Created in COMSOL Multiphysics 6.0



P-N Diode Circuit

This model is licensed under the COMSOL Software License Agreement 6.0. All trademarks are the property of their respective owners. See www.comsol.com/trademarks. 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 μ m and a depth of 7 μ m. The length of the diode has been set to 10 μ m (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).



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

- I In the Model Wizard window, click **9** 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

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

GEOMETRY I

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

Rectangle 1 (r1)

- I In the Geometry toolbar, click 📃 Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type w_diode/2.
- 4 In the **Height** text field, type d_diode.
- 5 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)

- I In the Geometry toolbar, click Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type w_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.
- 6 | P-N DIODE CIRCUIT

Fillet I (fill)

- I 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 I (ptI)

- I In the Geometry toolbar, click Point.
- 2 In the Settings window for Point, locate the Point section.
- **3** In the **x** text field, type w_anode/2.
- 4 Click 📗 Build All Objects.

Load the semiconductor material properties for silicon.

ADD MATERIAL

- I In the Home toolbar, click 🙀 Add Material to open the Add Material window.
- 2 Go to the Add Material window.

- 3 In the tree, select Semiconductors>Si Silicon.
- 4 Click Add to Component in the window toolbar.
- 5 In the Home toolbar, click 🙀 Add Material to close the Add Material window.

SEMICONDUCTOR (SEMI)

Analytic Doping Model I

- I In the Model Builder window, under Component I (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

- I In the Physics toolbar, click 🔵 Domains and choose Analytic Doping Model.
- 2 In the Settings window for Analytic Doping Model, locate the Domain Selection section.
- **3** From the Selection list, choose All domains.
- 4 Locate the Distribution section. From the list, choose Box.
- **5** Locate the **Uniform Region** section. Specify the r_0 vector as

0[um] Х -d 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/adml).

Analytic Doping Model 3

- I In the Physics toolbar, click 🔵 Domains and choose Analytic Doping Model.
- 2 In the Settings window for Analytic Doping Model, locate the Domain Selection section.
- 3 From the Selection list, choose All domains.
- 4 Locate the **Distribution** section. From the list, choose **Box**.
- 5 Locate the Uniform Region section. In the W text field, type w_anode/2.
- **6** In the *D* text field, type d_p.
- 8 | P-N DIODE CIRCUIT

- 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/adml).

Trap-Assisted Recombination 1

- I In the Physics toolbar, click 🔵 Domains and choose Trap-Assisted Recombination.
- **2** In the Settings window for Trap-Assisted Recombination, locate the Domain Selection section.
- **3** From the Selection list, choose All domains.

Metal Contact 1

- I In the Physics toolbar, click Boundaries and choose Metal Contact.
- 2 Select Boundary 5 only.
- 3 In the Settings window for Metal Contact, locate the Terminal section.
- 4 From the Terminal type list, choose Circuit (current).

Metal Contact 2

- I 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 I (compl) click Electrical Circuit (cir).

Resistor I (RI)

I 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
Р	1
n	0

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

Voltage Source I (VI)

I 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
Р	2
n	0

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

5 In the $v_{\rm src}$ text field, type Vac.

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

External I vs. U I (IvsUI)

I In the Electrical Circuit toolbar, click III 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
Ρ	2
n	1

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

STUDY I

Step 1: Time Dependent

I In the Model Builder window, under Study I click Step I: 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 (soll)

- I In the Study toolbar, click **Show Default Solver**.
- 2 In the Model Builder window, expand the Solution I (soll) node, then click Time-Dependent Solver I.
- **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.
- 10 | P-N DIODE CIRCUIT

- 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

I 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 I 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)

- I Right-click Component I (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
Р	1
n	0

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

- **5** In the $v_{\rm src}$ text field, type Vac.
- **6** In the *f* text field, type **f**.

Use the diode large scale model with the following parameters.

Diode I (DI)

- 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
Ρ	1
n	2

 ${\bf 4}\,$ Locate the Model Parameters section. In the $I_{\rm S}$ text field, type 10.

5 In the *N* text field, type eta.

Resistor I (RI)

I 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
Р	2
n	0

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

ADD STUDY

I In the Home toolbar, click $\stackrel{\text{res}}{\longrightarrow}$ Add Study to open the Add Study window.

- 2 Go to the Add Study window.
- 3 Find the Studies subsection. In the Select Study tree, select General Studies> Time Dependent.
- 4 Click Add Study in the window toolbar.
- 5 In the Home toolbar, click $\stackrel{\sim}{\longrightarrow}$ Add Study to close the Add Study window.

STUDY 2

Step 1: Time Dependent

- I 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

- I In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.
- 2 Right-click ID Plot Group 4 and choose Rename.
- **3** In the **Rename ID Plot Group** dialog box, type Voltage probes in the **New label** text field.
- 4 Click OK.
- 5 In the Settings window for ID Plot Group, locate the Data section.
- 6 From the Dataset list, choose None.
- 7 Click to expand the Title section. From the Title type list, choose None.
- 8 Locate the Plot Settings section. Select the x-axis label check box.
- 9 In the associated text field, type time (s).

IO Select the y-axis label check box.

- **II** In the associated text field, type Voltage (V).
- 12 Locate the Legend section. From the Position list, choose Lower right.

Global I

- I 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 I/Solution I (soll).
- 4 Click Replace Expression in the upper-right corner of the y-Axis Data section. From the menu, choose Component I (compl)>Electrical Circuit>Devices>Vl>cir.Vl_v Voltage across device Vl 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

Expression	Unit	Description
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.

Global 2

- I 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 In the Voltage probes toolbar, click **O** Plot.