Created in COMSOL Multiphysics 6.0

Reverse Recovery of a PIN Diode

This tutorial simulates the turn-off transient (reverse recovery) of a simple PIN diode with an inductive load, loosely based on the book "Fundamentals of Power Semiconductor Devices" by B.J. Baliga (p. 256, 2008 edition; Ref. 1). Unlike the book, which assumes an initial constant current ramp rate followed by an abrupt bottoming out of the device voltage at the supplied reverse voltage, this model uses the circuit capability of COMSOL Multiphysics to simulate the inductive load with a flyback diode in a more realistic fashion. The resulting time evolutions of the current, voltages, and carrier concentrations compare well to those shown in the book (Figs. 5.42–5.45).

Introduction

The PIN diode structure is an important building block for power electronic circuits. The process of switching the PIN rectifier from the on-state to the blocking state is referred to as the reverse recovery. The diode remains in the forward-biased mode for a short period of time after the applied voltage is switched, due to the large amount of stored charge carriers in the drift region during the on-state. This tutorial combines a simple 1D model of a PIN diode with a circuit of a voltage source, an inductive load, and a flyback diode to simulate the time evolutions of the current, voltages, and carrier concentrations.

Model Definition

The model simulates a diode of 80 μm length and 1 mm cross-section area. Important effects such as the Fletcher mobility model, Slotboom band gap narrowing, and Shockley-Read-Hall recombination are included. The device is grounded at the right endpoint and connected to the circuit at the left endpoint.

The circuit contains two voltage sources in parallel, but only one is activated at a time in the study steps. One voltage source is for the on-state and the other for the switching process. A **Global Equation** is used in the circuit to improve numerical stability, in the case when the time derivative of a dependent variable is used in the definition of a variable.

The **Events** interface is used to mark the abrupt changes in the slope of the applied voltage, so that the time-dependent solver can take appropriate actions.

The **Semiconductor Equilibrium** study step is used for the initial condition. Then the steady on-state is obtained by using a **Stationary** study step with the Auxiliary sweep, ramping the applied voltage from a small value up to the on-state voltage. Finally a **Time Dependent** study step is used to simulate the reverse recovery, first maintaining the applied voltage at the on-state value for 10 ns, then ramping it down to the reverse voltage in the next 10 ns, and then holding it there for the next 500 ns. The time axis is shifted so that at time = 0, the applied voltage has just been ramped down to the full reverse voltage.

Results and Discussion

Figure 1, Figure 2, and Figure 3 show the hole concentration at a few selected time points, the time evolution of the applied voltage and the device voltage, and the time evolution of the current, respectively. They exhibit the typical behavior as expected from the inductive load and the dissipation of the stored charge carriers in the drift region. The figures compare well with the corresponding figures in Ref. 1.

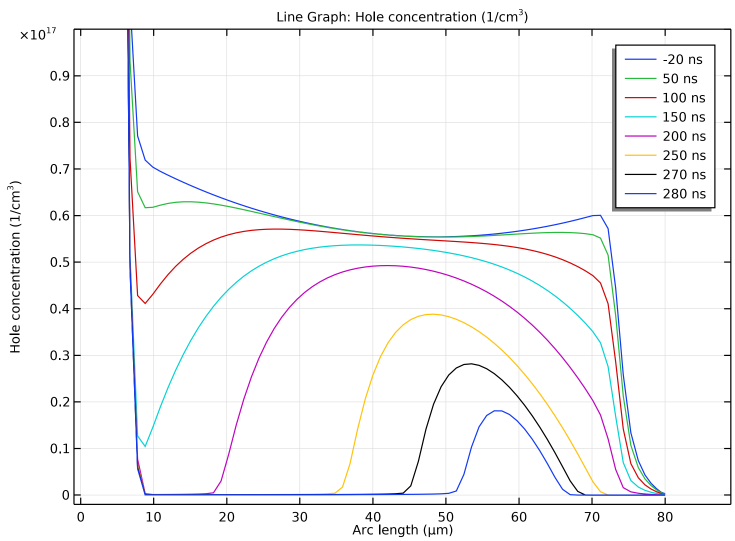


Figure 1: Hole concentration at a few selected time points.

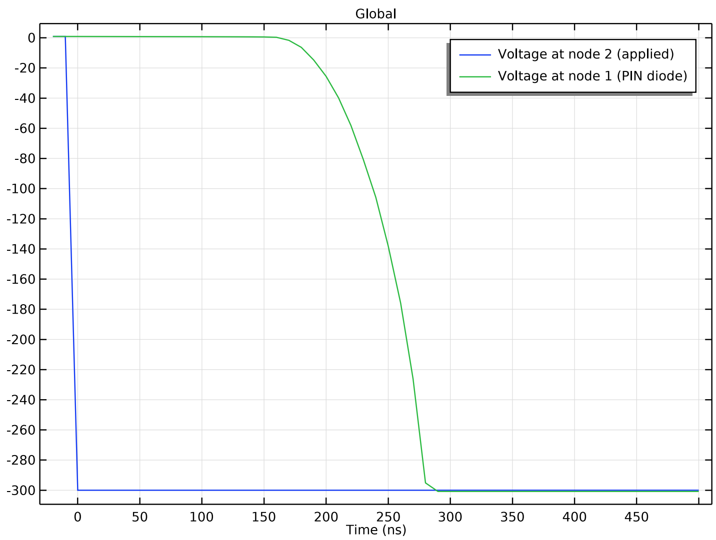


Figure 2: Time evolution of the applied voltage and the device voltage.

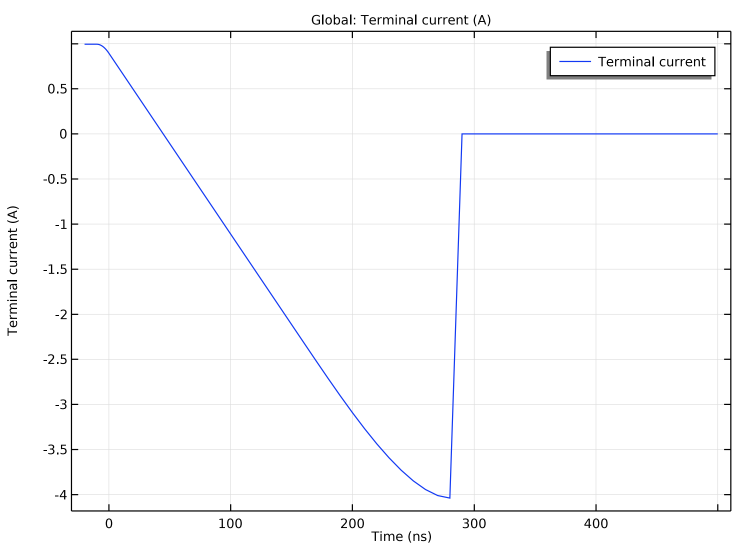


Figure 3: Time evolution of the current.

Reference

1. B.J. Baliga, *Fundamentals of Power Semiconductor Devices*, 2008 ed., Springer, pp. 242–243.

Application Library path: `Semiconductor_Module/Device_Building_Blocks/pin_reverse_recovery`

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 **1D**.

2 In the **Select Physics** tree, select **Semiconductor>Semiconductor (semi)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **AC/DC>Electrical Circuit (cir)**.

5 Click **Add**.

Add an **Events** interface to capture the abrupt change in the slope of the applied voltage.

6 In the **Select Physics** tree, select **Mathematics>ODE and DAE Interfaces>Events (ev)**.

7 Click **Add**.

8 Click **Study**.

The **Semiconductor Equilibrium** study can be used to obtain a good initial condition.

9 In the **Select Study** tree, select **Preset Studies for Some Physics Interfaces>Semiconductor Equilibrium**.

10 Click **Done**.

GEOMETRY 1

The Model Wizard exits and starts the COMSOL Desktop at the Geometry node. We can set the length scale here right away.

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**.

Import global parameters from a text file.

GLOBAL DEFINITIONS

Parameters 1

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

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `pin_reverse_recovery.txt`.

Build a simple 1D line interval of 80 um long.

GEOMETRY 1

Interval 1 (i1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Geometry 1** and choose **Interval**.

2 In the **Settings** window for **Interval**, locate the **Interval** section.

3 In the table, enter the following settings:

Coordinates (µm)
0
80

Create a ramp function for turning off the bias voltage.

GLOBAL DEFINITIONS

Ramp 1 (rm1)

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Ramp**.

2 In the **Settings** window for **Ramp**, locate the **Parameters** section.

3 Select the **Cutoff** check box.

Add silicon material.

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)

Enter the cross-section area for the 1D model.

SEMICONDUCTOR (SEMI)

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Semiconductor (semi)**.
2. In the **Settings** window for **Semiconductor**, locate the **Cross-Section Area** section.
3. In the A text field, type `area`.

Add a mobility model to account for the carrier-carrier scattering effect.

Semiconductor Material Model 1

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

Fletcher Mobility Model (C) 1

In the **Physics** toolbar, click **Attributes** and choose **Fletcher Mobility Model (C)**.

Note that the above step merely makes the mobility model available. To use it, we have to select it in the drop down menus in the Semiconductor Material Model node.

Semiconductor Material Model 1

1. In the **Model Builder** window, click **Semiconductor Material Model 1**.
2. In the **Settings** window for **Semiconductor Material Model**, locate the **Mobility Model** section.
3. From the μ_n list, choose **Electron mobility, Fletcher (semi/smm1/mmfl1)**.
4. From the μ_p list, choose **Hole mobility, Fletcher (semi/smm1/mmfl1)**.

   Add band gap narrowing effect.
5. Click to expand the **Band Gap Narrowing** section. From the **Band gap narrowing** list, choose **Slotboom**.

Set up doping - first the background doping.

Analytic Doping Model 1

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

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

3 From the **Selection** list, choose **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 `5e13[1/cm^3]`.

Then the P and N doping.

Geometric Doping Model 1

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

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

3 From the **Selection** list, choose **All domains**.

4 Locate the **Impurity** section. In the N_{A0} text field, type `1e19[1/cm^3]`.

5 Locate the **Profile** section. In the d_j text field, type `10[um]`.

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

Boundary Selection for Doping Profile 1

1 In the **Model Builder** window, expand the **Geometric Doping Model 1** node, then click **Boundary Selection for Doping Profile 1**.

2 Select Boundary 1 only.

Geometric Doping Model 2

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

2 In the **Settings** window for **Geometric 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 `1e19[1/cm^3]`.

6 Locate the **Profile** section. In the d_j text field, type `10[um]`.

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

Boundary Selection for Doping Profile 1

1 In the **Model Builder** window, expand the **Geometric Doping Model 2** node, then click **Boundary Selection for Doping Profile 1**.

2 Select Boundary 2 only.

Add ohmic contacts.

Metal Contact 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Metal Contact**.
2 Select Boundary 2 only.

Metal Contact 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Metal Contact**.
2 Select Boundary 1 only.
3 In the **Settings** window for **Metal Contact**, locate the **Terminal** section.
4 In the V_0 text field, type `V_circ`.

   The variable `V_circ` is not yet defined and turns to yellow colored. It will be defined later in the Electrical Circuit interface.

Add SRH recombination.

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**.
4 Locate the **Shockley-Read-Hall Recombination** section. From the τ_n list, choose **User defined**. In the associated text field, type `1[us]`.
5 From the τ_p list, choose **User defined**. In the associated text field, type `1[us]`.

Now set up the electrical circuit - first a "current source" representing the 1D PIN diode model. The bidirectional coupling between the electrical circuit and the semiconductor model is done in two parts: 1) The current of the current source is specified by the terminal current from the Semiconductor interface. 2) The voltage of the semiconductor metal contact is specified by the voltage of the current source.

ELECTRICAL CIRCUIT (CIR)

In the **Model Builder** window, under **Component 1 (comp1)** click **Electrical Circuit (cir)**.

Current Source 1 (I1)

1 In the **Electrical Circuit** toolbar, click **Current Source**.
2 In the **Settings** window for **Current Source**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	**Node names**
n	0

4 Locate the **Device Parameters** section. In the i_{src} text field, type `semi.I0_2`.

This completes part 1).

For part 2), turn on Advanced Physics Options to make Global Equation available. In certain situations where the time derivative of a dependent variable is used in the definition of a variable, it helps to create an intermediate variable using a Global Equation, as shown in the following steps.

5 Click the **Show More Options** button in the **Model Builder** toolbar.

6 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Equation-Based Contributions**.

7 Click **OK**.

Global Equations 1

1 In the **Electrical Circuit** toolbar, click **Global Equations**.

2 In the **Settings** window for **Global Equations**, locate the **Global Equations** section.

3 In the table, enter the following settings:

Name	**f(u,ut,utt,t) (1)**	**Initial value (u_0) (1)**	**Initial value (u_t0) (1/s)**	**Description**
`V_circ`	`cir.I1_v-V_circ`	0	0	

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog box, type `potential` in the text field.

6 Click **Filter**.

7 In the tree, select **Electromagnetics>Electric potential (V)**.

8 Click **OK**.

The Global Equation makes a copy of the voltage of the current source, `cir.I1_v`, into the intermediate variable `V_circ`, which has been used to specify the semiconductor metal contact voltage in a previous step. This completes part 2).

Add the load inductor and a generic flyback diode.

Inductor 1 (L1)

1 In the **Electrical Circuit** toolbar, click **Inductor**.

2 In the **Settings** window for **Inductor**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
n	1

4 Locate the **Device Parameters** section. In the L text field, type L0.

Diode 1 (D1)

1 In the **Electrical Circuit** toolbar, click **Diode**.

2 In the **Settings** window for **Diode**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	2
n	1

Add two voltage sources, one for the initial steady state and the other for the time-dependent study.

Voltage Source 1 (V1)

1 In the **Electrical Circuit** toolbar, click **Voltage Source**.

2 In the **Settings** window for **Voltage Source**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	2
n	0

4 Locate the **Device Parameters** section. In the v_{src} text field, type V0.

Voltage Source 2 (V2)

1 In the **Electrical Circuit** toolbar, click **Voltage Source**.

2 In the **Settings** window for **Voltage Source**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	2
n	0

4 Locate the **Device Parameters** section. In the v_{src} text field, type `Von+(Voff-Von)*rm1((t+10[ns])/10[ns])`.

   The voltage ramp is shifted 10 ns earlier so that at time = 0, the applied voltage is just ramped down to the full reverse voltage, `Voff`.

Add two events to mark the abrupt changes in the slope of the applied voltage, so that the time-dependent solver can take appropriate actions.

EVENTS (EV)

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

Explicit Event 1

1 In the **Physics** toolbar, click **Global** and choose **Explicit Event**.

2 In the **Settings** window for **Explicit Event**, locate the **Event Timings** section.

3 In the t_i text field, type `-10[ns]`.

Explicit Event 2

In the **Physics** toolbar, click **Global** and choose **Explicit Event**.

Exclude the Events interface and the 2nd voltage source from the initial study step.

STUDY 1

Step 1: Semiconductor Equilibrium

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

2 In the **Settings** window for **Semiconductor Equilibrium**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Events (ev)**.

4 Select the **Modify model configuration for study step** check box.

5 In the tree, select **Component 1 (Comp1)>Electrical Circuit (Cir)>Voltage Source 2 (v2)**.

6 Click **Disable**.

Add a stationary study step to ramp the applied voltage from a small value up to the on-state voltage `Von`. Exclude the Events interface and the 2nd voltage source from this study.

Stationary

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

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Events (ev)**.

4 Select the **Modify model configuration for study step** check box.

5 In the tree, select **Component 1 (Comp1)>Electrical Circuit (Cir)>Voltage Source 2 (v2)**.

6 Click **Disable**.

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

8 Click **Add**.

9 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
V0 (Applied DC voltage)	`0.1 0.3 0.5 0.7 Von`	V

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

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

Add a time dependent study step using the 2nd voltage source to start the applied voltage at the on state voltage `Von` for 10 ns, ramp it down to the reverse voltage `Voff` in the next 10 ns, and keep it at `Voff` for 500 ns afterward. Set the tolerance to 1e-5 for better accuracy.

Time Dependent

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

2 In the **Settings** window for **Time Dependent**, 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)>Electrical Circuit (Cir)>Voltage Source 1 (V1)**.

5 Click **Disable**.

6 Locate the **Study Settings** section. From the **Time unit** list, choose **ns**.

7 In the **Output times** text field, type `range(-20,1,0) range(10,10,500)`.

8 From the **Tolerance** list, choose **User controlled**.

9 In the **Relative tolerance** text field, type `1e-5`.

For study steps 2 and 3, where the solution from the previous study step is used as the initial condition, we can use **Initial value based** scaling of the dependent variables to obtain better error estimate and better convergence.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Dependent Variables 2**.

3 In the **Settings** window for **Dependent Variables**, locate the **Scaling** section.

4 From the **Method** list, choose **Initial value based**.

5 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)** click **Dependent Variables 3**.

6 In the **Settings** window for **Dependent Variables**, locate the **Scaling** section.

7 From the **Method** list, choose **Initial value based**.

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

Plot the hole concentration at a few selected time points to see its evolution.

RESULTS

Hole Concentration

1 In the **Model Builder** window, right-click **Carrier Concentrations (semi)** and choose **Duplicate**.

2 Click the **y-Axis Log Scale** button in the **Graphics** toolbar.

3 In the **Settings** window for **1D Plot Group**, type `Hole Concentration` in the **Label** text field.

4 Locate the **Data** section. From the **Time selection** list, choose **Manual**.

5 In the **Time indices (1-71)** text field, type `1 range(26,5,46) 48 49`.

6 Locate the **Plot Settings** section. Clear the **y-axis label** check box.

Set the axis limits to the range of interest.

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

8 In the **x maximum** text field, type `89`.

9 In the **y minimum** text field, type `-2e15`.

10 In the **y maximum** text field, type `1e17`.

Electron Concentration

1 In the **Model Builder** window, expand the **Hole Concentration** node.

2 Right-click **Results>Hole Concentration>Electron Concentration** and choose **Delete**.

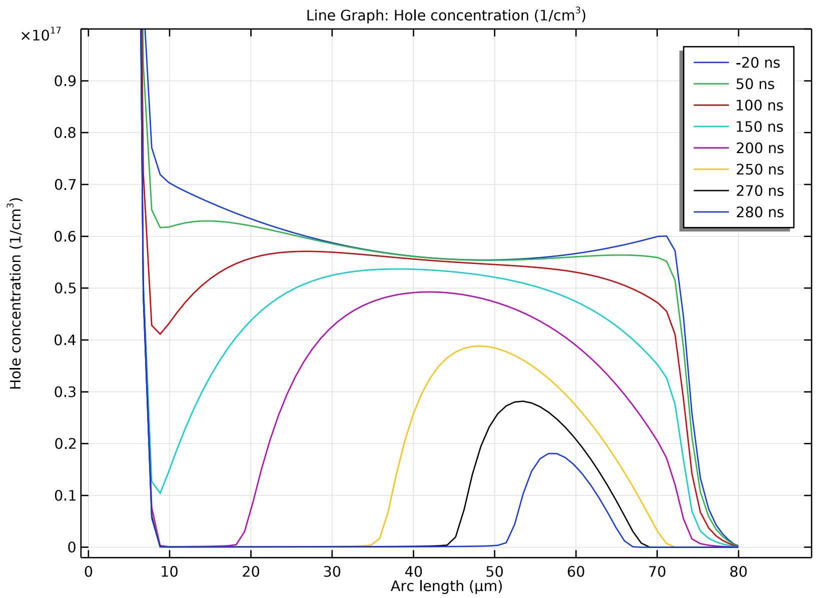
Hole Concentration

1 In the **Model Builder** window, under **Results>Hole Concentration** click **Hole Concentration**.

2 In the **Settings** window for **Line Graph**, click to expand the **Legends** section.

3 From the **Legends** list, choose **Automatic**.

4 In the **Hole Concentration** toolbar, click **Plot**.



Plot the time evolution of the applied voltage and the device voltage.

V(t)

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

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

Global 1

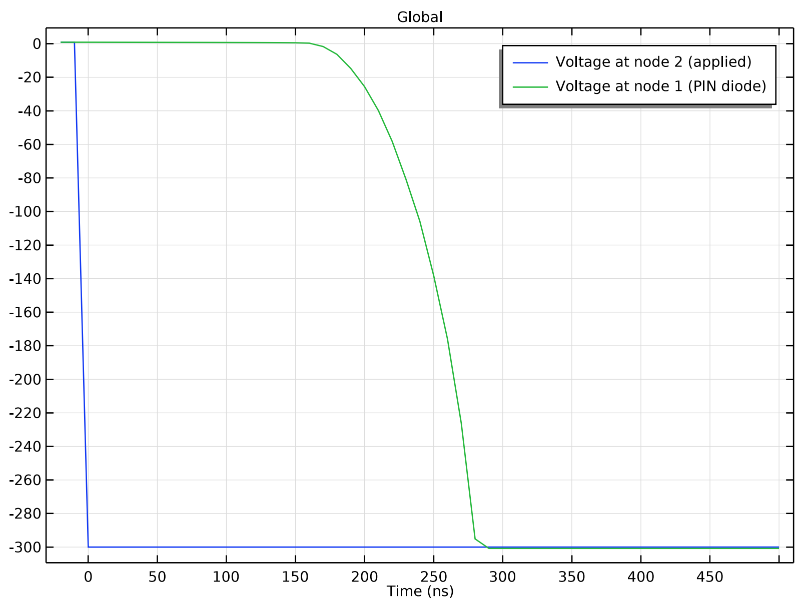
1 Right-click **V(t)** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`cir.v_2`	`V`	`Voltage at node 2 (applied)`
`cir.v_1`	`V`	`Voltage at node 1 (PIN diode)`

4 In the **V(t)** toolbar, click **Plot**.



Plot the time evolution of the current.

I(t)

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(t)` in the **Label** text field.

Global 1

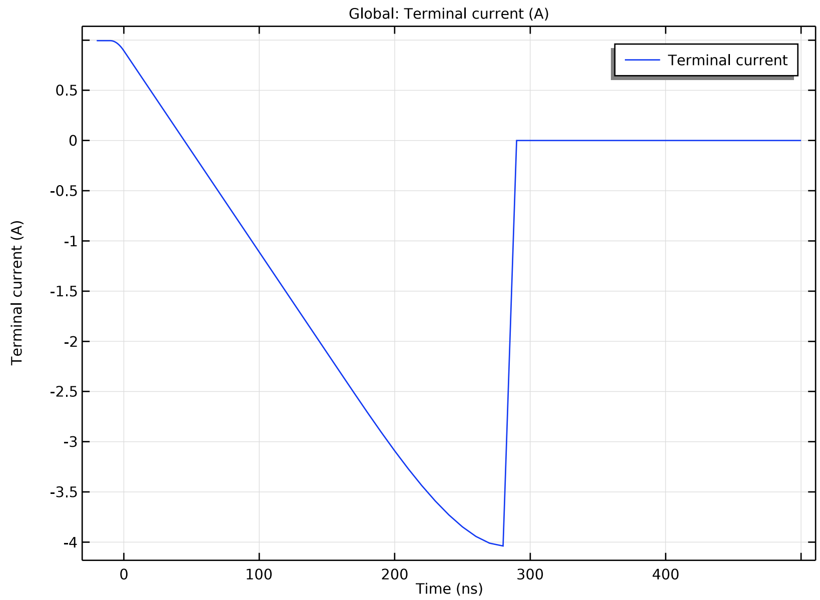
1 Right-click **I(t)** and choose **Global**.

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_2`	`A`	`Terminal current`

4 In the **I(t)** toolbar, click **Plot**.



Combine the voltage and current plots using 2 y-axes for the model thumbnail.

V(t) & I(t)

1 In the **Model Builder** window, right-click **V(t)** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `V(t) & I(t)` in the **Label** text field.

Global 1

In the **Model Builder** window, under **Results>I(t)** right-click **Global 1** and choose **Copy**.

Global 2

In the **Model Builder** window, right-click **V(t) & I(t)** and choose **Paste Global**.

V(t) & I(t)

1 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

2 From the **Title type** list, choose **None**.

3 Locate the **Plot Settings** section. Select the **y-axis label** check box.

4 In the associated text field, type `Applied and device voltage (V)`.

5 Select the **Two y-axes** check box.

6 In the table, select the **Plot on secondary y-axis** check box for **Global 2**.

7 Locate the **Legend** section. From the **Position** list, choose **Middle right**.

8 In the **V(t) & I(t)** toolbar, click **Plot**.