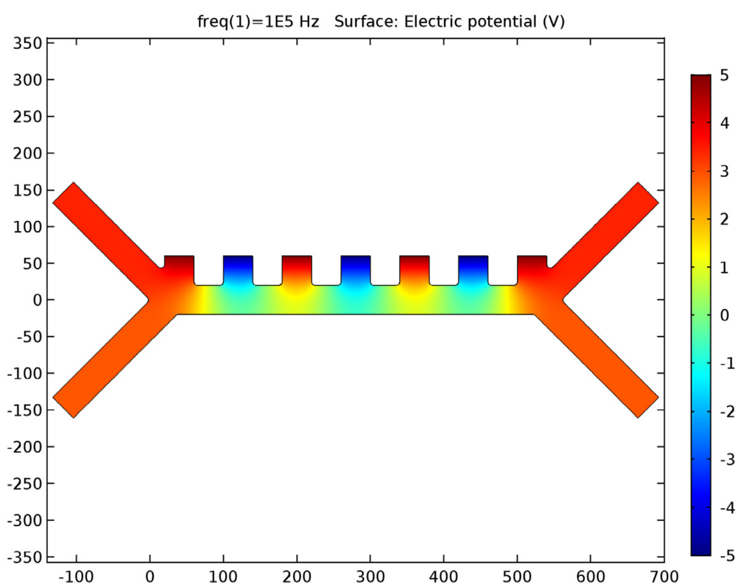

Figure 2: Spatial variation of the electric potential in the microfluidic channel.

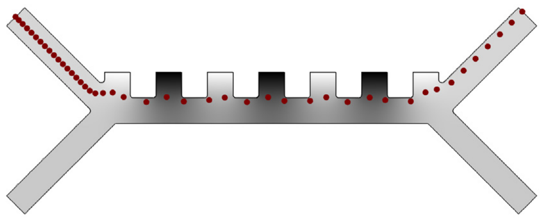

Figure 3: Particle trajectories without dielectrophoretic force applied. The RBCs are displayed in red and the platelets in blue. Since the particles are released at the same time and follow a similar path, the platelets are hidden behind the RBCs on the figure.

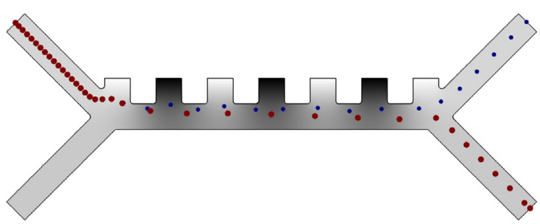

Figure 4: Particle trajectories with dielectrophoretic force applied. The RBCs are displayed in red and the platelets in blue. For sake of visualization, the relative size of the RBCs has been divided by two.

References

1. N. Piacentini, G. Mernier, R. Tornay, and P. Renaud, "Separation of platelets from other blood cells in continuous-flow by dielectrophoresis field-flow-fractionation," *Biomicrofluidics*, vol. 5, 034122, 2011.

2. M. Egger and E. Donath, "Electrorotation measurements of diamide-induced platelet activation changes," *Biophysical Journal*, vol. 68, pp. 364–372, 1995.

3. S. Park, Y. Zhang, T.H. Wang, and S. Yang, "Continuous dielectrophoretic bacterial separation and concentration from physiological media of high conductivity," Supplementary information, *Lab on a Chip*, vol. 11, pp. 2893–2900, 2011.

Application Library path: `Particle_Tracing_Module/Fluid_Flow/dielectrophoretic_separation`

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**.

2 In the **Select Physics** tree, select **AC/DC>Electric Currents (ec)**.

3 Click **Add**.

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

5 Click **Add**.

6 In the **Select Physics** tree, select **Fluid Flow>Particle Tracing>Particle Tracing for Fluid Flow (fpt)**.

7 Click **Add**.

8 Click **Done**.

GEOMETRY 1

Insert the prepared geometry sequence from file. You can read the instructions for creating the geometry in the appendix.

1 On the **Geometry** toolbar, click **Insert Sequence**.

2 Browse to the application's Application Libraries folder and double-click the file `dielectrophoretic_separation_geom_sequence.mph`.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

Load the model parameters from a file.

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

3 Click **Load from File**.

4 Browse to the application's Application Libraries folder and double-click the file `dielectrophoretic_separation_parameters.txt`.

ELECTRIC CURRENTS (EC)

Electric Potential 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Electric Potential**.

2 In the **Settings** window for Electric Potential, locate the **Electric Potential** section.

3 In the V_0 text field, type `5`.

4 Select Boundaries 8, 17, 26, and 34 only.

Electric Potential 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Electric Potential**.

2 In the **Settings** window for Electric Potential, locate the **Electric Potential** section.

3 In the V_0 text field, type `-5`.

4 Select Boundaries 13, 21, and 30 only.

CREEPING FLOW (SPF)

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

Inlet 1

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

2 Select Boundary 3 only.

3 In the **Settings** window for Inlet, locate the **Velocity** section.

4 In the U_0 text field, type `134[um/s]`.

Inlet 2

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

2 Select Boundary 1 only.

3 In the **Settings** window for Inlet, locate the **Velocity** section.

4 In the U_0 text field, type `853[um/s]`.

Outlet 1

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

2 Select Boundaries 40 and 41 only.

PARTICLE TRACING FOR FLUID FLOW (FPT)

Wall 1

1 In the **Model Builder** window, expand the **Particle Tracing for Fluid Flow (fpt)** node, then click **Wall 1**.

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

3 From the **Wall condition** list, choose **Bounce**.

Particle Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Particle Tracing for Fluid Flow (fpt)** click **Particle Properties 1**.

2 In the **Settings** window for Particle Properties, locate the **Particle Properties** section.

3 In the ρ_p text field, type `rho_p`.

4 In the d_p text field, type `dp1`.

5 In the **Model Builder** window, click **Particle Tracing for Fluid Flow (fpt)**.

Override Properties 1

1 On the **Physics** toolbar, click **Global** and choose **Override Properties**.

2 In the **Settings** window for Override Properties, locate the **Particle Properties** section.

3 In the ρ_p text field, type `rho_p`.

4 In the d_p text field, type `dp2`.

Inlet 1

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

2 Select Boundary 3 only.

3 In the **Settings** window for Inlet, locate the **Release Times** section.

4 In the **Release times** text field, type `range(0,0.05,3)`.

5 Locate the **Initial Position** section. From the **Initial position** list, choose **Uniform distribution**.

6 Locate the **Initial Velocity** section. From the **u** list, choose **Velocity field (spf)**.

Inlet 2

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

2 In the **Settings** window for Inlet, click to expand the **Inherit properties** section.

3 Locate the **Inherit Properties** section. From the **Inherit properties** list, choose **Override Properties 1**.

Outlet 1

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

2 Select Boundaries 40 and 41 only.

Drag Force 1

1 On the **Physics** toolbar, click **Domains** and choose **Drag Force**.

2 Select Domain 1 only.

3 In the **Settings** window for Drag Force, locate the **Drag Force** section.

4 From the **u** list, choose **Velocity field (spf)**.

MATERIALS

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

Material 1 (mat1)

1 In the **Settings** window for Material, locate the **Material Contents** section.

2 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Electrical conductivity	sigma	`sigma_f`	S/m	Basic
Relative permittivity	epsilonr	`epsilon_f`	1	Basic
Density	rho	`rho_f`	kg/m³	Basic
Dynamic viscosity	mu	`mu_f`	Pa·s	Basic

Add a **Stationary** and a **Frequency Domain** study step to respectively solve the fluid flow and electric potential in the channel.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Electric Currents (ec)** and **Particle Tracing for Fluid Flow (fpt)**.

5 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Add Study** in the window toolbar.

7 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 1

Frequency Domain

On the **Study** toolbar, click **Study Steps** and choose **Frequency Domain>Frequency Domain**.

Step 2: Frequency Domain

1 In the **Settings** window for Frequency Domain, locate the **Study Settings** section.

2 In the **Frequencies** text field, type `f0`.

3 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for **Creeping Flow (spf)** and **Particle Tracing for Fluid Flow (fpt)**.

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

RESULTS

Electric Potential (ec)

Click the **Zoom Extents** button on the **Graphics** toolbar. Compare the resulting plot to Figure 2.

Add a **Time Dependent** study to compute the trajectories of the particles without the **Dielectrophoretic Force** feature.

ADD STUDY

1 On the **Study** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Electric Currents (ec)** and **Creeping Flow (spf)**.

5 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Time Dependent**.

6 Click **Add Study** in the window toolbar.

7 On the **Study** toolbar, click **Add Study** to close the **Add Study** window.

Change the relative tolerance for better accuracy.

STUDY 2

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 2** click **Step 1: Time Dependent**.

2 In the **Settings** window for Time Dependent, locate the **Study Settings** section.

3 In the **Times** text field, type `range(0,0.05,3)`.

4 Select the **Relative tolerance** check box.

5 In the associated text field, type `1.0E-3`.

Check the values of variables not solve for in order to get access to the velocity field and electric potential computed in the first study.

6 Click to expand the **Values of dependent variables** section. Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

7 From the **Method** list, choose **Solution**.

8 From the **Study** list, choose **Study 1, Frequency Domain**.

Get the initial solution. The purpose is to generate a plot of the particule trajectories and use it to plot the particles while solving for them.

Particle Trajectories (fpt)

On the **Study** toolbar, click **Get Initial Value**.

Modify the plot to display the particle size and electric potential.

RESULTS

Particle Trajectories (fpt)

For clearer visualization use an `if` statement to display the RBCs with a diameter two times smaller than their real size.

Particle Trajectories 1

1 In the **Model Builder** window, expand the **Results>Particle Trajectories (fpt)** node, then click **Particle Trajectories 1**.

2 In the **Settings** window for Particle Trajectories, locate the **Coloring and Style** section.

3 Find the **Point style** subsection. In the **Point radius expression** text field, type `if(fpt.dp==dp2,dp2/2,dp1)`.

Color Expression 1

1 In the **Model Builder** window, expand the **Particle Trajectories 1** node, then click **Color Expression 1**.

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

3 In the **Expression** text field, type `fpt.dp`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Wave**.

Particle Trajectories (fpt)

In the **Model Builder** window, under **Results** right-click **Particle Trajectories (fpt)** and choose **Surface**.

Surface 1

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

2 From the **Color table** list, choose **GrayScale**.

3 Click the **Zoom Extents** button on the **Graphics** toolbar.

Plot the particle trajectories while solving. Note that the dielectrophoretic force isn't applied in this study, so all of the particles appear follow approximately the same trajectory.

STUDY 2

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 2** click **Step 1: Time Dependent**.

2 In the **Settings** window for Time Dependent, click to expand the **Results while solving** section.

3 Locate the **Results While Solving** section. Select the **Plot** check box.

4 From the **Plot group** list, choose **Particle Trajectories (fpt)**.

5 On the **Study** toolbar, click **Compute**.

RESULTS

Particle Trajectories (fpt)

Click the **Zoom Extents** button on the **Graphics** toolbar. The plot should look like Figure 3.

Now add another **Time Dependent** study to compute the effect of the dielectrophoretic force on the particle trajectories.

ADD STUDY

1. On the **Study** toolbar, click **Add Study** to open the **Add Study** window.
2. Go to the **Add Study** window.
3. Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies**.
4. Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Electric Currents (ec)** and **Creeping Flow (spf)**.
5. Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Time Dependent**.
6. Click **Add Study** in the window toolbar.
7. On the **Study** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 3

Step 1: Time Dependent

1. In the **Model Builder** window, under **Study 3** click **Step 1: Time Dependent**.
2. In the **Settings** window for Time Dependent, locate the **Study Settings** section.
3. In the **Times** text field, type `range(0,0.05,3)`.
4. Select the **Relative tolerance** check box.
5. In the associated text field, type `1.0E-3`.
6. Click to expand the **Values of dependent variables** section. Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.
7. From the **Method** list, choose **Solution**.
8. From the **Study** list, choose **Study 1, Frequency Domain**.

COMPONENT 1 (COMP1)

In the **Model Builder** window, expand the **Component 1 (comp1)** node.

PARTICLE TRACING FOR FLUID FLOW (FPT)

Dielectrophoretic Force 1

1 On the **Physics** toolbar, click **Domains** and choose **Dielectrophoretic Force**.

2 Select Domain 1 only.

3 In the **Settings** window for Dielectrophoretic Force, locate the **Dielectrophoretic Force** section.

4 From the $\mathbf{E}$ list, choose **Electric field (ec/cucn1)**.

5 Locate the **Particle Properties** section. In the $\varepsilon_{r,\ p}$ text field, type `epsilon_p1`.

6 In the σ_p text field, type `sigma_p1`.

7 Locate the **Advanced Settings** section. Select the **Use piecewise polynomial recovery on field** check box.

8 From the **Particles to affect** list, choose **Single species**.

Shell 1

1 On the **Physics** toolbar, click **Attributes** and choose **Shell**.

2 In the **Settings** window for Shell, locate the **Shell Properties** section.

3 In the t_s text field, type `th_s1`.

4 In the $\varepsilon_{r,\ s}$ text field, type `epsilon_s1`.

5 In the σ_s text field, type `sigma_s1`.

Dielectrophoretic Force 1

In the **Model Builder** window, under **Component 1 (comp1)>Particle Tracing for Fluid Flow (fpt)** right-click **Dielectrophoretic Force 1** and choose **Duplicate**.

Dielectrophoretic Force 2

1 In the **Settings** window for Dielectrophoretic Force, locate the **Particle Properties** section.

2 In the $\varepsilon_{r,\ p}$ text field, type `epsilon_p2`.

3 In the σ_p text field, type `sigma_p2`.

4 Locate the **Advanced Settings** section. From the **Affected particle properties** list, choose **Override Properties 1**.

Shell 1

1 In the **Model Builder** window, expand the **Dielectrophoretic Force 2** node, then click **Shell 1**.

2 In the **Settings** window for Shell, locate the **Shell Properties** section.

3 In the t_s text field, type `th_s2`.

4 In the $\varepsilon_{r,\ s}$ text field, type `epsilon_s2`.

5 In the σ_s text field, type `sigma_s2`.

Get the initial solution in order to view the particle trajectories while running the study.

STUDY 3

Particle Trajectories (fpt) 1

On the **Study** toolbar, click **Get Initial Value**.

RESULTS

Particle Trajectories 1

1 In the **Model Builder** window, expand the **Particle Trajectories (fpt) 1** node, then click **Particle Trajectories 1**.

2 In the **Settings** window for Particle Trajectories, locate the **Coloring and Style** section.

3 Find the **Point style** subsection. In the **Point radius expression** text field, type `if(fpt.dp==dp2,dp2/2,dp1)`.

Color Expression 1

1 In the **Model Builder** window, expand the **Particle Trajectories 1** node, then click **Color Expression 1**.

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

3 In the **Expression** text field, type `fpt.dp`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Wave**.

Particle Trajectories (fpt) 1

In the **Model Builder** window, under **Results** right-click **Particle Trajectories (fpt) 1** and choose **Surface**.

Surface 1

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

2 From the **Color table** list, choose **GrayScale**.

3 On the **Particle Trajectories (fpt) 1** toolbar, click **Plot**.

4 Click the **Zoom Extents** button on the **Graphics** toolbar.

Plot the particle trajectories while solving. Note that the dielectrophoretic force separates the particles.

STUDY 3

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 3** click **Step 1: Time Dependent**.

2 In the **Settings** window for Time Dependent, click to expand the **Results while solving** section.

3 Locate the **Results While Solving** section. Select the **Plot** check box.

4 From the **Plot group** list, choose **Particle Trajectories (fpt) 1**.

5 On the **Home** toolbar, click **Compute**.

RESULTS

Particle Trajectories (fpt) 1

1 In the **Model Builder** window, click **Particle Trajectories (fpt) 1**.

2 On the **Particle Trajectories (fpt) 1** toolbar, click **Plot**.

3 Click the **Zoom Extents** button on the **Graphics** toolbar. Compare the resulting plot to Figure 4.

Appendix - Geometry Instructions

On the **Home** toolbar, click **Add Component** and choose **2D**.

GEOMETRY 1

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

2 In the **Settings** window for Geometry, locate the **Units** section.

3 From the **Length unit** list, choose **µm**.

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

3 In the **Width** text field, type `560/2`.

4 In the **Height** text field, type `40`.

5 Locate the **Position** section. In the **y** text field, type `-20`.

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

3 In the **Width** text field, type 40.
4 In the **Height** text field, type 200.
5 Locate the **Position** section. In the **x** text field, type 9.
6 In the **y** text field, type -9.
7 Locate the **Rotation Angle** section. In the **Rotation** text field, type 45.

Mirror 1 (mir1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Mirror**.
2 Select the object **r2** only.
3 In the **Settings** window for Mirror, locate the **Normal Vector to Line of Reflection** section.
4 In the **x** text field, type 0.
5 In the **y** text field, type 1.
6 Locate the **Input** section. Select the **Keep input objects** check box.

Mirror 2 (mir2)

1 On the **Geometry** toolbar, click **Transforms** and choose **Mirror**.
2 Click in the **Graphics** window and then press Ctrl+A to select all objects.
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 **x** text field, type 560/2.

Square 1 (sq1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Square**.
2 In the **Settings** window for Square, locate the **Size** section.
3 In the **Side length** text field, type 40.
4 Locate the **Position** section. In the **x** text field, type 20.
5 In the **y** text field, type 20.

Array 1 (arr1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Array**.
2 Select the object **sq1** only.
3 In the **Settings** window for Array, locate the **Size** section.
4 In the **x size** text field, type 7.
5 Locate the **Displacement** section. In the **x** text field, type 80.

Union 1 (uni1)

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

Use the select box icon to select all the geometry objects.

2 Click the **Select Box** button on the **Graphics** toolbar.

3 Click in the **Graphics** window and then press Ctrl+A to select all objects.

4 In the **Settings** window for Union, locate the **Union** section.

5 Clear the **Keep interior boundaries** check box.

Fillet 1 (fil1)

1 On the **Geometry** toolbar, click **Fillet**.

2 On the object **uni1**, select Points 5, 6, 8, 9, 11, 13, 15, 17, 19, 22, 24, 26, 28, 30, 32, 34, 35, and 37 only.

3 In the **Settings** window for Fillet, locate the **Radius** section.

4 In the **Radius** text field, type 5.

5 Click **Build All Objects**.