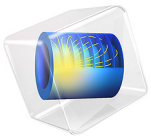
Created in COMSOL Multiphysics 6.0



Trench-Gate IGBT 2D

In this first half of a two-part example, a 2D model of a trench-gate IGBT is built, which will be extended to 3D in the second half. In general, it is the most efficient to start with a 2D model to make sure everything works as expected, before extending it to 3D. The Caughey-Thomas mobility model is combined with the Klaassen unified mobility model to account for velocity saturation and phonon, impurity, and carrier-carrier scattering. The contact resistance option of metal contact boundary conditions is used to implement the mixed-mode simulation with parasitic resistance at the collector and emitter as mentioned in the reference paper. The computed collector current density as a function of the collector voltage agrees reasonably well with the published result.

Introduction

In Ref. 1, Watanabe et al. studied the effect of three-dimensional current flow on the simulation result by comparing 2D and 3D models of trench-gate IGBTs. They found that the 3D model reveals a nonuniform current distribution in the third dimension in the high current regime, where the current in the MOS channel region is limited by the electron supply from the n^+-emitter. This nonuniform current distribution explains the reason why while the 3D model agrees well with measured result, the 2D model is off by the factor of the ratio of the n^+-emitter length to the total emitter length.

In this example, we start with the 2D model.

Model Definition

The model structure is detailed in Ref. 1, with additional details in Ref. 2.

Following the reference paper, the symmetry of the physics is used and only half of the cell is drawn in the geometry. Some thin regions are created under the gate and the emitter surface, in order to mesh those high-gradient regions with thin rectangles or isosceles trapezoids.

The **Klaassen Unified Mobility Model** and **Caughey-Thomas Mobility Model** are used. The band gap, effective density of states, and the band-gap narrowing reference concentration are modified according to Ref. 2. The **Contact resistance** option of metal contact boundary conditions is used to implement the mixed-mode simulation with parasitic resistance at the collector and emitter as mentioned in the reference paper.

See the comments in the section Modeling Instructions for more detailed discussions on the model construction, solution processes, and result visualization.

Results and Discussion

Figure 1 and Figure 2 show the collector current density as a function of the collector voltage, to be compared with Fig. 4(a) and (b) in Ref. 1. Reasonable agreement is seen.

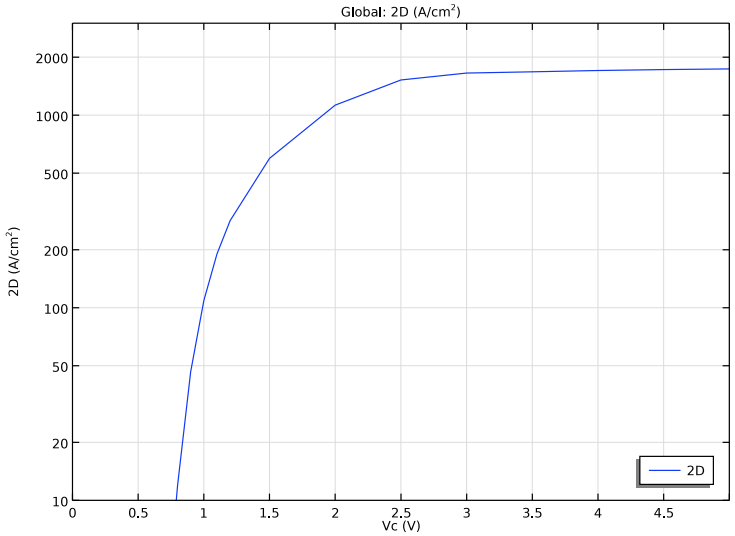


Figure 1: Collector current density as a function of the collector voltage, log scale.

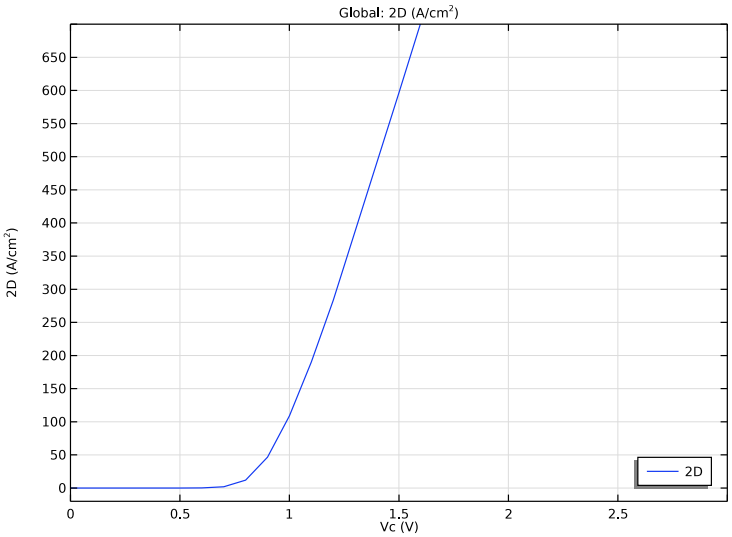


Figure 2: Collector current density as a function of the collector voltage, linear scale.

References

1. M. Watanabe and others, "Impact of three-dimensional current flow on accurate TCAD simulation for trench-gate IGBTs," *31st International Symposium on Power Semiconductor Devices and ICs (ISPSD)*, pp. 311–314, 2019, doi: 10.1109/ISPSD.2019.8757640.

2. N. Shigyo and others, "Modeling and Simulation of Si IGBTs," *2020 International Conference on Simulation of Semiconductor Processes and Devices (SISPAD)*, pp. 129–132, 2020, doi: 10.23919/SISPAD49475.2020.9241627.

Application Library path: `Semiconductor_Module/Transistors/trench_gate_igbt_2d`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.

2 In the **Select Physics** tree, select **Semiconductor>Semiconductor (semi)**.

3 Click **Add**.

4 Click **Study**.

It is usually a good practice to start the first study with a **Semiconductor Equilibrium** study step, which is easier to converge and provides a good initial value for subsequent study steps.

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

6 Click **Done**.

GEOMETRY 1

Set the length unit to a convenient one, in this model, micrometer.

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**.

Enter the device model parameters for the "k=3" case in the reference paper. Hide most of the parameters that will not be used parametric sweeps (in the first **Parameters** node). Use a second **Parameters** node for the terminal voltages that may be swept.

GLOBAL DEFINITIONS

Parameters 1

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
W	16[um]	1.6E-5 m	Device width
S	1[um]	1E-6 m	Mesa width
DT	2[um]	2E-6 m	Trench depth
WT	0.33[um]	3.3E-7 m	Trench width
Wewin	0.16[um]	1.6E-7 m	Emitter contact window
Dp	1.4[um]	1.4E-6 m	p-base depth
Dn	0.13[um]	1.3E-7 m	n+ emitter depth
tOX	33[nm]	3.3E-8 m	Oxide thickness
Dnb	120[um]	1.2E-4 m	n-base depth
Ln	1.5[um]	1.5E-6 m	n+ emitter length
Lp	1.5[um]	1.5E-6 m	p+ emitter length
Nab	3.8e17[cm^-3]	3.8E23 1/m³	p-base peak doping
Ndb	8.5e13[cm^-3]	8.5E19 1/m³	n-base doping
Ndbf	9e15[cm^-3]	9E21 1/m³	n-buffer doping
Nac	3.7e18[cm^-3]	3.7E24 1/m³	p+ collector doping
Nae	1e20[cm^-3]	1E26 1/m³	p+ emitter doping
Nde	1e20[cm^-3]	1E26 1/m³	n+ emitter doping
tau0	10[us]	1E-5 s	Carrier lifetime
tnbf	5[um]	5E-6 m	n-buffer thickness

Name	Expression	Value	Description
Dsub	10[um]	1E-5 m	p+ collector thickness
t0	Dp+Dnb+tnbf+Dsub	1.364E-4 m	Device thickness
d0	(Ln+Lp)/2	1.5E-6 m	Out-of-plan thickness
rhoCC	1.6e-4[ohm*cm^2]	1.6E-8 Ω·m²	p+ substrate equivalent resistivity
rhoCE	4.7e-6[ohm*cm^2]	4.7E-10 Ω·m²	Emitter contact resistivity
T0	300[K]	300 K	Lattice temperature
Eg0	1.1743[V]	1.1743 V	Band gap Shigyo
Nv_	(T0/300[K])^(3/2)* 1.04e19[1/cm^3]	1.04E25 1/m³	Valence band DOS COMSOL
Nc_	(T0/300[K])^(3/2)* 2.8e19[1/cm^3]	2.8E25 1/m³	Conduction band DOS COMSOL
ni0	1e10[cm^-3]	1E16 1/m³	Intrinsic carrier concentration
Nvcfac	sqrt(ni0^2/Nv_/ Nc_/exp(-Eg0* e_const/T0/ k_B_const))	4.2814	DOS factor to match ni
Nv0	Nvcfac*Nv_	4.4527E25 1/m³	Valence band DOS Shigyo
Nc0	Nvcfac*Nc_	1.1988E26 1/m³	Conduction band DOS Shigyo
Nref0	6.5e16[cm^-3]	6.5E22 1/m³	Band gap narrowing reference concentration Shigyo

4 Click to expand the **Visibility** section. Clear the **Show in parameter selections** check box.

Parameters 2

1 In the **Home** toolbar, click **Parameters** and choose **Add>Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
Vc	0[V]	0 V	Collector voltage
Vg	5[V]	5 V	Gate voltage

Build the geometry. We will take advantage of the symmetry of the physics, so draw only half of the cell. Make the emitter contact window slightly wider to ensure good contact to both the n+ and p+ doped regions due to their 2D arrangement. Create thin regions under the gate and the emitter surface, in order to mesh those high-gradient regions with thin rectangles or isosceles trapezoids. Those regions can still be merged with the rest of the geometry by using **Mesh Control Edges**.

GEOMETRY 1

Rectangle 1 - Device outline (half cell)

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

2 In the **Settings** window for **Rectangle**, type `Rectangle 1 - Device outline (half cell)` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `W/2`.

4 In the **Height** text field, type `t0`.

5 Locate the **Position** section. In the **y** text field, type `-t0`.

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

Layer name	Thickness (μm)
`Layer 1`	`Dsub`
`Layer 2`	`tnbf`

Rectangle 2 - Trench

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

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

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

4 In the **Height** text field, type `DT-WT/2`.

5 Locate the **Position** section. In the **x** text field, type `S/2`.

6 In the **y** text field, type `-(DT-WT/2)`.

Circle 1 - Trench

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

2 In the **Settings** window for **Circle**, type `Circle 1 - Trench` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Radius** text field, type `WT/2`.

4 Locate the **Position** section. In the **x** text field, type `S/2+WT/2`.

5 In the **y** text field, type `-(DT-WT/2)`.

Difference 1 - Device outline minus trench

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 In the **Settings** window for **Difference**, type `Difference 1 - Device outline minus trench` in the **Label** text field.

3 Select the object **r1** only.

4 Locate the **Difference** section. Find the **Objects to subtract** subsection. Click to select the **Activate Selection** toggle button.

5 Select the objects **c1** and **r2** only.

Point 1 - Emitter contact & doping boundary

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

2 In the **Settings** window for **Point**, type `Point 1 - Emitter contact & doping boundary` in the **Label** text field.

3 Locate the **Point** section. In the **x** text field, type `S/4-Wewin/1.5 S/4 S/4+Wewin/1.5`.

4 In the **y** text field, type `0 0 0`.

Rectangle 3 - Mesh help lines

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

2 In the **Settings** window for **Rectangle**, type `Rectangle 3 - Mesh help lines` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `S/2`.

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

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

Rectangle 4 - Mesh help lines

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

2 In the **Settings** window for **Rectangle**, type `Rectangle 4 - Mesh help lines` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `30[nm]`.

4 In the **Height** text field, type `DT-WT/2`.

5 Locate the **Position** section. In the **x** text field, type `S/2-30[nm]`.

6 In the **y** text field, type `-(DT-WT/2)`.

Rectangle 5- Mesh help lines

1 Right-click **Rectangle 4 - Mesh help lines** and choose **Duplicate**.

2 In the **Settings** window for **Rectangle**, type `Rectangle 5- Mesh help lines` in the **Label** text field.

3 Locate the **Position** section. In the **x** text field, type `S/2+WT`.

Circle 2 - Mesh help curves

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Circle 1 - Trench (c1)** and choose **Duplicate**.

2 In the **Settings** window for **Circle**, type `Circle 2 - Mesh help curves` in the **Label** text field.

3 Locate the **Object Type** section. From the **Type** list, choose **Curve**.

4 Locate the **Size and Shape** section. In the **Radius** text field, type `WT/2+30[nm]`.

5 In the **Sector angle** text field, type `180`.

6 Locate the **Rotation Angle** section. In the **Rotation** text field, type `180`.

Delete Entities 1 (del1)

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

2 On the object **c2**, select Boundaries 3 and 4 only.

Line Segment 1 - Mesh help line

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

2 In the **Settings** window for **Line Segment**, type `Line Segment 1 - Mesh help line` in the **Label** text field.

3 Locate the **Starting Point** section. From the **Specify** list, choose **Coordinates**.

4 In the **y** text field, type `-DT*8`.

5 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

6 In the **y** text field, type `-DT*8`.

7 In the **x** text field, type `W/2`.

Mesh Control Edges 1 (mce1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Mesh Control Edges**.

2 On the object **fin**, select Boundaries 8, 10, 15–18, 23, 25, 31, and 33 only.

3 In the **Geometry** toolbar, click **Build All**.

Add the built-in silicon material. Some properties will be replaced in the physics settings according to the reference paper.

ADD MATERIAL

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

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

3 In the tree, select **Semiconductors>Si - Silicon**.

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

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

Set up the physics. After creating the mobility subnodes, remember to select the desired mobility model in the **Semiconductor Material Model** parent node.

SEMICONDUCTOR (SEMI)

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

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

3 In the d text field, type `d0`.

4 Locate the **Model Properties** section. From the **Carrier statistics** list, choose **Fermi-Dirac**.

Semiconductor Material Model 1

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

Klaassen Unified Mobility Model (LIC) 1

In the **Physics** toolbar, click **Attributes** and choose **Klaassen Unified Mobility Model (LIC)**.

Semiconductor Material Model 1

In the **Model Builder** window, click **Semiconductor Material Model 1**.

Caughey-Thomas Mobility Model (E) 1

1 In the **Physics** toolbar, click **Attributes** and choose **Caughey-Thomas Mobility Model (E)**.

2 In the **Settings** window for **Caughey-Thomas Mobility Model (E)**, locate the **Input Mobilities** section.

3 From the $\mu_{n,in}$ list, choose **Electron mobility, Klaassen unified (semi/smm1/mmkl1)**.

4 From the $\mu_{p,in}$ list, choose **Hole mobility, Klaassen unified (semi/smm1/mmkl1)**.

Semiconductor Material Model 1

1 In the **Model Builder** window, click **Semiconductor Material Model 1**.

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

3 In the T text field, type `T0`.

4 Locate the **Material Properties** section. From the $E_{g,0}$ list, choose **User defined**. In the associated text field, type `Eg0`.

5 From the N_v list, choose **User defined**. In the associated text field, type `Nv0`.

6 From the N_c list, choose **User defined**. In the associated text field, type `Nc0`.

7 Locate the **Mobility Model** section. From the μ_n list, choose **Electron mobility, Caughey-Thomas (semi/smm1/mmct1)**.

8 From the μ_p list, choose **Hole mobility, Caughey-Thomas (semi/smm1/mmct1)**.

9 Click to expand the **Band Gap Narrowing** section. From the **Band gap narrowing** list, choose **Slotboom**.

10 From the N_{ref} list, choose **User defined**. In the associated text field, type `Nref0`.

Analytic Doping Model - n-base

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

2 In the **Settings** window for **Analytic Doping Model**, type `Analytic Doping Model - n-base` in the **Label** text field.

3 Select Domain 3 only.

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

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

Analytic Doping Model - n-buffer

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

2 In the **Settings** window for **Analytic Doping Model**, type `Analytic Doping Model - n-buffer` in the **Label** text field.

3 Select Domain 2 only.

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

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

Analytic Doping Model - p+ collector

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

2 In the **Settings** window for **Analytic Doping Model**, type `Analytic Doping Model - p+ collector` in the **Label** text field.

3 Select Domain 1 only.

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

Geometric Doping Model - p-base

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

2 In the **Settings** window for **Geometric Doping Model**, type `Geometric Doping Model - p-base` in the **Label** text field.

3 Select Domain 3 only.

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

5 Locate the **Profile** section. In the d_j text field, type `Dp`.

6 In the N_b text field, type `Ndb`.

Boundary Selection for Doping Profile 1

1 In the **Model Builder** window, expand the **Geometric Doping Model - p-base** node, then click **Boundary Selection for Doping Profile 1**.

2 Select Boundaries 7–10 and 13 only.

Geometric Doping Model - p+ emitter

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

2 In the **Settings** window for **Geometric Doping Model**, type `Geometric Doping Model - p+ emitter` in the **Label** text field.

3 Select Domain 3 only.

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

5 Locate the **Profile** section. In the d_j text field, type `Dn`.

6 In the N_b text field, type `Ndb`.

Boundary Selection for Doping Profile 1

1 In the **Model Builder** window, expand the **Geometric Doping Model - p+ emitter** node, then click **Boundary Selection for Doping Profile 1**.

2 Select Boundaries 7 and 8 only.

Geometric Doping Model - n+ emitter

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

2 In the **Settings** window for **Geometric Doping Model**, type `Geometric Doping Model - n+ emitter` in the **Label** text field.

3 Select Domain 3 only.

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

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

6 Locate the **Profile** section. In the d_j text field, type `Dn`.

7 In the N_b text field, type `Ndb`.

Boundary Selection for Doping Profile 1

1 In the **Model Builder** window, expand the **Geometric Doping Model - n+ emitter** node, then click **Boundary Selection for Doping Profile 1**.

2 Select Boundaries 9 and 10 only.

Trap-Assisted Recombination 1

1 In the **Physics** toolbar, click **Domains** and choose **Trap-Assisted Recombination**.

2 In the **Settings** window for **Trap-Assisted Recombination**, locate the **Domain Selection** section.

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

4 Locate the **Shockley-Read-Hall Recombination** section. From the τ_n list, choose **User defined**. In the associated text field, type `tau0`.

5 From the τ_p list, choose **User defined**. In the associated text field, type `tau0`.

Metal Contact - Emitter

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

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

3 Select Boundaries 8 and 9 only.

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

5 Select the **Contact resistance** check box.

6 In the ρ_c text field, type `rhoCE`.

Metal Contact - Collector

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

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

3 Select Boundary 2 only.

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

5 In the V_0 text field, type `Vc`.

6 Select the **Contact resistance** check box.

7 In the ρ_c text field, type `rhoCC`.

Thin Insulator Gate 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Thin Insulator Gate**.

2 Select Boundaries 11 and 12 only.

3 In the **Settings** window for **Thin Insulator Gate**, locate the **Terminal** section.

4 In the **Terminal name** text field, type `G`.

5 In the V_0 text field, type `Vg`.

6 Locate the **Gate Contact** section. In the ε_{ins} text field, type `3.9`.

7 In the d_{ins} text field, type `tOX`.

Create the mesh. To reduce computation time, a relatively coarse mesh is used in this example.

MESH 1

Edge 1 - Metal contact

1 In the **Mesh** toolbar, click  **Edge**.

2 In the **Settings** window for **Edge**, type `Edge 1 - Metal contact` in the **Label** text field.

3 Select Boundaries 8 and 9 only.

4 Click to expand the **Control Entities** section. Clear the **Smooth across removed control entities** check box.

Distribution 1

1 Right-click **Edge 1 - Metal contact** and choose **Distribution**.

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

3 From the **Distribution type** list, choose **Predefined**.

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

5 In the **Element ratio** text field, type `4`.

6 Select the **Symmetric distribution** check box.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.

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

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `9.1`.

5 In the **Minimum element size** text field, type `0.04`.

6 In the **Maximum element growth rate** text field, type `1.25`.

7 In the **Curvature factor** text field, type `0.35`.

8 In the **Resolution of narrow regions** text field, type `1.1`.

Edge 2 - Emitter surface

1 In the **Mesh** toolbar, click  **Edge**.

2 In the **Settings** window for **Edge**, type `Edge 2 - Emitter surface` in the **Label** text field.

3 Select Boundary 7 only.

4 Locate the **Control Entities** section. Clear the **Smooth across removed control entities** check box.

Distribution 1

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

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

3 From the **Distribution type** list, choose **Predefined**.

4 In the **Number of elements** text field, type `3`.

5 In the **Element ratio** text field, type `6`.

6 Select the **Reverse direction** check box.

Edge 3 - Emitter surface

1 In the **Mesh** toolbar, click  **Edge**.

2 In the **Settings** window for **Edge**, type `Edge 3 - Emitter surface` in the **Label** text field.

3 Select Boundary 10 only.

4 Locate the **Control Entities** section. Clear the **Smooth across removed control entities** check box.

Distribution 1

1 Right-click **Edge 3 - Emitter surface** and choose **Distribution**.

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

3 From the **Distribution type** list, choose **Predefined**.

4 In the **Element ratio** text field, type `3`.

5 Select the **Symmetric distribution** check box.

Copy Edge 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Copying Operations> Copy Edge**.

2 Select Boundaries 7–10 only.

3 In the **Settings** window for **Copy Edge**, locate the **Destination Boundaries** section.

4 Click to select the **Activate Selection** toggle button.

5 Select Boundary 26 only.

6 Click to expand the **Control Entities** section. Clear the **Smooth across removed control entities** check box.

Mapped 1 - Emitter depth

1 In the **Mesh** toolbar, click **Mapped**.

2 In the **Settings** window for **Mapped**, type `Mapped 1 - Emitter depth` in the **Label** text field.

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

4 Select Domain 9 only.

5 Click to expand the **Control Entities** section. Clear the **Smooth across removed control entities** check box.

6 Click to expand the **Reduce Element Skewness** section. Select the **Adjust edge mesh** check box.

Distribution 1

1 Right-click **Mapped 1 - Emitter depth** and choose **Distribution**.

2 Select Boundaries 18 and 29 only.

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

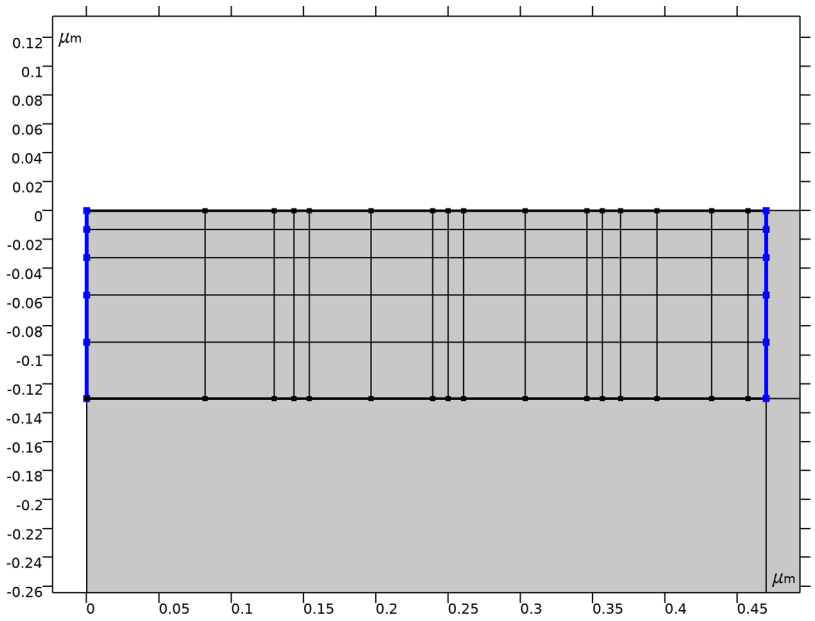
4 From the **Distribution type** list, choose **Predefined**.

5 In the **Element ratio** text field, type 3.

6 Select the **Reverse direction** check box.

7 Click the **Zoom to Selection** button in the **Graphics** toolbar.

8 Click **Build Selected**.



Mapped 2 - Gate depth

1 In the **Mesh** toolbar, click **Mapped**.

2 In the **Settings** window for **Mapped**, type `Mapped 2 - Gate depth` in the **Label** text field.

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

4 Select Domains 5–8 only.

5 Locate the **Control Entities** section. Clear the **Smooth across removed control entities** check box.

6 Locate the **Reduce Element Skewness** section. Select the **Adjust edge mesh** check box.

Distribution 1 - Left depth

1 Right-click **Mapped 2 - Gate depth** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, type `Distribution 1 - Left depth` in the **Label** text field.

3 Select Boundaries 19, 28, and 30 only.

4 Locate the **Distribution** section. From the **Distribution type** list, choose **Predefined**.

5 In the **Element ratio** text field, type `10`.

6 Select the **Reverse direction** check box.

Distribution 2 - Right depth

1 In the **Model Builder** window, right-click **Mapped 2 - Gate depth** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, type `Distribution 2 - Right depth` in the **Label** text field.

3 Select Boundaries 13 and 31 only.

4 Locate the **Distribution** section. From the **Distribution type** list, choose **Predefined**.

5 In the **Element ratio** text field, type `10`.

Distribution 3 - Left surface

1 Right-click **Mapped 2 - Gate depth** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, type `Distribution 3 - Left surface` in the **Label** text field.

3 Select Boundaries 20 and 27 only.

4 Locate the **Distribution** section. From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type `20`.

6 In the **Element ratio** text field, type `3`.

7 Select the **Symmetric distribution** check box.

Distribution 4 - Right surface

1 Right-click **Mapped 2 - Gate depth** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, type `Distribution 4 - Right surface` in the **Label** text field.

3 Select Boundaries 22 and 32 only.

4 Locate the **Distribution** section. From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type `6`.

6 In the **Element ratio** text field, type `4`.

Distribution 5 - Bottom surface

1 Right-click **Mapped 2 - Gate depth** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, type `Distribution 5 - Bottom surface` in the **Label** text field.

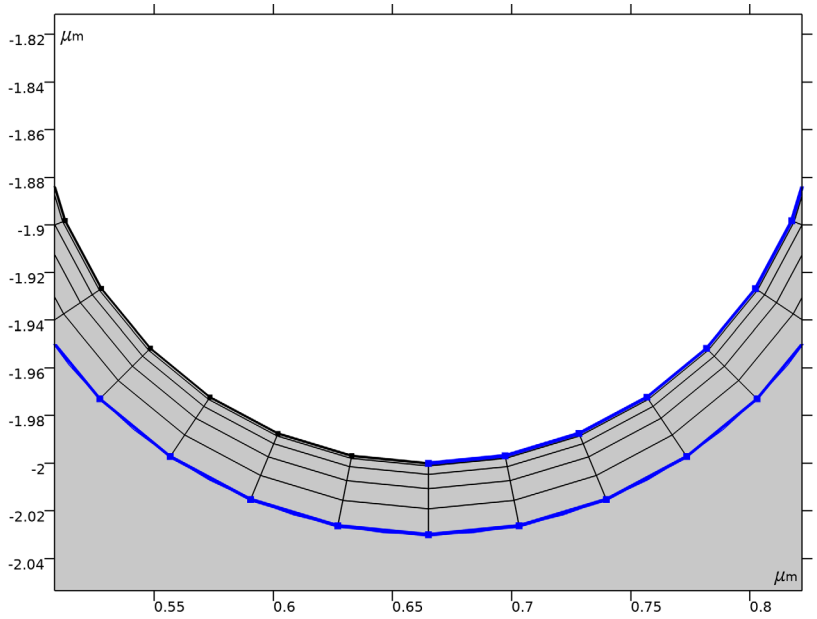
3 Select Boundaries 12, 21, 33, and 34 only.

4 Locate the **Distribution** section. From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type `8`.

6 Click the  **Zoom to Selection** button in the **Graphics** toolbar.

7 Click **Build Selected**.



Free Triangular 1

1 In the **Mesh** toolbar, click **Free Triangular**.

2 In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.

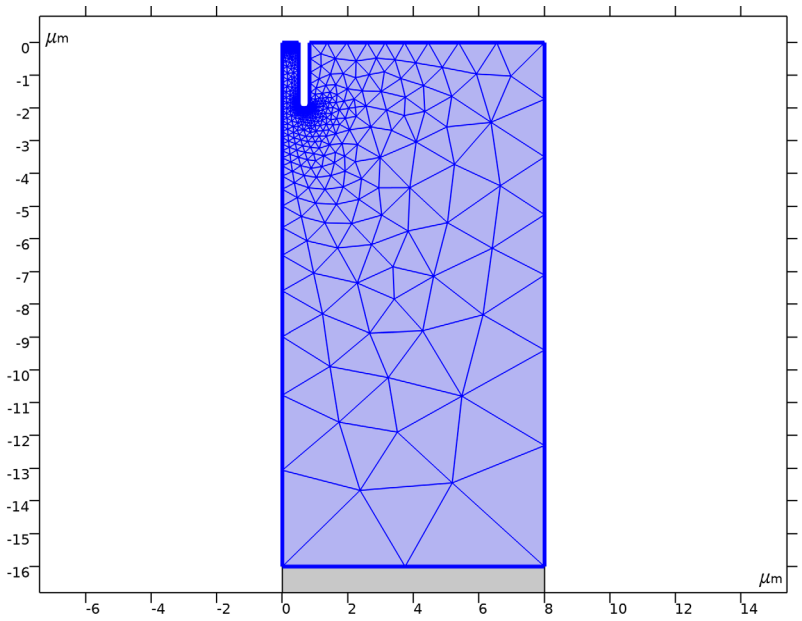
3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 4 only.

5 Click to expand the **Control Entities** section. Clear the **Smooth across removed control entities** check box.

6 Click the **Zoom to Selection** button in the **Graphics** toolbar.

7 Click **Build Selected**.



Copy Edge 2

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Copying Operations> Copy Edge**.

2 Select Boundary 25 only.

3 In the **Settings** window for **Copy Edge**, locate the **Destination Boundaries** section.

4 Click to select the **Activate Selection** toggle button.

5 Select Boundaries 2, 4, and 6 only.

6 Locate the **Control Entities** section. Clear the **Smooth across removed control entities** check box.

Mapped 3

1 In the **Mesh** toolbar, click **Mapped**.

2 In the **Settings** window for **Mapped**, locate the **Control Entities** section.

3 Clear the **Smooth across removed control entities** check box.

4 Locate the **Reduce Element Skewness** section. Select the **Adjust edge mesh** check box.

Distribution 1 - n-base

1 Right-click **Mapped 3** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, type `Distribution 1 - n-base` in the **Label** text field.

3 Select Boundaries 5 and 24 only.

4 Locate the **Distribution** section. From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type `10`.

6 In the **Element ratio** text field, type `5`.

7 Select the **Symmetric distribution** check box.

Distribution 2 - n-buffer

1 In the **Model Builder** window, right-click **Mapped 3** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, type `Distribution 2 - n-buffer` in the **Label** text field.

3 Select Boundaries 3 and 15 only.

4 Locate the **Distribution** section. From the **Distribution type** list, choose **Predefined**.

5 In the **Element ratio** text field, type `10`.

Distribution 3 - p+ collector

1 Right-click **Mapped 3** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, type `Distribution 3 - p+ collector` in the **Label** text field.

3 Select Boundaries 1 and 14 only.

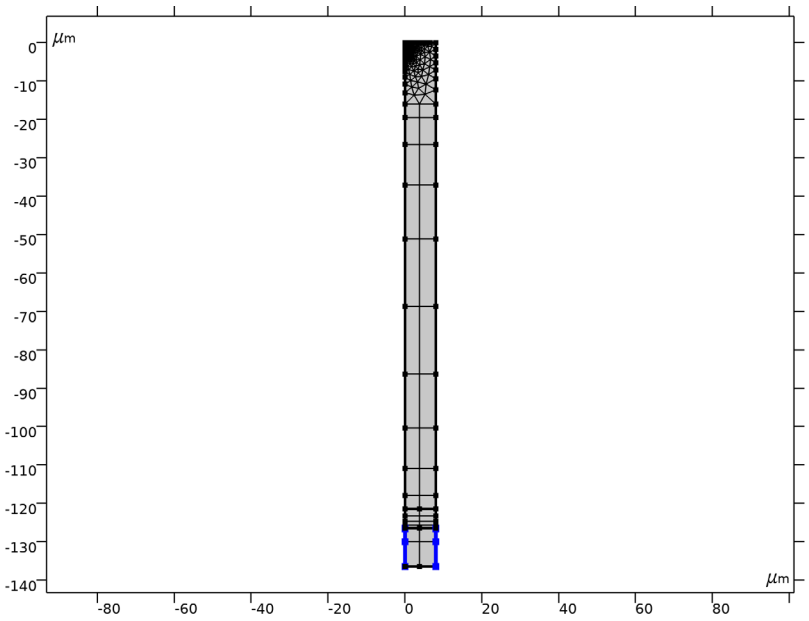
4 Locate the **Distribution** section. From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type `3`.

6 In the **Element ratio** text field, type `30`.

7 Select the **Reverse direction** check box.

8 Click **Build All**.



Add a Stationary study step to sweep the collector voltage from 0 to 5 V. For this second study step, use initial value based scaling and manual scaling for better error estimate.

STUDY 1

Stationary

1 In the **Study** toolbar, click **Study Steps** and choose **Stationary>Stationary**.

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

3 Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Vc (Collector voltage)	`range(0,0.1,1.2)` `range(1.5,0.5,5)`	V

6 In the **Study** toolbar, click **Get Initial Value**.

Solver Configurations

In the **Model Builder** window, expand the **Study 1>Solver Configurations** node.

Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)** node, then click **Dependent Variables 2**.

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

3 From the **Method** list, choose **Initial value based**.

4 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Dependent Variables 2** node, then click **Voltage drop across contact (comp1.semi.V_dae)**.

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

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

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

Plot the Jc-Vc curve in log and linear scales to compare with Fig. 4(a) and (b) in the reference paper.

RESULTS

J-V (log) - Fig.4(a)

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

2 In the **Settings** window for **1D Plot Group**, type `J-V (log) - Fig.4(a)` in the **Label** text field.

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

4 In the **x minimum** text field, type `0`.

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

6 In the **y minimum** text field, type `10`.

7 In the **y maximum** text field, type `3000`.

8 Select the **y-axis log scale** check box.

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

Global 1 - 2D

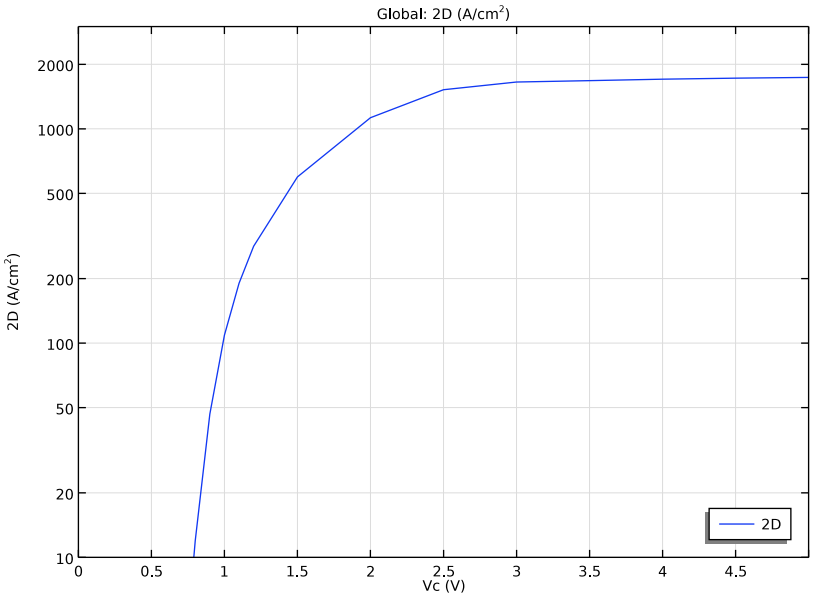
1 Right-click **J-V (log) - Fig.4(a)** and choose **Global**.

2 In the **Settings** window for **Global**, type `Global 1 - 2D` in the **Label** text field.

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

Expression	Unit	Description
`semi.J0_C`	`A/cm^2`	2D

4 In the **J-V (log) - Fig.4(a)** toolbar, click **Plot**.



J-V (linear) - Fig.4(b)

1 In the **Model Builder** window, right-click **J-V (log) - Fig.4(a)** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `J-V (linear) - Fig.4(b)` in the **Label** text field.

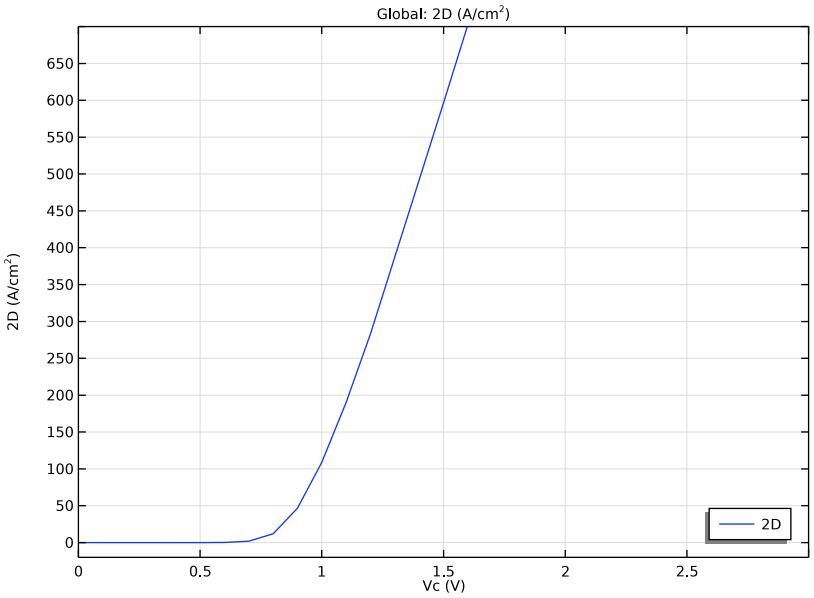
3 Locate the **Axis** section. In the **x maximum** text field, type `3`.

4 In the **y minimum** text field, type `-20`.

5 In the **y maximum** text field, type `700`.

6 Clear the **y-axis log scale** check box.

7 In the **J-V (linear) - Fig.4(b)** toolbar, click **Plot**.



Finally plot the electron and hole current streamlines on top of the electron concentration as the model thumbnail.

Electron Concentration & Current Streamlines

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

2 In the **Settings** window for **2D Plot Group**, type `Electron Concentration & Current Streamlines` in the **Label** text field.

3 Locate the **Plot Settings** section. Click **Go to Source**.

DEFINITIONS

Axis

1 In the **Model Builder** window, expand the **View 1** node, then click **Axis**.

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

3 In the **x minimum** text field, type `-0.1`.

4 In the **x maximum** text field, type `0.6`.

5 In the **y minimum** text field, type `-0.8`.

6 In the **y maximum** text field, type `0.05`.

7 Click **Update**.

RESULTS

Streamline 1 - Electron current

1 In the **Model Builder** window, right-click **Electron Concentration & Current Streamlines** and choose **Streamline**.

2 In the **Settings** window for **Streamline**, type `Streamline 1 - Electron current` in the **Label** text field.

3 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Semiconductor>Currents and charge>Electron current>semi.JnX,semi.JnY - Electron current density**.

4 Locate the **Streamline Positioning** section. In the **Number** text field, type `10`.

5 Select Boundary 9 only.

Streamline 2 - Hole current

1 Right-click **Electron Concentration & Current Streamlines** and choose **Streamline**.

2 In the **Settings** window for **Streamline**, type `Streamline 2 - Hole current` in the **Label** text field.

3 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Semiconductor>Currents and charge>Hole current>semi.JpX,semi.JpY - Hole current density**.

4 Locate the **Streamline Positioning** section. In the **Number** text field, type `10`.

5 Select Boundary 8 only.

6 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Color** list, choose **White**.

7 In the **Electron Concentration & Current Streamlines** toolbar, click **Plot**.