Created in COMSOL Multiphysics 6.1



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 simple n-p-n bipolar transistor simulation. For this particular model, the current voltage characteristics in the static common-emitter configuration are computed and the common emitter current gain is determined.

Introduction

The invention of the Bipolar Transistor at Bell labs in the late 1940s was arguably the most significant technological development of the 20th century. Bipolar transistors were the workhorses of the first integrated circuits and consequently have had an enormous impact on the modern world. Although bipolar transistors have largely been replaced by field-effect devices in modern integrated circuits, they are still important, particularly in high-frequency applications.

The bipolar transistor is basically a three-terminal device whose resistance between two terminals is controlled by the third. In this example, we are going to show the static characteristics of a n-p-n bipolar transistor in common-emitter mode — that is, when the base and collector voltages are measured relative to the grounded emitter. Figure 1 shows the biasing configuration of a n-p-n transistor in this configuration.



Figure 1: A n-p-n transistor in common-emitter configuration.

In the common-emitter mode, the base terminal of the transistor serves as the input, the collector is the output, and the emitter is common to both (in our case grounded). N-p- n transistors in common-emitter configuration give an inverted output and can have a high gain which is a strong function of the base current.

The model presented here shows how to perform a detailed characterization of a bipolar transistor in the common-emitter configuration.

Model Definition

The structure is half of the cross section of a simplified n-p-n bipolar transistor; see Figure 2. The model uses the following dimensions: an emitter contact length of 1.2/ 2 μ m, a base contact length of 0.3 μ m separated from the emitter contact by 0.35 μ m, and a collector contact length of 2.5/2 μ m. The total depth of the transistor is 1 μ m with a width of 2.5/2 μ m. The modeled doping profile is shown in Figure 2. 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: Top: The cross-section of the modeled transistor. The differently doped regions are labeled. The red line shows the axis of symmetry used to simplified the model geometry. Bottom: A line graph, taken along the axis of symmetry, showing the doping concentration. The labeled regions (n+, n, p) correspond with those in the geometry cross-section.

This model uses several studies to characterize the DC response of the bipolar transistor current output under both voltage and current driven configurations. A Semiconductor Initialization study step is also used, which automatically refines the default mesh where

the gradient of the semiconductor doping profile is large. The refined mesh is then used throughout the rest of the modeling sequence.

First, a study sweeps the base voltage with a fixed collector voltage. This allows the currents at all three terminals to be plotted as a function of $V_{\rm BE}$ 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 ($I_{\rm C}/I_{\rm E}$), 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 remaining studies are used to calculate the collector current as a function of the collector voltage for two different values for the base current. This is achieved using two studies that sweep the collector voltage, one with a fixed base current of $I_B = 2 \mu A$ and one with a fixed base current of $I_B = 10 \mu A$. These simulations are current driven because they are performed with a set base current applied. When solving current driven problems in COMSOL it is often necessary to provide an initial condition which is calculated from a suitable voltage driven study. In order to generate the required initial conditions for these studies a voltage to 0, which is the initial value it has in the current driven studies, and sweeps the base voltage. The solution which generates base currents closest to 2 μA is then used as the initial condition for the collector voltage sweeps.

Results and Discussion

Figure 3 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 software's sign convention: outward current from the semiconductor material is positive and inward current is negative. The figure also shows that the current is conserved. This can be seen as the difference between the emitter and collector currents, represented as magenta circles, matches the base current, shown in green. Thus the current is conserved, as

$$I_B = -I_E - I_C$$

Figure 4 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 5 shows 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 6 shows the collector current as a function of collector voltage for two different values of base current. This figure shows how a small base current can be used to control a larger collector current by changing the resistance between the emitter and collector terminals.

Figure 7 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 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 3: Terminal currents as a function of the base-emitter voltage (V_{BE}) for a fixed collector-emitter voltage $(V_{CE} = 0.5 V)$.



Figure 4: Gummel plot showing the magnitude of the collector and base current as a function of the base voltage.



Figure 5: Current gain as a function of collector current for a fixed base voltage of $V_{\rm BE}$ =0.5 V.



Figure 6: Plot of collector current vs. collector voltage for $I_B = 2 \mu A$ and $I_B = 10 \mu A$. Note that varying the base current controls the resistance between the emitter and collector.



Figure 7: 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 which 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

Modeling Instructions

From the File menu, choose New.

NEW

In the New window, click Solution Model Wizard.

MODEL WIZARD

I In the Model Wizard window, click 🤏 2D.

- 2 In the Select Physics tree, select Semiconductor>Semiconductor (semi).
- 3 Click Add.
- 4 Click \bigcirc Study.
- 5 In the Select Study tree, select Preset Studies for Selected Physics Interfaces> Semiconductor Initialization.
- 6 Click 🗹 Done.

GLOBAL DEFINITIONS

Parameters 1

- I In the Model Builder window, under Global Definitions click Parameters I.
- 2 In the Settings window for Parameters, locate the Parameters section.
- **3** Click **b** Load from File.
- **4** Browse to the model's Application Libraries folder and double-click the file bipolar_transistor_parameters.txt.

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**.

Rectangle 1 (r1)

- I In the **Geometry** toolbar, click **Rectangle**.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type w_BJT/2.
- **4** In the **Height** text field, type d_BJT.
- 5 Locate the Position section. In the y text field, type -d_BJT.

Define the emitter ohmic contact.

Point I (ptl)

- I In the **Geometry** toolbar, click **Point**.
- 2 In the Settings window for Point, locate the Point section.
- **3** In the **x** text field, type w_cE.

Define the base ohmic contact.

Point 2 (pt2)

I In the **Geometry** toolbar, click • **Point**.

- 2 In the Settings window for Point, locate the Point section.
- 3 In the x text field, type w_BJT/2-w_cB.
- 4 Click 🟢 Build All Objects.

Load the semiconductor material properties for silicon.

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.

SEMICONDUCTOR (SEMI)

- I In the Model Builder window, under Component I (compl) click Semiconductor (semi).
- 2 In the Settings window for Semiconductor, locate the Thickness section.
- **3** In the *d* text field, type 1_BJT.

Semiconductor Material Model I

- I In the Model Builder window, under Component I (compl)>Semiconductor (semi) click Semiconductor Material Model I.
- 2 In the Settings window for Semiconductor Material Model, locate the Model Input section.
- **3** In the *T* text field, type T0.

Add the doping profiles. First the epitaxial layer background doping.

Analytic Doping Model I

- 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 Impurity section. From the Impurity type list, choose Donor doping (n-type).
- **5** In the N_{D0} text field, type N_epi.

Analytic Doping Model 2

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 **Uniform Region** section. Specify the r_0 vector as

0[um]	х

-d_E Y

- **6** In the *W* text field, type $w_BJT/2$.
- **7** In the *D* text field, type d_E .
- 8 Locate the Impurity section. In the N_{A0} text field, type N_B+N_epi.
- **9** Locate the **Profile** section. In the d_i text field, type d_E.

IO From the N_b list, choose Donor concentration (semi/adml).

Analytic Doping Model 3

- 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 Uniform Region section. In the W text field, type w_E/2-d_E.
- 6 Locate the Impurity section. From the Impurity type list, choose Donor doping (n-type).
- 7 In the N_{D0} text field, type N_E+N_B.
- 8 Locate the **Profile** section. In the d_i text field, type d_E.
- **9** In the N_b text field, type N_B.

Analytic Doping Model 4

- 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 **Uniform Region** section. Specify the r_0 vector as

0[um] X

-d_BJT-d_C Y

6 In the *W* text field, type $w_BJT/2$.

- 7 In the *D* text field, type d_C.
- 8 Locate the Impurity section. From the Impurity type list, choose Donor doping (n-type).
- **9** In the N_{D0} text field, type N_C.
- **IO** Locate the **Profile** section. In the d_j text field, type 1.3*d_C.
- II From the N_b list, choose Donor concentration (semi/adml).

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.

Emitter 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_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 5 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 2 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.

Add an additional boundary condition which specify the current for the base contact. Depending on the requirements of each study, either the voltage or the current boundary condition will be disabled.

Base Current

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

Generate a suitable mesh using a **Semiconductor Initialization** study. This study automatically refines the mesh around the gradient of the dopant concentration, which results in a fine mesh in the regions of the p-n junctions.

First set the element size of the default mesh to Course then view the initial mesh generated by the physics induced sequence. This sequence generates a mesh which is finer near the metal contact boundaries.

MESH I

- I In the Model Builder window, under Component I (compl) click Mesh I.
- 2 In the Settings window for Mesh, locate the Physics-Controlled Mesh section.
- 3 From the Element size list, choose Coarse.
- 4 Click 📗 Build All.

The default physics induced mesh will look like this:



13 | BIPOLAR TRANSISTOR

Now generate the refined mesh, which is automatically created by the Semiconductor Initialization study.

MESH REFINEMENT STUDY

- I In the Model Builder window, click Study I.
- 2 In the Settings window for Study, type Mesh Refinement Study in the Label text field.
- 3 Locate the Study Settings section. Clear the Generate default plots check box.
- **4** In the **Home** toolbar, click **= Compute**.

View the refined mesh.

LEVEL I ADAPTED MESH I

- I In the Model Builder window, expand the Component I (compl)>Meshes node.
- 2 Right-click Component I (compl)>Meshes>Level I Adapted Mesh I and choose Build All.

The refined mesh should look like this:



Add a Stationary Study to perform a sweep of the base voltage.

ADD STUDY

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

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

- I In the Model Builder window, click Study 2.
- 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.

Before computing the full solution, get the initial solution in order to plot the doping profile to be sure it agrees with what you want to simulate.

- 3 Locate the Study Settings section. Clear the Generate default plots check box.
- **4** In the Study toolbar, click $\underset{t=0}{\bigcup}$ Get Initial Value.

RESULTS

ID Plot Group I

- I In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for ID Plot Group, locate the Data section.
- 3 From the Dataset list, choose V_B Sweep, V_C=0.5 V, V_E=0 V/Solution 5 (sol5).
- 4 From the Time selection list, choose First.

Line Graph I

I Right-click ID Plot Group I and choose Line Graph.

Select the center boundary and plot the logarithm of the absolute value of the signed dopant concentration.

- 2 Select Boundary 1 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. In the associated text field, type log10(abs(Ndoping)).

Plot the doping profile along -y to display the profile from the center of the emitter to the collector.

- 6 Locate the x-Axis Data section. From the Parameter list, choose Reversed arc length.
- 7 In the ID Plot Group I toolbar, click 💽 Plot.

Doping Profile Center

- I In the Model Builder window, click ID Plot Group I.
- 2 In the Settings window for ID Plot Group, click to expand the Title section.
- **3** From the **Title type** list, choose **None**.
- 4 Locate the Plot Settings section. Select the x-axis label check box.
- **5** Select the **y-axis label** check box.
- 6 In the x-axis label text field, type Reversed arc length (um).
- 7 In the y-axis label text field, type log10(abs(Ndoping (1/cm³))).
- 8 In the Label text field, type Doping Profile Center.
- **9** In the **Doping Profile Center** toolbar, click **I** Plot.

Define the sweep for the full base voltage (V_B) study.

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

```
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 Click to expand the Study Extensions section. Select the Auxiliary sweep check box.
- 7 Click + Add.
- **8** In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
V_B (Applied voltage: base)	range(0,0.025,1.1)	V

Decrease the relative tolerance to ensure that very small currents are calculated accurately.

Solver Configurations

In the Model Builder window, expand the V_B Sweep, V_C=0.5 V, V_E=0 V> Solver Configurations node.

Solution 5 (sol5)

- I In the Model Builder window, expand the V_B Sweep, V_C=0.5 V, V_E=0 V> Solver Configurations>Solution 5 (sol5) node, then click Stationary Solver I.
- 2 In the Settings window for Stationary Solver, locate the General section.
- **3** In the **Relative tolerance** text field, type 1.0E-10.

Step 1: Stationary In the **Home** toolbar, click **= Compute**.

RESULTS

Doping Profile Center

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.

ID Plot Group 2

- I In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for ID Plot Group, locate the Data section.
- 3 From the Dataset list, choose V_B Sweep, V_C=0.5 V, V_E=0 V/Solution 5 (sol5).

Global I

- I Right-click ID Plot Group 2 and choose Global.
- In the Settings window for Global, click Replace Expression in the upper-right corner of the y-Axis Data section. From the menu, choose Component I (compl)>Semiconductor> Terminals>semi.l0_l Terminal current A.
- 3 Click Add Expression in the upper-right corner of the y-Axis Data section. From the menu, choose Component I (comp1)>Semiconductor>Terminals>semi.10_2 Terminal current A.
- 4 Click Add Expression in the upper-right corner of the y-Axis Data section. From the menu, choose Component I (comp1)>Semiconductor>Terminals>semi.I0_3 Terminal current A.
- 5 Locate the y-Axis Data section. 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

6 In the ID Plot Group 2 toolbar, click 🗿 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 ID Plot Group 2 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 Circle.
- 7 From the **Positioning** list, choose **Interpolated**.
- 8 In the Number text field, type 30.

I_E, I_B and I_C as a function of V_BE

- I In the Model Builder window, click ID Plot Group 2.
- 2 In the Settings window for ID Plot Group, locate the Title section.
- 3 From the Title type list, choose None.
- 4 Locate the Plot Settings section. Select the x-axis label check box.
- 5 Select the y-axis label check box.
- **6** In the **x-axis label** text field, type V_BE (V).
- 7 In the y-axis label text field, type I (mA).
- 8 Locate the Legend section. From the Position list, choose Lower left.
- 9 In the ID Plot Group 2 toolbar, click 💿 Plot.

IO In the **Label** text field, type I_E, I_B and I_C as a function of V_BE.

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.

ID Plot Group 3

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

- 2 In the Settings window for ID Plot Group, locate the Data section.
- 3 From the Dataset list, choose V_B Sweep, V_C=0.5 V, V_E=0 V/Solution 5 (sol5).

Global I

- I Right-click ID Plot Group 3 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)	Α	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_BE

- I In the Model Builder window, click ID Plot Group 3.
- 2 In the Settings window for ID Plot Group, locate the Legend section.
- 3 From the Position list, choose Upper left.
- 4 Locate the Title section. From the Title type list, choose Manual.
- 5 In the Title text area, type Gummel Plot.
- 6 Locate the Plot Settings section.
- 7 Select the x-axis label check box. In the associated text field, type Base Voltage (V).
- 8 Select the y-axis label check box. In the associated text field, type Current (A).
- 9 Click the **y-Axis Log Scale** button in the **Graphics** toolbar.

Restrict the data range to exclude very small current values.

- IO Locate the Data section. From the Parameter selection (V_B) list, choose Manual.
- II In the Parameter indices (1-45) text field, type range(17,1,45).
- 12 In the 1D Plot Group 3 toolbar, click 💿 Plot.

IS In the Label text field, type Gummel Plot, I_C and I_B as a function of V_BE.

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.

ID Plot Group 4

- I In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for ID Plot Group, locate the Data section.
- 3 From the Dataset list, choose V_B Sweep, V_C=0.5 V, V_E=0 V/Solution 5 (sol5).
- 4 Locate the Axis section. Select the x-axis log scale check box.

Global I

- I Right-click ID Plot Group 4 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 ID Plot Group 4.
- 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 current (A).
- **4** Select the **y-axis label** check box. In the associated text field, type Current Gain. Restrict the data range to exclude very small current values.
- 5 Locate the Data section. From the Parameter selection (V_B) list, choose Manual.
- 6 In the Parameter indices (1-45) text field, type range(17,1,45).
- 7 In the ID Plot Group 4 toolbar, click 💽 Plot.
- 8 In the Label text field, type Current Gain.

The collector current as a function of the collector voltage can be controlled by applying a base current. In this way, the output characteristics of the device can be controlled by an input current from, say, a sensing circuit. This effect can be seen by plotting the collector current as a function of the collector voltage for two different values of the base current.

As the required studies are current driven it is helpful to first perform a voltage driven initialization study in order to generate suitable initial conditions. The initialization study will sweep the base voltage with the collector and emitter voltage both set to 0 V. The subsequent sweeps of the collector voltage in the current driven studies will begin from $V_C = 0 V$. A suitable solution from the initialization study, in which the base current from the voltage driven study is close to the initial base current chosen for the current driven study, can then be selected.

ADD STUDY

I In the Home toolbar, click $\stackrel{\sim}{\sim}$ 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 3.
- 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 Locate the Study Extensions section. Select the Auxiliary sweep check box.
- 7 From the Sweep type list, choose All combinations.
- 8 Click + Add.
- **9** In the table, enter the following settings:

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

10 Click + Add.

II 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

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

I3 In the **Home** toolbar, click **= Compute**.

To investigate the base current as a function of base voltage a Global Evaluation can be used to tabulate the results from the initialization study.

RESULTS

Global Evaluation I

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 6 (sol6).
- 4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
semi.IO_2	A	Terminal current

5 Click **= Evaluate**.

TABLE

I Go to the Table window.

The first current driven sweep of the collector voltage will use a base current of 2 uA. The table shows that a base voltage of 0.8 V results in a current which is close to 2 uA. Therefore, select the corresponding solution as the initial condition.

Also, set the **Initial voltage** for the base contact to the same value (0.8 V).

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 **0.8**[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 4.

- 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**, click to expand the **Values of Dependent Variables** section.
- **3** Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.
- 4 From the Method list, choose Solution.
- 5 From the Study list, choose Initialization Study, Stationary.
- 6 From the Parameter value (V_B (V),V_C (V)) list, choose 13: V_B=0.8 V, V_C=0 V.
- 7 Locate the Study Extensions section. Select the Auxiliary sweep check box.
- 8 From the Sweep type list, choose All combinations.
- 9 Click + Add.

IO In the table, enter the following settings:

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

II Click + Add.

12 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) range(0.3,0.1,1.5)	V

I3 From the **Run continuation for** list, choose **No parameter**.

I4 From the **Reuse solution from previous step** list, choose **Yes**.

I5 In the **Home** toolbar, click **= Compute**.

To investigate the effect that a larger base current has on the response of the collector current to the collector voltage sweep a similar study will be performed with a base current of 10 uA.

The study is the same as the previous one, but the base current (I_B) is set to a value of 10 uA instead of 2 uA.

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.

V_C SWEEP, $V_E=0$ V, FOR $I_B=I0[UA]$

- I In the Model Builder window, click Study 5.
- 2 In the Settings window for Study, type V_C Sweep, V_E=0 V, for I_B=10[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=10[uA] click Step I: Stationary.
- 2 In the Settings window for Stationary, locate the Values of Dependent Variables section.
- **3** Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.
- 4 From the Method list, choose Solution.
- 5 From the Study list, choose Initialization Study, Stationary.
- 6 From the Parameter value (V_B (V),V_C (V)) list, choose 15: V_B=0.84 V, V_C=0 V.
- 7 Locate the Study Extensions section. Select the Auxiliary sweep check box.
- 8 From the Sweep type list, choose All combinations.
- 9 Click + Add.

IO In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
I_B (Inward applied current: base)	10[uA]	A

II Click + Add.

12 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
V_C (Applied voltage:	range(0,0.05,0.4)	V
collector)	range(0.5,0.1,1.5)	

I3 From the **Run continuation for** list, choose **No parameter**.

I4 From the **Reuse solution from previous step** list, choose **Yes**.

Again, set the **Initial voltage** for the base contact to the same value as the selected initial condition (0.84 V).

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 **0.84**[V].

V_C SWEEP, $V_E=0$ V, FOR $I_B=I0[UA]$

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

Plotting both curves on the same graph allows for direct comparison.

RESULTS

Common-emitter output characteristics

- I In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.
- 2 Right-click ID Plot Group 5 and choose Rename.
- **3** In the **Rename ID Plot Group** dialog box, type Common-emitter output characteristics in the **New label** text field.
- 4 Click OK.

Global I

- I Right-click Common-emitter output characteristics and choose Global.
- 2 In the Settings window for Global, locate the Data section.
- 3 From the Dataset list, choose V_C Sweep, V_E=0 V, for I_B=2[uA]/Solution 7 (sol7).

4 Locate the y-Axis Data section. In the table, enter the following settings:

Expression	Unit	Description
abs(semi.IO_3)	uA	Collector Current I_B=2uA

5 Click to expand the Legends section. From the Legends list, choose Manual.

6 In the table, enter the following settings:

Legends I_B=2 uA

Global 2

- I In the Model Builder window, right-click Common-emitter output characteristics and choose Global.
- 2 In the Settings window for Global, locate the Data section.
- 3 From the Dataset list, choose V_C Sweep, V_E=0 V, for I_B=10[uA]/Solution 8 (sol8).
- 4 Locate the y-Axis Data section. In the table, enter the following settings:

Expression	Unit	Description
abs(semi.IO_3)	uA	Collector Current I_B=10 uA

- 5 Locate the Legends section. From the Legends list, choose Manual.
- 6 In the table, enter the following settings:

Legends

I_B=10 uA

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 Title section.
- 3 From the Title type list, choose Manual.
- **4** In the **Title** text area, type Collector Current as a function of Collector Voltage.
- 5 Locate the Plot Settings section.
- 6 Select the x-axis label check box. In the associated text field, type Collector Voltage (V).
- 7 Select the **y-axis label** check box. In the associated text field, type Collector Current (uA).

- 8 Locate the Legend section. From the Position list, choose Middle right.
- 9 In the Common-emitter output characteristics toolbar, click 🗿 Plot.

Finally, the current density for each kind of carrier can be visualized using a 2D arrow plot.

Voltage & Current Density

- I In the Home toolbar, click 🚛 Add Plot Group and choose 2D Plot Group.
- 2 In the Settings window for 2D Plot Group, type Voltage & 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 7 (sol7).

Surface 1

- I Right-click Voltage & Current Density and choose Surface.
- 2 In the Settings window for Surface, locate the Expression section.
- 3 In the Expression text field, type V.
- 4 Click to expand the Quality section. From the Resolution list, choose No refinement.
- 5 From the Smoothing list, choose Everywhere.
- 6 In the Voltage & Current Density toolbar, click 💿 Plot.

Arrow Surface 1

- I In the Model Builder window, right-click Voltage & Current Density and choose Arrow Surface.
- 2 In the Settings window for Arrow Surface, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I (compl)> Semiconductor>Currents and charge>Electron current>semi.JnX,semi.JnY Electron current density.
- 3 Locate the Coloring and Style section. From the Arrow length list, choose Logarithmic.
- 4 From the Color list, choose Black.

Arrow Surface 2

- I Right-click Arrow Surface I and choose Duplicate.
- 2 In the Settings window for Arrow Surface, 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 Locate the Coloring and Style section. From the Color list, choose White.

Voltage & Current Density

- I In the Model Builder window, click Voltage & Current Density.
- 2 In the Settings window for 2D Plot Group, click to expand the Title section.
- 3 From the Title type list, choose Manual.
- 4 In the Title text area, type Voltage and Current Density.
- 5 In the Voltage & Current Density toolbar, click 💽 Plot.
- 6 Click the 🕂 Zoom Extents button in the Graphics toolbar.

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 **Voltage and Current Density** in the Model Builder, locate the **Data** section of the 2D 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.