Created in COMSOL Multiphysics 6.1

P-N Diode Circuit

This model compares a full device level simulation with a lumped circuit model to simulate a half-wave rectifier.

Introduction

The p-n diode is of great importance in modern electronic applications. It is often used as a rectifier to convert alternative currents (AC) to direct currents (DC) by blocking either the positive or negative half of the AC wave. The present example simulates the transient behavior of a p-n diode used as the active component of a half-wave rectifier circuit -see Figure 1

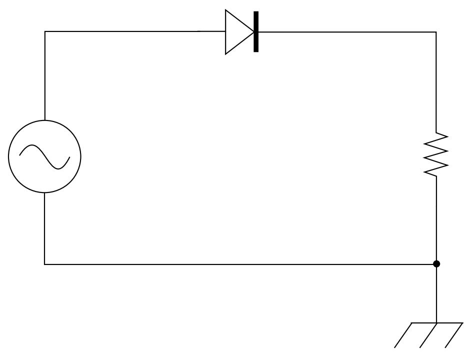


Figure 1: A basic half-wave rectifier circuit. An AC voltage source is connected to the anode of a p-n diode. The resistor represents the load of the circuit.

In this example, a full level device simulation is made by connecting a 2D meshed p-n junction to a circuit containing a sinusoidal source, a resistor, and a ground (the half-wave rectifier circuit is displayed in Figure 1). In order to validate the results, the outputs of the full device simulation are compared to the circuit response obtained using a large signal diode model (see the electric circuit).

Model Definition

Figure 2 shows the modeled device cross section and doping profile. The diode has a width of 10 µm and a depth of 7 µm. The length of the diode has been set to 10 µm (not meshed). A Shockley-Read-Hall recombination is also added to the model in order to simulate the type of recombination usually observed in indirect band-gap semiconductor such as silicon, which is the material used in this example. The meshed diode is connected to the half wave circuit using an ohmic terminal. For the large signal diode model, the saturation current and ideality factor have been set to values fitting the I-V curve of the modeled diode.

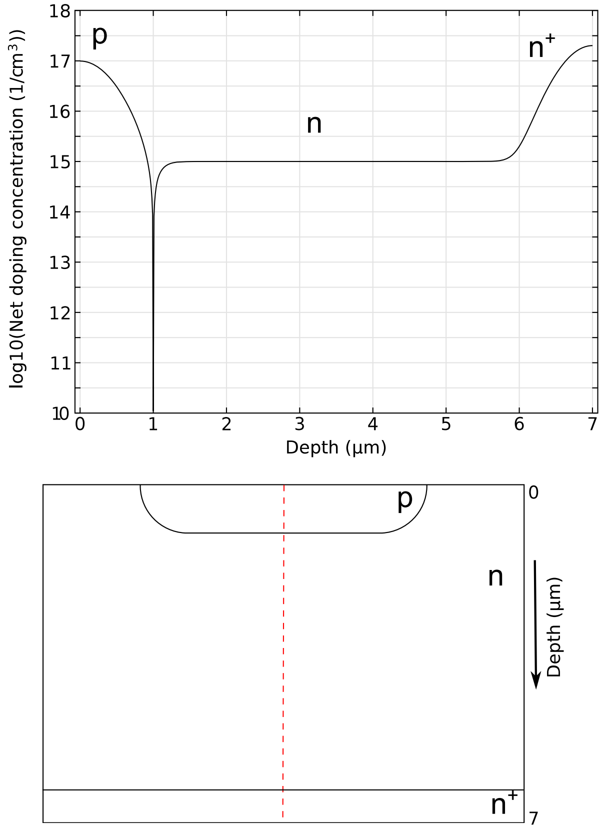


Figure 2: Top: net doping concentration along the symmetry line (center of the diode cross section). Bottom: cross section of the simulated device. To save computation time, only half of the diode is meshed, that is, the right side delimited by the axis of symmetry (red dashed line).

If the entire time history is of interest, then the time-dependent study should use a physical solution as the initial condition. Usually, this is done by adding a **Stationary** study step before the time dependent study step. In addition, it is sometimes necessary to turn off the **Consistent initialization** under the **Time Stepping** section of the settings window for the **Time-Dependent Solver** node under the **Solver Configurations** tree structure.

For simplicity, this example model has omitted all the above steps.

For a more detailed discussion on setting up transient studies, see the blog post

https://www.comsol.com/blogs/how-to-simulate-the-carrier-dynamics-in-semiconductor-devices/

and the two example models:

Forward Recovery of a PIN Diode (Application Library path **Semiconductor_Module/Device_Building_Blocks/pin_forward_recovery**)

Reverse Recovery of a PIN Diode (Application Library path **Semiconductor_Module/Device_Building_Blocks/pin_reverse_recovery**)

Results and Discussion

Figure 3 shows the output voltages obtained from both the full level simulation and large signal model. As expected from the reverse operation of a p-n diode, clipping occurs on the negative half of the wave in the diode.

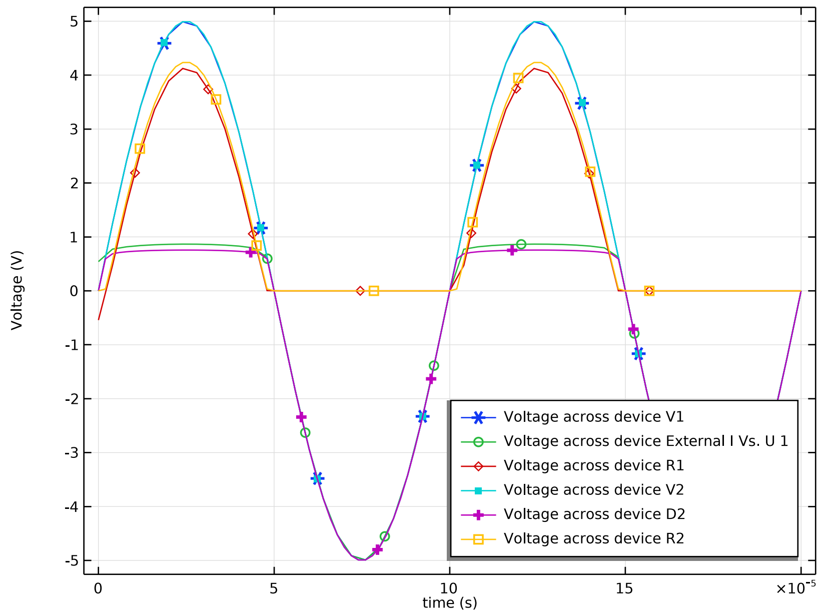


Figure 3: Output voltages obtained from both the full level simulation and large signal model. Voltages have been monitored at the source, diode, and load ends.

Application Library path: `Semiconductor_Module/Device_Building_Blocks/pn_diode_circuit`

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 **2D**.

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**.

6 Click **Study**.

7 In the **Select Study** tree, select **General Studies>Time Dependent**.

8 Click **Done**.

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 `pn_diode_circuit_parameters.txt`.

GEOMETRY 1

1 In the **Geometry** toolbar, click **Sketch**.

2 In the **Model Builder** window, click **Geometry 1**.

3 In the **Settings** window for **Geometry**, locate the **Units** section.

4 From the **Length unit** list, choose **µm**.

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Model Builder** window, click **Rectangle 1 (r1)**.

3 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

4 In the **Width** text field, type `w_diode/2`.

5 In the **Height** text field, type `d_diode`.

6 Locate the **Position** section. In the **y** text field, type `-d_diode`.

The doping profiles will be created in the semiconductor interface.However, in order to have a finer mesh in the junction vicinities, it is wise to create geometry objects defining the doping regions in the semiconducting material.

Rectangle 2 (r2)

1 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_anode/2+d_p`.

4 In the **Height** text field, type `d_p`.

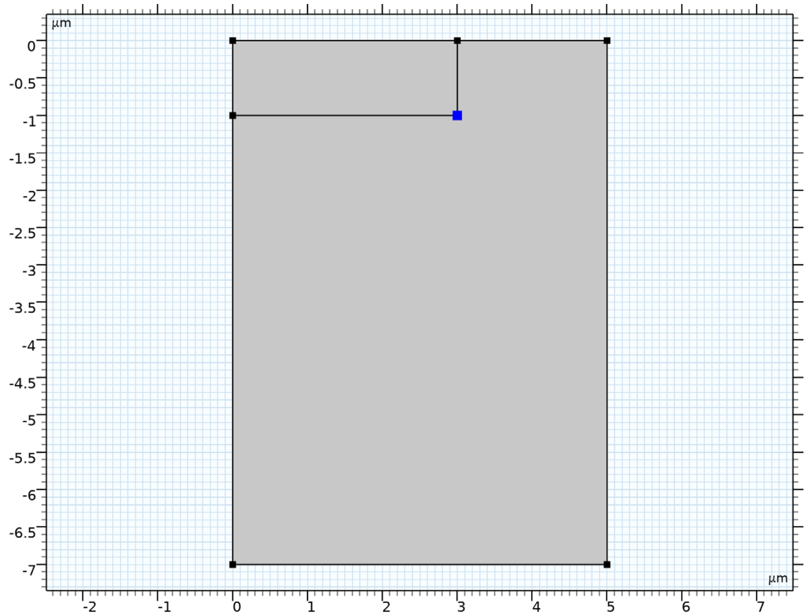
5 Locate the **Position** section. In the **y** text field, type `-d_p`.

Fillet 1 (fil1)

1 In the **Geometry** toolbar, click **Fillet**.

2 On the object **r2**, select Point 2 only.

It might be easier to select the correct point by using the **Selection List** window. To open this window, in the **Home** toolbar click **Windows** and choose **Selection List**. (If you are running the cross-platform desktop, you find **Windows** in the main menu.)



3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type `d_p`.

Point 1 (pt1)

1 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_anode/2`.

4 Click **Build All Objects**.

Load the semiconductor material properties for silicon.

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.

SEMICONDUCTOR (SEMI)

Analytic Doping Model 1

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 `Nd_back`.

Analytic Doping Model 2

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

0[um]	X
-d_diode	Y

6. In the W text field, type `w_diode/2`.
7. Locate the **Impurity** section. From the **Impurity type** list, choose **Donor doping (n-type)**.
8. In the N_{D0} text field, type `Nd_max`.
9. Locate the **Profile** section. From the N_b list, choose **Donor concentration (semi/adm1)**.

Analytic Doping Model 3

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 **Uniform Region** section. In the W text field, type `w_anode/2`.

6 In the D text field, type `d_p`.

7 Locate the **Impurity** section. In the N_{A0} text field, type `Na_max`.

8 Locate the **Profile** section. From the N_b list, choose **Donor concentration (semi/adm1)**.

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**.

Metal Contact 1

1 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 From the **Terminal type** list, choose **Circuit (current)**.

Metal Contact 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Metal Contact**.

2 Select Boundary 2 only.

ELECTRICAL CIRCUIT (CIR)

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

Resistor 1 (R1)

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

Use a 100 kOhm load resistor to limit the current in the circuit.

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

3 In the table, enter the following settings:

Label	Node names
p	1
n	0

4 Locate the **Device Parameters** section. In the R text field, type `100[kohm]`.

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. From the **Source type** list, choose **Sine source**.

5 In the v_{src} text field, type `Vac`.

6 In the f text field, type `f`.

External I vs. U 1 (IvsU1)

1 In the **Electrical Circuit** toolbar, click **External I vs. U**.

2 In the **Settings** window for **External I vs. U**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	2
n	1

4 Locate the **External Device** section. From the V list, choose **Terminal voltage (semi/mc1)**.

STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 Click **Range**.

4 In the **Range** dialog box, type `tmax/50` in the **Step** text field.

5 In the **Stop** text field, type `tmax`.

6 Click **Replace**.

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 **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

4 From the **Steps taken by solver** list, choose **Manual**.

5 In the **Time step** text field, type `tmax/100`.

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

RESULTS

Electron Concentration (semi)

By adding another circuit model, you can compare our coupled model with a full circuit model (using a large-signal diode model).

ADD PHYSICS

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

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

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

4 Click **Add to Component 1** in the window toolbar.

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

ELECTRICAL CIRCUIT 2 (CIR2)

Voltage Source 1 (V1)

1 Right-click **Component 1 (comp1)>Electrical Circuit 2 (cir2)** and choose **Voltage Source**.

2 In the **Settings** window for **Voltage Source**, type `V2` in the **Name** text field.

3 Locate the **Node Connections** section. In the table, enter the following settings:

Label	Node names
p	1
n	0

4 Locate the **Device Parameters** section. From the **Source type** list, choose **Sine source**.

5 In the v_{src} text field, type `Vac`.

6 In the f text field, type `f`.

Use the diode large scale model with the following parameters.

Diode 1 (D1)

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

2 In the **Settings** window for **Diode**, type D2 in the **Name** text field.

3 Locate the **Node Connections** section. In the table, enter the following settings:

Label	Node names
p	1
n	2

4 Locate the **Model Parameters** section. In the I_S text field, type I0.

5 In the N text field, type eta.

Resistor 1 (R1)

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

2 In the **Settings** window for **Resistor**, type R2 in the **Name** text field.

3 Locate the **Node Connections** section. In the table, enter the following settings:

Label	Node names
p	2
n	0

4 Locate the **Device Parameters** section. In the R text field, type 100[kohm].

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> Time Dependent**.

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

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

STUDY 2

Step 1: Time Dependent

1 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

2 Click **Range**.

3 In the **Range** dialog box, type tmax/100 in the **Step** text field.

4 In the **Stop** text field, type tmax.

5 Click **Replace**.

6 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

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

8 In the **Relative tolerance** text field, type `0.001`.

9 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check boxes for **Semiconductor (semi)** and **Electrical Circuit (cir)**.

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

RESULTS

Voltage probes

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **None**.

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

5 Locate the **Plot Settings** section.

6 Select the **x-axis label** check box. In the associated text field, type `time (s)`.

7 Select the **y-axis label** check box. In the associated text field, type `Voltage (V)`.

8 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

Global 1

1 Right-click **Voltage probes** and choose **Global**.

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

3 From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.

4 Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Electrical Circuit>Devices>V1>cir.V1_v - Voltage across device V1 - V**.

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

Expression	Unit	Description
`cir.V1_v`	V	Voltage across device V1
`cir.IvsU1_v`	V	Voltage across device External I Vs. U 1
`cir.R1_v`	V	Voltage across device R1

6 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

7 From the **Positioning** list, choose **Interpolated**.

Global 2

1 In the **Model Builder** window, right-click **Voltage probes** and choose **Global**.
2 In the **Settings** window for **Global**, locate the **Data** section.
3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.
4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`cir2.V2_v`	`V`	`Voltage across device V2`
`cir2.D2_v`	`V`	`Voltage across device D2`
`cir2.R2_v`	`V`	`Voltage across device R2`

5 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.
6 From the **Positioning** list, choose **Interpolated**.
7 In the **Voltage probes** toolbar, click **Plot**.