Electrolyte-Gated Organic Field-Effect Transistor

Introduction

An Electrolyte-Gated Organic Field-Effect Transistor, for short EGOFET, is a type of field-effect transistor where the gate dielectric layer is replaced by an electrolyte and the semiconductor is an organic material. Just like a traditional field-effect transistor, an EGOFET comprises tree electrodes, including source, drain, and gate electrode.

In an EGOFET constructed with a p-type organic semiconductor, the gate is negatively polarized, causing the positive ions of the electrolyte to accumulate at the gate–electrolyte interface. Concurrently, the negative ions accumulate at the electrolyte–organic semiconductor interface. This ends up in the formation of electrical double layer at the two aforementioned interfaces.

In this example, we are going to show how to model the static characteristics of this device based on a general Drift–Diffusion model.

Model Definition

The materials and geometry of this model are a simplified version of the device described in Ref. 1. The model uses the following configuration: source and drain electrodes positioned on the two side ends of the organic semiconductor domain and the gate electrode placed at the top boundary of the electrolyte domain. The total depth of the transistor is 1 cm with a width of 30 μm. Figure 1 shows the configuration of the modeled device.

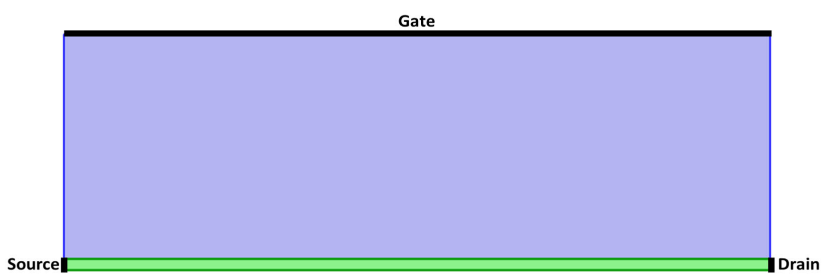

Figure 1: Configuration of the modeled EGOFET, with the organic semiconductor domain in green and the electrolyte in blue.

In a p-type organic semiconductor, the charge carriers are holes, while in the electrolyte domain, ions serve as the charge carriers. The model uses the Stabilized Convection–Diffusion Equation interface for defining transport phenomena (Equation 1) and the

Electrostatics interface to describe the motion of the charges under the applied electric field (Equation 2):

$$\nabla \cdot J_c = \nabla \cdot \left(-D_c \nabla c - z_c \frac{D_c}{RT} F c \nabla V\right) = 0 \tag{1}$$

$$-\nabla \cdot (\varepsilon_0 \varepsilon_r \nabla V) = c \times F \tag{2}$$

In the equations above, J_c, D_c, and z_c, are flux, diffusion coefficient, and charge of the charge carriers, respectively. R is the ideal gas constant, T the temperature, F the Faraday constant, and V the electric potential. ε_0 and ε_r are the permittivity of vacuum and relative permittivity of the material, respectively.

Since there are three different types of charge carriers in this device, we use the same set of equations for each, namely the holes, positive ions and negative ions. The procedure of the implementation is described in detail in the Modeling Instructions section.

Results and Discussion

Figure 2 shows the electric field profile at the cross-section of the device (along the y direction) for zero drain voltage and gate voltage varying from 0 to –0.5 V. The profile shows a high electric field at the two interfaces, corresponding to the gate–electrolyte and electrolyte–organic semiconductor interface. This is due to the accumulation of charges at these interfaces.

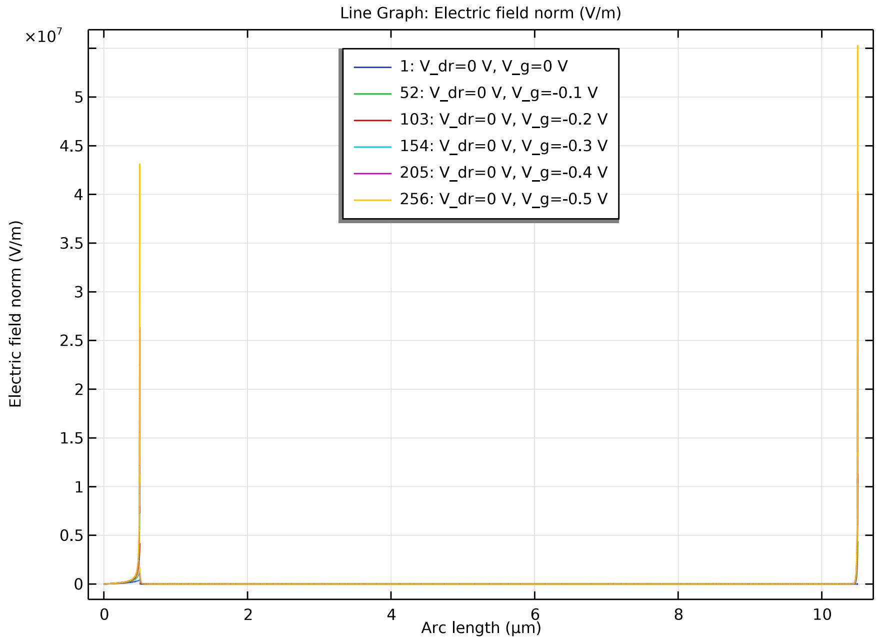

Figure 2: Electric field along the cross section of the EGOFET.

Figure 3 shows the transfer curve of the EGOFET which is drain current versus the gate voltage.

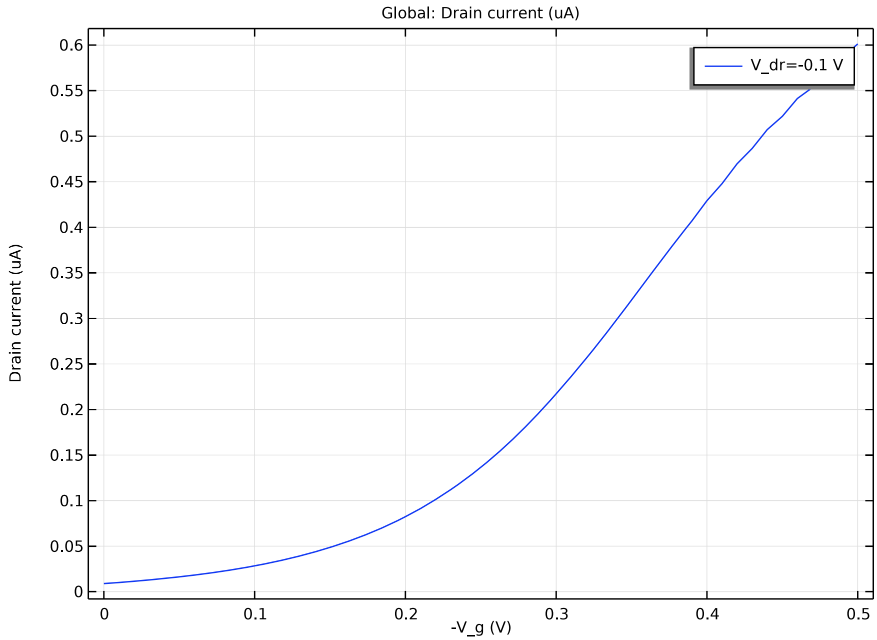

Figure 3: Drain current versus gate voltage curve.

Reference

1. N. Delavari, K. Tybrandt, M. Berggren, B. Piro, V. Noël, G.Mattana, and I. Zozoulenko, “Nernst–Planck–Poisson analysis of electrolyte-gated organic field-effect transistors,” *J. Phys. D: Appl. Phys.*, vol. 54, no. 41, 415101, 2021.

Application Library path: `Semiconductor_Module/Transistors/egofet`

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 **Mathematics>Classical PDEs>Stabilized Convection-Diffusion Equation (scdeq)**.
3 Click **Add**.
4 In the **Select Physics** tree, select **Mathematics>Classical PDEs>Stabilized Convection-Diffusion Equation (scdeq)**.
5 Click **Add**.
6 In the **Select Physics** tree, select **Mathematics>Classical PDEs>Stabilized Convection-Diffusion Equation (scdeq)**.
7 Click **Add**.
8 In the **Select Physics** tree, select **AC/DC>Electric Fields and Currents>Electrostatics (es)**.
9 Click **Add**.
10 Click **Study**.
11 In the **Select Study** tree, select **General Studies>Stationary**.
12 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Value	Description
w_dom	30[um]	3E-5 m	Domain width
h_semi	500[nm]	5E-7 m	Organic semiconductor thickness
h_electrolyte	10[um]	1E-5 m	Electrolyte thickness
V_dr	0[V]	0 V	Drain electrode Voltage
V_g	0[V]	0 V	Gate electrode voltage

Name	Expression	Value	Description
D_ch	1e-6[m^2/s]	1E-6 m²/s	Diffusion coefficient for holes
D_c	15e-10[m^2/s]	1.5E-9 m²/s	Diffusion coefficient for ions
T	300[K]	300 K	Temperature
d	1[cm]	0.01 m	depth
z_ch	+1	1	charge of hole
z_cp	+1	1	charge of positive ions
z_cn	-1	-1	charge of negative ions
ch0	1e-3[mol/m^3]	0.001 mol/m³	Holes initial concentration
cp0	1[mol/m^3]	1 mol/m³	Positive ions initial concentration
cn0	1[mol/m^3]	1 mol/m³	Negative ions initial concentration

GEOMETRY 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Geometry 1** node, then 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 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 `w_dom`.

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

5 Right-click **Rectangle 1 (r1)** and choose **Duplicate**.

Rectangle 2 (r2)

1 In the **Model Builder** window, click **Rectangle 2 (r2)**.

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

3 Clear the **Height** text field.

4 Locate the **Position** section. In the **y** text field, type h_semi.

5 Locate the **Size and Shape** section. In the **Height** text field, type h_electrolyte.

6 Click **Build All Objects**.

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 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type 6 in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Integration**, locate the **Advanced** section.

8 From the **Method** list, choose **Summation over nodes**.

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
I_dr	intop1(-reacf(ch))[mol/(m*s)]* F_const*d		Drain current

DRIFT-DIFFUSION EQUATION FOR HOLES

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Stabilized Convection-Diffusion Equation (scdeq)**.

2 In the **Settings** window for **Stabilized Convection-Diffusion Equation**, type Drift-Diffusion Equation for Holes in the **Label** text field.

3 Locate the **Domain Selection** section. In the list, select **2**.

4 Click **Remove from Selection**.

5 Select Domain 1 only.

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

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

Dependent variable quantity	Unit
Custom unit	mol/m^3

8 Click **Define Source Term Unit**.

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

Source term quantity	Unit
Custom unit	mol/(m^3*s)

10 Click to expand the **Dependent Variables** section. In the **Dependent variable (mol/m³)** text field, type `ch`.

Convection-Diffusion Equation 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Drift-Diffusion Equation for Holes (scdeq)** click **Convection-Diffusion Equation 1**.

2 In the **Settings** window for **Convection-Diffusion Equation**, locate the **Diffusion Coefficient** section.

3 In the c text field, type `D_ch`.

4 Click to expand the **Conservative Flux Convection Coefficient** section. Specify the α vector as

-z_ch*D_ch/(R_const*T)*F_const*Vx	x
-z_ch*D_ch/(R_const*T)*F_const*Vy	y

5 Click to collapse the **Convection Coefficient** section. Locate the **Source Term** section. In the f text field, type `0`.

Initial Values 1

1 In the **Model Builder** window, click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the ch text field, type `ch0`.

No Flux 1

1 In the **Physics** toolbar, click **Boundaries** and choose **No Flux**.

2 In the **Settings** window for **No Flux**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

Dirichlet Boundary Condition 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Dirichlet Boundary Condition**.
2 In the **Settings** window for **Dirichlet Boundary Condition**, locate the **Boundary Selection** section.
3 Click **Paste Selection**.
4 In the **Paste Selection** dialog box, type `1 6` in the **Selection** text field.
5 Click **OK**.
6 In the **Settings** window for **Dirichlet Boundary Condition**, locate the **Value on Boundary** section.
7 In the *r* text field, type `ch0`.

DRIFT-DIFFUSION EQUATION FOR POSITIVE IONS

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Stabilized Convection-Diffusion Equation 2 (scdeq2)**.
2 In the **Settings** window for **Stabilized Convection-Diffusion Equation**, type `Drift-Diffusion Equation for Positive Ions` in the **Label** text field.
3 Locate the **Domain Selection** section. In the list, select **1**.
4 Click **Remove from Selection**.
5 Select Domain 2 only.
6 Locate the **Units** section. Click **Define Dependent Variable Unit**.
7 In the **Dependent variable quantity** table, enter the following settings:

Dependent variable quantity	Unit
Custom unit	mol/m^3

8 Click **Define Source Term Unit**.
9 In the **Source term quantity** table, enter the following settings:

Source term quantity	Unit
Custom unit	mol/(m^3*s)

10 Locate the **Dependent Variables** section. In the **Dependent variable (mol/m³)** text field, type `cp`.

Convection-Diffusion Equation 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Drift-Diffusion Equation for Positive Ions (scdeq2)** click **Convection-Diffusion Equation 1**.

2 In the **Settings** window for **Convection-Diffusion Equation**, locate the **Diffusion Coefficient** section.

3 In the *c* text field, type `D_c`.

4 Locate the **Conservative Flux Convection Coefficient** section. Specify the α vector as

`-z_cp*D_c/(R_const*T)*F_const*Vx`	x
`-z_cp*D_c/(R_const*T)*F_const*Vy`	y

5 Locate the **Source Term** section. In the *f* text field, type `0`.

Initial Values 1

1 In the **Model Builder** window, click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the *cp* text field, type `cp0`.

No Flux 1

1 In the **Physics** toolbar, click **Boundaries** and choose **No Flux**.

2 In the **Settings** window for **No Flux**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

Dirichlet Boundary Condition 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Dirichlet Boundary Condition**.

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

3 Select Boundaries 3 and 7 only.

4 In the **Settings** window for **Dirichlet Boundary Condition**, locate the **Value on Boundary** section.

5 In the *r* text field, type `cp0`.

DRIFT-DIFFUSION EQUATION FOR NEGATIVE IONS

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Stabilized Convection-Diffusion Equation 3 (scdeq3)**.

2 In the **Settings** window for **Stabilized Convection-Diffusion Equation**, type `Drift-Diffusion Equation for Negative Ions` in the **Label** text field.

3 Locate the **Domain Selection** section. In the list, select **1**.

4 Click **Remove from Selection**.

5 Select Domain 2 only.

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

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

Dependent variable quantity	Unit
Custom unit	`mol/m^3`

8 Click **Define Source Term Unit**.

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

Source term quantity	Unit
Custom unit	`mol/(m^3*s)`

10 Locate the **Dependent Variables** section. In the **Dependent variable (mol/m³)** text field, type `cn`.

Convection-Diffusion Equation 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Drift-Diffusion Equation for Negative Ions (scdeq3)** click **Convection-Diffusion Equation 1**.

2 In the **Settings** window for **Convection-Diffusion Equation**, locate the **Diffusion Coefficient** section.

3 In the *c* text field, type `D_c`.

4 Locate the **Conservative Flux Convection Coefficient** section. Specify the α vector as

`-z_cn*D_c/(R_const*T)*F_const*Vx`	x
`-z_cn*D_c/(R_const*T)*F_const*Vy`	y

5 Locate the **Source Term** section. In the *f* text field, type `0`.

Initial Values 1

1 In the **Model Builder** window, click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the *cn* text field, type `cn0`.

No Flux 1

1 In the **Physics** toolbar, click **Boundaries** and choose **No Flux**.

2 In the **Settings** window for **No Flux**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

Dirichlet Boundary Condition 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Dirichlet Boundary Condition**.

2 Select Boundaries 3 and 7 only.

3 In the **Settings** window for **Dirichlet Boundary Condition**, locate the **Value on Boundary** section.

4 In the *r* text field, type `cn0`.

ELECTROSTATICS (ES)

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

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

3 In the *d* text field, type `d`.

4 Click to expand the **Dependent Variables** section.

Charge Conservation 1

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

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

3 In the list, select **2**.

4 Locate the **Constitutive Relation D-E** section. From the ε_r list, choose **User defined**. Locate the **Domain Selection** section. In the list, select **1**.

5 Locate the **Constitutive Relation D-E** section. In the ε_r text field, type `3`.

6 Right-click **Component 1 (comp1)>Electrostatics (es)>Charge Conservation 1** and choose **Duplicate**.

Charge Conservation 2

1 In the **Model Builder** window, click **Charge Conservation 2**.

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

3 In the list, select **1**.

4 Click **Remove from Selection**.

5 Select Domain 2 only.

6 Locate the **Constitutive Relation D-E** section. In the ε_r text field, type `79`.

V_s

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

2 In the **Settings** window for **Electric Potential**, type `V_s` in the **Label** text field.

3 Locate the **Boundary Selection** section. Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `1` in the **Selection** text field.

5 Click **OK**.

6 Right-click **V_s** and choose **Duplicate**.

V_dr

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

2 In the **Settings** window for **Electric Potential**, type V_dr in the **Label** text field.

3 Locate the **Boundary Selection** section. Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type 6 in the **Selection** text field.

6 Click **OK**.

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

8 In the V_0 text field, type V_dr.

9 Right-click **V_dr** and choose **Duplicate**.

V_g

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

2 In the **Settings** window for **Electric Potential**, type V_g in the **Label** text field.

3 Locate the **Boundary Selection** section. Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type 5 in the **Selection** text field.

6 Click **OK**.

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

8 In the V_0 text field, type V_g.

Space Charge Density 1

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

2 Select Domain 1 only.

3 In the **Settings** window for **Space Charge Density**, locate the **Space Charge Density** section.

4 In the ρ_v text field, type ch*F_const.

5 Right-click **Space Charge Density 1** and choose **Duplicate**.

Space Charge Density 2

1 In the **Model Builder** window, click **Space Charge Density 2**.

2 In the **Settings** window for **Space Charge Density**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 2 only.

5 Locate the **Space Charge Density** section. In the ρ_v text field, type `(cp-cn)*F_const`.

MESH 1

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

Size 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Mesh 1>Free Triangular 1** node.

2 Right-click **Free Triangular 1** and choose **Size**.

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

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

5 Click **Paste Selection**.

6 In the **Paste Selection** dialog box, type `1` in the **Selection** text field.

7 Click **OK**.

8 In the **Settings** window for **Size**, locate the **Element Size** section.

9 Click the **Custom** button.

10 Locate the **Element Size Parameters** section.

11 Select the **Maximum element size** check box. In the associated text field, type `1e-7[m]`.

Distribution 1

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `5` in the **Selection** text field.

5 Click **OK**.

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

7 In the **Number of elements** text field, type `500`.

8 Right-click **Distribution 1** and choose **Duplicate**.

Distribution 2

1 In the **Model Builder** window, click **Distribution 2**.

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

3 Click **Clear Selection**.

4 Click Paste Selection.

5 In the **Paste Selection** dialog box, type `1 6` in the **Selection** text field.

6 Click **OK**.

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

8 In the **Number of elements** text field, type `100`.

Boundary Layers 1

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

Boundary Layer Properties

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `4 5` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Boundary Layer Properties**, locate the **Layers** section.

7 In the **Stretching factor** text field, type `1.05`.

8 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

STUDY 1

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 Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, click to select the cell at row number 1 and column number 2.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
V_g (Gate electrode voltage)		V

7 Click **Range**.

8 In the **Range** dialog box, type `0` in the **Start** text field.

9 In the **Step** text field, type `-0.1`.

10 In the **Stop** text field, type `-0.5`.

11 Click **Replace**.

12 In the **Settings** window for **Stationary**, locate the **Study Extensions** section.

13 Click **Add**.

14 In the table, click to select the cell at row number 2 and column number 2.

15 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
V_dr (Drain electrode Voltage)		V

16 From the **Sweep type** list, choose **All combinations**.

17 Click **Range**.

18 In the **Range** dialog box, type `0` in the **Start** text field.

19 In the **Step** text field, type `-0.01`.

20 In the **Stop** text field, type `-0.5`.

21 Click **Replace**.

22 In the **Home** toolbar, click **Compute**.

RESULTS

Cut Line 2D 1

1 In the **Results** toolbar, click **Cut Line 2D**.

2 In the **Settings** window for **Cut Line 2D**, locate the **Line Data** section.

3 From the **Line entry method** list, choose **Point and direction**.

4 Find the **Point** subsection. In the **x** text field, type `w_dom/2`.

5 Find the **Direction** subsection. In the **x** text field, type `0`.

6 In the **y** text field, type `1`.

7 Click **Plot**.

V-y

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `V-y` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter selection (V_dr)** list, choose **First**.

Line Graph 1

Right-click **V-y** and choose **Line Graph**.

V-y

From the **Dataset** list, choose **Cut Line 2D 1**.

Line Graph 1

1 In the **Model Builder** window, click **Line Graph 1**.

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

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

4 In the **V-y** toolbar, click **Plot**.

V-y

In the **Model Builder** window, right-click **V-y** and choose **Duplicate**.

E-y

1 In the **Model Builder** window, under **Results** click **V-y 1**.

2 In the **Settings** window for **1D Plot Group**, type `E-y` in the **Label** text field.

3 Locate the **Legend** section. From the **Position** list, choose **Upper middle**.

Line Graph 1

1 In the **Model Builder** window, expand the **E-y** node, then click **Line Graph 1**.

2 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Electrostatics>Electric>es.normE - Electric field norm - V/m**.

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

4 In the **E-y** toolbar, click **Plot**.

E-y

In the **Model Builder** window, right-click **E-y** and choose **Duplicate**.

ch-y

1 In the **Model Builder** window, under **Results** click **E-y 1**.

2 In the **Settings** window for **1D Plot Group**, type `ch-y` in the **Label** text field.

Line Graph 1

1 In the **Model Builder** window, expand the **ch-y** node, then click **Line Graph 1**.

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

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

4 In the **ch-y** toolbar, click **Plot**.

5 Click the **y-Axis Log Scale** button in the **Graphics** toolbar.

ch-y

In the **Model Builder** window, right-click **ch-y** and choose **Duplicate**.

cp-y

1 In the **Model Builder** window, under **Results** click **ch-y 1**.

2 In the **Settings** window for **1D Plot Group**, type `cp-y` in the **Label** text field.

Line Graph 1

1 In the **Model Builder** window, expand the **cp-y** node, then click **Line Graph 1**.

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

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

4 In the **cp-y** toolbar, click **Plot**.

cp-y

In the **Model Builder** window, right-click **cp-y** and choose **Duplicate**.

cn-y

1 In the **Model Builder** window, under **Results** click **cp-y 1**.

2 In the **Settings** window for **1D Plot Group**, type `cn-y` in the **Label** text field.

Line Graph 1

1 In the **Model Builder** window, expand the **cn-y** node, then click **Line Graph 1**.

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

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

4 In the **cn-y** toolbar, click **Plot**.

I_dr-V_dr

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

2 In the **Settings** window for **1D Plot Group**, type `I_dr-V_dr` in the **Label** text field.

Global 1

1 Right-click **I_dr-V_dr** 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**
`I_dr`	`uA`	`Drain current`

4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

5 In the **Expression** text field, type `-V_dr`.

6 Click to expand the **Legends** section. Find the **Include** subsection. Clear the **Description** check box.

7 In the **I_dr-V_dr** toolbar, click **Plot**.

ADD STUDY

1 In 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 **General Studies>Stationary**.

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

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

STUDY 2_SWEEPING V_G

1 In the **Model Builder** window, click **Study 2**.

2 In the **Settings** window for **Study**, type `Study 2_Sweeping V_g` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Step 1: Stationary

1 In the **Model Builder** window, under **Study 2_Sweeping V_g** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Study Extensions** section.

3 Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
V_dr (Drain electrode Voltage)	`-0.1`	V

6 Click **Add**.

7 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
V_g (Gate electrode voltage)		V

8 From the **Sweep type** list, choose **All combinations**.

9 Click **Range**.

10 In the **Range** dialog box, type `0` in the **Start** text field.

11 In the **Step** text field, type `-0.01`.

12 In the **Stop** text field, type `-0.5`.

13 Click **Replace**.

14 In the **Home** toolbar, click **Compute**.

RESULTS

I_dr-V_dr

In the **Model Builder** window, under **Results** right-click **I_dr-V_dr** and choose **Duplicate**.

I_dr-V_g

1. In the **Model Builder** window, under **Results** click **I_dr-V_dr 1**.
2. In the **Settings** window for **1D Plot Group**, type `I_dr-V_g` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Study 2_Sweeping V_g/ Solution 2 (sol2)**.

Global 1

1. In the **Model Builder** window, expand the **I_dr-V_g** node, then click **Global 1**.
2. In the **Settings** window for **Global**, locate the **y-Axis Data** section.
3. In the table, enter the following settings:

Expression	Unit	Description
`I_dr`	`uA`	`Drain current`

4. Locate the **x-Axis Data** section. In the **Expression** text field, type `-V_g`.
5. In the **I_dr-V_g** toolbar, click **Plot**.