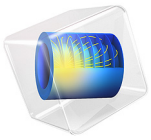
Created in COMSOL Multiphysics 6.2



3D Density-Gradient Simulation of a Nanowire MOSFET

This 3D model of a nanowire MOSFET employs the density-gradient theory to add the effect of quantum confinement to the conventional drift–diffusion formulation, without requiring excessively high computational costs. The oxide layer is simulated explicitly with geometric domains, and quantum confinement at the silicon–oxide interface is accounted for via a dedicated boundary condition. The density-gradient effective mass is anisotropic. Various selection utilities are used to simplify the assignment of physics settings and plot selections. The result matches well with the Id–Vg curves and electron density profiles published in the reference paper.

Introduction

As the semiconductor fabrication technology progresses to produce smaller and smaller devices, the effect of quantum confinement becomes more and more important. This tutorial follows the approach described in Ref. 1 to compute the electron density distribution and drain current as a function of the gate voltage of a Si nanowire MOSFET using the density-gradient theory (as reviewed in Ref. 2).

Model Definition

The model computes the DC characteristics and the electron density profile along the transverse and longitudinal centerlines of a Si nanowire MOSFET, to be compared with Fig. 2, 4, and 5 in Ref. 1. The effects of changing the value of the longitudinal effective mass are shown in those figures.

Not all simulation details are given in the paper. Nevertheless, the approach described in the paper is followed as closely as possible. In particular, the treatment of the potential barrier at the silicon–oxide interface follows the one discussed in Ref. 3. When a parameter is not listed, a typical value found in the literature is used in the model. The silicon material data built in the Semiconductor Module are used, with constant mobility. The gate metal work function and the drain voltage are selected to best match the Id–Vg curves in the reference paper. The section Modeling Instructions lists all the parameters used in the model.

The channel of the simulated structure is formed by a Si nanowire with a 3.2 nm square cross section, surrounded by an oxide layer of thickness 0.8 nm. The length of the channel is 4 nm. (See Fig. 1 in Ref. 1.) The source and drain lengths are arbitrarily set at 15 nm, long enough to provide the comparison with the figures in the reference paper.

The oxide layer is modeled explicitly as domains using the **Charge Conservation** domain condition. The quantum confinement effect for the silicon–oxide interface is added by

selecting the **Potential barrier** option for the **Insulator Interface** boundary condition, based on Ref. 3. The temperature is assumed to be 300 K. Maxwell–Boltzmann statistics is used, as indicated by Eq. (2) in Ref. 1.

The density-gradient effective mass is anisotropic in this model. This is done by selecting the **Diagonal** option under the section **Material Properties, Density-Gradient** in the settings window for the **Semiconductor Material Model** domain condition.

For the ease of meshing and plotting along the centerlines, some extra layers are created in the 3D geometry. Various **Selection** functionalities under both the **Geometry** and the **Definitions** nodes are then used to group the multiple geometric entities into different regions for convenient assignment of domain conditions, boundary conditions, and plot selections.

To reduce the computation time and file size, a relatively coarse mesh is used in this tutorial. Interested users are encouraged to perform mesh refinement studies by parameterizing the settings of the **Distribution** nodes used in the meshing sequence.

See the comments in the section Modeling Instructions for more detailed discussions on the model construction, meshing techniques, solution processes, and result visualization.

Results and Discussion

Figure 1 shows the drain current as a function of the gate voltage for a set of longitudinal effective mass values, to be compared with Fig. 2 in Ref. 1. The general trend and subthreshold slopes compare well, with some minor differences in the magnitude of the current.

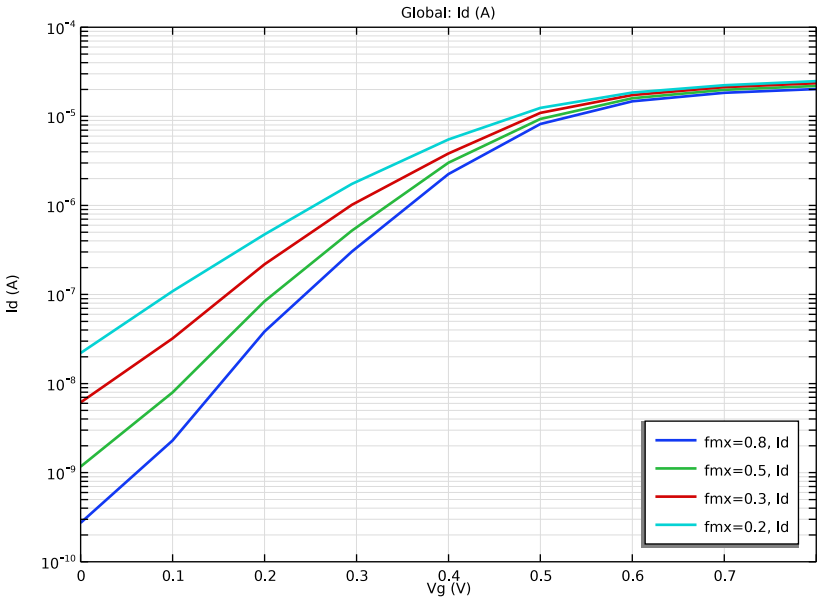


Figure 1: Drain current versus gate voltage curves for a few effective mass values.

Figure 2 and Figure 3 plot the electron concentration profiles along the longitudinal and transverse centerlines, which compares well with Fig. 4 and 5 in the reference paper.

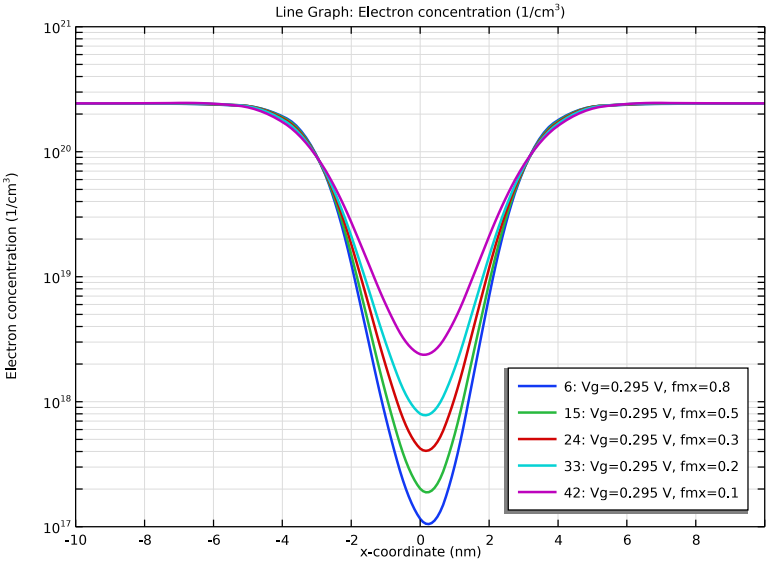


Figure 2: Electron concentration profile along the longitudinal centerline.

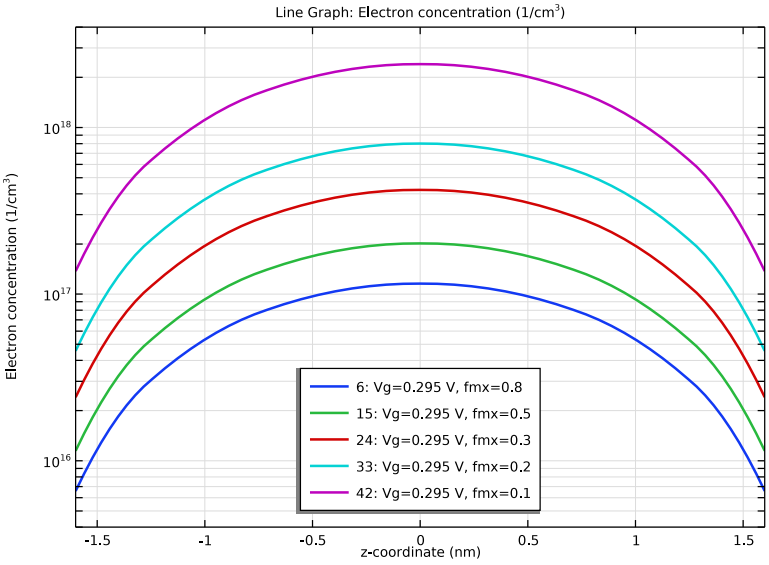


Figure 3: Electron concentration profile along the transverse centerline.

References

1. A.R. Brown, A. Martinez, N. Seoane, and A. Asenov, "Comparison of Density Gradient and NEGF for 3D Simulation of a Nanowire MOSFET," *Proc. 2009 Spanish Conf. Elec. Dev.*, p. 140, Feb. 11–13, 2009.

2. M.G. Ancona, "Density-gradient theory: a macroscopic approach to quantum confinement and tunneling in semiconductor devices," *J. Comput. Electron.*, vol. 10, p. 65, 2011.

3. S. Jin, Y.J. Park, and H.S. Min, "Simulation of Quantum Effects in the Nano-scale Semiconductor Device," *J. Semicond. Tech. Sci.*, vol. 4, no. 1, p. 32, 2004.

Application Library path: `Semiconductor_Module/Transistors/nanowire_density_gradient_3d`

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 **3D**.
2. In the **Select Physics** tree, select **Semiconductor>Semiconductor (semi)**.
3. Click **Add**.
4. Click **Study**.

   The **Semiconductor Equilibrium** study step is suitable either to solve for systems known to be in thermal equilibrium, or, in this case, to provide a good initial condition for subsequent gate voltage sweeps.
5. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces> Semiconductor Equilibrium**.
6. Click **Done**.

GEOMETRY 1

The Model Wizard starts the COMSOL Desktop at the **Geometry** node. Use this chance to select a convenient length unit. Then enter some dimensional parameters for building the geometry.

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

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

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

GLOBAL DEFINITIONS

Parameters 1: Geometry

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, type `Parameters 1: Geometry` in the **Label** text field.

3 Locate the **Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
Lgate	4[nm]	4E-9 m	Gate length
Wchannel	3.2[nm]	3.2E-9 m	Channel width
dOx	0.8[nm]	8E-10 m	Oxide thickness
Lsource	15[nm]	1.5E-8 m	Source length
Ldrain	15[nm]	1.5E-8 m	Drain length
Ltot	Lsource+Lgate+Ldrain	3.4E-8 m	Total length
Wtot	Wchannel+dOx*2	4.8E-9 m	Total width

Start building the geometry. Add some extra layers and lines for meshing and plotting purposes. Use the various **Selection** functionalities under both the **Geometry** and the **Definitions** nodes to group the multiple geometric entities into different regions for convenient assignment of domain conditions, boundary conditions, and plot selections.

GEOMETRY 1

Work Plane 1 (wp1)

1 In the **Geometry** toolbar, click  **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **yz-plane**.

4 In the **x-coordinate** text field, type `-Lsource-Lgate/2`.

Work Plane 1 (wp1)>Plane Geometry

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

Work Plane 1 (wp1)>Square 1 (sq1)

1 In the **Work Plane** toolbar, click  **Square**.

2 In the **Settings** window for **Square**, locate the **Size** section.

3 In the **Side length** text field, type `Wtot`.

4 Locate the **Position** section. From the **Base** list, choose **Center**.

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

Layer name	Thickness (nm)
`Layer 1`	`Wtot/2`

6 Select the **Layers to the left** check box.

Work Plane 1 (wp1)>Square 2 (sq2)

1 In the **Work Plane** toolbar, click  **Square**.

2 In the **Settings** window for **Square**, locate the **Size** section.

3 In the **Side length** text field, type `Wchannel`.

4 Locate the **Position** section. From the **Base** list, choose **Center**.

Extrude 1: Source

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Extrude**.

2 In the **Settings** window for **Extrude**, type `Extrude 1: Source` in the **Label** text field.

3 Locate the **General** section. From the **Extrude from** list, choose **Faces**.

4 On the object **wp1**, select Boundaries 3, 4, 6, and 7 only.

5 From the **Input object handling** list, choose **Keep**.

6 Locate the **Distances** section. In the table, enter the following settings:

Distances (nm)
`Lsource`

7 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

8 From the **Show in physics** list, choose **All levels**.

9 Find the **Cumulative selection** subsection. Click **New**.

10 In the **New Cumulative Selection** dialog box, type `Si` in the **Name** text field.

11 Click **OK**.

12 In the **Settings** window for **Extrude**, click **Build Selected**.

13 Right-click **Extrude 1: Source** and choose **Duplicate**.

Extrude 2: Channel

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Extrude 1: Source 1 (ext2)**.

2 In the **Settings** window for **Extrude**, type `Extrude 2: Channel` in the **Label** text field.

3 Locate the **General** section. Click the **Clear Selection** button for **Input faces**.

4 On the object **ext1**, select Boundaries 17–20 only.

5 Locate the **Distances** section. Click **Clear Table**.

6 In the table, enter the following settings:

Distances (nm)
`Lgate/2`
`Lgate`

7 Locate the **Selections of Resulting Entities** section. From the **Show in physics** list, choose **Domain selection**.

8 Click **Build Selected**.

9 Right-click **Extrude 2: Channel** and choose **Duplicate**.

Extrude 3: Drain

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Extrude 2: Channel 1 (ext3)**.

2 In the **Settings** window for **Extrude**, type `Extrude 3: Drain` in the **Label** text field.

3 Locate the **General** section. Click the **Clear Selection** button for **Input faces**.

4 On the object **ext2**, select Boundaries 33–36 only.

5 Locate the **Distances** section. Click **Clear Table**.

6 In the table, enter the following settings:

Distances (nm)
`Ldrain`

7 Locate the **Selections of Resulting Entities** section. From the **Show in physics** list, choose **All levels**.

8 Click **Build Selected**.

Extrude 1: Source (ext1)

In the **Model Builder** window, right-click **Extrude 1: Source (ext1)** and choose **Duplicate**.

Extrude 4: Source oxide

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Extrude 1: Source 1 (ext4)**.

2 In the **Settings** window for **Extrude**, type `Extrude 4: Source oxide` in the **Label** text field.

3 Locate the **General** section. Click the **Clear Selection** button for **Input faces**.

4 On the object **wp1**, select Boundaries 1, 2, 5, and 8 only.

5 Locate the **Selections of Resulting Entities** section. From the **Show in physics** list, choose **Off**.

6 Find the **Cumulative selection** subsection. Click **New**.

7 In the **New Cumulative Selection** dialog box, type `Oxide` in the **Name** text field.

8 Click **OK**.

9 In the **Settings** window for **Extrude**, click **Build Selected**.

Extrude 2: Channel (ext2)

In the **Model Builder** window, right-click **Extrude 2: Channel (ext2)** and choose **Duplicate**.

Extrude 5: Gate oxide

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Extrude 2: Channel 1 (ext5)**.

2 In the **Settings** window for **Extrude**, type `Extrude 5: Gate oxide` in the **Label** text field.

3 Locate the **General** section. Click the **Clear Selection** button for **Input faces**.

4 On the object **ext4**, select Boundaries 25–28 only.

5 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Oxide**.

6 Click **Build Selected**.

Extrude 3: Drain (ext3)

In the **Model Builder** window, right-click **Extrude 3: Drain (ext3)** and choose **Duplicate**.

Extrude 6: Drain oxide

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Extrude 3: Drain 1 (ext6)**.

2 In the **Settings** window for **Extrude**, type `Extrude 6: Drain oxide` in the **Label** text field.

3 Locate the **General** section. Click the **Clear Selection** button for **Input faces**.

4 On the object **ext5**, select Boundaries 49–52 only.

5 Locate the **Selections of Resulting Entities** section. From the **Show in physics** list, choose **Off**.

6 Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Oxide**.

7 Click **Build All Objects**.

DEFINITIONS

Adjacent 1

1 In the **Definitions** toolbar, click **Adjacent**.

2 In the **Settings** window for **Adjacent**, locate the **Input Entities** section.

3 Under **Input selections**, click **Add**.

4 In the **Add** dialog box, select **Extrude 5: Gate oxide** in the **Input selections** list.

5 Click **OK**.

6 Right-click **Adjacent 1** and choose **Duplicate**.

Adjacent 2

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

2 In the **Settings** window for **Adjacent**, locate the **Input Entities** section.

3 In the **Input selections** list, select **Extrude 5: Gate oxide**.

4 Under **Input selections**, click **Delete**.

5 Under **Input selections**, click **Add**.

6 In the **Add** dialog box, select **Oxide** in the **Input selections** list.

7 Click **OK**.

Gate

1 In the **Definitions** toolbar, click **Intersection**.

2 In the **Settings** window for **Intersection**, locate the **Geometric Entity Level** section.

3 From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to intersect**, click **Add**.

5 In the **Add** dialog box, select **Adjacent 1** in the **Selections to intersect** list.

6 Click **OK**.

7 In the **Settings** window for **Intersection**, locate the **Input Entities** section.

8 Under **Selections to intersect**, click + **Add**.

9 In the **Add** dialog box, select **Adjacent 2** in the **Selections to intersect** list.

10 Click **OK**.

11 In the **Settings** window for **Intersection**, type `Gate` in the **Label** text field.

Source+Drain

1 In the **Definitions** toolbar, click  **Union**.

2 In the **Settings** window for **Union**, type `Source+Drain` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to add**, click + **Add**.

4 In the **Add** dialog box, select **Extrude 1: Source** in the **Selections to add** list.

5 Click **OK**.

6 In the **Settings** window for **Union**, locate the **Input Entities** section.

7 Under **Selections to add**, click + **Add**.

8 In the **Add** dialog box, select **Extrude 3: Drain** in the **Selections to add** list.

9 Click **OK**.

Box 1: x axis

1 In the **Definitions** toolbar, click  **Box**.

2 In the **Settings** window for **Box**, type `Box 1: x axis` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Edge**.

4 Locate the **Box Limits** section. In the **y minimum** text field, type `-0.1`.

5 In the **y maximum** text field, type `0.1`.

6 In the **z minimum** text field, type `-0.1`.

7 In the **z maximum** text field, type `0.1`.

8 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

9 Right-click **Box 1: x axis** and choose **Duplicate**.

Box 2: z axis

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions>Selections** click **Box 1: x axis 1**.

2 In the **Settings** window for **Box**, type `Box 2: z axis` in the **Label** text field.

3 Locate the **Box Limits** section. In the **x minimum** text field, type `-0.1`.

4 In the **x maximum** text field, type `0.1`.

5 In the **z minimum** text field, type `-inf`.

6 In the **z maximum** text field, type `inf`.

Use the built-in silicon material for the silicon domains.

ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.

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

3 In the tree, select **Semiconductors>Si - Silicon**.

4 Click **Add to Component** in the window toolbar.

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

MATERIALS

Si - Silicon (mat1)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Selection** list, choose **Si**.

GLOBAL DEFINITIONS

Parameters 2: Physics

1 In the **Home** toolbar, click **Parameters** and choose **Add>Parameters**.

Configure physics settings (first enter some physics parameters). Select the density-gradient formulation from the **Discretization** section of the main physics settings window. Specify the anisotropic density-gradient effective mass in the settings window for the **Semiconductor Material Model** domain feature.

2 In the **Settings** window for **Parameters**, type `Parameters 2: Physics` in the **Label** text field.

3 Locate the **Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
T0	300[K]	300 K	Temperature
fmx	0.8	0.8	Longitudinal DG effective mass
fmy	0.12	0.12	Transverse DG effective mass

Name	Expression	Value	Description
mx	fmx*me_const	7.2875E-31 kg	Longitudinal DG effective mass
my	fmy*me_const	1.0931E-31 kg	Transverse DG effective mass
mz	my	1.0931E-31 kg	Transverse DG effective mass
epsrOx	3.9	3.9	Oxide dielectric constant
mox	0.5*me_const	4.5547E-31 kg	Oxide DG effective mass
moxstar	0.22*me_const	2.0041E-31 kg	Oxide DG effective mass
PhiBox	3.15[V]	3.15 V	Oxide potential barrier
Nd	1e20[cm^-3]	1E26 1/m³	Doping
cp	1	1	Continuation parameter for doping
Vd	0.05[V]	0.05 V	Drain voltage
Phig	4.5[V]	4.5 V	Gate metal work function
Vg	0.8[V]	0.8 V	Gate voltage

SEMICONDUCTOR (SEMI)

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

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

3 From the **Solution** list, choose **Majority carriers only**.

4 Click to expand the **Discretization** section. From the **Formulation** list, choose **Finite element density-gradient (quadratic shape function)**.

Semiconductor Material Model 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Semiconductor (semi)** click **Semiconductor Material Model 1**.

2 In the **Settings** window for **Semiconductor Material Model**, locate the **Model Input** section.

3 In the T text field, type `T0`.

4 Locate the **Material Properties, Density-Gradient** section. From the list, choose **Diagonal**.

5 In the $\mathbf{m}_e^{DG}$ table, enter the following settings:

mx	0	0
0	my	0
0	0	mz

Apply the **Charge Conservation** domain condition to model the oxide layer explicitly. Use the **Potential barrier** option to include quantum confinement effects at the silicon-oxide interfaces.

Charge Conservation 1

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

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

3 From the **Selection** list, choose **Oxide**.

4 Locate the **Electric Field** section. From the ε_r list, choose **User defined**. In the associated text field, type `epsrOx`.

Insulator Interface 1

1 In the **Model Builder** window, click **Insulator Interface 1**.

2 In the **Settings** window for **Insulator Interface**, locate the **Density-Gradient** section.

3 From the **Formulation** list, choose **Potential barrier**.

4 In the $\mathbf{m}_e^{Ox}$ text field, type `mox`.

5 In the $\mathbf{m}_e^{Ox*}$ text field, type `moxstar`.

6 In the Φ_n^{Ox} text field, type `PhiBox`.

Add doping and metal contacts.

Analytic Doping Model 1

1 In the **Physics** toolbar, click **Domains** and choose **Analytic Doping Model**.

2 In the **Settings** window for **Analytic Doping Model**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Source+Drain**.

4 Locate the **Impurity** section. From the **Impurity type** list, choose **Donor doping (n-type)**.

5 In the N_{D0} text field, type `Nd*cp`.

Metal Contact 1: Source

1 In the **Physics** toolbar, click **Boundaries** and choose **Metal Contact**.

2 In the **Settings** window for **Metal Contact**, type `Metal Contact 1: Source` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Extrude 1: Source**.

Metal Contact 2: Drain

1 In the **Physics** toolbar, click **Boundaries** and choose **Metal Contact**.

2 In the **Settings** window for **Metal Contact**, type `Metal Contact 2: Drain` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Extrude 3: Drain**.

4 Locate the **Terminal** section. In the V_0 text field, type `Vd`.

Terminal 1: Gate

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

2 In the **Settings** window for **Terminal**, type `Terminal 1: Gate` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Gate**.

4 Locate the **Terminal** section. In the Φ_e text field, type `Phig`.

5 From the **Terminal type** list, choose **Voltage**.

6 In the V_0 text field, type `Vg`.

Create the mesh. Use the **Distribution** feature to arrange an efficient **Swept** mesh that minimizes the number of mesh elements while still resolving large gradients in the solution fields. To reduce the computation time and file size, a relatively coarse mesh is used in this tutorial. Interested users are encouraged to perform mesh refinement studies by parameterizing the settings of the **Distribution** nodes.

MESH 1

Mapped 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Mapped**.

2 Select Boundaries 1, 4, 8, 11, 15, 18, 21, and 24 only.

3 In the **Settings** window for **Mapped**, click to expand the **Reduce Element Skewness** section.

4 Select the **Adjust edge mesh** check box.

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Edges 9, 20, 21, 24, 27, and 31 only.

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

4 From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type `3`.

6 In the **Element ratio** text field, type 2.

7 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 **Edge Selection** section.

3 Click  **Clear Selection**.

4 Select Edges 10, 12, 13, 15, 23, and 33 only.

5 Locate the **Distribution** section. Select the **Reverse direction** check box.

Distribution 3

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

2 Select Edges 5, 17, 26, and 34 only.

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

4 In the **Number of elements** text field, type 1.

Swept 1

In the **Mesh** toolbar, click  **Swept**.

Distribution 1

1 Right-click **Swept 1** and choose **Distribution**.

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

3 From the **Selection** list, choose **Extrude 2: Channel**.

4 Locate the **Distribution** section. In the **Number of elements** text field, type 3.

Distribution 2

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

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

3 From the **Selection** list, choose **Source+Drain**.

4 Locate the **Distribution** section. From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type 10.

6 In the **Element ratio** text field, type 3.

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

8 Click  **Build All**.

The **Semiconductor Equilibrium** study step provides a good initial condition for subsequent gate voltage sweeps. Use the **Auxiliary sweep** feature to ramp up the doping concentrations

from small values in order to achieve convergence. Add a **Stationary** study step to sweep the gate voltage for a set of longitudinal DG effective mass values.

STUDY 1

Step 1: Semiconductor Equilibrium

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

2 In the **Settings** window for **Semiconductor Equilibrium**, click to expand 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
cp (Continuation parameter for doping)	`10^range(-12,4,0)`	

6 In the table, click to select the cell at row number 1 and column number 3.

Step 2: Stationary

1 In the **Study** toolbar, click **Study Steps** and choose **Stationary>Stationary**.

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

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

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

5 Click + **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
fmx (Longitudinal DG effective mass)	`0.8 0.5 0.3 0.2 0.1`	

7 In the table, click to select the cell at row number 1 and column number 3.

8 Click + **Add**.

9 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Vg (Gate voltage)	`range(0.8,-0.1,0.4) 0.295 range(0.2,-0.1,-0.01)`	V

10 In the table, click to select the cell at row number 2 and column number 3.

11 From the **Reuse solution from previous step** list, choose **Auto**.

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

Plot the Id-Vg curve to be compared with Fig. 2 in the reference paper.

RESULTS

Id-Vg

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

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

3 Locate the **Data** section. From the **Parameter selection (fmx)** list, choose **Manual**.

4 In the **Parameter indices (1-5)** text field, type `range(1,4)`.

5 Locate the **Axis** section. Select the **Manual axis limits** check box.

6 In the **x minimum** text field, type `0`.

7 In the **x maximum** text field, type `0.8`.

8 In the **y minimum** text field, type `1e-10`.

9 In the **y maximum** text field, type `1e-4`.

10 Select the **y-axis log scale** check box.

11 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

Global 1

1 Right-click **Id-Vg** and choose **Global**.

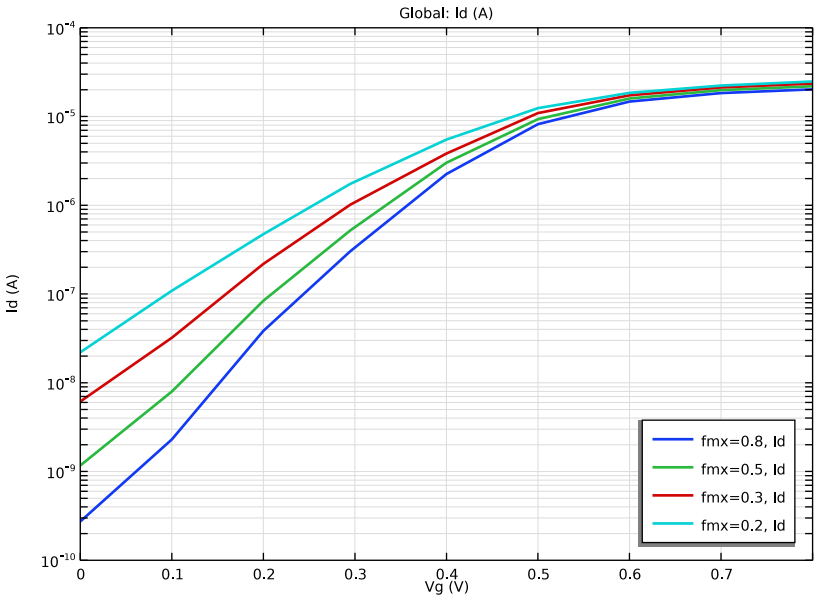
2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`semi.IO_2`	`A`	`Id`

4 Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.

5 In the **Id-Vg** toolbar, click **Plot**.



Plot the electron concentration along the longitudinal centerline to be compared with Fig. 4 in the reference paper.

n(x)

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

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

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

4 In the **Parameter values (Vg (V))** list, select **0.295**.

5 Locate the **Axis** section. Select the **Manual axis limits** check box.

6 In the **x minimum** text field, type `-10`.

7 In the **x maximum** text field, type `10`.

8 In the **y minimum** text field, type `1e17`.

9 In the **y maximum** text field, type `1e21`.

10 Select the **y-axis log scale** check box.

11 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

Line Graph 1

1 Right-click **n(x)** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **Selection** section.

3 From the **Selection** list, choose **Box 1: x axis**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `semi.N`.

5 In the **Unit** field, type `1/cm^3`.

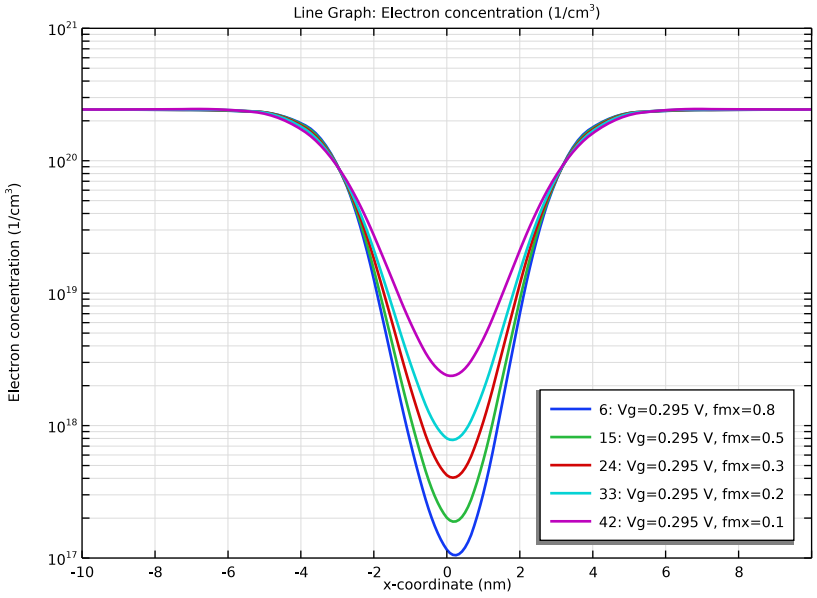
6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

7 In the **Expression** text field, type `x`.

8 Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.

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

10 In the **n(x)** toolbar, click **Plot**.



Plot the electron concentration along the transverse centerline to be compared with Fig. 4 in the reference paper.

n(x)

In the **Model Builder** window, right-click **n(x)** and choose **Duplicate**.

n(z)

1 In the **Model Builder** window, expand the **Results>n(x) 1** node, then click **n(x) 1**.

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

3 Locate the **Axis** section. In the **x minimum** text field, type `-1.6`.

4 In the **x maximum** text field, type `1.6`.

5 In the **y minimum** text field, type `4e15`.

6 In the **y maximum** text field, type `4e18`.

7 Locate the **Legend** section. From the **Position** list, choose **Lower middle**.

Line Graph 1

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

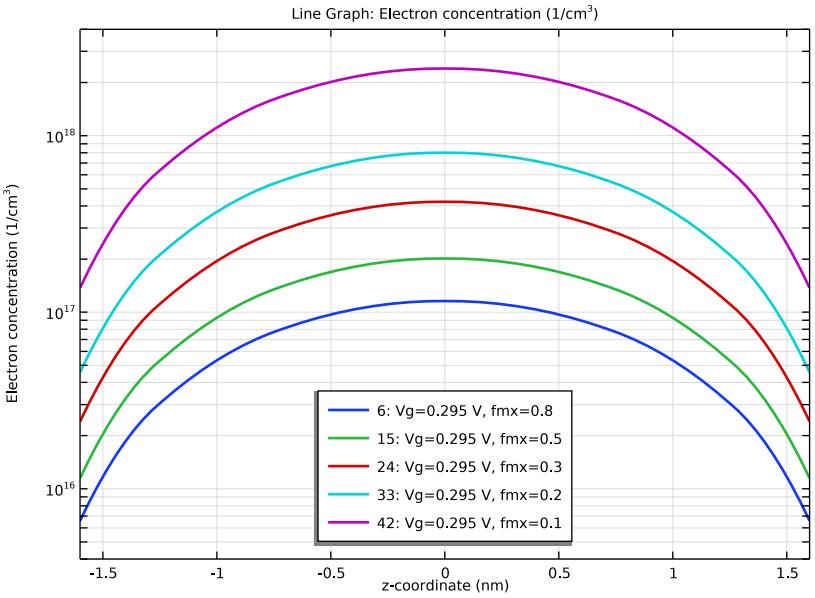
2 In the **Settings** window for **Line Graph**, locate the **Selection** section.

3 Click **Clear Selection**.

4 From the **Selection** list, choose **Box 2: z axis**.

5 Locate the **x-Axis Data** section. In the **Expression** text field, type `z`.

6 In the **n(z)** toolbar, click **Plot**.



Create a fancy 3D plot for the model thumbnail.

Electron Concentration (semi)

1 In the **Model Builder** window, under **Results** click **Electron Concentration (semi)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Parameter value (fmx)** list, choose **0.8**.

4 From the **Parameter value (Vg (V))** list, choose **0.5**.

5 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Volume 1

1 In the **Model Builder** window, expand the **Electron Concentration (semi)** node.

2 Right-click **Volume 1** and choose **Disable**.

Slice 1

1 In the **Model Builder** window, right-click **Electron Concentration (semi)** and choose **Slice**.

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

3 In the **Expression** text field, type `semi.log10N`.

4 Locate the **Plane Data** section. From the **Plane** list, choose **ZX-planes**.

5 In the **Planes** text field, type `1`.

6 Right-click **Slice 1** and choose **Duplicate**.

Slice 2

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

2 In the **Settings** window for **Slice**, locate the **Plane Data** section.

3 From the **Plane** list, choose **YZ-planes**.

4 In the **Planes** text field, type `7`.

5 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Slice 1**.

Arrow Volume 1

1 In the **Model Builder** window, right-click **Electron Concentration (semi)** and choose **Arrow Volume**.

2 In the **Settings** window for **Arrow Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Semiconductor>Currents and charge>semi.JX,...,semi.JZ - Total current density, nodal value**.

3 Locate the **Arrow Positioning** section. Find the **Y grid points** subsection. In the **Points** text field, type `5`.

4 Find the **Z grid points** subsection. In the **Points** text field, type `19`.

5 Locate the **Coloring and Style** section. From the **Color** list, choose **Black**.

Isosurface 1

1 Right-click **Electron Concentration (semi)** and choose **Isosurface**.

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

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

4 Locate the **Levels** section. In the **Total levels** text field, type `10`.

5 Locate the **Coloring and Style** section. Click **Change Color Table**.

6 In the **Color Table** dialog box, select **Linear>GrayScale** in the tree.

7 Click **OK**.

8 In the **Settings** window for **Isosurface**, locate the **Coloring and Style** section.

9 Clear the **Color legend** check box.

Filter 1

1 Right-click **Isosurface 1** and choose **Filter**.

2 In the **Settings** window for **Filter**, locate the **Element Selection** section.

3 In the **Logical expression for inclusion** text field, type `-5e-11<y`.

4 In the **Electron Concentration (semi)** toolbar, click **Plot**.