Created in COMSOL Multiphysics 6.2

3D Analysis of a Bipolar Transistor

This model shows how to set up a 3D simulation of a NPN 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 two current driven studies are 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.

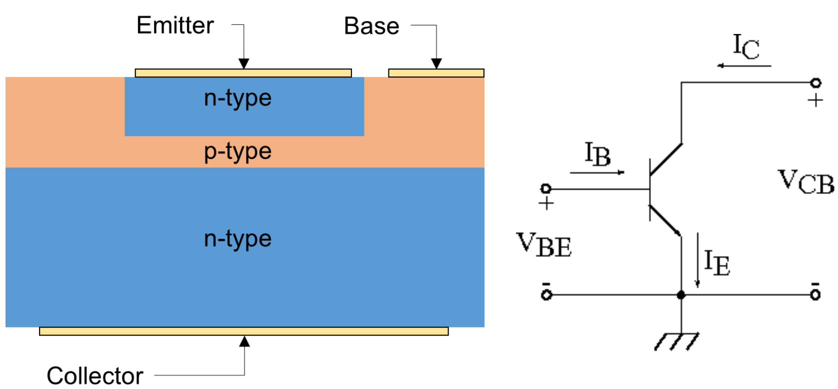


Figure 1: Left: Simplified cross section through a bipolar transistor showing the structure of the device. Right: Circuit diagram showing the common emitter configuration.

A bipolar transistor consists of three regions known as the emitter, base, and collector. In an NPN 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 base–collector 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 fixed currents 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.

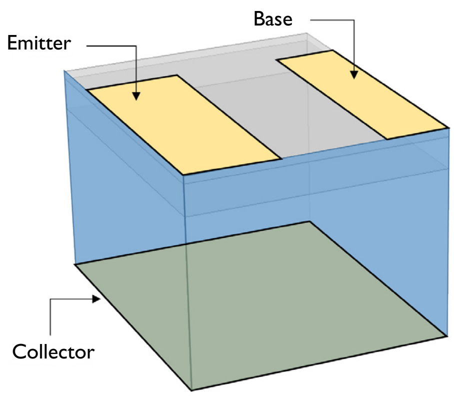


Figure 2: Model geometry, the symmetry planes are highlighted in blue and the boundaries to which the three electric contacts are applied are labeled.

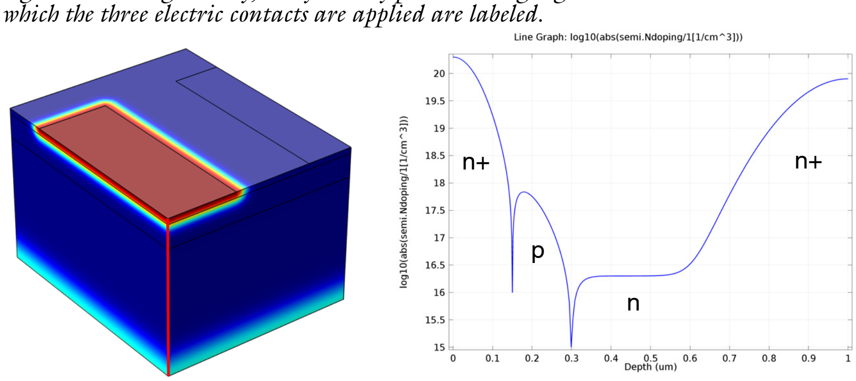


Figure 3: Dopant distribution for the bipolar transistor device. Left: Volume plot showing the total net dopant concentration, the emitter region can be clearly seen in red; the boundary between the base and collector is not apparent due to the large magnitude of the concentration in the n+ regions. 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 physics and studies settings in this model are exactly the same as in the 2D model Bipolar Transistor.

Results and Discussion

Figure 4 displays the current at each terminal as a function of the base–emitter voltage (V_{BE}) for a fixed collector–emitter voltage (V_{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

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

The results are in good agreement with the 2D model Bipolar Transistor. Note that when comparing 2D and 3D results, the total current should be scaled with reference to the effective thickness.

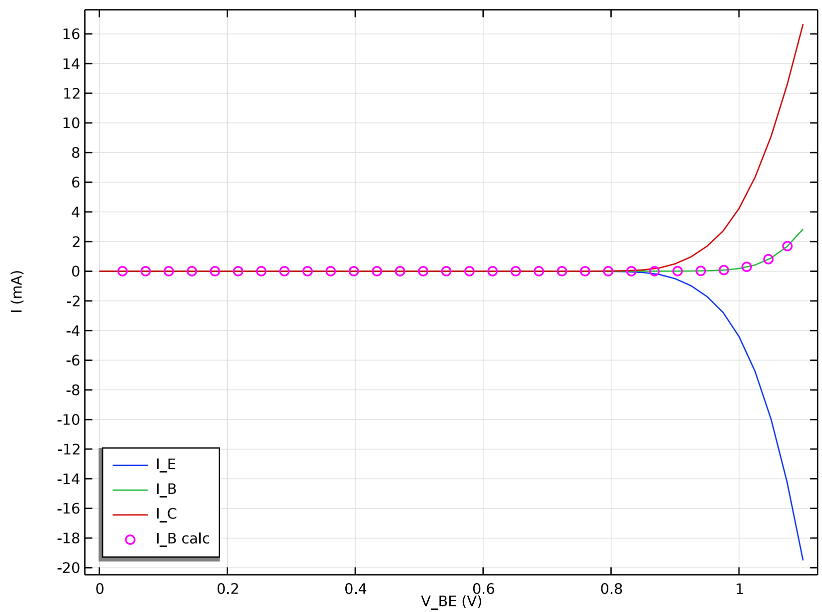


Figure 4: Terminal currents as a function of the base–emitter voltage (V_{BE}) for a fixed collector–emitter voltage (V_{CE} = 0.5 V).

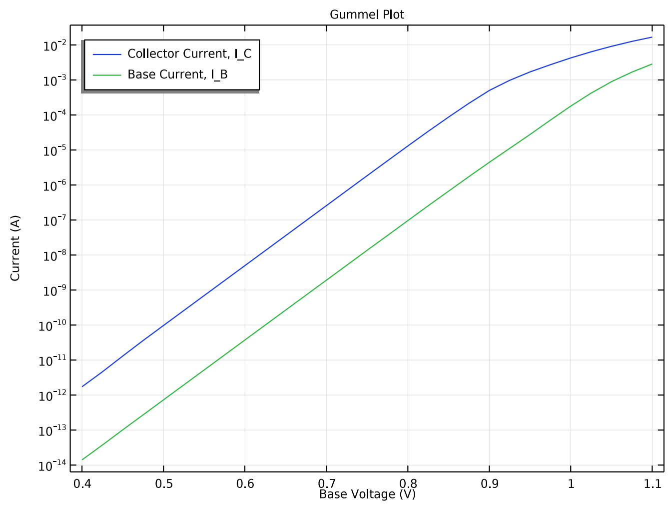


Figure 5: Gummel plot showing the magnitude of the collector and base current as a function of the base voltage.

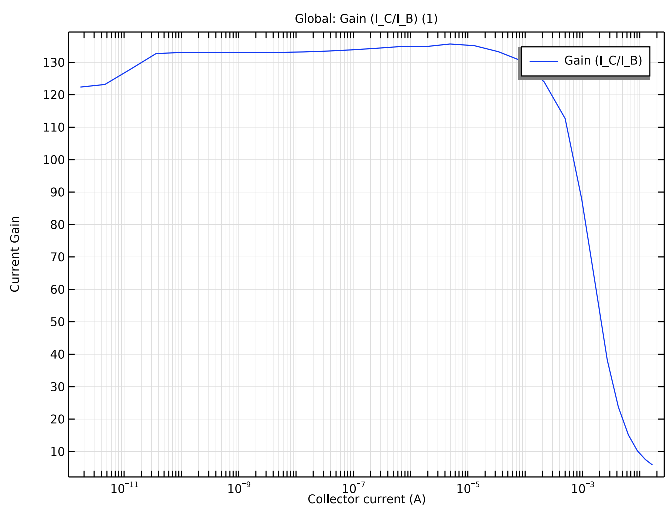


Figure 6: Current gain as a function of collector current for a fixed base voltage of V_{CE}=0.5 V.

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_{CE}=0.5 V.

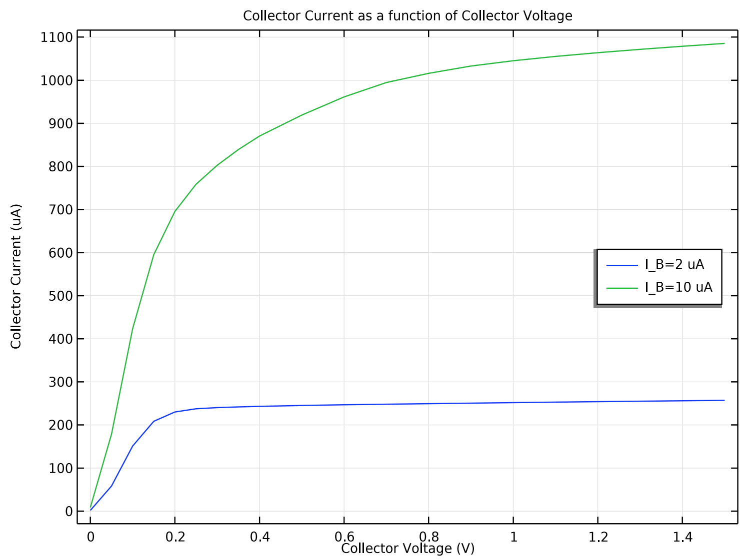


Figure 7: Plot of collector current vs. collector voltage for I_B = 2 μA and I_B = 10 μA. Note that varying the base current controls the resistance between the emitter and collector.

Figure 7 shows the collector current as a function of collector voltage for two different values of base current 2 μA and 10 μA. 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.

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.

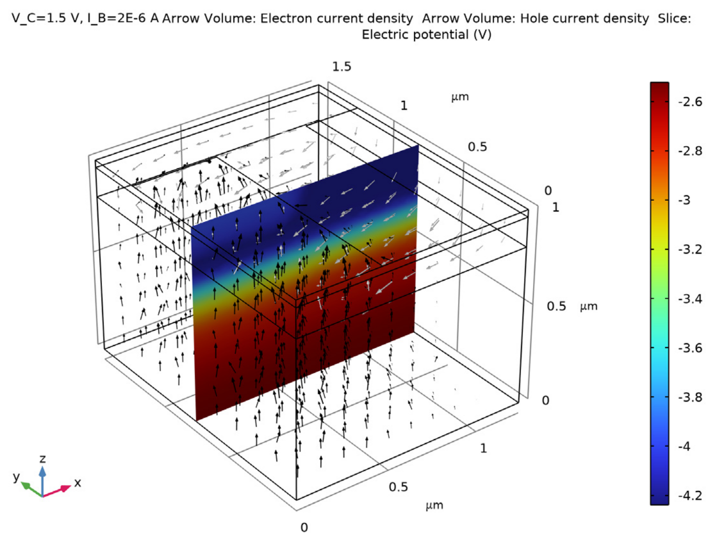


Figure 8: Voltage and current density for $I_B = 2\ \mu A$ and $V_{CE} = 1.5\ V$. The color shows the voltage and the arrows show the current density for electrons (black) and holes (white). Note that the hole current flows from the base to the emitter and does not enter the lower n-doped region, whilst 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

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1. In the **Model Wizard** window, click **3D**.
2. In the **Select Physics** tree, select **Semiconductor>Semiconductor (semi)**.
3. Click **Add**.
4. Click **Study**.
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`.

1. 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 1

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
2. In the **Settings** window for **Geometry**, locate the **Units** section.
3. From the **Length unit** list, choose **µm**.

Block 1 (blk1)

1. 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 `l_BJT/2`.
5. In the **Height** text field, type `d_BJT`.

Block 2 (blk2)

1. 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 `l_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 1 (wp1)

1 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 **Go to Plane Geometry**.

Work Plane 1 (wp1)>Rectangle 1 (r1)

1 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 `l_E/2-2*d_E`.

Work Plane 1 (wp1)>Rectangle 2 (r2)

1 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 `l_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 1** and choose **Build All**.

ADD MATERIAL

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

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

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

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

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

MATERIALS

Si - Silicon (mat1)

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

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

1 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[um]	X
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 `l_cB/2`.
9 In the H text field, type `d_E`.

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

11 From the N_b list, choose **Donor concentration (semi/adm1)**.

12 In the **Label** text field, type `Base p Doping`.

Emitter n Doping

1 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 `l_E/2-2*d_E`.

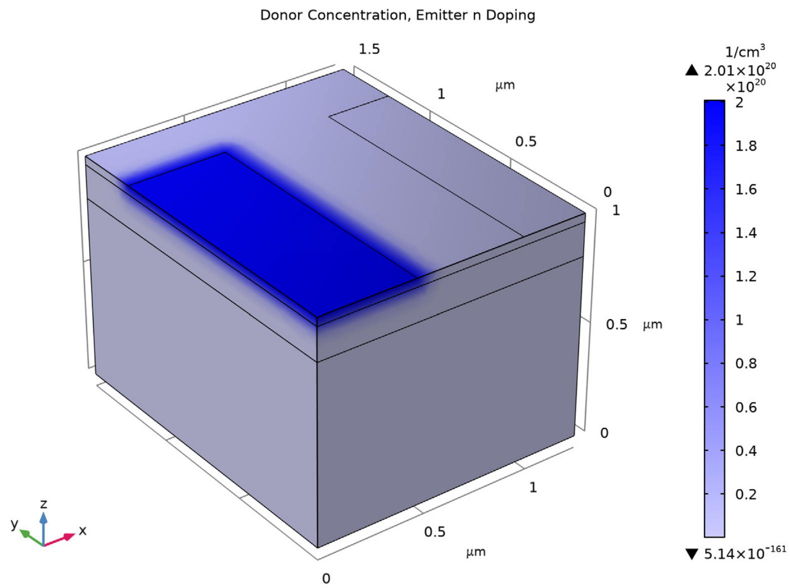
10 Locate the **Profile** section. In the d_j text field, type `d_E`.

11 In the N_b text field, type `N_B`.

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

Here you can plot the preview of the doping profile for the selected feature.

13 Click **Plot Doping Profile for Selected** in the window toolbar.



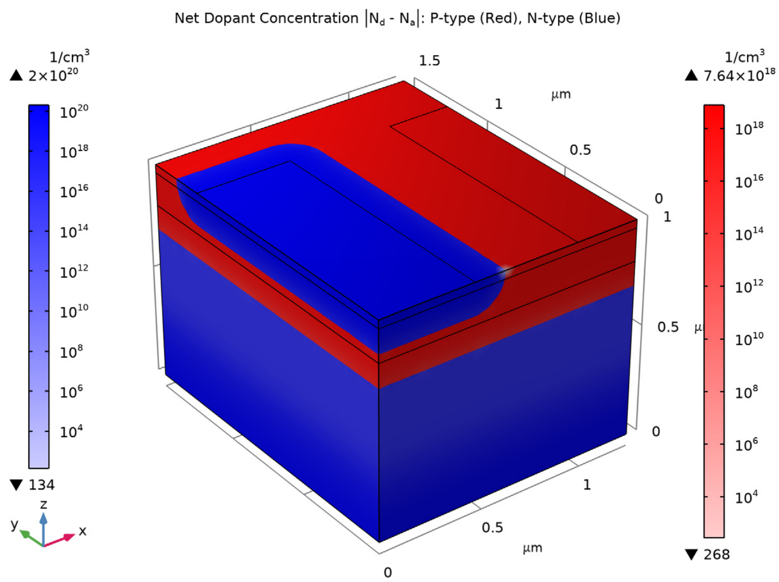
Add another n-type region for the collector.

Collector n Doping

1. In the **Physics** toolbar, click **Domains** and choose **Analytic Doping Model**.
2. In the **Settings** window for **Analytic Doping Model**, locate the **Domain Selection** section.
3. From the **Selection** list, choose **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 `l_BJT/2`.
9. Locate the **Profile** section. In the d_j text field, type `1.3*d_C`.
10. From the N_b list, choose **Donor concentration (semi/adm1)**.
11. In the **Label** text field, type `Collector n Doping`.

   Here you can plot the preview of the doping profile for all the features.

12 Click **Plot Net Doping Profile for All** in the window toolbar.



Add a **Trap-Assisted Recombination** feature to the model.

Trap-Assisted Recombination 1

1. 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

1. 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

1. 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

1 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

1 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

1 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 1

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

1 In the **Mesh** toolbar, click **Boundary** and choose **Free Triangular**.

2 Select Boundaries 10, 11, and 15 only.

Size

1 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 1

1 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 1

1 Right-click **Swept 1** 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

1 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 1

1 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

1 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 1

1 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`.

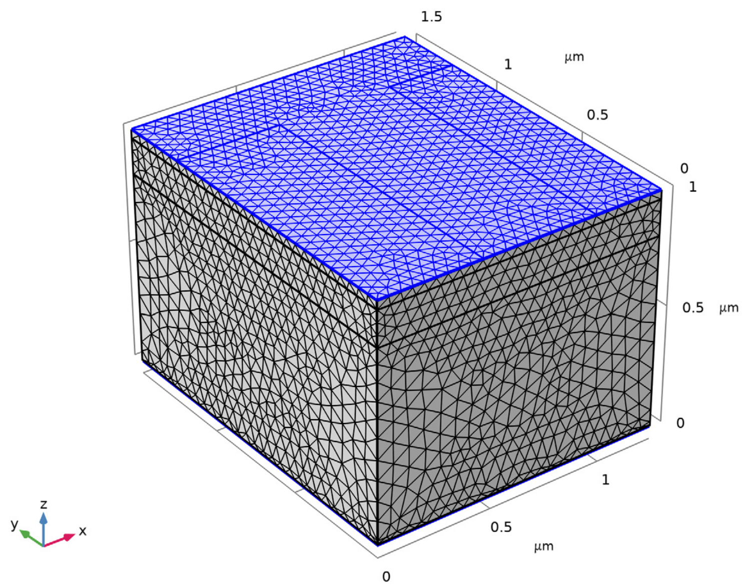
6 From the **Growth rate** list, choose **Exponential**.

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

8 Select the **Reverse direction** check box.

9 In the **Model Builder** window, right-click **Mesh 1** 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

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

1. In the **Model Builder** window, under **V_B Sweep, V_C=0.5 V, V_E=0 V** click **Step 1: 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 1 (comp1)>Semiconductor (semi)>Base Current**.

5 Click **Disable**.

6 In the tree, select **Component 1 (comp1)>Semiconductor (semi)>Collector Current**.

7 Click **Disable**.

8 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.

9 Click **Add**.

10 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.

11 In the **Study** toolbar, click **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

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

2 In the **Settings** window for **1D Plot Group**, type `Doping Profile` in the **Label** text field.

Line Graph 1

1 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

1 In the **Model Builder** window, click **Doping Profile**.

2 In the **Settings** window for **1D 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 **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

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

2 In the **Settings** window for **1D Plot Group**, type `I_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 1

1 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.I0_1	mA	I_E
semi.I0_2	mA	I_B
semi.I0_3	mA	I_C

4 In the **I_E, I_B and I_C as a Function of V_B** 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

1 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

1 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 **1D 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

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2 In the **Settings** window for **1D Plot Group**, type `Gummel Plot, I_C and I_B as a function of V_B` in the **Label** text field.

Global 1

1 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)`	`A`	`Base Current, I_B`

Gummel Plot, I_C and I_B as a function of V_B

1 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 **1D 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**.

10 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.

11 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)`.

13 In the **Gummel Plot, I_C and I_B as a function of V_B** toolbar, click **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

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

2 In the **Settings** window for **1D Plot Group**, type `Current Gain` in the **Label** text field.

Global 1

1 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

1 In the **Model Builder** window, click **Current Gain**.

2 In the **Settings** window for **1D 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.

10 Locate the **Data** section. From the **Parameter selection (V_B)** list, choose **Manual**.

11 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

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

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

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

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

INITIALIZATION STUDY

1 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

1 In the **Model Builder** window, under **Initialization Study** click **Step 1: 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 1 (comp1)>Semiconductor (semi)>Base Current**.

5 Click **Disable**.

6 In the tree, select **Component 1 (comp1)>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**.

10 From the **Study** list, choose **V_B Sweep, V_C=0.5 V, V_E=0 V, Stationary**.

11 From the **Parameter value (V_B (V))** list, choose **0 V**.

12 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

13 Click **Add**.

14 In the table, enter the following settings:

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

15 Click **Add**.

16 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

17 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

1 In the **Results** toolbar, click **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.I0_2`	μA	`Terminal current`

5 Click **Evaluate**.

TABLE 1

1 Go to the **Table 1** 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

1 In the **Model Builder** window, under **Component 1 (comp1)>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

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

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

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

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

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

1 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

1 In the **Model Builder** window, under **V_C Sweep, V_E=0 V, for I_B=2[uA]** click **Step 1: 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 1 (comp1)>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**.

10 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

11 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

13 Click Add.

14 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

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

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

17 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

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

2 In the **Settings** window for **1D 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 1

1 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.I0_3)`	uA	`Collector Current with I_B=2uA`

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

Common-Emitter Output Characteristics

1 In the **Model Builder** window, click **Common-Emitter Output Characteristics**.

2 In the **Settings** window for **1D 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

1. 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

1. 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**.
10. From the **Color** list, choose **Black**.

Arrow Volume 2

1. Right-click **Arrow Volume 1** 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

1 In the **Model Builder** window, right-click **Current Density** and choose **Slice**.

2 Click the **Go to Default 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.