Created in COMSOL Multiphysics 6.1



3D Analysis of a Bipolar Transistor

This model is licensed under the COMSOL Software License Agreement 6.1. All trademarks are the property of their respective owners. See www.comsol.com/trademarks. This model shows how to set up a 3D simulation of a n-p-n bipolar transistor. It is a 3D version of the device shown in the Bipolar Transistor model and demonstrates how to extend semiconductor modeling into 3D using COMSOL Multiphysics. As in the 2D version of this model, the device is simulated whilst operating in the common-emitter regime. A voltage-driven study is computed to characterize the current-voltage response of the device, and a current driven study is performed to simulate the device operating as an analog current amplifier.

Introduction

Bipolar transistors rely on both electron and hole currents in order to function whereas unipolar transistors, such as MOSFET devices, operate utilizing only one species of carrier. Bipolar transistors have largely been replaced in integrated circuits by field-effect devices; however, they are still important in analog electronics — particularly in power control circuitry where they can be used as switches and current amplifiers.



Figure 1: Left: Simplified cross section through a bipolar transistor showing the structure of the device. Right: Circuit diagram showing the common emitter configuration.

A bipolar transistor consists of three regions known as the emitter, base, and collector. In an n-p-n transistor the p-type base region is sandwiched between the n-type emitter and collector regions, as shown in the left panel of Figure 1. In the common emitter configuration the emitter contact is the common ground for both the base and collector contacts, that is, the base and collector voltages are measured relative to the emitter, which is grounded. This is shown schematically in the right panel of Figure 1.

In normal operation, the base-emitter junction is under forward bias and the basecollector junction is under reverse bias. Electrons are injected over the forward bias p-n junction from the emitter into the base. They then diffuse through the base region as minority carriers. Those electrons which reach the base-collector junction are swept to the collector contact by the electric field of the depleted region near the reverse bias p-n junction.

The effective resistance between the emitter and collector can be varied by applying a current to the base. In this way, the collector-emitter current can be controlled by a smaller base-emitter current. In this configuration the device functions as a current amplifier, as the collector-emitter current (at a given collector-emitter voltage) is proportional to the base-emitter current. Typically, the current gain can have values of the order of 100 which makes bipolar transistors attractive in a wide range of power management circuitry. For example, a small current from some sensing circuitry, such as a photodiode or temperature probe, could be used to control a larger current needed to operate a motor or a heating element.

The model presented here preforms a detailed DC current-voltage characterization of the bipolar transistor device. The current gain is computed as a function of the collector current, along with an emitter-collector I-V curve for a fixed current applied to the base.

Model Definition

The model geometry is shown in Figure 2. Due to the symmetry of the device only one quarter of the whole structure is explicitly modeled. The modeled doping profile is shown in Figure 3. As is typical of the profile used in silicon bipolar transistors, it consists of four regions (n+, p, n, and n+), described in detail in Modeling Instructions.



Figure 2: Model geometry, the symmetry planes are highlighted in blue and the boundaries to which the three electric contacts are applied are labeled.



Figure 3: Dopant distribution for the bipolar transistor device. Left: Volume plot showing the total net dopant concentration, the emitter region can be clearly seen in red; the boundary between the base and collector is not apparent due to the large magnitude of the concentration in the n+ regions. Right: Line cut of the total net dopant concentration taken along the red line shown in the left-hand pane. The p-type base region can be seen in this plot.

The first study in this model sweeps the base voltage whilst applying a constant voltage of 0.5 V to the collector contact, where both voltages are measured relative to the grounded emitter. This setup allows the currents at all three terminals to be plotted as a function of the base voltage and demonstrates that the simulation conserves current. This study is also used to create a Gummel plot, which shows the current at the collector and base contacts as a function of the base voltage. The same data is then used to calculate the current gain, defined as the ratio of the collector and base currents (I_C/I_E), as a function of the collector current. The current gain curve is an important characteristic for current regulation and

power control applications, as it is used to calculate the expected collector output for a given base input.

The next two studies are used to compute the collector current as a function of collector voltage for an applied base current of $2 \mu A$. This is an example of a current driven problem, as a set current is being applied to one of the contacts. When solving current driven problems in COMSOL Multiphysics, it is often necessary to provide an initial condition that is calculated from a suitable voltage driven study, in which only voltages are applied at contacts. In order to generate the required initial conditions for these studies a voltage driven initialization study is performed. This initialization study sets the collector voltage to 0, which is the initial value it will have in the current driven studies, and sweeps the base voltage. A solution that generates a base current of suitable magnitude can then be used as the initial conditions for the current driven study.

Results and Discussion

Figure 4 displays the current at each terminal as a function of the base-emitter voltage $(V_{\rm BE})$ for a fixed collector-emitter voltage $(V_{\rm CE}$ =0.5 V). Note that the figure shows the terminal currents using the COMSOL Multiphysics sign convention: current that flows from the contact into the semiconductor is positive, and current that flows out of the semiconductor into the contact is negative. The figure also shows that the current is conserved. This can be seen as the sum of the base and collector currents have equal magnitude and opposite sign to the emitter current, i.e: the base current can be calculated from the other currents using

$$I_B = -(I_E + I_C)$$



Figure 4: Terminal currents as a function of the base-emitter voltage (V_{BE}) for a fixed collector-emitter voltage $(V_{CE} = 0.5 V)$.



Figure 5: Gummel plot showing the magnitude of the collector and base current as a function of the base voltage.



Figure 6: Current gain as a function of collector current for a fixed base voltage of $V_{\rm BE}$ =0.5 V.

Figure 5 shows the Gummel plot for the modeled bipolar transistor. The Gummel plot shows the magnitude of the collector and base currents, plotted on a logarithmic scale, as a function of the base voltage.



Figure 6 shows the current gain, defined as I_C/I_B , as a function of the collector current at a fixed base voltage of V_{BE} =0.5 V.

Figure 7: Plot of collector current vs. collector voltage for $I_B = 2 \mu A$.

Figure 7 shows the collector current as a function of collector voltage for a fixed applied base current of 2 μ A. This figure shows the collector I-V curve for the device in the common emitter configuration. Initially the current increases linearly with increasing emitter-collector voltage, before reaching a saturation level. The gradient of the linear regime and the magnitude of the saturation current depend on the base current. As the model is computationally intensive only one value of base current is simulated, however see Bipolar Transistor for a comparison between the different applied base currents.

Figure 8 shows the voltage and carrier current densities throughout the device. With V_{CE} = 1.5 V the device is in the forward-active regime. In this regime the emitter-base junction is forward biased and the base-collector junction is reverse biased. Electrons are injected from the emitter into the base through the forward biased junction. These electrons then diffuse through the p-type base region as minority carriers. Those that make it to the reverse biased base-collector junction are swept toward the collector terminal by the junction electric field. The thickness of the base region must be small enough to allow the electrons to diffuse through with high probability. Holes can travel easily from the base to the emitter regions through the forward biased emitter-base junction, but they cannot

traverse the reverse biased base-collector junction. Hence the hole current flows between the emitter and base terminals without entering the lower n-doped region, and the electron current flows between the emitter and collector terminals.



Figure 8: Voltage and current density for $I_B = 2 \mu A$ and $V_{CE} = 1.5 V$. The color shows the voltage and the arrows show the current density for electrons (black) and holes (white). Note that the hole current flows from the base to the emitter and does not enter the lower n-doped region, whilst the electron current flows between the collector and emitter. This current pattern is due to the two p-n junctions that form the device. The electric field is largest around the junctions, as can be seen by the rapid change in voltage between the differently doped regions.

Application Library path: Semiconductor_Module/Transistors/ bipolar_transistor_3d

Modeling Instructions

Note: This model is computationally expensive and may require more resources than are available on a typical desktop machine. It is likely to take several hours to solve all the studies in the model. A 2D version of this model is available (Bipolar Transistor), which typically can be solved on a normal desktop machine in a few minutes.

From the File menu, choose New.

NEW

In the New window, click 🙆 Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click 间 3D.
- 2 In the Select Physics tree, select Semiconductor>Semiconductor (semi).
- 3 Click Add.
- 4 Click \bigcirc Study.
- 5 In the Select Study tree, select General Studies>Stationary.
- 6 Click 🗹 Done.

GLOBAL DEFINITIONS

Import the model parameters from bipolar_transistor_3d_parameters.txt.

- I In the Model Builder window, click Global Definitions.
- 2 In the Settings window for Parameters, locate the Parameters section.
- **3** Browse to the model's Application Libraries folder and double-click the file bipolar_transistor_3d_parameters.txt.

Create the model geometry, the device consists of two Blocks and a Work Plane. The larger Block defines the volume of the device, and the smaller block is used when generating the structured mesh. The Work Plan is used to define the geometry of the electric contacts on the top surface. Note that, as the device has planes of symmetry in the *xz*- and *yz*-planes passing through the origin, it is only necessary to model one quarter of the device.

GEOMETRY I

I In the Model Builder window, under Component I (compl) click Geometry I.

- 2 In the Settings window for Geometry, locate the Units section.
- **3** From the **Length unit** list, choose **µm**.

Block I (blkI)

- I In the **Geometry** toolbar, click 🗍 **Block**.
 - Add Silicon as the material for the device.
- 2 In the Settings window for Block, locate the Size and Shape section.
- **3** In the **Width** text field, type w_BJT/2.
- 4 In the **Depth** text field, type 1_BJT/2.
- **5** In the **Height** text field, type d_BJT.

Block 2 (blk2)

- I In the **Geometry** toolbar, click 🗍 Block.
- 2 In the Settings window for Block, locate the Size and Shape section.
- 3 In the Width text field, type w_BJT/2.
- 4 In the **Depth** text field, type 1_BJT/2.
- 5 In the Height text field, type 1*d_E.
- 6 Locate the Position section. In the z text field, type d_BJT-1.25*d_E.

Work Plane I (wp1)

- I In the Geometry toolbar, click 🖶 Work Plane.
- 2 In the Settings window for Work Plane, locate the Plane Definition section.
- 3 In the z-coordinate text field, type d_BJT.
- 4 Click 📥 Show Work Plane.

Work Plane I (wpl)>Rectangle I (rl)

- I In the Work Plane toolbar, click 📃 Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type w_cE.
- 4 In the **Height** text field, type 1_E/2-2*d_E.

Work Plane I (wp1)>Rectangle 2 (r2)

- I In the Work Plane toolbar, click 📃 Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- **3** In the **Width** text field, type w_BJT/2-w_EB-w_E/2.
- 4 In the **Height** text field, type 1_cB/2-2*d_E.

- 5 Locate the Position section. In the xw text field, type w_BJT/2-w_cB.
- 6 In the Model Builder window, right-click Geometry I and choose Build All.

ADD MATERIAL

- I 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)

Now the physics can be configured for the model. The first step is to create the required dopant distribution. This is achieved using four **Analytic Doping Model** features, one to specify a constant background level and then one for each of the emitter, base, and collector regions.

Add a constant background n-doping to the device.

SEMICONDUCTOR (SEMI)

Constant Background n Doping

- I In the Model Builder window, under Component I (compl) right-click Semiconductor (semi) and choose Doping>Analytic Doping Model.
- 2 In the Settings window for Analytic Doping Model, locate the Domain Selection section.
- **3** From the Selection list, choose All domains.
- 4 Locate the Impurity section. From the Impurity type list, choose Donor doping (n-type).
- **5** In the N_{D0} text field, type N_epi.
- 6 In the Label text field, type Constant Background n Doping.

Base p Doping

I In the Physics toolbar, click 🔚 Domains and choose Analytic Doping Model.

Add a layer of p-type doping to for the base region.

- 2 In the Settings window for Analytic Doping Model, locate the Domain Selection section.
- **3** From the Selection list, choose All domains.
- 4 Locate the Distribution section. From the list, choose Box.

- **5** Locate the **Impurity** section. In the N_{A0} text field, type N_B+N_epi.
- 6 Locate the Uniform Region section. Specify the r_0 vector as

0[am]	^
0[um]	Y
d_BJT-d_E	Z

- **7** In the *W* text field, type $w_BJT/2$.
- **8** In the *D* text field, type $1_cB/2$.
- **9** In the *H* text field, type d_E.
- IO Locate the **Profile** section. In the d_j text field, type d_E.
- II From the N_b list, choose Donor concentration (semi/adml).
- 12 In the Label text field, type Base p Doping.

Emitter n Doping

I In the Physics toolbar, click 🔚 Domains and choose Analytic Doping Model.

Add an n-type region for the emitter.

- 2 In the Settings window for Analytic Doping Model, locate the Domain Selection section.
- **3** From the Selection list, choose All domains.
- 4 Locate the Distribution section. From the list, choose Box.
- 5 Locate the Impurity section. From the Impurity type list, choose Donor doping (n-type).
- **6** In the N_{D0} text field, type N_E+N_B.
- 7 Locate the **Uniform Region** section. Specify the r_0 vector as

0[um]	X
-------	---

0[um] Y

d_BJT Z

8 In the *W* text field, type $w_E/2 - d_E$.

9 In the *D* text field, type $1_E/2-2*d_E$.

IO Locate the **Profile** section. In the d_j text field, type d_E.

II In the N_b text field, type N_B.

12 In the Label text field, type Emitter n Doping.

Add another n-type region for the collector.

Collector n Doping

- I 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 All domains.
- 4 Locate the Distribution section. From the list, choose Box.
- 5 Locate the Impurity section. From the Impurity type list, choose Donor doping (n-type).
- **6** In the N_{D0} text field, type N_C.
- 7 Locate the **Uniform Region** section. In the *W* text field, type $w_BJT/2$.
- **8** In the *D* text field, type $1_BJT/2$.
- **9** Locate the **Profile** section. In the d_i text field, type 1.3*d_C.
- **IO** From the N_b list, choose **Donor concentration (semi/adml)**.
- II In the Label text field, type Collector n Doping.

Add a Trap-Assisted Recombination feature to the model.

Trap-Assisted Recombination 1

- I In the Physics toolbar, click 🔚 Domains and choose Trap-Assisted Recombination.
- **2** In the Settings window for Trap-Assisted Recombination, locate the Domain Selection section.
- 3 From the Selection list, choose All domains.

Next add Metal Contact features to define the emitter, base, and collector contacts.

Emitter Voltage

- I In the Physics toolbar, click 📄 Boundaries and choose Metal Contact.
- 2 Select Boundary 10 only.
- 3 In the Settings window for Metal Contact, locate the Terminal section.
- **4** In the V_0 text field, type V_E.
- **5** In the **Label** text field, type Emitter Voltage.

Base Voltage

- I In the Physics toolbar, click 🔚 Boundaries and choose Metal Contact.
- 2 Select Boundary 15 only.
- 3 In the Settings window for Metal Contact, locate the Terminal section.
- **4** In the V_0 text field, type V_B.
- 5 In the Label text field, type Base Voltage.

Collector Voltage

- I In the Physics toolbar, click 📄 Boundaries and choose Metal Contact.
- 2 Select Boundary 3 only.
- 3 In the Settings window for Metal Contact, locate the Terminal section.
- **4** In the V_0 text field, type V_C.
- 5 In the Label text field, type Collector Voltage.

As well as applying a voltage to all three contacts, this model also requires the application of a current to the base and collector contacts. This is achieved by duplicating each of the respective voltage-applying contacts and selecting to apply a current. In each study the relevant contact boundary conditions are chosen by selectively disabling the features which are not required.

Base Current

- I In the Model Builder window, right-click Base Voltage and choose Duplicate.
- 2 In the Settings window for Metal Contact, locate the Terminal section.
- 3 From the Terminal type list, choose Current.
- **4** In the I_0 text field, type I_B.
- 5 In the Label text field, type Base Current.

Collector Current

- I In the Model Builder window, right-click Collector Voltage and choose Duplicate.
- 2 In the Settings window for Metal Contact, locate the Terminal section.
- 3 From the Terminal type list, choose Current.
- **4** In the *I*⁰ text field, type **I**_C.
- **5** In the V_{init} text field, type V_B.
- 6 In the Label text field, type Collector Current.

The next step is to configure a suitable mesh. For three dimensional semiconductor models a structured swept mesh is recommended. This is achieved by creating a free triangular mesh on the top surface of the device and then sweeping it down through the rest of the geometry.

MESH I

First create the free triangular mesh on the top surface of the device. The size node is set to calibrate the mesh density for semiconductor physics with the predefined **finer** settings.

Free Triangular 1

- I In the Mesh toolbar, click \bigwedge Boundary and choose Free Triangular.
- **2** Select Boundaries 10, 11, and 15 only.

Size

- I In the Model Builder window, click Size.
- 2 In the Settings window for Size, locate the Element Size section.
- **3** From the **Calibrate for** list, choose **Semiconductor**.
- 4 From the Predefined list, choose Finer.
- 5 Click 📗 Build All.

Next the mesh is swept down through the three domains. A **distribution** node is added to each swept mesh in order to control the mesh density in the z direction. The mesh is created such that it is finest around the emitter-base and base-collector junctions.

Swept I

- I In the Mesh toolbar, click 🎪 Swept.
- 2 In the Settings window for Swept, locate the Domain Selection section.
- 3 From the Geometric entity level list, choose Domain.
- **4** Select Domain 3 only.

Distribution I

- I Right-click Swept I and choose Distribution.
- 2 In the Settings window for Distribution, locate the Distribution section.
- 3 In the Number of elements text field, type ceil(mfac*10*(d_BJT-(d_BJT-1.5*d_E+ 1.25*d_E))/d_BJT).

Swept 2

- I In the Mesh toolbar, click 🦓 Swept.
- 2 In the Settings window for Swept, locate the Domain Selection section.
- 3 From the Geometric entity level list, choose Domain.
- **4** Select Domain 2 only.

Distribution I

- I Right-click Swept 2 and choose Distribution.
- 2 In the Settings window for Distribution, locate the Distribution section.
- 3 From the Distribution type list, choose Predefined.

- 4 In the Number of elements text field, type ceil(mfac*4*(1.25*d_E/d_BJT)).
- 5 In the Element ratio text field, type 0.25.
- 6 From the Growth rate list, choose Exponential.
- 7 Select the Symmetric distribution check box.
- **8** Select the **Reverse direction** check box.

Swept 3

- I In the Mesh toolbar, click 🦓 Swept.
- 2 In the Settings window for Swept, locate the Domain Selection section.
- **3** From the **Geometric entity level** list, choose **Domain**.
- 4 Select Domain 1 only.

Distribution I

- I Right-click Swept 3 and choose Distribution.
- 2 In the Settings window for Distribution, locate the Distribution section.
- **3** From the **Distribution type** list, choose **Predefined**.
- 4 In the Number of elements text field, type ceil(mfac*1.25*(d_BJT-1.25*d_E-(d_BJT-(d_BJT-1.5*d_E+1.25*d_E)))/d_BJT).
- 5 In the Element ratio text field, type 0.25.
- 6 From the Growth rate list, choose Exponential.
- 7 Select the Symmetric distribution check box.
- 8 Select the **Reverse direction** check box.



9 In the Model Builder window, right-click Mesh I and choose Build All. The resulting mesh should look like this:

Configure the first study, this study will sweep the base voltage with $V_C = 0.5$ V and $V_E = 0$ V. As this is a voltage driven study with no applied currents the two current contacts are disabled. The range of voltages to apply is chosen to ensure a good distribution of data points in both the Current-Voltage graphs and the Current Gain graphs.

$V_B SWEEP, V_C=0.5 V, V_E=0 V$

- I In the Model Builder window, click Study I.
- 2 In the Settings window for Study, type V_B Sweep, V_C=0.5 V, V_E=0 V in the Label text field.
- 3 Locate the Study Settings section. Clear the Generate default plots check box.

Step 1: Stationary

- I In the Model Builder window, under V_B Sweep, V_C=0.5 V, V_E=0 V click Step I: Stationary.
- 2 In the Settings window for Stationary, locate the Physics and Variables Selection section.
- **3** Select the Modify model configuration for study step check box.

- 4 In the tree, select Component I (compl)>Semiconductor (semi)>Base Current.
- 5 Click 🖉 Disable.
- 6 In the tree, select Component I (compl)>Semiconductor (semi)>Collector Current.
- 7 Click 🖉 Disable.
- 8 Click to expand the Study Extensions section. Select the Auxiliary sweep check box.
- 9 Click + Add.

IO In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
V_B (Applied voltage: base)	range(0,0.05,1.1)	V

Before computing the solution, it is a good idea to get the initial value and confirm that the doping profile is set up correctly.

II In the Study toolbar, click $\underset{=}{\overset{\cup}{t=0}}$ Get Initial Value.

Plot the doping profile on a 1D Line Graph taken along a vertical line cut with (x,y) coordinates of (0,0). This corresponds to the front left edge of the model geometry, and is at the center of the device. This position samples the emitter, base, and collector doping.

RESULTS

Doping Profile

- I In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for ID Plot Group, type Doping Profile in the Label text field.

Line Graph I

- I Right-click Doping Profile and choose Line Graph.
- **2** Select Edges 1, 4, and 7 only.
- 3 In the Settings window for Line Graph, locate the y-Axis Data section.
- 4 In the Expression text field, type log10(abs((semi.Nd-semi.Na)/1[1/cm^3])).
- **5** Select the **Description** check box.
- 6 Locate the x-Axis Data section. From the Parameter list, choose Reversed arc length.
- 7 In the Doping Profile toolbar, click 💿 Plot.

Doping Profile

- I In the Model Builder window, click Doping Profile.
- 2 In the Settings window for ID Plot Group, locate the Plot Settings section.

- 3 Select the x-axis label check box. In the associated text field, type Depth (um).
- **4** In the **Doping Profile** toolbar, click **I** Plot.

The correct doping profile is in the right hand panel of Figure 3.

Now compute the solution to the study.

V_B SWEEP, $V_C=0.5$ V, $V_E=0$ V

Step 1: Stationary

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

Plot the current at each terminal as a function of the base voltage (V_B). Note that currents which flow from the contact into the semiconductor have positive sign and those which flow from the semiconductor into a contact have negative sign.

RESULTS

I_E, I_B and I_C as a Function of V_B

- I In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for ID Plot Group, type I_E, I_B and I_C as a Function of V_B in the Label text field.
- 3 Click to expand the Title section. From the Title type list, choose None.
- 4 Locate the Legend section. From the Position list, choose Lower left.

Global I

- I Right-click I_E, I_B and I_C as a Function of V_B 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_1	mA	I_E
semi.IO_2	mA	I_B
semi.IO_3	mA	I_C

4 In the I_E, I_B and I_C as a Function of V_B toolbar, click in Plot.

The current is conserved. To see this, add a plot of I_B=-I_E-I_C to the graph, which, due to the software sign convention, corresponds with emitter current being equal to the sum of the base and collector currents.

Global 2

- I In the Model Builder window, right-click I_E, I_B and I_C as a Function of V_B 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_1-semi.IO_3	mA	I_B calc

- 4 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.
- 5 From the Color list, choose Magenta.
- 6 Find the Line markers subsection. From the Marker list, choose Cycle.
- 7 From the Positioning list, choose Interpolated.
- 8 In the Number text field, type 30.
- 9 In the I_E, I_B and I_C as a Function of V_B toolbar, click 🗿 Plot.

I_E, I_B and I_C as a Function of V_B

- I In the Model Builder window, click I_E, I_B and I_C as a Function of V_B.
- 2 In the Settings window for ID Plot Group, locate the Plot Settings section.
- 3 Select the x-axis label check box. In the associated text field, type Base voltage (V).
- 4 Select the y-axis label check box. In the associated text field, type Current (mA).
- 5 In the I_E, I_B and I_C as a Function of V_B toolbar, click 💽 Plot.

Plot the collector and base currents as a function of the base-emitter voltage. This kind of plot is known as a Gummel plot, and is useful in device characterization.

Gummel Plot, I_C and I_B as a function of V_B

I In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.

2 In the Settings window for ID Plot Group, type Gummel Plot, I_C and I_B as a function of V_B in the Label text field.

Global I

- I Right-click Gummel Plot, I_C and I_B as a function of V_B 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
abs(semi.IO_3)	A	Collector Current, I_C
abs(semi.IO_2)	А	Base Current, I_B

Gummel Plot, I_C and I_B as a function of V_B

- I In the Model Builder window, click Gummel Plot, I_C and I_B as a function of V_B.
- 2 In the Settings window for ID Plot Group, locate the Title section.
- **3** From the **Title type** list, choose **Manual**.
- 4 In the Title text area, type Gummel Plot.
- 5 Locate the Plot Settings section.
- 6 Select the x-axis label check box. In the associated text field, type Base Voltage (V).
- 7 Select the y-axis label check box. In the associated text field, type Current (A).
- 8 Locate the Axis section. Select the y-axis log scale check box.
- 9 Locate the Legend section. From the Position list, choose Upper left.
- IO In the Gummel Plot, I_C and I_B as a function of V_B toolbar, click 🗿 Plot.

Notice that the very small currents for low values of base voltage are not reliable, this is due to the tolerance of the solver and the small magnitude of the current. Restrict the data to the range where the results are reliable.

- II Locate the Data section. From the Parameter selection (V_B) list, choose Manual.
- 12 In the Parameter indices (1-23) text field, type range(11,1,23).
- **I3** In the **Gummel Plot**, **I_C** and **I_B** as a function of **V_B** toolbar, click **O** Plot.

Another useful characterization quantity is the DC current gain curve, which is the ratio of the collector to base current (I_C/I_B) as a function of collector current.

Current Gain

I In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.

2 In the Settings window for ID Plot Group, type Current Gain in the Label text field.

Global I

- I Right-click Current Gain 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
abs(semi.IO_3/semi.IO_2)	1	Gain (I_C/I_B)

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

5 In the Expression text field, type semi.IO_3.

Current Gain

- I In the Model Builder window, click Current Gain.
- 2 In the Settings window for ID Plot Group, locate the Title section.
- 3 From the Title type list, choose Manual.
- 4 In the Title text area, type Current Gain (I_C/I_B).
- 5 Locate the Plot Settings section.
- 6 Select the x-axis label check box. In the associated text field, type Collector Current (A).
- 7 Select the y-axis label check box. In the associated text field, type Current Gain.
- 8 Locate the Axis section. Select the x-axis log scale check box.
- 9 In the Current Gain toolbar, click 💽 Plot.

As with the previous graph, restrict the data range to show only the reliable values.

- IO Locate the Data section. From the Parameter selection (V_B) list, choose Manual.
- II In the Parameter indices (1-23) text field, type range(11,1,23).

12 In the Current Gain toolbar, click 💿 Plot.

Bipolar transistors can be used to regulate current in analog circuits. In the common emitter configuration the effective resistance between the collector and emitter can be controlled by applying a current to the base. This allows the collector current, at a given voltage difference between the collector and emitter, to be controlled by an input current from a sensing circuit. In this way, a small current from a low power sensor applied to the base can be used to control a larger current which is output from the collector.

The next pair of studies compute the collector current as a function of collector voltage when a current of 2 uA is applied to the base. When a current is applied to a contact the problem is said to be current-driven. For computing current-driven problems in COMSOL it is often a good idea to first perform a suitable voltage-driven study, in which only voltages are applied to contacts, in order to generate appropriate initial values for the

dependent variables. In this case, this is achieved using a study that sweeps the base voltage with the collector and emitter voltages both set to 0 V.

ADD STUDY

- I In the Home toolbar, click $\stackrel{\text{res}}{\longrightarrow}$ 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 \sim Add Study to close the Add Study window.

INITIALIZATION STUDY

- I In the Model Builder window, click Study 2.
- 2 In the Settings window for Study, type Initialization Study in the Label text field.
- 3 Locate the Study Settings section. Clear the Generate default plots check box.

Step 1: Stationary

- I In the Model Builder window, under Initialization Study click Step I: Stationary.
- 2 In the Settings window for Stationary, locate the Physics and Variables Selection section.
- 3 Select the Modify model configuration for study step check box.
- 4 In the tree, select Component I (compl)>Semiconductor (semi)>Base Current.
- 5 Click 🕖 Disable.
- 6 In the tree, select Component I (compl)>Semiconductor (semi)>Collector Current.
- 7 Click 🖉 Disable.
- 8 Click to expand the Values of Dependent Variables section. Find the Initial values of variables solved for subsection. From the Settings list, choose User controlled.
- 9 From the Method list, choose Solution.
- IO From the Study list, choose V_B Sweep, V_C=0.5 V, V_E=0 V, Stationary.
- II From the Parameter value (V_B (V)) list, choose 0 V.
- 12 Locate the Study Extensions section. Select the Auxiliary sweep check box.
- I3 Click + Add.

I4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
V_C (Applied voltage: collector)	0	V

15 Click + Add.

I6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
V_B (Applied voltage: base)	range(0,0.1,0.7) range(0.72,0.02,0.9)	V

I7 From the Sweep type list, choose All combinations.

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

19 In the **Home** toolbar, click **= Compute**.

In order to select the appropriate initial values for the current driven problem, a solution with a base current close to 2 uA should be selected. A Global Evaluation of the base current from the Initialization Study allows such a solution to be identified.

RESULTS

Global Evaluation 1

- I In the Results toolbar, click (8.5) Global Evaluation.
- 2 In the Settings window for Global Evaluation, locate the Data section.
- 3 From the Dataset list, choose Initialization Study/Solution 2 (sol2).

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
semi.IO_2	μA	Terminal current

5 Click **= Evaluate**.

TABLE

I Go to the Table window.

The table shows that a base voltage of $V_B = 0.82$ V corresponds with a base current of $I_B = 2.8$ uA, which is suitably close to the desired 2 uA current for the current-driven study.

SEMICONDUCTOR (SEMI)

Base Current

- I In the Model Builder window, under Component I (compl)>Semiconductor (semi) click Base Current.
- 2 In the Settings window for Metal Contact, locate the Terminal section.

3 In the V_{init} text field, type **0.82**[V].

ADD STUDY

- I In the Home toolbar, click $\stackrel{\text{res}}{\longrightarrow}$ 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.

V_C SWEEP, $V_E=0$ V, FOR $I_B=2[UA]$

- I In the Model Builder window, click Study 3.
- 2 In the Settings window for Study, type V_C Sweep, V_E=0 V, for I_B=2[uA] in the Label text field.
- 3 Locate the Study Settings section. Clear the Generate default plots check box.

Step 1: Stationary

- I In the Model Builder window, under V_C Sweep, V_E=0 V, for I_B=2[uA] click Step I: Stationary.
- 2 In the Settings window for Stationary, locate the Physics and Variables Selection section.
- **3** Select the Modify model configuration for study step check box.
- 4 In the tree, select Component I (compl)>Semiconductor (semi)>Collector Current.
- 5 Click 🖉 Disable.
- 6 Locate the Values of Dependent Variables section. Find the Initial values of variables solved for subsection. From the Settings list, choose User controlled.
- 7 From the Method list, choose Solution.
- 8 From the Study list, choose Initialization Study, Stationary.
- 9 From the Parameter value (V_B (V),V_C (V)) list, choose 14: V_B=0.82 V, V_C=0 V.
- IO Locate the Study Extensions section. Select the Auxiliary sweep check box.
- II Click + Add.

12 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
I_B (Inward applied current: base)	2[uA]	A

I3 Click + Add.

I4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
V_C (Applied voltage: collector)	range(0,0.05,0.2)	V
	range(0.3,0.1,1.5)	

I5 From the Sweep type list, choose All combinations.

16 From the Run continuation for list, choose No parameter.

I7 From the **Reuse solution from previous step** list, choose **Yes**.

18 In the **Home** toolbar, click **= Compute**.

Plot the collector current as a function of collector voltage to give the I-V curve for an applied base current of 2 uA.

RESULTS

Common-Emitter Output Characteristics

- I In the Home toolbar, click 🔎 Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for ID Plot Group, type Common-Emitter Output Characteristics in the Label text field.
- 3 Locate the Data section. From the Dataset list, choose V_C Sweep, V_E=0 V, for I_B=2[uA]/Solution 3 (sol3).

Global I

- I Right-click Common-Emitter Output Characteristics 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
abs(semi.IO_3)	uA	Collector Current with I_B=2uA

4 In the Common-Emitter Output Characteristics toolbar, click 💿 Plot.

Common-Emitter Output Characteristics

- I In the Model Builder window, click Common-Emitter Output Characteristics.
- 2 In the Settings window for ID Plot Group, locate the Plot Settings section.
- 3 Select the x-axis label check box. In the associated text field, type Collector Voltage (V).

- 4 Select the **y-axis label** check box. In the associated text field, type Collector Current (uA).
- 5 Locate the Title section. From the Title type list, choose Manual.
- **6** In the **Title** text area, type Collector current as a function of collector voltage.
- 7 Locate the Legend section. Clear the Show legends check box.
- 8 In the Common-Emitter Output Characteristics toolbar, click Plot.
 Finally, the current density for each kind of carrier can be visualized using a 3D arrow plot.

Current Density

- I In the Home toolbar, click 🚛 Add Plot Group and choose 3D Plot Group.
- 2 In the Settings window for 3D Plot Group, type Current Density in the Label text field.
- 3 Locate the Data section. From the Dataset list, choose V_C Sweep, V_E=0 V, for I_B=2[uA]/Solution 3 (sol3).

Arrow Volume 1

- I Right-click Current Density and choose Arrow Volume.
- 2 In the Settings window for Arrow Volume, locate the Expression section.
- 3 In the X component text field, type semi.JnX.
- 4 In the Y component text field, type semi.JnY.
- 5 In the Z component text field, type semi.JnZ.
- 6 Locate the Arrow Positioning section. Find the X grid points subsection. In the Points text field, type 10.
- 7 Find the Y grid points subsection. In the Points text field, type 10.
- 8 Find the Z grid points subsection. In the Points text field, type 10.
- 9 Locate the Coloring and Style section. From the Arrow length list, choose Logarithmic.

IO From the **Color** list, choose **Black**.

Arrow Volume 2

- I Right-click Arrow Volume I and choose Duplicate.
- 2 In the Settings window for Arrow Volume, locate the Expression section.
- 3 In the X component text field, type semi.JpX.
- 4 In the Y component text field, type semi.JpY.
- 5 In the Z component text field, type semi.JpZ.

6 Locate the Coloring and Style section. From the Color list, choose White.

To aid in understanding the current flow throughout the device it is useful to add a slice plot of the voltage to highlight the emitter-base and base-collector junctions.

Slice 1

- I In the Model Builder window, right-click Current Density and choose Slice.
- 2 Click the View button in the Graphics toolbar.
- 3 In the Settings window for Slice, locate the Plane Data section.
- 4 From the Plane list, choose ZX-planes.
- 5 In the Planes text field, type 1.
- 6 Locate the Expression section. In the Expression text field, type V.
- 7 In the Current Density toolbar, click 💽 Plot.

The value of V_C for which the final plot group is plotted can be changed in order to investigate the operation of the bipolar transistor. To do this, click on **Current Density** in the Model Builder, locate the **Data** section of the **3D plot group** panel and change the value of **Parameter value** (**V_C**).

At $V_C = 0$ V the electron and hole currents flow in unison from the base contact to both the collector and emitter contacts. This is expected, as the device is being driven by a base current. The net collector current is very small as the electron and hole currents are nearly balanced.

At $V_C = 1.5$ V the device is operating in the saturation regime. The hole current flows mainly from the base to the emitter and the electron current flows mainly from the collector to the emitter. This results in a large net current at the collector contact.