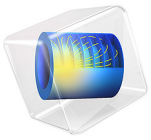
Created in COMSOL Multiphysics 6.0



Forward Recovery of a PIN Diode

This tutorial simulates the turn-on transient (forward recovery) of a simple PIN diode, based on the book "Fundamentals of Power Semiconductor Devices" by B.J. Baliga (p. 242, 2008 edition; Ref. 1). The diode is current driven with a constant ramp rate of $1{\cdot}10^9$, $2{\cdot}10^9$, and $1{\cdot}10^{10}$ A/cm^2/s and a steady state current density of 100 A/cm^2. The resulting time evolution of the device voltage and electron concentration compare well to those shown in the book (Figs. 5.30–5.31). For a more sophisticated example including band gap narrowing, carrier-carrier scattering, and external load circuit elements, see the tutorial *Reverse Recovery of a PIN Diode*.

Introduction

The PIN diode structure is an important building block for power electronic circuits. The process of switching the PIN rectifier from the off-state to the on-state is referred to as the forward recovery. During the turn-on time interval with a specified current input, the device voltage shows an initial spike, due to the amount of time required to accumulate extra charge carriers in the drift region for the diode to be fully turned on. This tutorial follows the approach in Ref. 1, where an idealized linearly ramped-on current input (with a specified max on-current) is used. For a more sophisticate example that uses the circuit capability of COMSOL Multiphysics to simulate the inductive load with a flyback diode in a more realistic fashion, see the tutorial *Reverse Recovery of a PIN Diode* That tutorial also includes band gap narrowing and carrier-carrier scattering effects.

Model Definition

The model simulates a diode of 80 μm length and 1 mm cross-section area. The background n-doping is $5{\cdot}10^{13}$/cm^3 and the peak n- and p-doping at the exterior boundaries is $1{\cdot}10^{19}$/cm^3. **Shockley-Read-Hall recombination** is included. The device is grounded at the right endpoint and current driven from the left endpoint.

A **Ramp** function with a **Cutoff** is used to specify the linear current ramp up to the max on-current. The **Events** interface is used to mark the abrupt change in the slope of the applied current at the end of the current ramp, so that the time-dependent solver can take appropriate actions.

The **Semiconductor Equilibrium** study step is used for the initial condition. Then a **Time Dependent** study step is used to simulate the forward recovery, with an **Auxiliary sweep** to study a few different current ramp rates.

Results and Discussion

Figure 1 and Figure 2 show the time evolution of the device voltage and the electron concentration at a few selected time points, respectively. They exhibit the typical behavior as expected from the initial accumulation of extra charge carriers in the drift region. The figures compare well with the corresponding figures in Ref. 1.

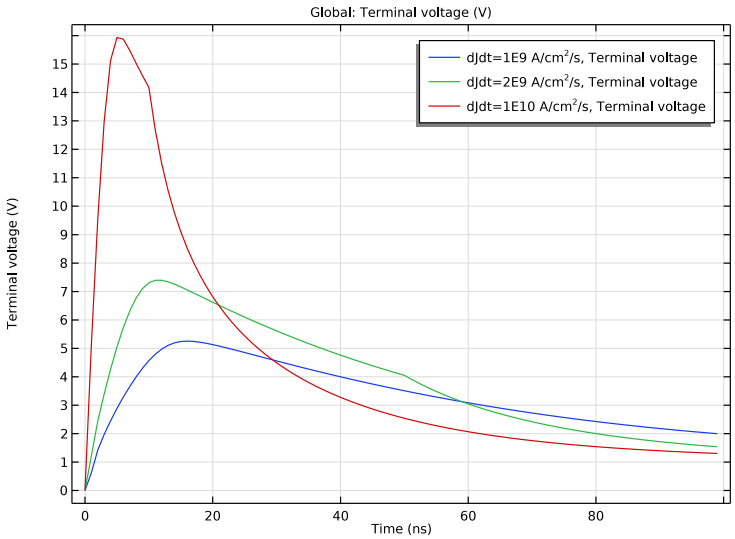


Figure 1: Time evolution of the device voltage.

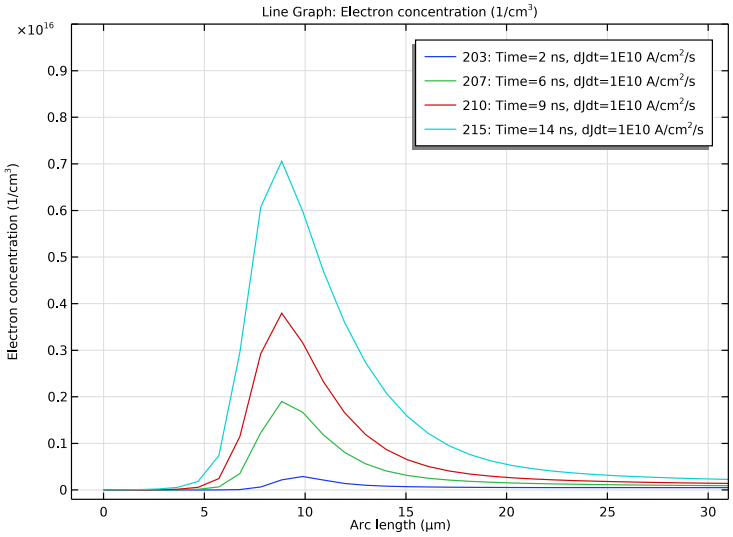


Figure 2: Time evolution of the electron concentration.

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_forward_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**.

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

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

5 Click **Add**.

6 Click **Study**.

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

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

8 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_forward_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

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`.

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 From the **Terminal type** list, choose **Current**.

5 In the I_0 text field, type `J_on*area*rm1(t/t_on)`.

The ramp function `rm1` is not yet defined and turns the expression to yellow colored. Let us define it now.

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 SRH recombination.

SEMICONDUCTOR (SEMI)

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]`.

Add an event to mark the end of the applied current ramp, 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 `t_on`.

Exclude the Events interface 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)**.

Add a time dependent study step with Auxiliary sweep, to simulate a few different current ramp rates.

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 **Study Settings** section.

3 From the **Time unit** list, choose **ns**.

4 In the **Output times** text field, type `range(0,1,99)`.

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

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

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
dJdt (Desired current density ramp rate)	`1e9 2e9 1e10`	`A/cm^2/s`

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

Plot the time evolution of the device voltage, to see the different amount of spiking for the different current ramp rates.

RESULTS

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

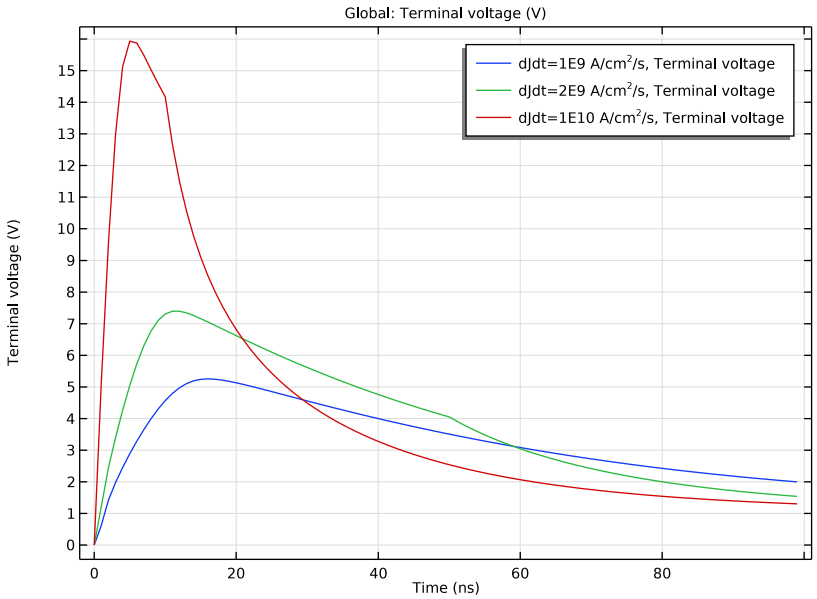
1 Right-click **V(t)** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`semi.VO_2`	`V`	`Terminal voltage`

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



Plot the electron concentration at a few selected time points, to see the initial buildup of the charge carriers in the drift region.

Electron 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 `Electron Concentration` in the **Label** text field.

4 Locate the **Data** section. From the **Parameter selection (dJdt)** list, choose **Last**.

5 From the **Time selection** list, choose **Manual**.

6 In the **Parameter indices (1-100)** text field, type `3 7 10 15`.

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

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

9 In the **x maximum** text field, type `31`.

10 In the **y minimum** text field, type `-2e14`.

11 In the **y maximum** text field, type `1e16`.

Electron Concentration

1 In the **Model Builder** window, expand the **Electron Concentration** node, then click **Electron Concentration**.

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

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

Hole Concentration

In the **Model Builder** window, under **Results>Electron Concentration** right-click **Hole Concentration** and choose **Delete**.

Electron Concentration

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

2 In the **Electron Concentration** toolbar, click **Plot**.