Model created in COMSOL Multiphysics 6.4

Silicon Carbide Diode Breakdown

Introduction

Silicon carbide (SiC) diodes have several advantages over silicon diodes, which makes them very useful in power electronic applications. They can operate at high temperatures and have a high breakdown voltage.

By applying a noticeably high reverse-biased voltage across the diode, the charge carriers get accelerated due to the high electric field. As a result of impact ionization, avalanche process in the SiC diode starts. Collision of high velocity carriers with the lattice atoms, leads to generation of additional electron–hole pairs. These generated electron–hole pairs lead to additional collisions leading to generation of more electron–hole pairs. Therefore, avalanche breakdown in diode, allows the flow of a large current in its reverse mode.

This example shows how to model the avalanche breakdown due to impact ionization in a SiC diode.

Model Definition

Figure 1 shows the model geometry, indicating different domains, doping profiles, and the field plates. The device is 60 μm in width and 26 μm in height, and is modeled in 2D axisymmetric. An impact ionization generation is defined as the mechanism responsible for the avalanche breakdown. To account for the recombination, a trap-assisted recombination is added using Shockley–Read–Hall trapping model. The material used for diode part is silicon carbide [solid, 4H Polytype] and the field plates are silicon oxide as available in the Semiconductors material library.

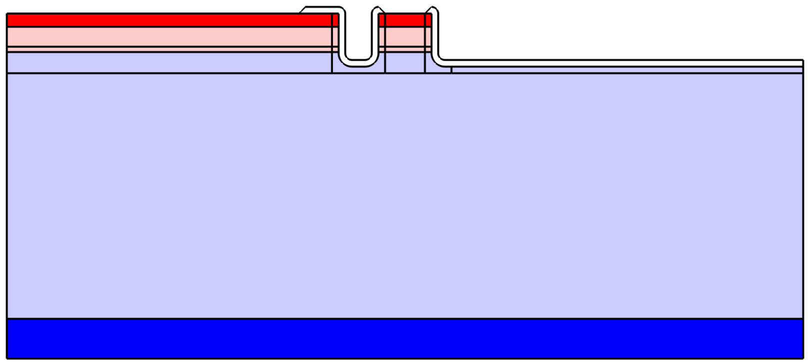

Figure 1: Model geometry showing the doping levels. p-type and n-type domains are shown in red and blue color, respectively. The field plates are placed over the device shown in white.

The procedure of the implementation is described in detail in the Modeling Instructions section.

Results and Discussion

Figure 2 shows the reverse I–V characteristics of the SiC diode.

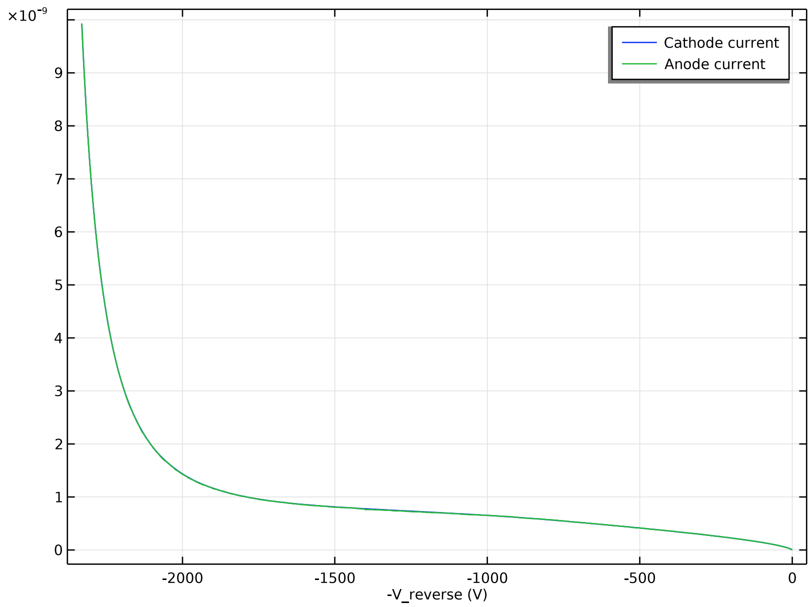

Figure 2: Current–voltage Characteristics of the SiC diode in reverse bias.

Figure 3 shows the high breakdown electric field of the SiC diode.

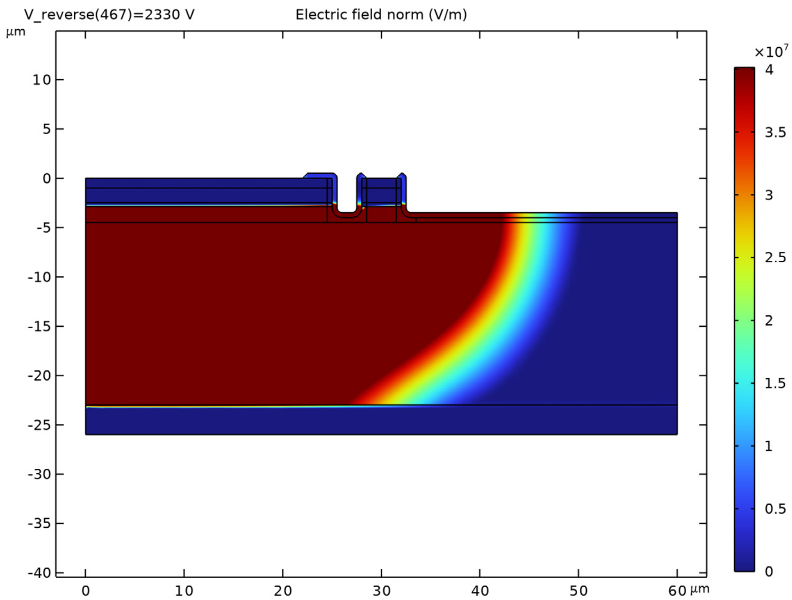

Figure 3: Electric field distribution of the SiC diode in breakdown.

Figure 4 shows the carrier generation terms demonstrating the paths of breakdown current.

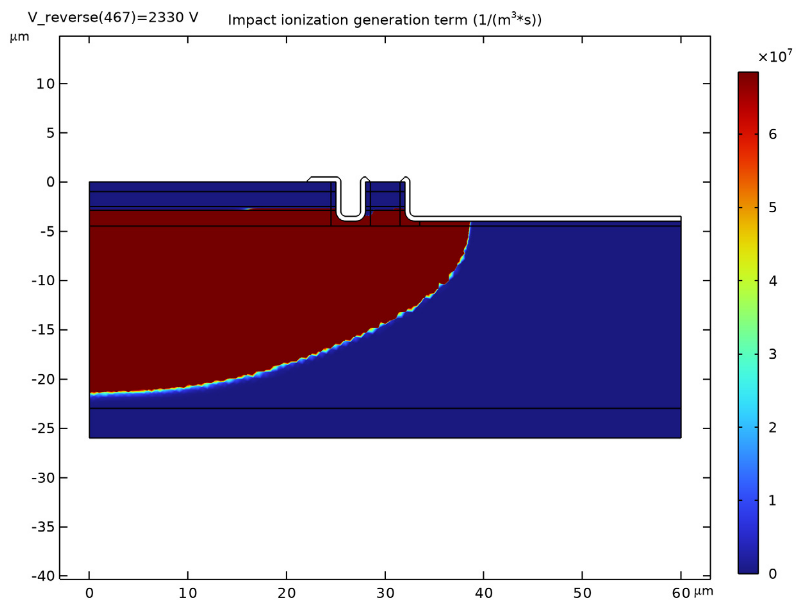

Figure 4: 2D demonstration of the carrier generation of SiC diode in breakdown.

Application Library path: `Semiconductor_Module/Device_Building_Blocks/sic_diode_breakdown`

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 Axisymmetric**.

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 for Selected Physics Interfaces** > **Semiconductor Equilibrium**.

6 Click **Done**.

GLOBAL DEFINITIONS

Simulation Parameters

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

2 In the **Settings** window for **Parameters**, type `Simulation Parameters` in the **Label** text field.

3 In the **Model Builder** window, click **Simulation Parameters**.

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

Name	**Expression**	**Value**	**Description**
V_reverse	0[V]	0 V	Reverse bias
T_lattice	900[K]	900 K	Lattice temperature
I_stop	10[nA]	1E-8 A	Current level for stop condition
d_cont2edg	3[µm]	3E-6 m	Distance between contact and edge
W_anode	25[µm]	2.5E-5 m	Width of anode
W_device	60[µm]	6E-5 m	Width of device
H_device	26[µm]	2.6E-5 m	Height of device
W_etch1	3[µm]	3E-6 m	Width of first etch
D_etch1	4[µm]	4E-6 m	
W_ring1	4[µm]	4E-6 m	Width of first ring
d_box	0.5[µm]	5E-7 m	
D_p_cont	1[µm]	1E-6 m	
D_p_anode	1.5[µm]	1.5E-6 m	
D_p_drift	0.4[µm]	4E-7 m	
R_final_etch	W_anode+W_etch1+ W_ring1	3.2E-5 m	Radial position of final etch
H_substrate	3[µm]	3E-6 m	Height of substrate

Name	Expression	Value	Description
D_p_region	D_p_cont+ D_p_anode+ D_p_drift	2.9E-6 m	Depth of pn-junction
t_ox	0.5[µm]	5E-7 m	Oxide thickness

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**.

Semiconductor

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

2 In the **Settings** window for **Rectangle**, type `Semiconductor` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `W_device`.

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

5 Locate the **Position** section. In the **z** text field, type `-H_device`.

6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (µm)
p-contact	D_p_cont
p-anode	D_p_anode
p-drift	D_p_drift
n-etch	D_etch1-D_p_cont-D_p_anode-D_p_drift+d_box
n-drift	H_device-H_substrate-D_etch1-d_box

7 Clear the **Layers on bottom** checkbox.

8 Select the **Layers on top** checkbox.

Etch box 1

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

2 In the **Settings** window for **Rectangle**, type `Etch box 1` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `W_etch1+2*d_box`.

4 In the **Height** text field, type `D_etch1+d_box`.

5 Locate the **Position** section. In the **r** text field, type `W_anode-d_box`.

6 In the **z** text field, type `-D_etch1-d_box`.

Etch 1

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

2 In the **Settings** window for **Rectangle**, type `Etch 1` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `W_etch1`.

4 In the **Height** text field, type `D_etch1`.

5 Locate the **Position** section. In the **r** text field, type `W_anode`.

6 In the **z** text field, type `-D_etch1`.

7 Locate the **Assigned Attributes** section. Select the **Construction geometry** checkbox.

Etch final

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

2 In the **Settings** window for **Rectangle**, type `Etch final` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `W_device-R_final_etch`.

4 In the **Height** text field, type `D_etch1`.

5 Locate the **Position** section. In the **r** text field, type `R_final_etch`.

6 In the **z** text field, type `-D_etch1`.

7 Locate the **Assigned Attributes** section. Select the **Construction geometry** checkbox.

Etch box 2

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

2 In the **Settings** window for **Rectangle**, type `Etch box 2` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `W_device+d_box-R_final_etch`.

4 In the **Height** text field, type `D_etch1+d_box`.

5 Locate the **Position** section. In the **r** text field, type `R_final_etch-d_box`.

6 In the **z** text field, type `-D_etch1-d_box`.

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

Layer name	Thickness (µm)
Layer 1	2*d_box+1

8 Select the **Layers to the left** checkbox.

9 Clear the **Layers on bottom** checkbox.

Fillet 1 (fil1)

1 In the **Geometry** toolbar, click **Fillet**.
2 On the object **r3**, select Points 1 and 2 only.
3 On the object **r4**, select Point 1 only.
4 In the **Settings** window for **Fillet**, locate the **Radius** section.
5 In the **Radius** text field, type `1`.

Difference 1 (dif1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.
2 Select the objects **r1**, **r2**, and **r5** only.
3 In the **Settings** window for **Difference**, locate the **Difference** section.
4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.
5 Select the objects **fil1(1)** and **fil1(2)** only.

Contact boundary

1 In the **Geometry** toolbar, click **Point**.
2 In the **Settings** window for **Point**, type `Contact boundary` in the **Label** text field.
3 Locate the **Point** section. In the **r** text field, type `W_anode-d_cont2edg`.

P-Contact

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.
2 In the **Settings** window for **Box Selection**, type `P-Contact` in the **Label** text field.
3 Locate the **Box Limits** section. In the **r minimum** text field, type `-1`.
4 In the **r maximum** text field, type `W_device+1`.
5 In the **z minimum** text field, type `-D_p_cont-0.1`.
6 In the **z maximum** text field, type `0.1`.

P-Anode

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.
2 In the **Settings** window for **Box Selection**, type `P-Anode` in the **Label** text field.
3 Locate the **Box Limits** section. In the **r minimum** text field, type `-1`.
4 In the **r maximum** text field, type `W_device+1`.
5 In the **z minimum** text field, type `-D_p_anode-D_p_drift-D_p_cont-0.1`.
6 In the **z maximum** text field, type `-D_p_cont+0.1`.

N-Drift

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.
2 In the **Settings** window for **Box Selection**, type `N-Drift` in the **Label** text field.
3 Locate the **Box Limits** section. In the **r minimum** text field, type `-1`.
4 In the **r maximum** text field, type `W_device+1`.
5 In the **z minimum** text field, type `-H_device+H_substrate-0.1`.
6 In the **z maximum** text field, type `-D_p_region+0.1`.

N-Substrate

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.
2 In the **Settings** window for **Box Selection**, type `N-Substrate` in the **Label** text field.
3 Locate the **Box Limits** section. In the **r minimum** text field, type `-1`.
4 In the **r maximum** text field, type `W_device+1`.
5 In the **z minimum** text field, type `-H_device-0.1`.
6 In the **z maximum** text field, type `-H_device+H_substrate+0.1`.
7 Click **Build All Objects**.

All domains before oxide

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.
2 In the **Settings** window for **Explicit Selection**, type `All domains before oxide` in the **Label** text field.
3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Object**.
4 Select the object **dif1** only.
5 Locate the **Resulting Selection** section. From the **Show in physics** list, choose **Off**.

All exterior boundaries before oxide

1 In the **Geometry** toolbar, click **Selections** and choose **Adjacent Selection**.
2 In the **Settings** window for **Adjacent Selection**, type `All exterior boundaries before oxide` in the **Label** text field.
3 Locate the **Input Entities** section. Click **Add**.
4 In the **Add** dialog, select **All domains before oxide** in the **Input selections** list.
5 Click **OK**.

Top boundaries box

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.

2 In the **Settings** window for **Box Selection**, type `Top boundaries box` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Box Limits** section. In the **r minimum** text field, type `1`.

5 In the **r maximum** text field, type `49`.

6 In the **z minimum** text field, type `-4.2`.

7 Locate the **Resulting Selection** section. From the **Show in physics** list, choose **Off**.

Top boundaries for oxide

1 In the **Geometry** toolbar, click **Selections** and choose **Intersection Selection**.

2 In the **Settings** window for **Intersection Selection**, type `Top boundaries for oxide` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog, select **All exterior boundaries before oxide** in the **Selections to intersect** list.

6 Click **OK**.

7 In the **Settings** window for **Intersection Selection**, locate the **Input Entities** section.

8 Click **Add**.

9 In the **Add** dialog, select **Top boundaries box** in the **Selections to intersect** list.

10 Click **OK**.

11 In the **Settings** window for **Intersection Selection**, locate the **Resulting Selection** section.

12 From the **Show in physics** list, choose **Off**.

Oxide offset

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

2 In the **Settings** window for **Offset**, type `Oxide offset` in the **Label** text field.

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

4 From the **Input entities** list, choose **Top boundaries for oxide**.

5 Locate the **Options** section. In the **Distance** text field, type `t_ox`.

Move 1 (mov1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Move**.

2 Select the object **dif1** only.

3 In the **Settings** window for **Move**, locate the **Displacement** section.

4 In the **z** text field, type `t_ox`.

5 Select the object **pt1** only.

6 Locate the **Input** section. Select the **Keep input objects** checkbox.

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 On the object **off1**, select Point 1 only.

3 In the **Settings** window for **Line Segment**, locate the **Endpoint** section.

4 Click to select the **Activate Selection** toggle button for **End vertex**.

5 On the object **dif1**, select Point 7 only.

Line Segment 2 (ls2)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 On the object **pt1**, select Point 1 only.

3 In the **Settings** window for **Line Segment**, locate the **Endpoint** section.

4 Click to select the **Activate Selection** toggle button for **End vertex**.

5 On the object **mov1**, select Point 1 only.

Line Segment 3 (ls3)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 On the object **off1**, select Point 17 only.

3 In the **Settings** window for **Line Segment**, locate the **Endpoint** section.

4 Click to select the **Activate Selection** toggle button for **End vertex**.

5 On the object **dif1**, select Point 29 only.

Line Segment 4 (ls4)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 On the object **off1**, select Point 18 only.

3 In the **Settings** window for **Line Segment**, locate the **Endpoint** section.

4 Click to select the **Activate Selection** toggle button for **End vertex**.

5 On the object **dif1**, select Point 34 only.

Line Segment 5 (ls5)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 On the object **off1**, select Point 27 only.

3 In the **Settings** window for **Line Segment**, locate the **Endpoint** section.

4 Click to select the **Activate Selection** toggle button for **End vertex**.

5 On the object **dif1**, select Point 46 only.

Line Segment 1 (ls1), Line Segment 2 (ls2), Line Segment 3 (ls3), Line Segment 4 (ls4), Line Segment 5 (ls5)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1**, Ctrl-click to select **Line Segment 1 (ls1)**, **Line Segment 2 (ls2)**, **Line Segment 3 (ls3)**, **Line Segment 4 (ls4)**, and **Line Segment 5 (ls5)**.

2 Right-click and choose **Group**.

Oxide connecting lines

In the **Settings** window for **Group**, type `Oxide connecting lines` in the **Label** text field.

Convert to Solid 1 (csol1)

1 In the **Geometry** toolbar, click **Conversions** and choose **Convert to Solid**.

2 Select the objects **dif1**, **ls1**, **ls2**, **ls3**, **ls4**, **ls5**, **mov1**, and **off1** only.

Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

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

4 On the object **csol1**, select Domains 7 and 20 only.

Chamfer 1 (cha1)

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

2 On the object **del1**, select Points 9, 47, and 53 only.

3 In the **Settings** window for **Chamfer**, locate the **Distance** section.

4 In the **Distance from vertex** text field, type `t_ox`.

All domains

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `All domains` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Object**.

4 Select the object **cha1** only.

5 Locate the **Resulting Selection** section. From the **Show in physics** list, choose **Off**.

All exterior boundaries

1 In the **Geometry** toolbar, click **Selections** and choose **Adjacent Selection**.

2 In the **Settings** window for **Adjacent Selection**, type `All exterior boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. Click **Add**.

4 In the **Add** dialog, select **All domains** in the **Input selections** list.

5 Click **OK**.

6 In the **Settings** window for **Adjacent Selection**, locate the **Resulting Selection** section.

7 From the **Show in physics** list, choose **Off**.

Anode contact box

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.

2 In the **Settings** window for **Box Selection**, type `Anode contact box` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Box Limits** section. In the **r minimum** text field, type `1`.

5 In the **r maximum** text field, type `26.5`.

6 In the **z minimum** text field, type `-3.8`.

7 Locate the **Resulting Selection** section. From the **Show in physics** list, choose **Off**.

Anode contact

1 In the **Geometry** toolbar, click **Selections** and choose **Intersection Selection**.

2 In the **Settings** window for **Intersection Selection**, type `Anode contact` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog, select **Anode contact box** in the **Selections to intersect** list.

6 Click **OK**.

7 In the **Settings** window for **Intersection Selection**, locate the **Input Entities** section.

8 Click **Add**.

9 In the **Add** dialog, select **All exterior boundaries** in the **Selections to intersect** list.

10 Click **OK**.

Field contact ring 1 box

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.

2 In the **Settings** window for **Box Selection**, type `Field contact ring 1 box` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Box Limits** section. In the **r minimum** text field, type 27.8.

5 In the **r maximum** text field, type 35.

6 In the **z minimum** text field, type -3.8.

7 In the **z maximum** text field, type 1.

8 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

9 Locate the **Resulting Selection** section. From the **Show in physics** list, choose **Off**.

Field contact for ring 1

1 In the **Geometry** toolbar, click **Selections** and choose **Intersection Selection**.

2 In the **Settings** window for **Intersection Selection**, type `Field contact for ring 1` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog, select **All exterior boundaries** in the **Selections to intersect** list.

6 Click **OK**.

7 In the **Settings** window for **Intersection Selection**, locate the **Input Entities** section.

8 Click **Add**.

9 In the **Add** dialog, select **Field contact ring 1 box** in the **Selections to intersect** list.

10 Click **OK**.

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** > **SiC - Silicon Carbide** > **SiC - Silicon Carbide [solid, 4H Polytype]**.

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

5 In the tree, select **MEMS** > **Insulators** > **SiO2 - Silicon oxide**.

6 Click the **Add to Component** button in the window toolbar.

7 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

SiO2 - Silicon oxide (mat2)

Select Domains 7 and 23 only.

SEMICONDUCTOR (SEMI)

1 Click the **Show More Options** button in the **Model Builder** toolbar.
2 In the **Show More Options** dialog, click **Select All**.
3 Click **OK**.
4 In the **Model Builder** window, under **Component 1 (comp1)** click **Semiconductor (semi)**.
5 In the **Settings** window for **Semiconductor**, click to expand the **Reference Temperature** section.
6 In the T_0 text field, type `T_lattice`.

Semiconductor Material Model 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Semiconductor (semi)** click **Semiconductor Material Model 1**.
2 In the **Settings** window for **Semiconductor Material Model**, locate the **Model Input** section.
3 From the T list, choose **Common model input**.
4 Click **Go to Source** for **Temperature**.

GLOBAL DEFINITIONS

Default Model Inputs

1 In the **Model Builder** window, under **Global Definitions** click **Default Model Inputs**.
2 In the **Settings** window for **Default Model Inputs**, locate the **Browse Model Inputs** section.
3 In the tree, select **General** > **Temperature (K) - minput.T**.
4 Find the **Expression for remaining selection** subsection. In the **Temperature** text field, type `T_lattice`.

SEMICONDUCTOR (SEMI)

Semiconductor Material Model 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Semiconductor (semi)** click **Semiconductor Material Model 1**.
2 In the **Settings** window for **Semiconductor Material Model**, click to expand the **Dopant Ionization** section.
3 From the **Dopant ionization** list, choose **Incomplete ionization**.

4 In the $\Delta E_d \equiv E_c - E_d$ text field, type `0.060[V]`.

5 In the $\Delta E_a \equiv E_a - E_v$ text field, type `0.190[V]`.

Arora Mobility Model (LI) 1

In the **Physics** toolbar, click **Attributes** and choose **Arora Mobility Model (LI)**.

Charge Conservation 1

1 In the **Physics** toolbar, click **Domains** and choose **Charge Conservation**.

2 Select Domains 7 and 23 only.

3 In the **Settings** window for **Charge Conservation**, locate the **Model Input** section.

4 From the T list, choose **Common model input**.

Impact Ionization Generation 1

1 In the **Physics** toolbar, click **Domains** and choose **Impact Ionization Generation**.

2 In the **Settings** window for **Impact Ionization Generation**, locate the **Domain Selection** section.

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

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**.

Impact Ionization Generation 1, Trap-Assisted Recombination 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Semiconductor (semi)**, Ctrl-click to select **Impact Ionization Generation 1** and **Trap-Assisted Recombination 1**.

2 Right-click and choose **Group**.

Generation and Recombination

In the **Settings** window for **Group**, type `Generation and Recombination` in the **Label** text field.

P-type contact doping

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

2 In the **Settings** window for **Analytic Doping Model**, type `P-type contact doping` in the **Label** text field.

3 Select Domains 6, 11, 14, 18, and 22 only.

4 Locate the **Impurity** section. In the N_{A0} text field, type `5e19[1/cm^3]`.

P-type anode doping

1 In the **Physics** toolbar, click **Domains** and choose **Analytic Doping Model**.
2 In the **Settings** window for **Analytic Doping Model**, type `P-type anode doping` in the **Label** text field.
3 Select Domains 4, 5, 9, 10, 12, 13, 16, 17, 20, and 21 only.
4 Locate the **Impurity** section. In the N_{A0} text field, type `1e18[1/cm^3]`.

N-type drift region

1 In the **Physics** toolbar, click **Domains** and choose **Analytic Doping Model**.
2 In the **Settings** window for **Analytic Doping Model**, type `N-type drift region` in the **Label** text field.
3 Select Domains 2, 3, 8, 15, 19, and 24 only.
4 Locate the **Impurity** section. From the **Impurity type** list, choose **Donor doping (n-type)**.
5 In the N_{D0} text field, type `2e15[1/cm^3]`.

N-type substrate

1 In the **Physics** toolbar, click **Domains** and choose **Analytic Doping Model**.
2 In the **Settings** window for **Analytic Doping Model**, type `N-type substrate` in the **Label** text field.
3 Select Domain 1 only.
4 Locate the **Impurity** section. From the **Impurity type** list, choose **Donor doping (n-type)**.
5 In the N_{D0} text field, type `2e18[1/cm^3]`.

N-type drift region, N-type substrate, P-type anode doping, P-type contact doping

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Semiconductor (semi)**, Ctrl-click to select **P-type contact doping**, **P-type anode doping**, **N-type drift region**, and **N-type substrate**.
2 Right-click and choose **Group**.

Doping

In the **Settings** window for **Group**, type `Doping` in the **Label** text field.

Anode Contact (voltage)

1 In the **Physics** toolbar, click **Boundaries** and choose **Metal Contact**.
2 In the **Settings** window for **Metal Contact**, type `Anode Contact (voltage)` in the **Label** text field.
3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Anode contact**.

4 In the list box, select **31 (not applicable)**.

5 Locate the **Terminal** section. In the **Terminal name** text field, type `anode`.

6 In the V_0 text field, type `-V_reverse`.

Anode field contact

1 In the **Physics** toolbar, click **Boundaries** and choose **Electric Potential**.

2 In the **Settings** window for **Electric Potential**, type `Anode field contact` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Anode contact**.

4 Locate the **Electric Potential** section. In the V_0 text field, type `-V_reverse`.

Cathode contact

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

2 In the **Settings** window for **Metal Contact**, type `Cathode contact` in the **Label** text field.

3 Select Boundary 2 only.

4 Locate the **Terminal** section. In the **Terminal name** text field, type `cathode`.

Field contact ring 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Electric Potential**.

2 In the **Settings** window for **Electric Potential**, type `Field contact ring 1` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Field contact for ring 1**.

4 Locate the **Electric Potential** section. In the V_0 text field, type `semi.V0_ring1`.

Contact ring 1

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

2 In the **Settings** window for **Metal Contact**, type `Contact ring 1` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Field contact for ring 1**.

4 Locate the **Terminal** section. In the **Terminal name** text field, type `ring1`.

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

MESH 1

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

Size 3

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

2 Drag and drop **Size 3** below **Size 2**.

3 Select Domains 4, 8–22, and 24 only.

4 In the **Settings** window for **Size**, locate the **Element Size** section.

5 From the **Predefined** list, choose **Finer**.

6 Click **Build All**.

HIGH-TEMPERATURE REVERSE SWEEP

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

2 In the **Settings** window for **Study**, type `High-Temperature Reverse Sweep` in the **Label** text field.

Step 2: Stationary

1 In the **Study** toolbar, click **Stationary**.

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

3 Select the **Auxiliary sweep** checkbox.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
V_reverse (Reverse bias)	`range(0,5,2500)`	V

Solution 1 (sol1)

1 In the **Model Builder** window, right-click **Solver Configurations** and choose **Show Default Solver**.

2 Expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, click **Dependent Variables 2**.

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

5 From the **Method** list, choose **Initial-value based**.

6 In the **Model Builder** window, click **Total Normal Current Density (comp1.semi.nJw)**.

7 In the **Settings** window for **Field**, locate the **Scaling** section.

8 From the **Method** list, choose **Manual**.

9 In the **Scale** text field, type `1e4`.

10 In the **Model Builder** window, click **Electric Potential (comp1.V)**.

11 In the **Settings** window for **Field**, locate the **Scaling** section.

12 From the **Method** list, choose **Manual**.

13 In the **Scale** text field, type 100.

14 In the **Model Builder** window, click **Terminal Voltage (comp1.semi.mc3.V0_ode)**.

15 In the **Settings** window for **State**, locate the **Scaling** section.

16 From the **Method** list, choose **Manual**.

17 In the **Scale** text field, type 10.

18 In the **Model Builder** window, expand the **High-Temperature Reverse Sweep** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 2** node, then click **Parametric 1**.

19 In the **Settings** window for **Parametric**, click to expand the **Continuation** section.

20 Select the **Tuning of step size** checkbox.

21 In the **Initial step size** text field, type 0.05.

22 In the **Minimum step size** text field, type 0.01.

23 In the **Maximum step size** text field, type 1.

24 Right-click **Parametric 1** and choose **Stop Condition**.

25 In the **Settings** window for **Stop Condition**, type Stop at breakdown current in the **Label** text field.

26 Locate the **Stop Expressions** section. Click + **Add**.

27 In the table, enter the following settings:

Stop expression	Stop if	Active	Description
abs(comp1.semi.I0_cathode)>I_stop	True (>=1)	√	Stop expression 1

28 Locate the **Output at Stop** section. From the **Add solution** list, choose **Step before stop**.

29 Clear the **Add information** checkbox.

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

RESULTS

I-V

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type I-V in the **Label** text field.

Global 1

1 Right-click **I-V** 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
abs(semi.I0_cathode)	A	Cathode current
abs(semi.I0_anode)	A	Anode current

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

5 In the **Expression** text field, type `-V_reverse`.

6 In the **I-V** toolbar, click **Plot**.

Electric Field

1 In the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Electric Field` in the **Label** text field.

Surface 1

1 Right-click **Electric Field** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1) > Semiconductor > Electric > semi.normE - Electric field norm - V/m**.

3 In the **Electric Field** toolbar, click **Plot**.

4 Click **Plot First**.

5 Click to expand the **Range** section. Select the **Manual color range** checkbox.

6 In the **Electric Field** toolbar, click **Plot**.

7 Click **Plot Last**.

Carrier Generation

1 In the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Carrier Generation` in the **Label** text field.

Surface 1

1 Right-click **Carrier Generation** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1) > Semiconductor > Generation and recombination > semi.Gii - Impact ionization generation term - 1/(m³·s)**.

3 In the **Carrier Generation** toolbar, click **Plot**.

4 Click to expand the **Range** section. Select the **Manual color range** checkbox.

5 In the **Maximum** text field, type `6.84739599908922E7`.

6 In the **Carrier Generation** toolbar, click **Plot**.