This tutorial is prepared by Lingyan Gong of Wenzhou University. It is aimed to help the readers to build the ICP-based preconcentration model using Comsol v5.2 software. Backgrounds, theories, methods, and key results are described in the following paper:

Zirui Li, Wei Liu, Lingyan Gong, Yudan Zhu, Yuantong Gu, Jongyoon Han, "Accurate multi physics numerical analysis of particle preconcentration based on ion concentration polarization", *International Journal of Applied Mechanics*, **9**(8), 1750107, 2017. http://dx.doi.org/10.1142/S1758825117501071

Modeling Instructions

From the **File** menu, choose **New**.

N E W

In the **New** window, click **Model Wizard**.

M O D E L W I Z A R D

1 In the **Model Wizard** window, click **2D**.

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

3 Click **Add.**

4 In the **Select Physics** tree, select **AC/DC>Electrostatics(es).**

5 Click **Add.**

6 In the **Select Physics** tree, select **Chemical Species Transport>Transport of Dilutes Species (tds).**

7 Click **Add.**

8 In the **Concentration** table, enter the following setting:

c1

9 In the **Select Physics** tree, select **Chemical Species Transport>Transport of Dilutes Species (tds).**

10 Click **Add.**

11 In the **Concentration** table, enter the following setting:

c2

12 In the **Select Physics** tree, select **Chemical Species Transport>Transport of Dilutes Species (tds).**

13 Click **Add.**

14 In the **Concentration** table, enter the following setting:

c3

15 Click **Study.**

16 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary.**

17 Click **Done**.

GLOBAL DEFINITIONS

Parameters

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

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

3 In the table, enter the following setting:

Name	Expression	Value	Description
W	120[um]	1.2E-4 m	Length of microchannel
H	2[um]	2E-6 m	Width of microchannel
sigma	-5[mC/m^2]	-0.005 C/m^2	Surface charge density
phi0	25.8[mV]	0.0258 V	Thermal voltage
VL	20	20	Voltage at inlet (in thermal voltage)
VR	0	0	Voltage at outlet (in thermal voltage)
vcm	0	0	Cross-membrane Voltage (in thermal voltage)
T	300[K]	300 K	Temperature
rho	1e3[kg/m^3]	1000 kg/m^3	Mass density of water
eta	1e-3[Pa*s]	0.001 Pa·s	Dynamic Viscosity
c0	1[mM]	1 mol/m^3	Concentration of K+
D1	1.97*1e-9[m^2/s]	1.97E-9 m^2/s	Diffusion coefficient of K+
D2	2.03*1e-9[m^2/s]	2.03E-9 m^2/s	Diffusion coefficient of Cl-
D3	0.51*1e-9[m^2/s]	5.10E-10 m^2/s	Diffusion coefficient of P2-
rc0	1e-7	1E-7	Ratio of particle concentration in the reservoir to that of Cl-

DEFINITIONS

Create an random function for the simulation of initial concentrations of ions and particles.

1 In the **Home** toolbar, click **Functions>Local>Random.**

2 In the **Function name** text field, type ran.

3 In the **Number of arguments** text field, type 2.

4 From the **Distribution** list, choose **Normal**.

GEOMETRY 1

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 **µm**.

4 Locate the **Advanced** section. From the **Default repair tolerance** list, choose **Relative**.

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 240.

4 In the **Height** text field, type H.

5 Locate the **Position** section. In the **y** text field, type -2.

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 60.

4 In the **Height** text field, type60.

5 Locate the **Position** section. In the **y** text field, type -60.

Rectangle 3 (r3)

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

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

8 In the **Width** text field, type 60.

9 In the **Height** text field, type 60.

10 Locate the **Position** section. In the **x** text field, type 180.

11 Locate the **Position** section. In the **y** text field, type -60.

Point 1 (pt1)

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

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

3 In the **x** text field, type 119.

Point 2 (pt2)

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

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

3 In the **x** text field, type 121.

Point 3 (pt3)

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

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

3 In the **x** text field, type 119.

4 In the **y** text field, type -H.

Point 4 (pt4)

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

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

3 In the **x** text field, type 121.

4 In the **y** text field, type -H.

Form Union (fin)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Form Union (fin)** and choose **Build Selected**.

CREEPING FLOW (SPF)

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

2 In the **Model Builder** window's toolbar, click the **Show** button and select **Stabilization** in the menu.

3 In the **Settings** window for Creeping Flow, type Creeping in the **Label** text field.

4 Click to expand the **Consistent Stabilization** section, then clear the **Crosswind diffusion** check box.

CREEPING (SPF)

Transport Properties 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Creeping (spf)** node, then click **Fluid Properties 1**.

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

3 From the ρ list, choose **User defined**. In the associated text field, type rho.

4 From the μ list, choose **User defined**. In the associated text field, type eta.

Volume Force 1

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

2 In the **Settings** window for Volume Force, locate the **Domain Selection** section. From the **Selection** list choose **All Domains**.

3 Locate the **Volume Force** section. in the **Volume force** table, enter the following setting:

(c1-c2-2*c3)*(es.Ex)*e_const*N_A_const	x

(c1-c2-2*c3)*(es.Ey)*e_const*N_A_const	y

Symmetry 1

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

2 Select boundaries 5, 9, 11, 13 and 18 only.

3 In the **Settings** window for Symmetry, locate the **Boundary Selection** section. Click the **Create Selection** button and type Center Line in the pop-up window. Click **OK**, then **Center Line** is shown under the **Definitions** node in the **Model Builder** window.

Inlet 1

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

2 Select boundaries 1 and 3 only.

3 In the **Settings** window for Inlet, locate the **Boundary Condition** section. From the list, choose **Pressure**.

Outlet 1

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

2 Select boundaries 19 and 20 only.

3 In the **Settings** window for Outlet, locate the **Pressure Condition** section, select the **Normal flow** check box.

Pressure Point Constraint 1

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

2 Select point 14 only.

ELECTROSTATICS(ES)

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

2 In the **Settings** window for Electrostatics, locate **Thickness** section. In the **Out-of-plane thickness** text field, type 10[um].

Charge Conservation 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)> Electrostatics (es)** node, then click **Charge Conservation 1**.

2 In the **Settings** window for Charge Conservation, locate the **Electric Field** section.

3 From the **Relative permittivity** list, choose **User defined**. In the associated text field, type 78.

Space Charge Density 1

1 On the **Physics** toolbar, click **Domains** and choose **Space Charge Density**.

2 In the **Settings** window for Space Charge Density, locate the **Domain Selection** section. From the Selection list, choose **All domains.**

3 Locate the **Space Charge Density** section. From the **Space charge density** list, choose **User defined**. In the associated text field, type (c1-c2-2*c3)*e_const*N_A_const.

Surface Charge Density 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Surface Charge Density**.

2 Select boundaries 8 and 12 only.

3 In the **Settings** window for Surface Charge Density, locate the **Surface Charge Density** section. In the **Surface charge density** text field, type sigma.

Electric Potential 1

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

2 Select boundaries 1 and 3 only.

3 In the **Settings** window for Electric Potential, locate the **Electric Potential** section. In the **Electric potential** text field, type VL*phi0.

Electric Potential 2

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

2 Select boundaries 19 and 20 only.

3 In the **Settings** window for Electric Potential, locate the **Electric Potential** section. In the **Electric potential** text field, type VR*phi0.

Electric Potential 3

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

2 Select boundary 10 only.

3 In the **Settings** window for Electric Potential, locate the **Electric Potential** section. In the **Electric potential** text field, type ((VL+VR)/2-vcm)*phi0.

TRANSPORT OF DILUTED SPECIES (TDS)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Transport of Diluted Species (tds)**.

2 In the **Settings** window for Transport of Diluted Species, type c1 Species in the **Label** text field.

3 Locate the **Transport Mechanism** section. Select the **Migration in electric field** check box.

4 Click to expand the **Consistent Stabilization** section, then clear the **Crosswind diffusion** check box.

5 Click to expand the **Advanced Settings** section. From the **Convective term** list, choose **Conservative form**.

6 Click to expand the **Discretization** section. From the **Concentration** list, choose **Quadratic**.

C1 SPECIES (TDS)

Transport Properties 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>c1 Species (tds)** node, then click **Transport Properties 1**.

2 In the **Settings** window for Transport Properties, locate the **Model Inputs** section. From the **Velocity field** list, choose **Velocity field (spf)**. From the **Electric potential** list, choose **Electric potential (es)**. In the **Temperature** text field, type T.

3 Locate the **Diffusion** section. In the **Diffusion coefficient** text field, type D1.

4 Locate the **Migration in Electric Field** section. In the **Charge number** text field, type 1.

Initial Values 1

1 In the **Model Builder** window, click **Component 1 (comp1)>c1 Species (tds)>Initial Values 1.**

2 In the **Settings** window for Initial Values, locate the **Initial Values** section. In the **Concentration** text field, type c0*(1+0.0001*ran(x[1/m],y[1/m])).

Inflow 1

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

2 Select boundaries 1 and 3 only.

3 In the **Settings** window for Inflow, locate the **Concentration** section. Type c0 in the text field.

Outflow 1

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

2 Select boundaries 19 and 20 only.

Symmetry 1

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

2 In the **Settings** window for Symmetry, locate the **Boundary Selection** section. From the **Selection list**, choose **Center Line**.

Concentration 1

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

2 Select boundaries 10 only.

3 In the **Settings** window for Concentration, locate the **Concentration** section, select the **Species c1** check box and type c0*2 in the associated text field.

TRANSPORT OF DILUTED SPECIES 2 (TDS2)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Transport of Diluted Species 2 (tds2)**.

2 In the **Settings** window for Transport of Diluted Species, type c2 Species in the **Label** text field.

3 Locate the **Transport Mechanism** section. Select the **Migration in electric field** check box.

4 Click to expand the **Consistent Stabilization** section, then clear the **Crosswind diffusion** check box.

5 Click to expand the **Advanced Settings** section. From the **Convective term** list, choose **Conservative form**.

6 Click to expand the **Discretization** section. From the **Concentration** list, choose **Quadratic**.

C2 SPECIES (TDS2)

Transport Properties 1

1. In the **Model Builder** window, expand the **Component 1 (comp1)>c2 Species (tds2)** node, then click **Transport Properties 1**.
2. In the **Settings** window for Transport Properties, locate the **Model Inputs** section. From the **Velocity field** list, choose **Velocity field (spf)**. From the **Electric potential** list, choose **Electric potential (es)**. In the **Temperature** text field, type T.
3. Locate the **Diffusion** section. In the **Diffusion coefficient** text field, type D2.
4. Locate the **Migration in Electric Field** section. In the **Charge number** text field, type -1.

Initial Values 1

1. In the **Model Builder** window, click **Component 1 (comp1)>c2 Species (tds2)>Initial Values 1.**
2. In the **Settings** window for Initial Values, locate the **Initial Values** section. In the **Concentration** text field, type c0*(1-2*rc0).

Inflow 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Inflow.**
2. Select boundaries 1 and 3 only.
3. In the **Settings** window for Inflow, locate the **Concentration** section. Type c0*(1-2*rc0) in the text field.

Outflow 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Outflow.**
2. Select boundaries 19 and 20 only.

Symmetry 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Symmetry.**
2. In the **Settings** window for Symmetry, locate the **Boundary Selection** section. From the **Selection list**, choose **Center Line**.

TRANSPORT OF DILUTED SPECIES 3 (TDS3)

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Transport of Diluted Species 3 (tds3)**.
2. In the **Settings** window for Transport of Diluted Species, type c3 Species in the **Label** text field.
3. Locate the **Transport Mechanism** section. Select the **Migration in electric field** check box.
4. Click to expand the **Consistent Stabilization** section, then clear the **Crosswind diffusion** check box.
5. Click to expand the **Advanced Settings** section. From the **Convective term** list, choose **Conservative form**.
6. Click to expand the **Discretization** section. From the **Concentration** list, choose **Quadratic**.

C3 SPECIES (TDS3)

Transport Properties 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>c3 Species (tds3)** node, then click **Transport Properties 1**.

2 In the **Settings** window for Transport Properties, locate the **Model Input** section. From the **Velocity field** list, choose **Velocity field (spf)**. From the **Electric potential** list, choose **Electric potential (es)**. In the **Temperature** text field, type T.

3 Locate the **Diffusion** section. In the **Diffusion coefficient** text field, type D3.

4 Locate the **Migration in Electric Field** section. In the **Charge number** text field, type -2.

Initial Values 1

1 In the **Model Builder** window, click **Component 1 (comp1)>c3 Species (tds3)>Initial Values 1.**

2 In the **Settings** window for Initial Values, locate the **Initial Values** section. In the **Concentration** text field, type c0 *rc0.

Inflow 1

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

2 Select boundaries 1 and 3 only.

3 In the **Settings** window for Inflow, locate the **Concentration** section. Type c0*rc0 in the text field.

Outflow 1

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

2 Select boundaries 19 and 20 only.

Symmetry 1

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

2 In the **Settings** window for Symmetry, locate the **Boundary Selection** section. From the **Selection list**, choose **Center Line**.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Edge**.

Edge 1

1 In the **Settings** window for Edge, locate the **Boundary Selection** section. Click the **Paste Selection** button, and type 2, 4-5, 8-13, 15, 17-18 in the text field of the pop-up window.

2 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Edge 1** and choose **Distribution**.

Distribution 1

1 Select Boundaries 8, 9, 12 and 13 only.

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

3 From the **Distribution properties** list, choose **Predefined distribution type**.

4 In the **Number of elements** text field, type 300.

5 In the **Element ratio** text field, type 300.

6 From the **Distribution method**, choose the **Geometric sequence**.

7 Select the **Symmetric distribution** check box.

Edge 1

In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Edge 1** and choose **Distribution**.

Distribution 2

1 Select Boundaries 10 and 11 only.

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

3 From the **Distribution properties** list, choose **Predefined distribution type**.

4 In the **Number of elements** text field, type 60.

5 In the **Element ratio** text field, type 30.

6 From the **Distribution method**, choose the **Geometric sequence**.

7 Select the **Symmetric distribution** check box.

Edge 1

In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Edge 1** and choose **Distribution**.

Distribution 3

1 Select Boundaries 15, 17 and 18 only.

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

3 From the **Distribution properties** list, choose **Predefined distribution type**.

4 In the **Number of elements** text field, type 80.

5 In the **Element ratio** text field, type 2000.

6 From the **Distribution method**, choose the **Geometric sequence**.

7 Select the **Reverse distribution** check box.

Edge 1

In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Edge 1** and choose **Distribution**.

Distribution 4

1 Select Boundaries 2, 4 and 5 only.

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

3 From the **Distribution properties** list, choose **Predefined distribution type**.

4 In the **Number of elements** text field, type 80.

5 In the **Element ratio** text field, type 2000.

6 From the **Distribution method**, choose the **Geometric sequence**.

Mapped 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Mapped**.

2 In the **Settings** window for Mapped, Locate the **Domain Selection** section. From the **Geometric entity level** list, choose **Domain**.

3 Select domains 2, 3 and 5 only.

4 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Mapped 1** and choose **Distribution**.

Distribution 1

1 Select boundaries 3, 7, 16 and 20 only.

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

3 From the **Distribution properties** list, choose **Predefined distribution type**.

4 In the **Number of elements** text field, type 40.

5 In the **Element ratio** text field, type 12000.

6 From the **Distribution method**, choose the **Geometric sequence**.

Mapped 2

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Mapped**.

2 In the **Settings** window for Mapped, Locate the **Domain Selection** section. From the **Geometric entity level** list, choose **Domain**.

3 Select domains 1 and 4 only.

4 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Mapped 2** and choose **Distribution**.

Distribution 1

1 Select boundaries 1, 6, 14 and 19 only.

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

3 From the **Distribution properties** list, choose **Predefined distribution type**.

4 In the **Number of elements** text field, type 40.

5 In the **Element ratio** text field, type 1200000.

6 From the **Distribution method**, choose the **Geometric sequence**.

7 Select the **Reverse distribution** check box.

8 Click **Build All**.

STUDY 1

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

2 In the **Settings** window for study, type Study 1 (reference initial) in the **Label** text field.

3 Locate the **Study Setting** section, clear the **Generate default plots** check box.

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.

2 In the **Settings** window for Stationary, click to expand the **Study extensions** section.

3 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

4 Click the **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
vcm (Cross-membrane Voltage (in thermal voltage))	2	

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

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>Stationary**.

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

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

STUDY 2

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Study 2**.

2 In the **Settings** window for study, type Study 2 (changing vcm) in the **Label** text field.

3 Locate the **Study Setting** section, clear the **Generate default plots** check box.

Step 1: Stationary

1. In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.
2. In the **Settings** window for Stationary, locate the **Values of Dependent Variables** section.
3. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.
4. From the **Method** list, choose **Solution**.
5. From the **Study** list, choose **Study 1 (reference initial), Stationary**.

6. From the **Parameter value (vcm)** list, choose **2**.
7. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.
8. From the **Method** list, choose **Solution**.
9. From the **Study** list, choose **Study 1 (reference initial), Stationary**.
10. From the **Parameter value (vcm)** list, choose **2**.
11. Locate the **Study Extensions** section.
12. Select the **Auxiliary sweep** check box.
13. Click the **Add**.
14. In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
vcm (Cross-membrane Voltage(in thermal voltage))	range(2,1,50)	

15. From the **Run continuation for** list, choose **No parameter**.
16. From the **Reuse solution from previous step** list, choose **Yes**.
17. On the **Study** toolbar, click **Compute**.

R E S U L T S

1D Plot Group 1

1. On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2. In the **Settings** window for 1D Plot Group, type Enrichment factor in the **Label** text field.
3. Locate the **Data** section. From the **Data set** list, choose **Study 2 (changing vcm)/Solution 2(sol2).**
4. From the **Parameter selection (vcm)** list, choose **From list**.
5. From the **Parameter values (vcm)** list, choose **4**, **10**, **14**, **17**, **20**, **23**, **26**, **34** and **50**.
6. Locate the **Plot Settings** section. Select the **x-axis label** check box and type x(μm) in the associated text field.
7. Locate the **Plot Settings** section. Select the **y-axis label** check box and type Enrichment factor in the associated text field.
8. Locate the **Axis** section, select the **y-axis log scale** check box.
9. Click to expand the **Legend** section. Select the **Show legends** check box.

Line Graph 1

1. On the **Enrichment factor** toolbar, click **Line Graph**.
2. In the **Settings** window for Line Graph, locate the **Selection** section.
3. From the **Selection** list, choose **Center Line**.

4. Locate the **y-Axis Data** section. In the **Expression** text field, type c3/(c0*rc0).

5. Click to expand the **Coloring and Style** section. In the **Width** text field, type 2.

6. Click to expand the **Legends** section. Select the **Show legends** check box.

7. From the **Legends** list, choose **Manual**.

8. In the **Legends** table, enter the following setting:

Legends
vcm=4 VT
vcm=10 VT
vcm=14 VT
vcm=17 VT
vcm=20 VT
vcm=23 VT
vcm=26 VT
vcm=34 VT
vcm=50 VT

9. On the **Enrichment factor** toolbar, click **Plot**.

1D Plot Group 2

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

2 In the **Settings** window for 1D Plot Group, type Concentration (a) in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 2 (changing vcm)/Solution 2(sol2).**

4 From the **Parameter selection (vcm)** list, choose **From list**.

5 From the **Parameter values (vcm)** list, choose **14.**

6 Locate the **Plot Settings** section. Select the **x-axis label** check box and type x(μm) in the associated text field.

7 Locate the **Axis** section, select the **y-axis log scale** check box.

8 Locate the **Legend** section, select the **Show legends** check box.

Line Graph 1

1 On the **Concentration (a)** toolbar, click **Line Graph**.

2 In the **Settings** window for Line Graph, type c1 in the **Label** text field.

3 Locate the **Selection** section. From the **Selection** list, choose **Center Line**.

4 Locate the **y-Axis Data** section. Click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Component 1>c1 Species>c1 - Concentration**.

5 Click to expand the **Coloring and Style** section. In the **Width** text field, type 2.

6 Click to expand the **Legends** section. Select the **Show legends** check box.

7 From the **Legends** list, choose **Manual**.

8 In the **Legends** table, enter the following setting:

Lengends
K+

Line Graph 2

1 On the **Concentration (a)** toolbar, click **Line Graph**.

2 In the **Settings** window for Line Graph, type c2 in the **Label** text field.

3 Locate the **Selection** section. From the **Selection** list, choose **Center Line**.

4 Locate the **y-Axis Data s**ection. Click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Component 1>c2 Species>c2 - Concentration**.

5 Click to expand the **Coloring and Style** section.

6 From the **Line** list, choose **Dashed**.

7 In the **Width** text field, type 2.

8 Click to expand the **Legends** section. Select the **Show legends** check box.

9 From the **Legends** list, choose **Manual**.

10 In the **Legends** table, enter the following setting:

Lengends
Cl-

Line Graph 3

1 On the **Concentration (a)** toolbar, click **Line Graph**.

2 In the **Settings** window for Line Graph, type c3 in the **Label** text field.

3 Locate the **Selection** section. From the **Selection** list, choose **Center line**.

4 Locate the **y-Axis Data** section. Click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Component 1>c3 Species>c3 - Concentration**.

5 Click to expand the **Coloring and Style** section. In the **Width** text field, type 2.

6 Click to expand the **Legends** section. Select the **Show legends** check box.

7 From the **Legends** list, choose **Manual**.

8 In the **Legends** table, enter the following setting:

Lengends
P2-

9 On the **Concentration (a)** toolbar, click **Plot**.

1D Plot Group 3

1 In the **Model Builder** window, right click **Results>Concentration (a).** Choose **Duplicate.**

2 In the **Settings** window for 1 D Plot Group, type Concentration (b) in the **Label** text field.

3 Locate the **Data** section. From the **Parameter values (vcm)** list, choose **34**.

4 On the **Concentration (b)** toolbar, click **Plot.**

1D Plot Group 4

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

2 In the **Settings** window for 1D Plot Group, type Potential in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 2 (changing vcm)/Solution 2(sol2).**

4 From the **Parameter selection (vcm)** list, choose **From list**.

5 From the **Parameter values (vcm)** list, choose **4**, **14** and **34**.

6 Locate the **Plot Settings** section. Select the **x-axis label** check box and type x(μm) in the associated text field.

7 Select the **y-axis label** check box and type Φ/V_T in the associated text field.

8 Locate the **Legend** section, select the **Show legends** check box.

Line Graph 1

1 On the **Potential** toolbar, click **Line Graph**.

2 Locate the **Selection** section. From the **Selection** list, choose **Center line**.

3 Locate the **y-Axis Data** section. In the **Expression** text field, type V/phi0.

4 Click to expand the **Coloring and Style** section. In the **Width** text field, type 2.

5 Click to expand the **Legends** section. Select the **Show legends** check box.

6 From the **Legends** list, choose **Manual**.

7 In the **Legends** table, enter the following setting:

Lengends
vcm=4 VT
vcm=14 VT
vcm=34 VT

8 On the **Potential** toolbar, click **Plot**.

1D Plot Group 5

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

2 In the **Settings** window for 1D Plot Group, type Ex in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 2 (changing vcm)/Solution 2(sol2).**

4 From the **Parameter selection (vcm)** list, choose **From list.**

5 From the **Parameter values (vcm)** list, choose **4**, **14** and **34**.

6 Locate the **Plot Settings** section. Select the **x-axis label** check box and type x(μm) in the associated text field.

7 Select the **y-axis label** check box and type Ex (V/m) in the associated text field.

8 Locate the **Legend** section, select the **Show legends** check box.

Line Graph 1

1 On the **Ex** toolbar, click **Line Graph**.

2 Locate the **Selection** section. From the **Selection** list, choose **Center line**.

3 Locate the **y-Axis Data** section. Click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Component 1>Electrostatics>Electric> Electric field>es.Ex – Electric field, x component.**

4 Click to expand the **Coloring and Style** section. In the **Width** text field, type 2.

5 Click to expand the **Legends** section. Select the **Show legends** check box.

6 From the **Legends** list, choose **Manual**.

7 In the **Legends** table, enter the following setting:

Legends
vcm=4 VT
vcm=14 VT
vcm=34 VT

8 On the **Ex** toolbar, click **Plot**.

1D Plot Group 6

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

2 In the **Settings** window for 1D Plot Group, type Pressure in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 2 (changing vcm)/Solution 2(sol2).**

4 From the **Parameter selection (vcm)** list, choose **From list**.

5 From the **Parameter values (vcm)** list, choose **4, 14, 20** and **34**.

6 Locate the **Plot Settings** section. Select the **x-axis label** check box and type x(μm) in the associated text field.

7 Locate the **Legend** section, select the **Show legends** check box.

Line Graph 1

1 On the **Pressure** toolbar, click **Line Graph**.

2 Locate the **Selection** section. From the **Selection** list, choose **Center line**.

3 Locate the **y-Axis Data** section. Click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Component 1>Creeping >Velocity and pressure>p – Pressure.**

4 Click to expand the **Coloring and Style** section. In the **Width** text field, type 2.

5 Click to expand the **Legends** section. Select the **Show legends** check box.

6 From the **Legends** list, choose **Manual**.

7 In the **Legends** table, enter the following setting:

Legends
vcm=4 VT

vcm=14 VT
vcm=20 VT
vcm=34 VT

8 On the **Pressure** toolbar, click **Plot**.

2D Plot Group 7

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

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

3 From the **Data set** list, choose **Study 2 (changing vcm)/Solution 2(sol2).**

4 From the **Parameter values (vcm)** list, choose **14**.

5 Locate the **Legend** section, select the **Show legends** check box.

Streamline 1

1 Right click **2D Plot Group 7** and choose **Streamline**.

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

3 From the **Positioning** list, choose **Magnitude controlled.**

4 In the **Density** text field, type 31.

5 On the **2D Plot Group 7** toolbar, click **Plot**.