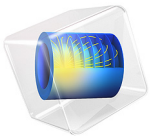
Created in COMSOL Multiphysics 5.3a



PN-Junction 1D

This model sets up a simple 1D benchmark of a pn-junction, or semiconductor diode, based on an example given in Ref. 1.

Introduction

A semiconductor diode consists of two regions with different doping: a p-type region with a dominant concentration of holes, and an n-type region with a dominant concentration of electrons. The anode contact is adjacent to the p-type region, and the cathode connects to the n-type region. Impurities that the manufacturing process adds to the semiconductor material determine each region's doping type. Donor impurities add additional electrons to the material conduction band and create n-type material. Acceptor impurities remove electrons from the valence band and create p-type material.

It is possible to derive a semiconductor model from Maxwell's equations and the Boltzmann equation for carrier transport. The three basic semiconductor equations (for a stationary problem) are given below:

$$\nabla \cdot (\varepsilon \nabla V) = -q(p - n + N_D^+ - N_A^-)$$
$$\nabla \cdot \mathbf{J}_n = -qR_{SRH}$$
$$\nabla \cdot \mathbf{J}_p = qR_{SRH}$$

where ε is the dielectric permittivity of the semiconductor, V is the electric potential, p and n are the electron and hole concentrations respectively, N_D^+ and N_A^- are the concentrations of ionized donors and acceptors respectively, J_n and J_p are the electron and hole currents, respectively, and R_{SRH} is the Shockley-Read-Hall recombination rate (see below for a description, in this case this term is assumed to be the only source of electron and hole recombination).

The electron and hole currents can be expressed in terms of V, n, and p in the following manner (assuming an isothermal, non-degenerate semiconductor with a constant band structure):

$$\mathbf{J}_n = -nq\mu_n \nabla V + \mu_n k_B T \nabla n$$
$$\mathbf{J}_p = -pq\mu_p \nabla V - \mu_p k_B T \nabla n$$

where μ_n and μ_p are the carrier mobilities, k_B is Boltzmann's constant, and T is the absolute temperature. In this model, the mobility is a complex function of the temperature and the donor and acceptor concentrations, as defined in Ref. 1.

The term R_{SRH} gives the rate at which electrons recombine (by a mechanism known as Shockley-Read-Hall recombination—which is often dominant in silicon). This recombination process involves traps in the forbidden band gap of the semiconductor. The recombination rate due to this mechanism is given by:

$$R_{SRH} = \frac{np - n_i^2}{\tau_p(n + n_1) + \tau_n(p + p_1)}$$

where n_i is the intrinsic carrier concentration (that is the carrier concentration in an undoped semiconductor), τ_n and τ_p are the carrier lifetimes, and n_1 and p_1 are parameters related to the trap energy level. If the trap level is located at the middle of the band gap (which is assumed in this model), then n_1 and p_1 equal n_i.

Model Definition

This model simulates the behavior of a pn-junction under reverse, equilibrium, and forward bias. The modeled junction has a length of 5 μm and a net doping concentration of 1×10^{15} cm^{-3} for both the p and n-doped side. A Shockley-Read-Hall recombination feature is also added to the model in order to simulate recombination usually observed in indirect band-gap semiconductor such as silicon, which is the material used in this model. The model uses the material parameters used in Ref. 1 and compares the carrier concentration profiles obtained from the computation with those obtained in the reference under different biasing conditions (–4 V, 0 V, and 0.5 V). Two different discretization methods are used to solve the model: the Finite Element Log Formation discretization; and the Finite Volume discretization. The results of these two computation methods are found to be in good agreement.

Results and Discussion

Figure 1shows the electric potential in the junction for different biasing conditions. A good agreement is shown between the finite element, finite volume and reference for all biasing conditions. Figure 2, Figure 4, and Figure 6 show the energy diagrams for each biasing conditions. On these figures, one can notice a perfect agreement between the finite element (using streamline diffusion stabilization) and finite volume. Comparing the carrier concentrations also shows a very good agreement between the reference, finite element and finite volume computations - See Figure 3, Figure 5, and Figure 7

.

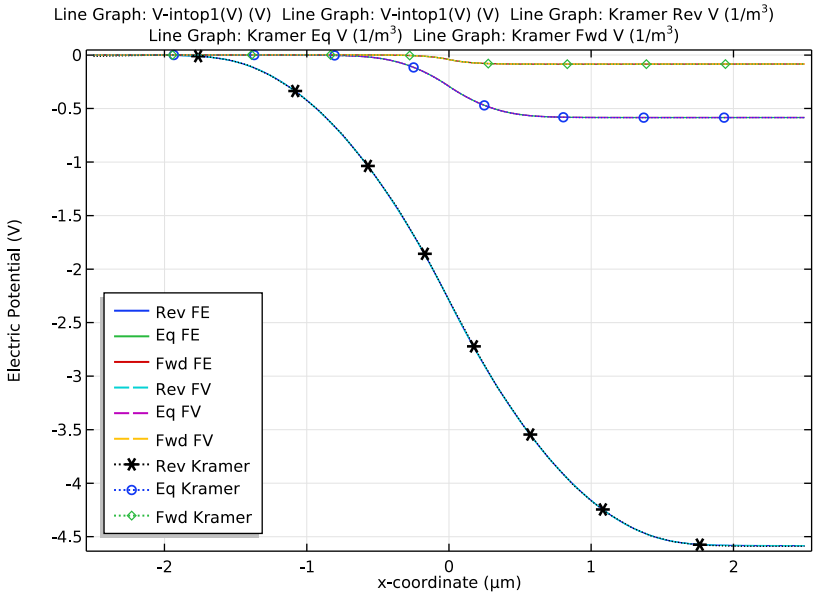


Figure 1: Electric potential obtained using the finite element (FE) and finite volume (FV) methods under different bias, that is, reverse (Rev), equilibrium (Eq), and forward (Fwd) bias. The results are compared to those calculated in Ref. 1 *(Kramer)*

.

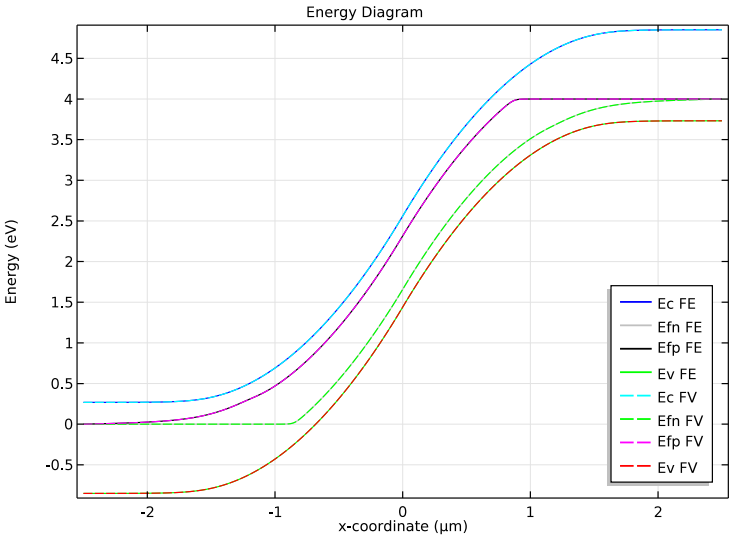


Figure 2: Energy diagram obtained using the finite element (FE) and finite volume (FV) methods under reverse bias (-4 V).

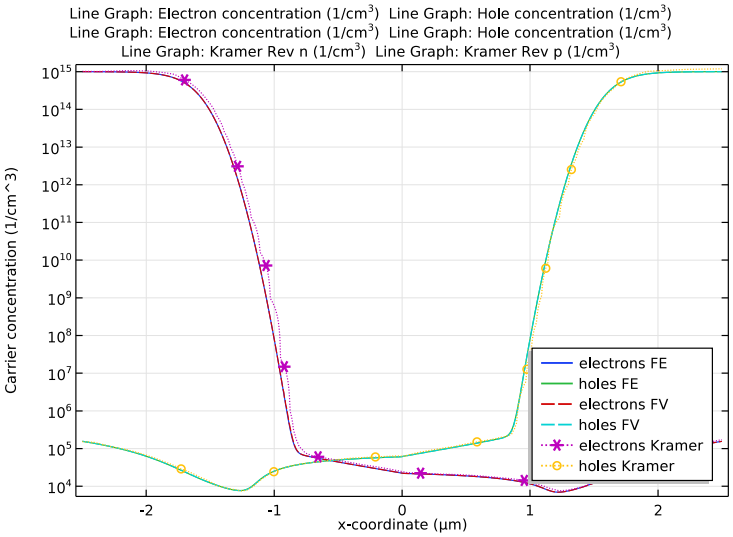


Figure 3: Electron and hole concentrations obtained using the finite element (FE) and finite volume (FV) methods under reverse bias (-4 V). The profiles are compared to those calculated in Ref. 1 *(Kramer)*

.

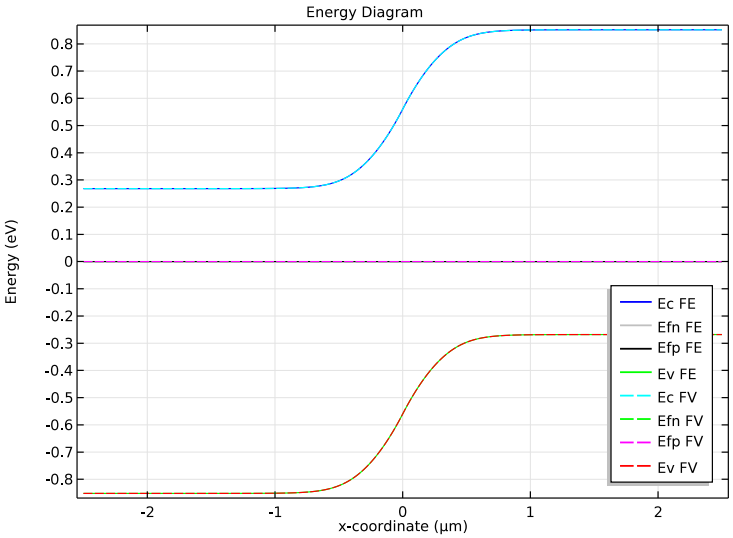


Figure 4: Energy diagram obtained using the finite element (FE) and finite volume (FV) methods at equilibrium (0 V).

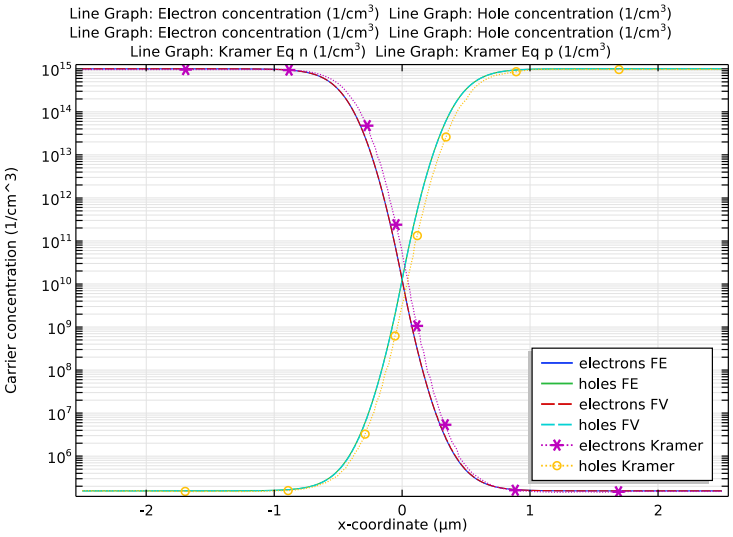


Figure 5: Electron and hole concentrations obtained using the finite element (FE) and finite volume (FV) methods at equilibrium (0 V). The profiles are compared to those calculated in Ref. 1 *(Kramer)*

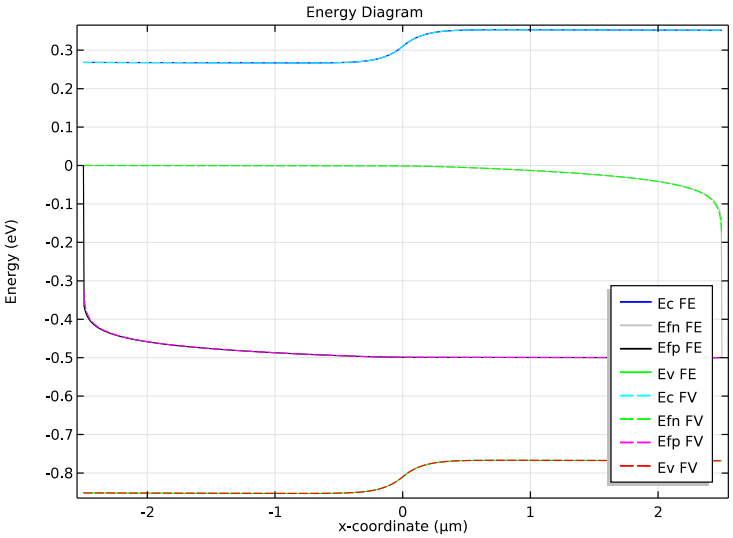


Figure 6: Energy diagram obtained using the finite element (FE) and finite volume (FV) methods under forward bias (0.5 V).

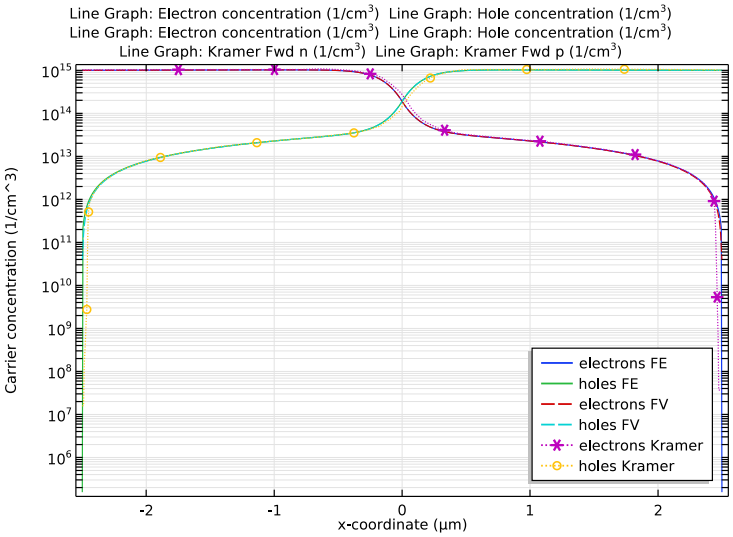


Figure 7: Electron and hole concentrations obtained using the finite element (FE) and finite volume (FV) methods under forward bias (0.5 V). The profiles are compared to those calculated in Ref. 1 *(Kramer)*

Reference

1. K.M. Kramer and W.N.G. Hitchon, *Semiconductor Devices a Simulation Approach,* Prentice Hall, Upper Saddle River, NJ, 1997.

Application Library path: `Semiconductor_Module/Verification_Examples/pn_junction_1d`

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. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies>Stationary**.
6. Click **Done**.

GEOMETRY 1

In the **Model Builder** window's toolbar, click the **Show** button and select **Discretization** in the menu.

SEMICONDUCTOR (SEMI)

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Semiconductor (semi)**.
2. In the **Settings** window for **Semiconductor**, click to expand the **Discretization** section.
3. From the **Formulation** list, choose **Finite element, log formulation (linear shape function)**.

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**.

Interval 1 (i1)

1 On the **Geometry** toolbar, click **Interval**.

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

3 In the **Left endpoint** text field, type `-2.5`.

4 In the **Right endpoint** text field, type `2.5`.

Point 1 (pt1)

1 On the **Geometry** toolbar, click **Point**.

2 In the **Settings** window for **Point**, click **Build All Objects**.

GLOBAL DEFINITIONS

Parameters

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

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_junction_1d_parameters.txt`.

DEFINITIONS

Variables 1

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

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

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `pn_junction_1d_variables.txt`.

Interpolation 1 (int1)

1 On the **Home** toolbar, click **Functions** and choose **Local>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 In the **Function name** text field, type `kramer_eq_V`.

4 Click **Load from File**.

5 Browse to the model's Application Libraries folder and double-click the file `pn_junction_1d_Kramer_eq_V.txt`.

6 Locate the **Units** section. In the **Arguments** text field, type `m`.

7 In the **Function** text field, type `1/m^3`.

8 Right-click **Interpolation 1 (int1)** and choose **Rename**.

9 In the **Rename Interpolation** dialog box, type `Kramer Eq V` in the **New label** text field.

10 Click **OK**.

Interpolation 2 (int2)

1 On the **Home** toolbar, click **Functions** and choose **Local>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 In the **Function name** text field, type `kramer_fwd_V`.

4 Click **Load from File**.

5 Browse to the model's Application Libraries folder and double-click the file `pn_junction_1d_Kramer_fwd_V.txt`.

6 Locate the **Units** section. In the **Arguments** text field, type `m`.

7 In the **Function** text field, type `1/m^3`.

8 Right-click **Interpolation 2 (int2)** and choose **Rename**.

9 In the **Rename Interpolation** dialog box, type `Kramer Fwd V` in the **New label** text field.

10 Click **OK**.

Interpolation 3 (int3)

1 On the **Home** toolbar, click **Functions** and choose **Local>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 In the **Function name** text field, type `kramer_rev_V`.

4 Click **Load from File**.

5 Browse to the model's Application Libraries folder and double-click the file `pn_junction_1d_Kramer_rev_V.txt`.

6 Locate the **Units** section. In the **Arguments** text field, type `m`.

7 In the **Function** text field, type `1/m^3`.

8 Right-click **Interpolation 3 (int3)** and choose **Rename**.

9 In the **Rename Interpolation** dialog box, type `Kramer Rev V` in the **New label** text field.

10 Click **OK**.

Interpolation 4 (int4)

1 On the **Home** toolbar, click **Functions** and choose **Local>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `pn_junction_1d_Kramer_eq_n.txt`.

5 In the **Function name** text field, type `kramer_eq_n`.

6 Locate the **Units** section. In the **Arguments** text field, type `m`.

7 In the **Function** text field, type `1/m^3`.

8 Right-click **Interpolation 4 (int4)** and choose **Rename**.

9 In the **Rename Interpolation** dialog box, type `Kramer Eq n` in the **New label** text field.

10 Click **OK**.

Interpolation 5 (int5)

1 On the **Home** toolbar, click **Functions** and choose **Local>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `pn_junction_1d_Kramer_eq_p.txt`.

5 In the **Function name** text field, type `kramer_eq_p`.

6 Locate the **Units** section. In the **Arguments** text field, type `m`.

7 In the **Function** text field, type `1/m^3`.

8 Right-click **Interpolation 5 (int5)** and choose **Rename**.

9 In the **Rename Interpolation** dialog box, type `Kramer Eq p` in the **New label** text field.

10 Click **OK**.

Interpolation 6 (int6)

1 On the **Home** toolbar, click **Functions** and choose **Local>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 In the **Function name** text field, type `kramer_fwd_n`.

4 Click **Load from File**.

5 Browse to the model's Application Libraries folder and double-click the file `pn_junction_1d_Kramer_fwd_n.txt`.

6 Locate the **Units** section. In the **Arguments** text field, type `m`.

7 In the **Function** text field, type `1/m^3`.

8 Right-click **Interpolation 6 (int6)** and choose **Rename**.

9 In the **Rename Interpolation** dialog box, type `Kramer Fwd n` in the **New label** text field.

10 Click **OK**.

Interpolation 7 (int7)

1 On the **Home** toolbar, click **Functions** and choose **Local>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 In the **Function name** text field, type `kramer_fwd_p`.

4 Click **Load from File**.

5 Browse to the model's Application Libraries folder and double-click the file `pn_junction_1d_Kramer_fwd_p.txt`.

6 Locate the **Units** section. In the **Arguments** text field, type `m`.

7 In the **Function** text field, type `1/m^3`.

8 Right-click **Interpolation 7 (int7)** and choose **Rename**.

9 In the **Rename Interpolation** dialog box, type `Kramer Fwd p` in the **New label** text field.

10 Click **OK**.

Interpolation 8 (int8)

1 On the **Home** toolbar, click **Functions** and choose **Local>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 In the **Function name** text field, type `kramer_rev_n`.

4 Click **Load from File**.

5 Browse to the model's Application Libraries folder and double-click the file `pn_junction_1d_Kramer_rev_n.txt`.

6 Locate the **Units** section. In the **Arguments** text field, type `m`.

7 In the **Function** text field, type `1/m^3`.

8 Right-click **Interpolation 8 (int8)** and choose **Rename**.

9 In the **Rename Interpolation** dialog box, type `Kramer Rev n` in the **New label** text field.

10 Click **OK**.

Interpolation 9 (int9)

1 On the **Home** toolbar, click **Functions** and choose **Local>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `pn_junction_1d_Kramer_rev_p.txt`.

5 In the **Function name** text field, type `kramer_rev_p`.

6 Locate the **Units** section. In the **Arguments** text field, type `m`.

7 In the **Function** text field, type `1/m^3`.

8 Right-click **Interpolation 9 (int9)** and choose **Rename**.

9 In the **Rename Interpolation** dialog box, type `Kramer Rev p` in the **New label** text field.

10 Click **OK**.

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 1 only.

SEMICONDUCTOR (SEMI)

Semiconductor Material Model 1

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

2 In the **Settings** window for **Semiconductor Material Model**, locate the **Model Inputs** section.

3 In the T text field, type `T1`.

4 Locate the **Mobility Model** section. From the μ_n list, choose **User defined**. In the associated text field, type `mu_n`.

5 From the μ_p list, choose **User defined**. In the associated text field, type `mu_p`.

Trap-Assisted Recombination 1

1 On 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**.

Analytic Doping Model 1

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

2 Select Domain 1 only.

3 In the **Settings** window for **Analytic Doping Model**, locate the **Impurity** section.

4 From the **Impurity type** list, choose **Donor doping (n-type)**.

5 In the N_{D0} text field, type `Nd`.

Analytic Doping Model 2

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

2 Select Domain 2 only.

3 In the **Settings** window for **Analytic Doping Model**, locate the **Impurity** section.

4 In the N_{A0} text field, type `Na`.

Metal Contact 1

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

2 Select Boundary 1 only.

Metal Contact 2

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

2 In the **Settings** window for **Metal Contact**, locate the **Terminal** section.

3 In the V_0 text field, type `Va`.

4 Select Boundary 3 only.

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Relative permittivity	epsilonr	`epsilonr_param`	1	Basic
Band gap	Eg0	`Eg0_param`	V	Semiconductor material
Electron affinity	chi0	`chi0_param`	V	Semiconductor material
Effective density of states, conduction band	Nc	`Nc_param`	1/m³	Semiconductor material
Effective density of states, valence band	Nv	`Nv_param`	1/m³	Semiconductor material

Property	Variable	Value	Unit	Property group
Electron lifetime, SRH	taun	taun_param	s	Shockley-Read-Hall recombination
Hole lifetime, SRH	taup	taup_param	s	Shockley-Read-Hall recombination

MESH 1

Distribution 1

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

2 Right-click **Edge 1** and choose **Distribution**.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type `500`.

Configure the first study. The finite element log formulation discretization requires the voltage applied to the device to be ramped on. This is achieved using the 'sweep' variable, which is ramped from 0 to 1 using the solver's continuation functionality. The voltage applied to the device, Va, is calculated using Va=bias*sweep, as can be seen in the parameters table within the Definitions node. The first study will be configured to apply three different biases, with each bias being ramped from 0 V to the intended value as 'sweep' increases from 0 to 1.

STUDY 1

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

2 In the **Settings** window for **Study**, type `Study 1 - Finite Element Log Formulation` in the **Label** text field.

STUDY 1 - FINITE ELEMENT LOG FORMULATION

Step 1: Stationary

1 In the **Model Builder** window, expand the **Study 1 - Finite Element Log Formulation** node, then click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Study extensions** section.

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

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

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
bias (Device bias)	-4 0 0.5	

7 Click **Add**.

8 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
sweep (Sweep parameter for gradually turning on bias)	0 1	

9 On the **Home** toolbar, click **Compute**.

RESULTS

Energy Levels (semi)

1 In the **Model Builder** window, under **Results** click **Energy Levels (semi)**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Parameter selection (bias)** list, choose **First**.

4 From the **Parameter selection (sweep)** list, choose **Last**.

Conduction Band Energy Level

1 In the **Model Builder** window, expand the **Energy Levels (semi)** node, then click **Conduction Band Energy Level**.

2 In the **Settings** window for **Line Graph**, locate the **x-Axis Data** section.

3 From the **Parameter** list, choose **Expression**.

4 In the **Expression** text field, type x.

5 Click to expand the **Legends** section. In the table, enter the following settings:

Legends
Ec FE

Electron_quasi-Fermi_energy_level

1 In the **Model Builder** window, under **Results>Energy Levels (semi)** click **Electron_quasi-Fermi_energy_level**.

2 In the **Settings** window for **Line Graph**, locate the **x-Axis Data** section.

3 From the **Parameter** list, choose **Expression**.

4 In the **Expression** text field, type x.

5 Locate the **Legends** section. In the table, enter the following settings:

Legends
Efn FE

6 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Solid**.

7 From the **Color** list, choose **Gray**.

Hole_quasi-Fermi_energy_level

1 In the **Model Builder** window, under **Results>Energy Levels (semi)** click **Hole_quasi-Fermi_energy_level**.

2 In the **Settings** window for **Line Graph**, locate the **x-Axis Data** section.

3 From the **Parameter** list, choose **Expression**.

4 In the **Expression** text field, type x.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Solid**.

6 Locate the **Legends** section. In the table, enter the following settings:

Legends
Efp FE

Valence Band Energy Level

1 In the **Model Builder** window, under **Results>Energy Levels (semi)** click **Valence Band Energy Level**.

2 In the **Settings** window for **Line Graph**, locate the **x-Axis Data** section.

3 From the **Parameter** list, choose **Expression**.

4 In the **Expression** text field, type x.

5 Locate the **Legends** section. In the table, enter the following settings:

Legends
Ev FE

Energy Levels (semi)

1 In the **Model Builder** window, under **Results** click **Energy Levels (semi)**.

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

3 From the **Position** list, choose **Lower right**.

4 Right-click **Results>Energy Levels (semi)** and choose **Rename**.

5 In the **Rename 1D Plot Group** dialog box, type `Energy Levels Reverse Bias` in the **New label** text field.

6 Click **OK**.

Carrier Concentrations (semi)

1 In the **Settings** window for **1D Plot Group**, locate the **Legend** section.

2 From the **Position** list, choose **Lower right**.

3 Right-click **Results>Carrier Concentrations (semi)** and choose **Rename**.

4 In the **Rename 1D Plot Group** dialog box, type `Carrier Concentrations Reverse Bias` in the **New label** text field.

5 Click **OK**.

Carrier Concentrations Reverse Bias

1 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

2 From the **Parameter selection (bias)** list, choose **First**.

3 From the **Parameter selection (sweep)** list, choose **Last**.

Electron Concentration

1 In the **Model Builder** window, expand the **Results>Carrier Concentrations Reverse Bias** node, then click **Electron Concentration**.

2 In the **Settings** window for **Line Graph**, locate the **x-Axis Data** section.

3 From the **Parameter** list, choose **Expression**.

4 In the **Expression** text field, type `x`.

5 Locate the **Legends** section. In the table, enter the following settings:

Legends
`electrons FE`

Hole Concentration

1 In the **Model Builder** window, under **Results>Carrier Concentrations Reverse Bias** click **Hole Concentration**.

2 In the **Settings** window for **Line Graph**, locate the **x-Axis Data** section.

3 From the **Parameter** list, choose **Expression**.

4 In the **Expression** text field, type `x`.

5 Locate the **Legends** section. In the table, enter the following settings:

Legends
holes FE

Line Graph 1

1 In the **Model Builder** window, expand the **Results>Electric Potential (semi)** node, then click **Line Graph 1**.

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

3 In the **Expression** text field, type `V-intop1(V)`.

4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

5 In the **Expression** text field, type `x`.

6 Click to expand the **Legends** section. Select the **Show legends** check box.

7 From the **Legends** list, choose **Manual**.

8 In the table, enter the following settings:

Legends
Rev FE
Eq FE
Fwd FE

Electric Potential (semi)

1 In the **Model Builder** window, under **Results** click **Electric Potential (semi)**.

2 In the **Settings** window for **1D Plot Group**, locate the **Legend** section.

3 From the **Position** list, choose **Lower left**.

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

In order to compare the finite element log formulation discretization with finite volume discretization, change the discretization and add a second study to re-solve the model with the new selection.

SEMICONDUCTOR (SEMI)

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

2 In the **Settings** window for **Semiconductor**, locate the **Discretization** section.

3 From the **Formulation** list, choose **Finite volume (constant shape function)**.

ADD STUDY

1 On 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 **Preset Studies>Stationary**.

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

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

STUDY 2

1 In the **Settings** window for **Study**, type `Study 2 - Finite Volume` in the **Label** text field.

2 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Unlike in the previous study, the finite volume discretization does not require that the voltage be ramped up gradually. Therefore the desired values of 'Va' can be set directly in the auxiliary sweep.

STUDY 2 - FINITE VOLUME

Step 1: Stationary

1 In the **Model Builder** window, under **Study 2 - Finite Volume** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Study extensions** section.

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

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Va	`-4 0 0.5`	

6 On the **Home** toolbar, click **Compute**.

RESULTS

Line Graph 2

1 In the **Model Builder** window, under **Results>Electric Potential (semi)** right-click **Line Graph 1** and choose **Duplicate**.

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

3 From the **Data set** list, choose **Study 2 - Finite Volume/Solution 2 (sol2)**.

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

5 Locate the **Legends** section. In the table, enter the following settings:

Legends
Rev FV
Eq FV
Fwd FV

Line Graph 3

1 Right-click **Line Graph 1** and choose **Duplicate**.

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

3 In the **Expression** text field, type `kramer_rev_V(x)`.

4 Locate the **Data** section. From the **Data set** list, choose **Study 1 - Finite Element Log Formulation/Solution 1 (sol1)**.

5 From the **Parameter selection (bias)** list, choose **First**.

6 From the **Parameter selection (sweep)** list, choose **Last**.

7 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dotted**.

8 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

9 Locate the **Legends** section. In the table, enter the following settings:

Legends
Rev Kramer

Line Graph 4

1 Right-click **Results>Electric Potential (semi)>Line Graph 3** and choose **Duplicate**.

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

3 In the **Expression** text field, type `kramer_eq_V(x)`.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
Eq Kramer

Line Graph 5

1 Right-click **Results>Electric Potential (semi)>Line Graph 4** and choose **Duplicate**.

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

3 In the **Expression** text field, type `kramer_fwd_V(x)`.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
Fwd Kramer

5 On the **Electric Potential (semi)** toolbar, click **Plot**.

6 In the **Model Builder** window, under **Results>Energy Levels Reverse Bias** right-click **Conduction Band Energy Level**, hold the 'shift' key and left click **Valence Band Energy Level** to select all four Line Graphs. Right click the selection and choose **Duplicate**.

Conduction Band Energy Level 1

1 In the **Model Builder** window, under **Results>Energy Levels Reverse Bias** click **Conduction Band Energy Level 1**.

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

3 From the **Data set** list, choose **Study 2 - Finite Volume/Solution 2 (sol2)**.

4 From the **Parameter selection (Va)** list, choose **First**.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

6 From the **Color** list, choose **Cyan**.

7 Locate the **Legends** section. In the table, enter the following settings:

Legends
Ec FV

Electron_quasi-Fermi_energy_level 1

1 In the **Model Builder** window, under **Results>Energy Levels Reverse Bias** click **Electron_quasi-Fermi_energy_level 1**.

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

3 From the **Data set** list, choose **Study 2 - Finite Volume/Solution 2 (sol2)**.

4 From the **Parameter selection (Va)** list, choose **First**.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

6 From the **Color** list, choose **Green**.

7 Locate the **Legends** section. In the table, enter the following settings:

Legends
Efn FV

Hole_quasi-Fermi_energy_level 1

1 In the **Model Builder** window, under **Results>Energy Levels Reverse Bias** click **Hole_quasi-Fermi_energy_level 1**.

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

3 From the **Data set** list, choose **Study 2 - Finite Volume/Solution 2 (sol2)**.

4 From the **Parameter selection (Va)** list, choose **First**.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

6 From the **Color** list, choose **Magenta**.

7 Locate the **Legends** section. In the table, enter the following settings:

Legends
Efp FV

Valence Band Energy Level 1

1 In the **Model Builder** window, under **Results>Energy Levels Reverse Bias** click **Valence Band Energy Level 1**.

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

3 From the **Data set** list, choose **Study 2 - Finite Volume/Solution 2 (sol2)**.

4 From the **Parameter selection (Va)** list, choose **First**.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

6 From the **Color** list, choose **Red**.

7 Locate the **Legends** section. In the table, enter the following settings:

Legends
Ev FV

8 On the **Energy Levels Reverse Bias** toolbar, click **Plot**.

Carrier Concentrations Reverse Bias

Left click **Electron Concentration**, hold the 'shift' key and left click **Hole Concentration** to select both Line Graphs. Right-click the selection and choose **Duplicate**.

Electron Concentration 1

1 In the **Model Builder** window, expand the **Carrier Concentrations Reverse Bias** node, then click **Electron Concentration 1**.

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

3 From the **Data set** list, choose **Study 2 - Finite Volume/Solution 2 (sol2)**.

4 From the **Parameter selection (Va)** list, choose **First**.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

6 Locate the **Legends** section. In the table, enter the following settings:

Legends
electrons FV

Hole Concentration 1

1 In the **Model Builder** window, under **Results>Carrier Concentrations Reverse Bias** click **Hole Concentration 1**.

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

3 From the **Data set** list, choose **Study 2 - Finite Volume/Solution 2 (sol2)**.

4 From the **Parameter selection (Va)** list, choose **First**.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

6 Locate the **Legends** section. In the table, enter the following settings:

Legends
holes FV

7 In the **Model Builder** window, under **Results>Carrier Concentrations Reverse Bias** left click **Electron Concentration 1**, hold the 'shift' key and left-click **Hole Concentration 1**. Right-click the selection and choose **Duplicate**.

Electron Concentration 1.1

1 In the **Model Builder** window, under **Results>Carrier Concentrations Reverse Bias** click **Electron Concentration 1.1**.

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

3 In the **Expression** text field, type `kramer_rev_n(x)`.

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dotted**.

5 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

6 Locate the **Legends** section. In the table, enter the following settings:

Legends
electrons Kramer

Hole Concentration 1.1

1 In the **Model Builder** window, under **Results>Carrier Concentrations Reverse Bias** click **Hole Concentration 1.1**.

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

3 In the **Expression** text field, type `kramer_rev_p(x)`.

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dotted**.

5 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

6 Locate the **Legends** section. In the table, enter the following settings:

Legends
holes Kramer

7 On the **Carrier Concentrations Reverse Bias** toolbar, click **Plot**.

8 In the **Model Builder** window, under **Results**, click **Energy Levels Reverse Bias**, hold the 'shift' key and click **Carrier Concentrations Reverse Bias**. Right-click the selection and choose **Duplicate**.

9 In the **Model Builder** window, under **Results**, click **Energy Levels Reverse Bias 1**, hold the 'shift' key and left-click **Carrier Concentrations Reverse Bias 1**. Right-click the selection and choose **Duplicate**.

Energy Levels Reverse Bias 1

1 In the **Model Builder** window, under **Results** right-click **Energy Levels Reverse Bias 1** and choose **Rename**.

2 In the **Rename 1D Plot Group** dialog box, type `Energy Levels Equilibrium` in the **New label** text field.

3 Click **OK**.

4 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

5 From the **Parameter selection (sweep)** list, choose **From list**.

6 In the **Parameter values (sweep)** list, select **0**.

Conduction Band Energy Level 1

1 In the **Model Builder** window, expand the **Results>Energy Levels Equilibrium** node, then click **Conduction Band Energy Level 1**.

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

3 From the **Parameter selection (Va)** list, choose **From list**.

4 In the **Parameter values (Va)** list, select **0**.

Electron_quasi-Fermi_energy_level 1

1 In the **Model Builder** window, under **Results>Energy Levels Equilibrium** click **Electron_quasi-Fermi_energy_level 1**.

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

3 From the **Parameter selection (Va)** list, choose **From list**.

4 In the **Parameter values (Va)** list, select **0**.

Hole_quasi-Fermi_energy_level 1

1 In the **Model Builder** window, under **Results>Energy Levels Equilibrium** click **Hole_quasi-Fermi_energy_level 1**.

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

3 From the **Parameter selection (Va)** list, choose **From list**.

4 In the **Parameter values (Va)** list, select **0**.

Valence Band Energy Level 1

1 In the **Model Builder** window, under **Results>Energy Levels Equilibrium** click **Valence Band Energy Level 1**.

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

3 From the **Parameter selection (Va)** list, choose **From list**.

4 In the **Parameter values (Va)** list, select **0**.

5 On the **Energy Levels Equilibrium** toolbar, click **Plot**.

Carrier Concentrations Reverse Bias 1

1 In the **Model Builder** window, under **Results** click **Carrier Concentrations Reverse Bias 1**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Parameter selection (sweep)** list, choose **From list**.

4 In the **Parameter values (sweep)** list, select **0**.

Electron Concentration 1

1 In the **Model Builder** window, expand the **Carrier Concentrations Reverse Bias 1** node, then click **Electron Concentration 1**.

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

3 From the **Parameter selection (Va)** list, choose **From list**.

4 In the **Parameter values (Va)** list, select **0**.

Hole Concentration 1

1 In the **Model Builder** window, under **Results>Carrier Concentrations Reverse Bias 1** click **Hole Concentration 1**.

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

3 From the **Parameter selection (Va)** list, choose **From list**.

4 In the **Parameter values (Va)** list, select **0**.

Electron Concentration 1.1

1 In the **Model Builder** window, under **Results>Carrier Concentrations Reverse Bias 1** click **Electron Concentration 1.1**.

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

3 In the **Expression** text field, type `kramer_eq_n(x)`.

Hole Concentration 1.1

1 In the **Model Builder** window, under **Results>Carrier Concentrations Reverse Bias 1** click **Hole Concentration 1.1**.

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

3 In the **Expression** text field, type `kramer_eq_p(x)`.

4 On the **Carrier Concentrations Reverse Bias 1** toolbar, click **Plot**.

Carrier Concentrations Reverse Bias 1

1 In the **Model Builder** window, under **Results** right-click **Carrier Concentrations Reverse Bias 1** and choose **Rename**.

2 In the **Rename 1D Plot Group** dialog box, type `Carrier Concentrations Equilibrium` in the **New label** text field.

3 Click **OK**.

Energy Levels Reverse Bias 1.1

1 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

2 From the **Parameter selection (bias)** list, choose **Last**.

Conduction Band Energy Level 1

1 In the **Model Builder** window, expand the **Energy Levels Reverse Bias 1.1** node, then click **Conduction Band Energy Level 1**.

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

3 From the **Parameter selection (Va)** list, choose **Last**.

Electron_quasi-Fermi_energy_level 1

1 In the **Model Builder** window, under **Results>Energy Levels Reverse Bias 1.1** click **Electron_quasi-Fermi_energy_level 1**.

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

3 From the **Parameter selection (Va)** list, choose **Last**.

Hole_quasi-Fermi_energy_level 1

1 In the **Model Builder** window, under **Results>Energy Levels Reverse Bias 1.1** click **Hole_quasi-Fermi_energy_level 1**.

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

3 From the **Parameter selection (Va)** list, choose **Last**.

Valence Band Energy Level 1

1 In the **Model Builder** window, under **Results>Energy Levels Reverse Bias 1.1** click **Valence Band Energy Level 1**.

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

3 From the **Parameter selection (Va)** list, choose **Last**.

4 On the **Energy Levels Reverse Bias 1.1** toolbar, click **Plot**.

Energy Levels Reverse Bias 1.1

1 In the **Model Builder** window, under **Results** right-click **Energy Levels Reverse Bias 1.1** and choose **Rename**.

2 In the **Rename 1D Plot Group** dialog box, type `Energy Levels Forward Bias` in the **New label** text field.

3 Click **OK**.

Carrier Concentrations Reverse Bias 1.1

1 In the **Model Builder** window, under **Results** right-click **Carrier Concentrations Reverse Bias 1.1** and choose **Rename**.

2 In the **Rename 1D Plot Group** dialog box, type `Carrier Concentrations Forward Bias` in the **New label** text field.

3 Click **OK**.

4 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

5 From the **Parameter selection (bias)** list, choose **Last**.

Electron Concentration 1

1 In the **Model Builder** window, expand the **Results>Carrier Concentrations Forward Bias** node, then click **Electron Concentration 1**.

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

3 From the **Parameter selection (Va)** list, choose **Last**.

Hole Concentration 1

1 In the **Model Builder** window, under **Results>Carrier Concentrations Forward Bias** click **Hole Concentration 1**.

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

3 From the **Parameter selection (Va)** list, choose **Last**.

Electron Concentration 1.1

1 In the **Model Builder** window, under **Results>Carrier Concentrations Forward Bias** click **Electron Concentration 1.1**.

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

3 In the **Expression** text field, type `kramer_fwd_n(x)`.

Hole Concentration 1.1

1 In the **Model Builder** window, under **Results>Carrier Concentrations Forward Bias** click **Hole Concentration 1.1**.

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

3 In the **Expression** text field, type `kramer_fwd_p(x)`.

4 On the **Carrier Concentrations Forward Bias** toolbar, click **Plot**.