

# INTRODUCTION TO Semiconductor Module

# Introduction to the Semiconductor Module

© 1998–2017 COMSOL

Protected by patents listed on [www.comsol.com/patents](http://www.comsol.com/patents), and U.S. Patents 7,519,518; 7,596,474; 7,623,991; 8,457,932; 8,954,302; 9,098,106; 9,146,652; 9,323,503; 9,372,673; and 9,454,625. Patents pending.

This Documentation and the Programs described herein are furnished under the COMSOL Software License Agreement ([www.comsol.com/comsol-license-agreement](http://www.comsol.com/comsol-license-agreement)) and may be used or copied only under the terms of the license agreement.

COMSOL, the COMSOL logo, COMSOL Multiphysics, COMSOL Desktop, COMSOL Server, and LiveLink are either registered trademarks or trademarks of COMSOL AB. All other trademarks are the property of their respective owners, and COMSOL AB and its subsidiaries and products are not affiliated with, endorsed by, sponsored by, or supported by those trademark owners. For a list of such trademark owners, see [www.comsol.com/trademarks](http://www.comsol.com/trademarks).

Version: COMSOL 5.3a

## Contact Information

Visit the Contact COMSOL page at [www.comsol.com/contact](http://www.comsol.com/contact) to submit general inquiries, contact Technical Support, or search for an address and phone number. You can also visit the Worldwide Sales Offices page at [www.comsol.com/contact/offices](http://www.comsol.com/contact/offices) for address and contact information.

If you need to contact Support, an online request form is located at the COMSOL Access page at [www.comsol.com/support/case](http://www.comsol.com/support/case). Other useful links include:

- Support Center: [www.comsol.com/support](http://www.comsol.com/support)
- Product Download: [www.comsol.com/product-download](http://www.comsol.com/product-download)
- Product Updates: [www.comsol.com/support/updates](http://www.comsol.com/support/updates)
- COMSOL Blog: [www.comsol.com/blogs](http://www.comsol.com/blogs)
- Discussion Forum: [www.comsol.com/community](http://www.comsol.com/community)
- Events: [www.comsol.com/events](http://www.comsol.com/events)
- COMSOL Video Gallery: [www.comsol.com/video](http://www.comsol.com/video)
- Support Knowledge Base: [www.comsol.com/support/knowledgebase](http://www.comsol.com/support/knowledgebase)

Part number: CM024103

# Contents

---

- Introduction .....5
- Semiconductor Devices .....7
- The Semiconductor Module Physics Interface Guide ..... I I
  - Physics Interface Guide by Space Dimension and Preset Study Type ..... I5
- Tutorial Example: DC Characteristics of a MOSFET..... I6



# Introduction

---

Device engineers and physicists use the Semiconductor Module to design and understand semiconductor devices. For many years semiconductor device design has been closely associated with the use of simulation tools, due to the high cost of prototyping new devices and processes. The advance of nanotechnology and organic semiconductors has helped create many novel devices. Researchers in these fields also use simulation to assist the fundamental understanding and design optimization. For all of the systems mentioned above, multiphysics effects often play an important role and COMSOL is the ideal platform for investigating these effects.

The Semiconductor Module enables the stationary and dynamic performance of devices to be modeled in one, two and three dimensions, together with circuit-based modeling of active and passive devices. In the frequency domain, it is possible to model devices driven by a combination of AC and DC signals. A predefined Semiconductor interface can be used to model a broad range of semiconductor devices and can be straightforwardly coupled with other physics interfaces to model phenomena such as heat generation and optoelectronic effects. A predefined Schrödinger Equation interface allows the modeling of quantum-confined systems such as quantum wells, wires, and dots.

The Semiconductor interface solves the Semiconductor equations by either the finite volume or the finite element method. The physics interface solves a set of coupled partial differential equations for the electric potential and for the electron and hole concentrations (or their logarithm in the case of the finite element method log formulation, or their quasi-Fermi levels in the case of the quasi-Fermi level formulation). The corresponding initial and boundary conditions are easily specified in the physics interface.

COMSOL's design emphasizes the physics by providing users with the equations solved by each feature and by offering full access to the underlying equation system. The Semiconductor Module User's Guide provides complete information on the theory underlying the Semiconductor interface. There is also tremendous flexibility to add user-defined equations and expressions to the system. For example, user-defined mobility models can be readily specified simply by typing appropriate expressions into the user defined feature, no scripting or coding is required. These user-defined models can be combined arbitrarily with the predefined models built into the software. When COMSOL Multiphysics compiles the equations, the complex couplings generated by these user-defined expressions are automatically included in the equation system. The equations are then solved using a range of state of the art solvers.

Once a solution is obtained, a large range of result analysis tools are available to interrogate the data, and predefined plots are automatically generated to show the device response. COMSOL Multiphysics offers the flexibility to evaluate a wide range of physical quantities including predefined quantities such as the electron and hole currents (including current components from drift, diffusion and thermal diffusion), the electric field, the generation/recombination rate, and the temperature, all available through easy-to-use menus, as well as arbitrary user-defined expressions.

To model a Semiconductor device, the geometry is first defined in the software. Then appropriate materials are selected and the Semiconductor interface is added. The dopant distribution can be computed separately using a diffusion equation calculation, imported from third party software or specified empirically using the built in doping features. Initial conditions and boundary conditions are set up within the physics interface. Next, the mesh is defined and a solver is selected. Finally the results are visualized using a wide range of post-processing tools. All of these steps are accessed from the COMSOL Desktop graphical user interface.

The Schrödinger Equation interface solves the Schrödinger equation for the wave function of a single particle in an external potential. This can be applied to general quantum mechanical problems, as well as for the electron and hole wave functions in quantum-confined systems under the envelope function approximation.

Appropriate boundary conditions and study types are implemented for the user to easily set up models to compute relevant quantities in various situations, such as the eigenenergies of bound states, the decay rates of quasi bound states, the transmission and reflection coefficients, the resonant tunneling condition, and the effective band gap of a superlattice structure. The Double Barrier 1D model and the Superlattice Band Gap Tool app in the Application Libraries illustrate the usage of the various built-in functionalities.

## Semiconductor Devices

The semiconductor module can be applied to solve a range of device simulation problems. The Semiconductor interface can be straightforwardly coupled with other physics interfaces, such as the Electromagnetic waves interfaces (using the predefined Semiconductor Optoelectronics multiphysics coupling), the Heat Transfer in Solids interface and the Electrical Circuits interface. Coupling to a circuit is straightforward using the terminals included with all the boundary conditions. Figure 1 shows results obtained from a 2D pn junction model in which a device model of a diode is coupled to an electrical circuit to produce a rectifier. The electron and hole concentrations are shown in the plots when different voltages are applied to the circuit.

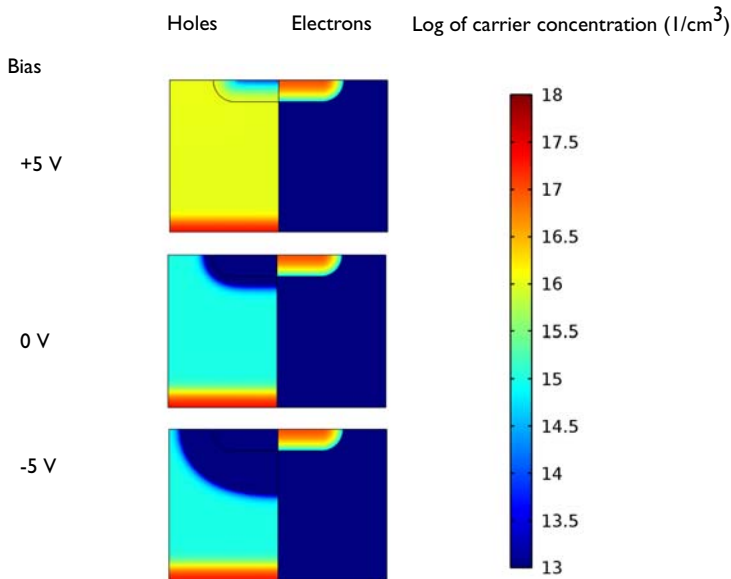


Figure 1: Electron and hole concentrations in a pn-junction diode connected to a series resistor under different bias conditions. This plot shows clearly the changing geometry and extent of the depletion region under reverse bias.

A range of common device types can be simulated with the module, including MOSFETs, MESFETs, JFETs, diodes, and bipolar transistors. These devices can be analyzed for the stationary case, in the time domain or in the frequency domain (with mixed DC and AC signals, using the small signal analysis study

type). A number of standard analyses are illustrated with the MOSFET model series. The first model in the MOSFET series is described in the section [Tutorial Example: DC Characteristics of a MOSFET](#), below. Figure 2 shows some of the results obtained from this analysis.

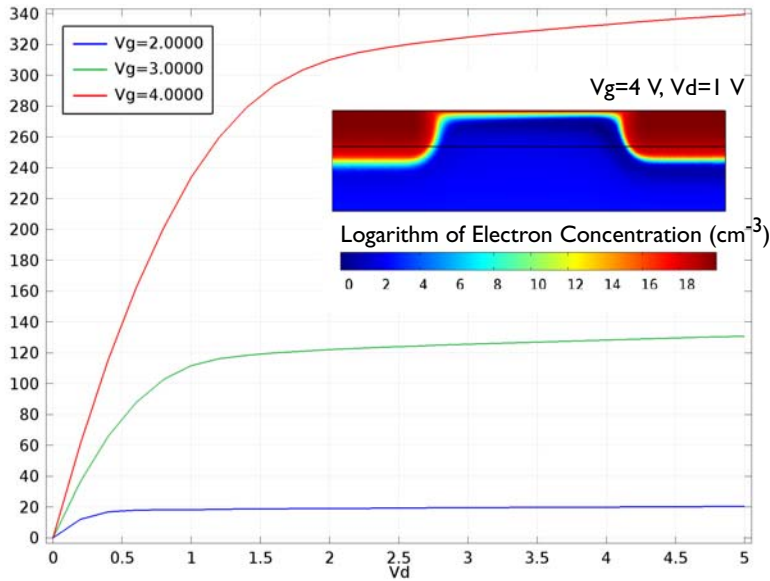


Figure 2: Stationary analysis of a MOSFET. The plot shows the drain current plotted against the drain voltage ( $V_d$ ) for a range of different values of the gate voltage ( $V_g$ ). The inset shows the logarithm of the electron concentration (in units of  $\text{cm}^{-3}$ ) for a gate voltage of 4 V and for a drain voltage of 1 V. The channel is clearly visible.

In addition to the MOSFET model series, which shows how to include a range of increasingly complicated semiconductor physics effects using features included within the Semiconductor interface, a wide array of other example models are available. The Bipolar Transistor model sequence gives an example of how to set up a multiphysics device simulation. Initially, a model of a 2D cross section of a bipolar transistor device is created using only the Semiconductor interface. This model is then extended by two other models in the sequence, one adds coupling to the Heat Transfer interface and the other demonstrates how to create a full 3D simulation of the same device. The ISFET model demonstrates the coupling to the electrochemistry physics. Several individual models are also provided, including a MESFET, an EEPROM device, and some LED models. Figure 3 shows the electron and hole currents flowing in a simple 2D Bipolar transistor.

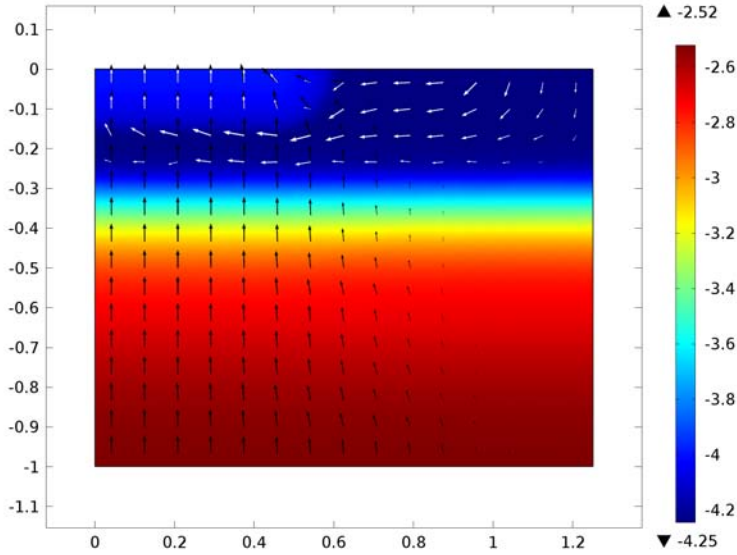
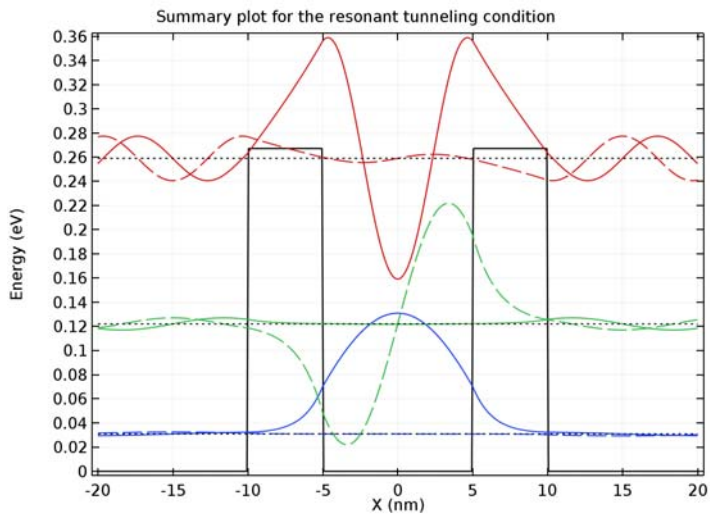


Figure 3: Log of the current density in  $A/cm^2$  (color) and direction of the current flow for electrons (black arrows) and holes (white arrows) in a simple 2D bipolar transistor.

The Semiconductor interface includes a wide range of predefined variables that can be used to access various current components, such as the electron and hole currents as well as the individual contributions to the current from the mobility/ electric field, from diffusion and from thermal diffusion.

The Schrödinger Equation interface enables the modeling of various quantum-confined systems. The following graph shows the wave functions shifted by the respective energy levels for the resonant tunneling conditions of a double barrier structure.



# The Semiconductor Module Physics Interface Guide

---

Each COMSOL Multiphysics physics interface (for example the Semiconductor interface, or the Schrödinger Equation interface) expresses the relevant physical phenomena in the form of sets of partial or ordinary differential equations, together with appropriate boundary and initial conditions. Each feature added to the physics interface represents a term or condition in the underlying equation set. These features are usually associated with a geometric entity within the model, such as a domain, boundary, edge, or point. [Figure 4](#) uses the MOSFET application library example to show the Model Builder and the Settings window for the selected Semiconductor Material Model 1 feature node. This node adds the semiconductor equations to the simulation within the domains selected. In the Model Inputs section the temperature of the material is specified. It is straightforward to link this temperature to a separate Heat Transfer interface to solve non-isothermal problems — the Semiconductor interface automatically defines an appropriate heat source term that can be readily accessed in a Heat Transfer interface. In the Material Properties section the Settings window indicate that the relative permittivity and the band gap are inherited from the material properties assigned to the domain. The material properties can be set up as functions of other dependent variables in the model, for example, the temperature. The dopant density is specified by means of multiple additive doping features, which can be used to combine Gaussian and user defined dopant density profiles to produce the desired profile. Several boundary conditions are also indicated in the model tree. The Ohmic Contact boundary condition is commonly used to model non-rectifying interconnects. The Thin Insulating Gate feature models a gate with a thickness smaller than the typical length scale of the mesh. It is also possible to model gates explicitly, solving Poisson's equation within the dielectric.

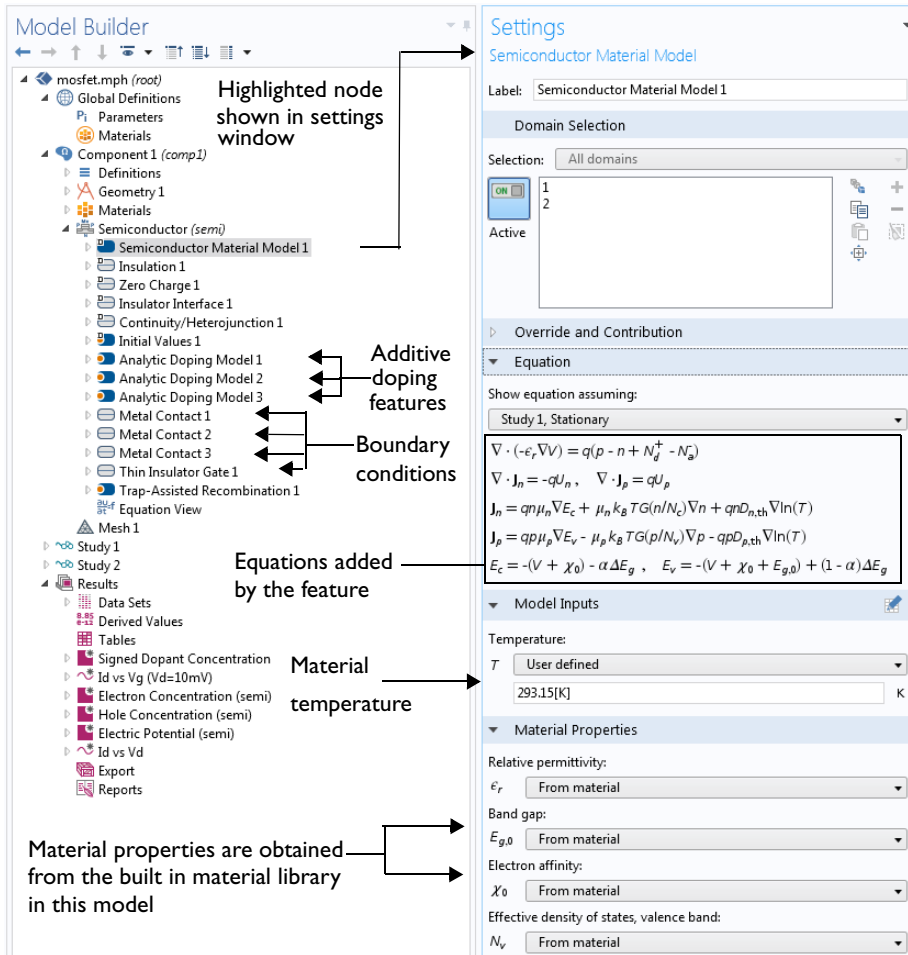


Figure 4: The Model Builder (to the left), and the Settings window for Semiconductor Material Model 1 for the selected feature node (to the right). The Equation section in the Settings window shows the model equations.

The Semiconductor interface is the starting point for most simulations. The Semiconductor Module also includes physics interfaces to enable modeling of different physical situations encountered in device design. When a new model is started, these physics interfaces are selected from the Model Wizard.

Figure 5 shows the physics interfaces included with the Semiconductor Module. The two Semiconductor Optoelectronics interfaces are only available with an additional Wave Optics module license.

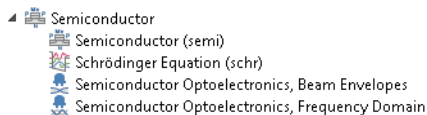




Figure 5: The Semiconductor Module interfaces as displayed in the Model Wizard for a 3D model. The Semiconductor Optoelectronics interfaces are only available with an additional Wave Optics Module license.

Also see [Physics Interface Guide by Space Dimension and Preset Study Type](#). Below, a brief overview of each of the Semiconductor Module physics interfaces is given.


## ELECTROSTATICS

The Electrostatics interface (  ), found under the AC/DC branch in the Model Wizard, solves for the electric potential given the charge distribution in the domain and the voltages applied to boundaries. It is used to model electrostatic devices under static or quasi-static conditions, that is at frequencies sufficiently low that wave propagation effects can be neglected. Many of the features of the Electrostatics interface are included in the Semiconductor interface, where they affect the solution of the electric potential.


## ELECTRICAL CIRCUIT

The Electrical Circuit interface (  ), found under the AC/DC branch in the Model Wizard, has the equations to model electrical circuits with or without connections to a distributed fields model. The interface solves for the voltages, currents, and charges associated with the circuit elements. Circuit models can contain passive elements like resistors, capacitors, and inductors as well as active elements such as diodes and transistors. Circuits can be imported from an existing SPICE net list. A typical application of this interface would be to consider the effect of series or parallel lumped components on device behavior.

## SEMICONDUCTOR


The Semiconductor interface (  ), found under the Semiconductor branch in the Model Wizard, solves the Semiconductor equations including the equations of electrostatics. The physics interface allows both insulating and semiconducting domains to be modeled. The equations account fully for thermal effects, and the interface can be coupled to a heat transfer interface using the temperature model input and the predefined heat source term. This physics interface is appropriate for modeling semiconductor devices.

## SEMICONDUCTOR OPTOELECTRONICS, BEAM ENVELOPES

The Semiconductor Optoelectronics, Beam Envelopes (  ) interface combines the Semiconductor interface with the Electromagnetic Waves, Beam Envelopes interface. The coupling occurs through the Optical Transitions feature, which adds a stimulated emission generation term (appropriate for direct band-gap materials) on domains in the Semiconductor interface. This term is proportional to the optical intensity in the corresponding Wave Equation, Beam Envelopes feature in the Electromagnetic Waves, Beam Envelopes interface. Additionally spontaneous emission (for direct band-gap materials) is accounted for. The effect of the light adsorption or emission is accounted for by a corresponding change in the complex permittivity or refractive index in the Wave Equation, Beam Envelopes feature.


The interface is used for modeling devices such as photodiodes, light emitting diodes and laser diodes without quantum wells in direct band gap materials.

## SEMICONDUCTOR OPTOELECTRONICS, FREQUENCY DOMAIN

The Semiconductor Optoelectronics, Frequency Domain (  ) interface combines the Semiconductor interface with the Electromagnetic Waves, Frequency Domain interface. The coupling occurs through the Optical Transitions feature, which adds a stimulated emission generation term (appropriate for direct band-gap materials) on domains in the Semiconductor interface. This term is proportional to the optical intensity in the corresponding Wave Equation, Electric feature in the Electromagnetic Waves, Frequency Domain interface. Additionally spontaneous emission (for direct band-gap materials) is accounted for. The effect of the light adsorption or emission is accounted for by a corresponding change in the complex permittivity or refractive index in the Wave Equation, Electric feature.









The interface is used for modeling devices such as photodiodes, light emitting diodes and laser diodes without quantum wells in direct band gap materials.

## SCHRÖDINGER EQUATION

The Schrödinger Equation interface (  ), found under the Semiconductor branch in the Model Wizard, solves the Schrödinger equation for a single particle in an external potential. This physics interface is useful for general quantum mechanical problems as well as for quantum confined systems such as quantum wells, wires, and dots.

## Physics Interface Guide by Space Dimension and Preset Study Type

The table below lists the physics interfaces available specifically with this module in addition to the COMSOL Multiphysics basic license.

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
 <b>AC/DC</b>				
Electrical Circuit		cir	Not space dependent	stationary; frequency domain; time dependent; small signal analysis, frequency domain
Electrostatics <sup>1</sup>		es	all dimensions	stationary; time dependent; stationary source sweep; eigenfrequency; frequency domain; small signal analysis, frequency domain
 <b>Semiconductor</b>				
Semiconductor		semi	all dimensions	small-signal analysis, frequency domain; stationary; time dependent; semiconductor equilibrium
Semiconductor Optoelectronics, Beam Envelopes <sup>2</sup>		—	3D, 2D, and 2D axisymmetric	frequency-stationary; frequency-transient; small-signal analysis, frequency domain
Semiconductor Optoelectronics, Frequency Domain <sup>2</sup>		—	3D, 2D, and 2D axisymmetric	frequency-stationary; frequency-transient; small-signal analysis, frequency domain
Schrödinger Equation		schr	all dimensions	eigenvalue; stationary; time dependent
<sup>1</sup> This physics interface is included with the core COMSOL package but has added functionality for this module.				
<sup>2</sup> Requires both the Wave Optics Module and the Semiconductor Module.				

## Tutorial Example: DC Characteristics of a MOSFET

---

This tutorial calculates the DC characteristics of a MOS (metal-oxide semiconductor) transistor. The MOSFET (metal oxide semiconductor field-effect transistor) is by far the most common semiconductor device, and the primary building block in all commercial processors, memories, and digital integrated circuits. Since the first microprocessors were introduced approximately 40 years ago this device has experienced tremendous development, and today it is being manufactured with feature sizes of 22 nm and smaller.

The MOSFET is essentially a miniaturized switch. In this example the source and drain contacts (the input and output of the switch) are both Ohmic (low resistance) contacts to heavily doped n-type regions of the device. Between these two contacts is a region of p-type semiconductor. The gate contact lies above the p-type semiconductor, slightly overlapping the two n-type regions. It is separated from the semiconductor by a thin layer of Silicon oxide, so that it forms a capacitor with the underlying semiconductor. Applying a voltage to the gate changes the local band structure beneath it through the Field Effect. A sufficiently high voltage can cause the semiconductor to change from p-type to n-type in a thin layer (the channel) underneath the gate. This is known as inversion and the channel is sometimes referred to as the inversion layer. The channel connects the two n-type regions of semiconductor with a thin n-type region under the gate. This region has a significantly lower resistance than the series resistance of the np/pn junctions that separated the source and the drain before the gate voltage produced the inversion layer. Consequently applying a gate voltage can be used to change the resistance of the device. The gate voltage where a significant current begins to flow is called the threshold or turn-on voltage. Figure 6 shows a schematic MOSFET with the main electrical connections highlighted. Figure 7 shows an electron microscope image of a modern MOSFET device.

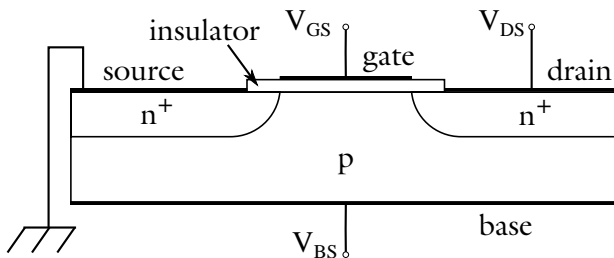


Figure 6: Schematic diagram of a typical MOSFET. The current flows from the source to the drain through a channel underneath the gate. The size of the channel is controlled by the gate voltage.

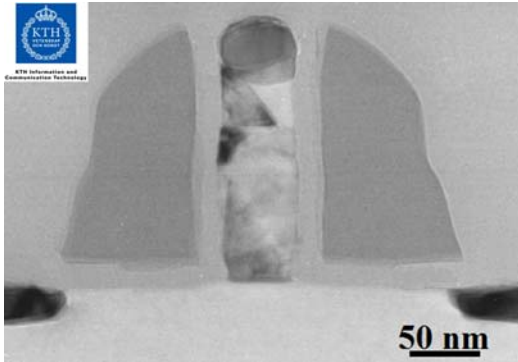


Figure 7: Cross-section TEM (transmission electron microscopy) image of a 50 nm gate length MOSFET fabricated at KTH Electrum laboratory by P.E. Hellström and co-workers within the ERC advanced grant OSIRIS research project headed by Prof. M. Östling.

As the voltage between the drain and the source is increased the current carried by the channel eventually saturates through a process known as pinch-off, in which the channel narrows at one end due to the effect of the field parallel to the surface. The channel width is controlled by the gate voltage. Typically a larger gate voltage results in wider channel and consequently a lower resistance for a given drain voltage. Additionally the saturation current is larger for a higher gate voltage.

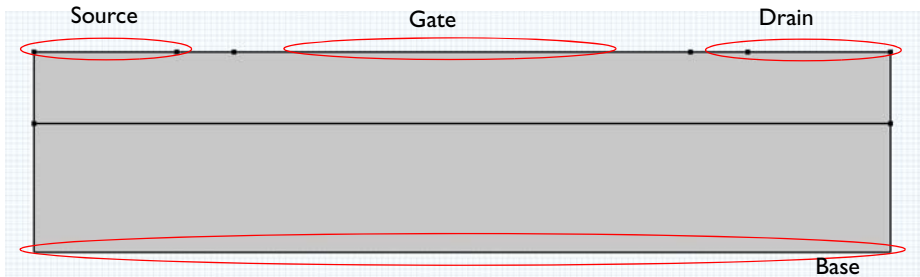


Figure 8: Model geometry showing the external connections.

Figure 8 shows the model geometry, indicating how the geometry elements correspond to features in Figure 6. In this model both the source and the base are connected to ground and the voltages applied to the drain and the gate are varied. In the first study a small voltage (50 mV) is applied to the Drain and the Gate voltage is swept from 0 to 5 V. A plot of the current flowing between the source and the drain is used to determine the turn-on voltage of the device. The second study sweeps the drain voltage from 0 to 5 V at three different values of the gate



voltage (2, 3 and 4 V). The drain current versus drain voltage is then plotted at several values of the gate voltage.

## Model Wizard






---

**Note:** These instructions are for the user interface on Windows but apply, with minor differences, also to Linux and Mac.

- 1 To start the software, double-click the COMSOL icon on the desktop. When the software opens, you can choose to use the Model Wizard to create a new COMSOL model or Blank Model to create one manually. For this tutorial, click the Model Wizard button.

If COMSOL is already open, you can start the Model Wizard by selecting New  from the File menu and then click Model Wizard .

The Model Wizard guides you through the first steps of setting up a model. The next window lets you select the dimension of the modeling space.

- 2 In the Space Dimension window, click the 2D button .
- 3 In the Select physics tree under Semiconductor, click Semiconductor (semi) .
- 4 Click Add. Click the Study button .
- 5 In the tree under Preset Studies, click Stationary .
- 6 Click Done .

## Global Definitions

---

### Parameters

- 1 On the Home toolbar click Parameters .

**Note:** On Linux and Mac, the Home toolbar refers to the specific set of controls near the top of the Desktop.


- 2 In the Settings window locate the Parameters section. In the table, enter the following settings:

NAME	EXPRESSION	DESCRIPTION
Vd	10[mV]	Drain Voltage
Vg	2[V]	Gate Voltage

## Geometry I



---

The geometry can be specified using COMSOL's built in tools. First choose to define geometry objects using micrometer units.

- 1 In the Model Builder under Component 1 click Geometry 1 .
- 2 In the Settings window for Geometry locate the Units section. From the Length unit list, choose  $\mu\text{m}$ .



### *Rectangle I*

Next create a rectangle to define the geometry extents.





- 1 In the Model Builder, right click Geometry 1  and choose Rectangle .
- 2 In the Settings window for Rectangle locate the Size and Shape section. In the Width text field, type 3. In the Height text field, type 0.7.

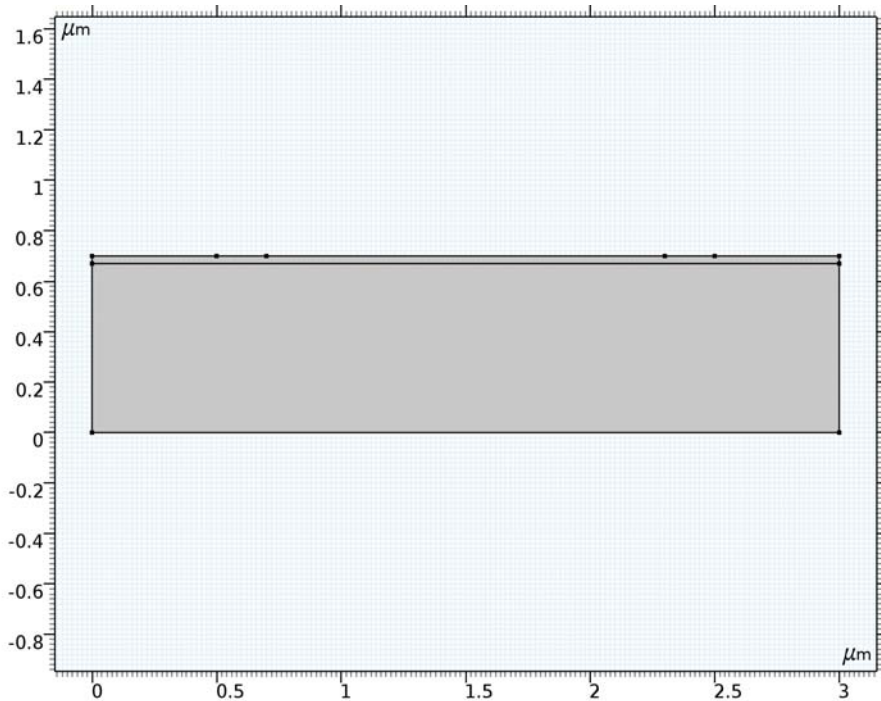
### *Polygon I*

Add a polygon that includes points to define the source, drain and gate contacts. It will also include a line to help create the mesh.

- 1 Right click Geometry 1  and choose Polygon .
- 2 In the Settings window for Polygon, locate the Object Type section. From the Type list, choose Closed curve.
- 3 Locate the Coordinates section.
  - In the x text field, type 0 0 0.5 0.7 2.3 2.5 3 3.
  - In the y text field, type 0.67 0.7 0.7 0.7 0.7 0.7 0.7 0.67.

### *Polygon I*

- 1 On the Geometry toolbar, click Virtual Operations  and choose Mesh Control Edges .
- 2 In the Settings window for Mesh Control Edges, locate the Input section. For the “Edges to include:” input panel, select Boundary 4 only. This tells the software that Boundary 4 is used for mesh generation only, and is ignored by the physics settings.
- 3 Click the Build All Objects button  to build the entire geometry. Use the Zoom Extents button  at the top of the Graphics window to zoom to the full geometry if desired.






## Materials

---

### Add Material


Next the material properties are added to the model.

- 1 On the Home toolbar click Add Material .
- 2 Go to the Add Material window. In the tree under Semiconductors, click Si .
- 3 Click  Add to Component.

## Semiconductor

---


Next define the physics settings. Start with the carrier statistics and doping.

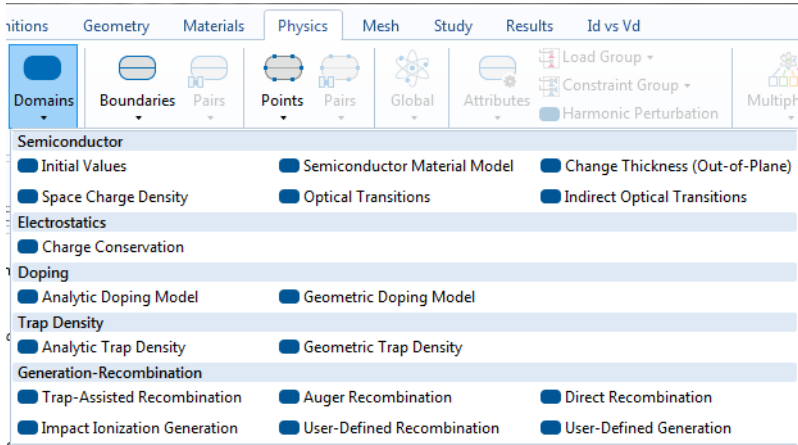
- 1 In the Model Builder window, click Semiconductor .

- 2 In the Settings window for Semiconductor locate the Model Properties section. From the Carrier statistics drop down menu, choose Fermi-Dirac.

### Analytic Doping Model 1

First a constant background acceptor concentration is defined.


- 1 On the Physics toolbar click the Domains menu and choose Analytic Doping Model .



- 2 In the Settings window for Semiconductor Doping Model locate the Domain Selection section. From the Selection list, choose All domains.
- 3 Locate the Impurity section.
  - From the Impurity type list, choose Acceptor doping (p-type).
  - In the  $N_{A0}$  text field, type  $1e17 [1/cm^3]$ .

### Analytic Doping Model 2

Add a second doping feature to define the implanted doping profile for the source.

- 1 On the Physics toolbar click the Domains menu and choose Analytic Doping Model .
- 2 In the Settings window for Analytic Doping Model locate the Domain Selection section. From the Selection list, choose All domains.
- 3 Locate the Distribution section. From the list, choose Box.

When defining a Gaussian doping distribution a rectangular region of constant doping is defined. The Gaussian drop off occurs away from the edges of this rectangular region.

Define the location of the lower left corner of the uniformly doped region.

1 Locate the Uniform Region section.

- Specify the  $r_0$  vector as

0[um]	X
0.6[um]	Y

2 Then define the width and height of the uniformly doped region.

- In the  $W$  text field, type 0.6[um].
- In the  $D$  text field, type 0.1[um].

Choose the dopant type and the doping level in the uniformly doped region.

3 Locate the Impurity section.

- From the Impurity type drop down menu, choose Donor doping (n-type).
- In the  $N_{D0}$  text field, type 1e20[1/cm^3].

Next specify the length scale over which the Gaussian drop off occurs. If doping into a background dopant distribution of opposite type (as in this case), this setting specifies the junction depth. In this model different length scales are used in the x and y directions.

4 Locate the Profile section. Select the Specify different length scales for each direction check box.

5 Specify the  $d_j$  vector as

0.2[um]	X
0.25[um]	Y

Finally specify the constant background doping level.

6 From the  $N_b$  list, choose Acceptor concentration (semi/adm1).

### Analytic Doping Model 3

Add a similar Gaussian doping profile for the drain.

1 Right-click Analytic Doping Model 2 and choose Duplicate .

2 In the Settings window for Analytic Doping Model, locate the Uniform Region section.


- Specify the  $r_0$  vector as

2.4[um]	X
0.6[um]	Y

Next set up boundary conditions for the contacts and gate.

### *Metal Contact 1*

First add a contact for the source.

- 1 On the Physics toolbar click the Boundaries menu and choose Metal Contact .


The Metal Contact feature is used to define Metal-Semiconductor interfaces of various types. In this instance use the default Ideal Ohmic contact to define the source.

- 2 Select Boundary 3 only.

**Note:** The Metal Contact feature in COMSOL Multiphysics is a so called Terminal boundary condition. By default a fixed potential of 0 V is applied, which is appropriate in this instance, since the source is grounded. The terminal can also be set up to specify an input current, input power, or to connect to a voltage or current source from an external circuit.

### *Metal Contact 2*

Add a second Metal Contact feature to define the drain.

- 1 On the Physics toolbar click the Boundaries menu and choose Metal Contact .


- 2 Select Boundary 7 only.

Set the drain voltage to be determined by the previously defined parameter.

- 3 In the Settings window for Metal Contact locate the Terminal section. In the  $V_0$  text field, type  $V_d$ .

### *Metal Contact 3*


Add a third Metal Contact to set the body voltage to 0 V.

- 1 On the Physics toolbar click the Boundaries menu and choose Metal Contact .

- 2 Select Boundary 2 only.

### *Thin Insulator Gate 1*

Set up the Gate. The Gate dielectric is not explicitly represented in the model, instead the Thin Insulator Gate boundary condition represents both the gate contact and the thin layer of oxide.

- 1 On the Physics toolbar click the Boundaries menu and choose Thin Insulator Gate .

The Thin Insulator Gate feature is also a terminal, but in this instance it is possible to fix the voltage or charge on the terminal, as well as to connect it to a circuit.


**Note:** The charge setting determines the charge on the conductor and does not relate to trapped charge at the oxide-semiconductor interface.

The voltage applied to the gate is determined by the parameter added previously.

- 2 In the Settings window for Thin Insulator Gate, locate the Terminal section. In the  $V_0$  text field, type  $V_g$ .
- 3 Locate the Gate Contact section.
  - In the  $\epsilon_{ins}$  text field, type 4.5.
  - In the  $d_{ins}$  text field, type 30[nm].
- 4 Select Boundary 5 only.

### *Trap-Assisted Recombination I*


A range of recombination/generation mechanisms are available to be added to the model. In this case, we simply add trap-assisted recombination, using the default Shockley-Read-Hall model.

- 1 On the Physics toolbar click the Domains menu and choose Generation-Recombination section > Trap-Assisted Recombination .
- 2 In the Settings window for Trap-Assisted Recombination locate the Domain Selection section. From the Selection list, choose All domains.


## Mesh I

---




We will use a user-defined mesh for this model.

- 1 In the Model Builder window, under Component 1 (comp1) click Mesh 1 .
- 2 In the Settings window for Mesh, locate the Mesh Settings section. From the Sequence type drop-down menu, choose User-controlled mesh.


### *Size*

- 1 In the Model Builder window, under Component 1 (comp1)>Mesh 1 click Size .
- 2 In the Settings window for Size, locate the Element Size section. Click the Custom button.
- 3 Locate the Element Size Parameters section. In the “Maximum element growth rate” text field, type 1.05.


### *Size 1, Size 2 and Free Triangular 1*

- 1 In the Model Builder window, under Component 1 (comp1)>Mesh 1 right-click Size 1  and choose Delete.
- 2 Right-click Size 2  and choose Delete.
- 3 Right-click Free Triangular 1  and choose Delete.


### *Edge 1*

- 1 Right-click Mesh 1 and choose More Operations>Edge .
- 2 In the Settings window for Edge, locate the “Boundary Selection” section. Select Boundaries 3–7 only.
- 3 Click to expand the “Control Entities” section. Clear the “Smooth across removed control entities” check box.


### *Size 1*

- 1 Right-click Edge 1 and choose Size .
- 2 In the Settings window for Size, locate the Element Size section.
  - From the “Calibrate for” drop down menu, choose Semiconductor.
  - Click the Custom button.
- 3 Locate the Element Size Parameters section.
  - Select the “Maximum element size” check box.
  - In the associated text field, type 0.03.

### *Mapped 1*



- 1 In the Model Builder window, right-click Mesh 1 and choose Mapped .
- 2 In the Settings window for Mapped, locate the Domain Selection section.
  - From the “Geometric entity level” drop down menu, choose Domain.
  - Select Domain 2 only.
- 3 Click to expand the “Control Entities” section. Clear the “Smooth across removed control entities” check box.
- 4 Click to expand the Advanced Settings section. Select the “Adjust evenly distributed edge mesh” check box.

### *Distribution 1*

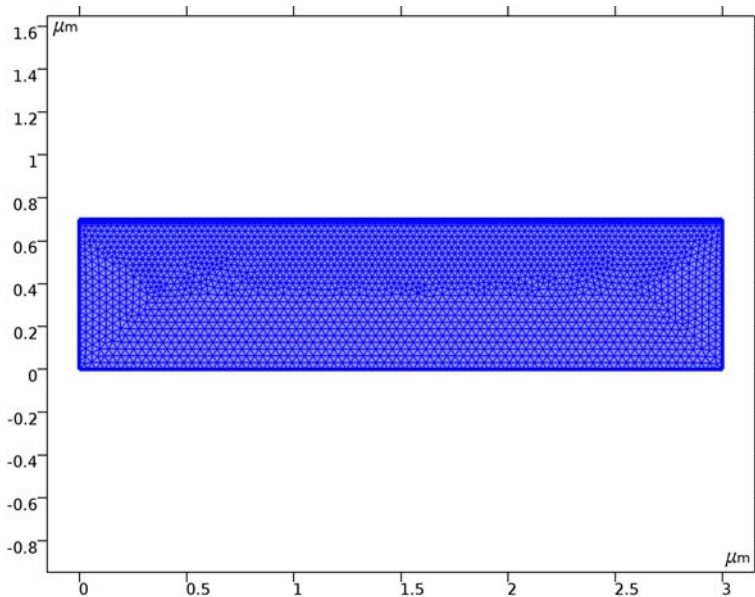
- 1 Right-click Mapped 1 and choose Distribution .
- 2 Select Boundary 9 only.

- 3 In the Settings window for Distribution, locate the Distribution section.
- From the “Distribution properties” drop down menu, choose “Predefined distribution type”.
  - In the Number of elements text field, type 8.
  - In the Element ratio text field, type 9.
  - From the “Distribution method” drop down menu, choose “Geometric sequence”.
  - Select the “Reverse direction” check box.

### *Free Triangular 1*

- 1 In the Model Builder window, right-click Mesh 1 and choose Free Triangular .
- 2 In the Settings window for Free Triangular, click to expand the “Control entities” section. Clear the “Smooth across removed control entities” check box.
- 3 Click Build all .



The user-defined mesh is shown in the image below. The mapped mesh with the specific distribution helps create layers of thin elements underneath the gate, where the large gradient of the carrier concentration needs to be resolved by the mesh.



## Study 1

---

Before setting up the study, check that the doping was set up correctly. To do this, first get the initial value for the study.






- 1 In the Model Builder click Study 1 .  
Disable the default plots since these are not required.
- 2 In the Settings window for Study locate the Study Settings section. Click to clear the Generate default plots check box.
- 3 On the Study toolbar click Get Initial Value .

## Results

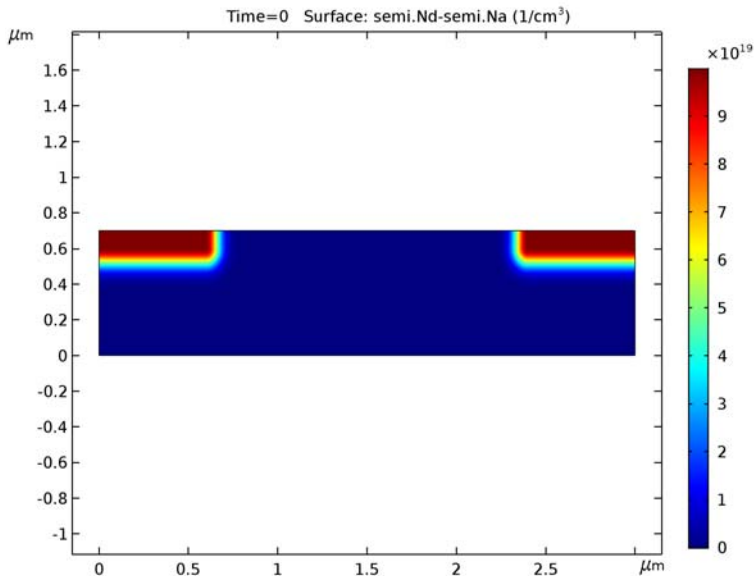
---

### 2D Plot Group 1

Add a 2D plot group to check the dopant distribution in the model.

- 1 On the Home toolbar click Add Plot Group  and choose 2D Plot Group .
- 2 Rename the Plot Group by typing “Signed Dopant Concentration” in the Label field.
- 3 On the Signed Dopant Concentration toolbar click Surface .  
Plot the signed dopant concentration (Nd-Na). This quantity is positive for net donor doping and negative for net acceptor doping.
- 4 In the Model Builder under Results>Signed Dopant Concentration, click Surface 1 .
- 5 In the Settings window for Surface, locate the Expression section. In the Expression text field, type `semi.Nd-semi.Na`.
- 6 Click the Plot button .

The doping distribution is shown below.




## Study I

---


### Step 1: Stationary

Now set up a Stationary study to determine the turn-on voltage for the transistor. In this study we set  $V_d$  to 10 mV and sweep over  $V_g$ .

- 1 In the Model Builder under Study 1, click Step 1: Stationary .
- 2 In the Settings window for Stationary click to expand the Study extensions section.



The study extensions panel can be used to set up a parametric sweep.

- 3 Locate the Study Extensions section. Select the Auxiliary sweep check box.
- 4 From the Sweep type list, choose All combinations. This sweeps over every combination of the specified parameters.

- 5 Click Add . In the table, enter the following settings:

PARAMETER NAME	PARAMETER VALUE LIST
Vd	0.01

The drain voltage is fixed at a constant value.




- 6 Click Add .
- 7 In the table cell for the parameter value list for Vg, enter range(0,0.2,1.4)  
2 3 4. The gate voltage is swept between 0 and 4V with uneven step sizes to reduce computation time and file size.
- By default the “Run continuation for” setting is set to Last parameter (Vg in this case). This setting configures the solver to use the solution for the previous value of Vg as the initial guess for the solution when solving for the next value of Vg. It also allows the solver to take intermediate steps at values of Vg not specified in the parameter value list if necessary.
- 8 On the Home toolbar click Compute .

## Results

---


### 1D Plot Group 2

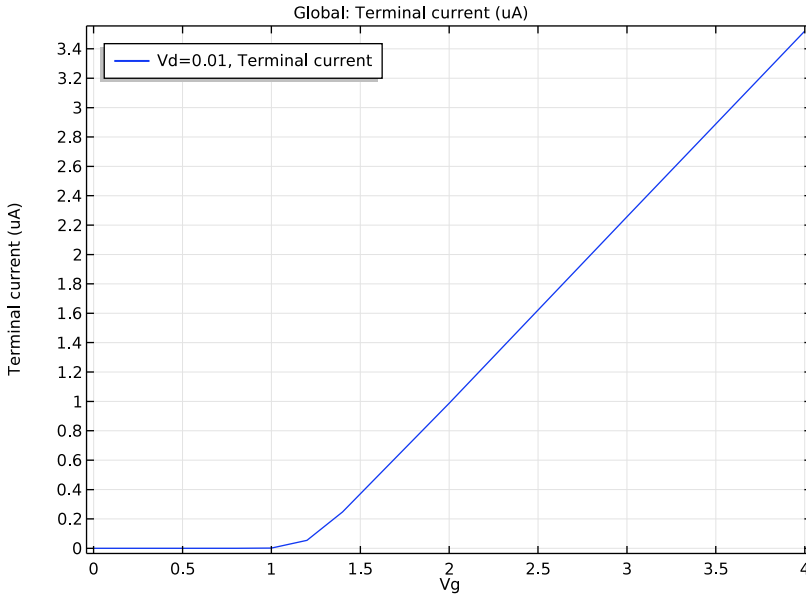
Add a 1D plot group to plot the source current versus the gate voltage.

- 1 On the Home toolbar click Add Plot Group  and choose 1D Plot Group .
- 2 Rename the Plot Group by typing “Id vsVg (Vd=10mV)” in the Label field.
- 3 On the 1D Plot Group toolbar click Global .

The post-processing menus contain a wide range of quantities available for plotting. Choose the current flowing into terminal 2.

- 4 In the Settings window for Global  click Replace Expression  (it is in the upper-right corner of the y-Axis Data section). In the menu, double-click Model>Component1>Semiconductor>Terminals>semi.I0\_2-Terminal Current.

5 Change the Unit to uA and click the Plot button .



From the plot it is clear that the threshold voltage,  $V_T$ , of the transistor is approximately 1.2 V. It is possible to compare this value with the theoretical value given by (S. M. Sze and K. K. Ng, *Physics of Semiconductor Devices*, Wiley, Hoboken, New Jersey, pp. 305–306, 2007):

$$V_T \equiv V_{FB} + 2\psi_B + \frac{d_{ox}(4\epsilon_{r,s}\epsilon_0 q N_a \psi_B)^{\frac{1}{2}}}{\epsilon_{r,ox}\epsilon_0}$$

where  $d_{ox}$  is the thickness of the oxide and  $\epsilon_{r,ox}$  is its relative permittivity,  $\epsilon_0$  is the permittivity of free space,  $\epsilon_{r,s}$  is the relative permittivity of the semiconductor,  $q$  is the electron charge and  $N_a$  is the acceptor concentration under the gate. The flat band voltage  $V_{FB}$  and the potential difference between the intrinsic level and the Fermi-level,  $\psi_B$ , are given by the following equations:

$$V_{FB} = \Phi_m - \chi + \frac{k_B T}{q} \ln\left(\frac{n_{eq}}{N_c}\right)$$

$$\psi_B = \frac{k_B T}{q} \ln\left(\frac{p_{eq}}{n_i}\right)$$

where  $\Phi_m$  is the work function of the metal contact,  $\chi$  is the semiconductor electron affinity,  $k_B$  is Boltzmann's constant,  $T$  is the absolute temperature,  $N_c$  is

the semiconductor density of states in the conduction band and  $n_i$  is the intrinsic carrier density. The equilibrium electron ( $n_{eq}$ ) and hole ( $p_{eq}$ ) densities are given by:

$$n_{eq} = \frac{1}{2}(N_d - N_a) + \frac{1}{2}\sqrt{(N_d - N_a)^2 + 4n_i}$$





$$p_{eq} = -\frac{1}{2}(N_d - N_a) \pm \frac{1}{2}\sqrt{(N_d - N_a)^2 + 4n_i}$$

Where  $N_d$  is the donor concentration under the gate. Note that these equations assume both complete ionization and Maxwell Boltzmann statistics (reasonable assumptions near the threshold condition). These equations give a threshold voltage of 1.20 V, in good agreement with the simulation result.



## Study 2

---


Now add an additional study to plot the source current as a function of drain voltage at a range of different gate voltages.



- 1 On the Home toolbar click Add Study .
- 2 In the Add Study window, find the Studies subsection. In the tree under Preset Studies, click Stationary .
- 3 Click Add Study  in the window toolbar.
- 4 On the Home toolbar click Add Study  to close the Add Study window.

### Step 1: Stationary

- 1 In the Model Builder under Study 2, click Step 1: Stationary .
- 2 In the Settings window for Stationary locate the Study Extensions section.
- 3 Select the Auxiliary sweep check box.
- 4 From the Sweep type list choose All combinations. Click Add . Select Vg from the Parameter name drop down menu.

Now set up the study to sweep over Vg.

- 5 Click Range . Go to the Range dialog box.
  - In the Start text field, type 2.
  - In the Step text field, type 1.
  - In the Stop text field, type 4.
- 6 Click Replace.



- 7 Click Add . In this case the default Auxiliary Parameter Vd is the desired parameter for the sweep.
- 8 In the table cell for the parameter value list for Vd, enter range (0,0.25,1.5)  
2 3 4 5.  
The *inner* sweep over Vd, the last parameter, should be used for the continuation solver (this is the default), since the solution will only change slightly between close values of Vd.  
Configure the solver to reuse the solution from the last step of the continuation parameter sweep as the initial value for the next step for the *outer* sweep parameter Vg.
- 9 From the “Reuse solution for previous step” drop down menu, choose Auto.
- 10 On the Home toolbar click Compute .

## Results




---

By looking at the electron concentration at Vd values of 0, 1 and 5 V the pinch-off of the channel can be clearly seen.

### *Electron Concentration (semi)*

- 1 In the Model Builder under Results, click Electron Concentration (semi) .  
By default the plot shows the results for the case Vd=5 V and Vg=4 V.
- 2 In the Settings window for 2D Plot Group locate the Data section. Change the Vd Parameter value to 1 V and then to 0 V, each time selecting the Plot button  to see how the results change.

### *Electric Potential (semi)*

- 1 In the Model Builder under Results, click Electric Potential (semi) .
- 2 Look at the plot for Vd values of 5 V, 1 V and 0 V.  
The plots in [Figure 9](#) show the results. The pinch off effect is apparent at 5 V.  
Add another 1D plot group to plot the drain current vs the drain voltage.
- 3 On the Home toolbar select Add Plot Group , and choose 1D Plot Group .
- 4 Rename the plot group by typing “Id vs Vd” in the Label field.
- 5 In the Settings window for 1D Plot Group locate the Data section. From the Data set list, choose Study 2/Solution 2.

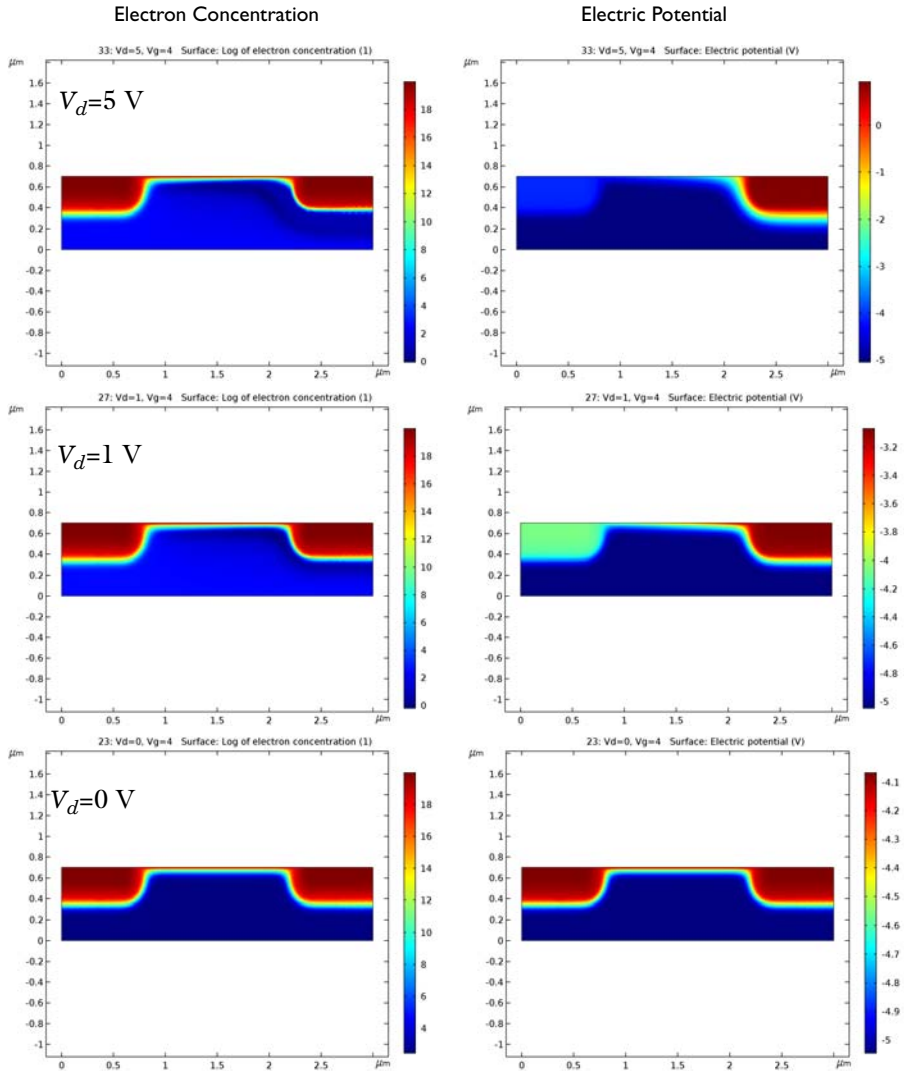


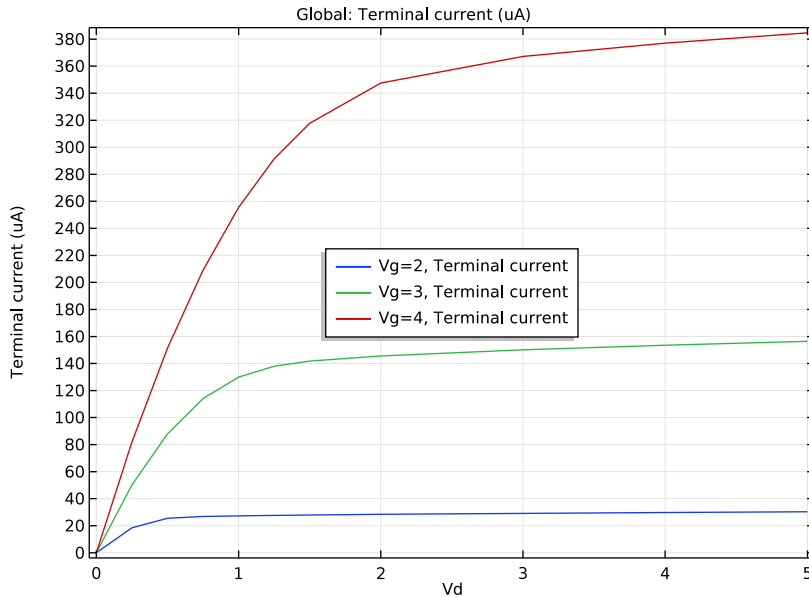


Figure 9: The pinch-off of the channel is apparent in the plots of the electron concentration and the potential distribution at  $V_d$  values of 0, 1 and 5 V.

### ID Plot Group 6

- 1 From the Id vs  $V_d$  toolbar click Global .
- 2 In the Model Builder under Results>Id vs  $V_d$ , click Global 1 .

- 3 In the Settings window for Global click Replace Expression  in the upper-right corner of the y-Axis Data section. In the menu, double click on Model>Component1>Semiconductor>Terminals>semi.I0\_2-Terminal Current.
- 4 Change the Unit to uA for the current.
- 5 Click the Plot button .



The drain current vs drain voltage diagram shows a linear region for low bias, followed by a nonlinear region. The drain current increases slightly with voltage in the saturation region at higher drain voltages, as a result of short channel effects. Short channel effects mean that the standard analytic expressions for the saturation voltage and current do not apply, but the saturation voltages and currents are of a similar magnitude to those predicted by the simple theory (S. M. Sze and K. K. Ng, *Physics of Semiconductor Devices*, Wiley, Hoboken, New Jersey, pp. 305–306, 2007).