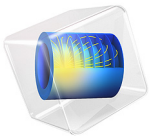
Created in COMSOL Multiphysics 6.2



A Cross-Bridge Kelvin Resistor Model for the Extraction of Specific Contact Resistivity

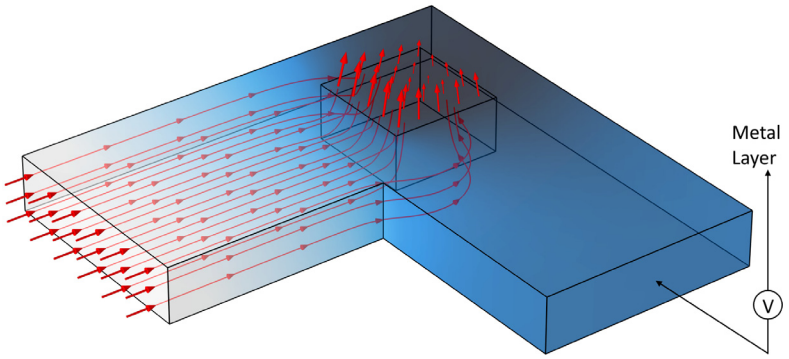
This benchmark example builds two models of a cross-bridge Kelvin resistor used for extracting the specific contact resistivity. The first model simulates the system in 3D, using the contact resistance feature in the Semiconductor interface. The other model is a 2D approximation of the system developed in the reference paper, implemented using a boundary PDE math interface. The computed Kelvin contact resistance as a function of the contact window size and the diffusion tap width correlates well between the two models, and agrees well with the result shown in the reference paper.

Introduction

Cross-bridge Kelvin resistor structures are commonly used to characterize the contact resistance of metal–semiconductor contacts. This tutorial examines a specific series of test structures described in the reference paper (Ref. 1), using a 3D semiconductor model and a 2D approximate model (the latter being developed in Ref. 2). The simulated Kelvin contact resistance as a function of the contact window size and the diffusion tap width will be compared between the two models and will be compared with the experimental and simulated results from Ref. 1.

Model Definition

The test structure is shown in the sketch below.

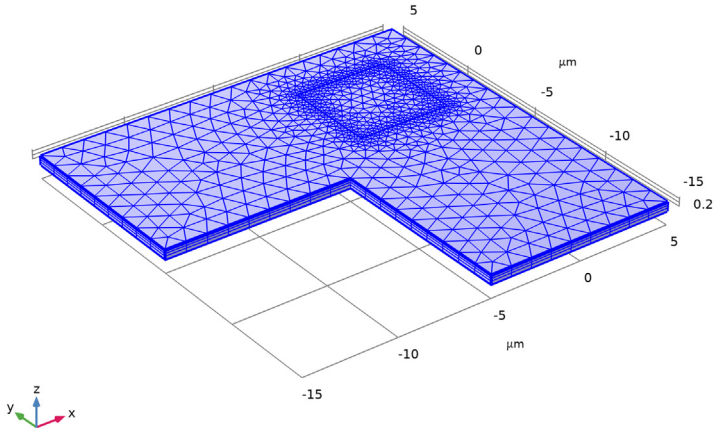


A 5000 Å thick n^+ doped polysilicon layer is patterned to an "L" shape as shown in the sketch above. Current is injected into the left arm of the "L" and exits through a square contact window to a metal layer above, as indicated by the red arrows (the metal layer is not shown in the sketch). A voltmeter measures the voltage drop between the right arm of the "L" and the metal layer. The *Kelvin contact resistance* R_c is defined as this voltage drop divided by the total current flowing through the contact window (SI unit: Ω).

The metal–semiconductor contact is characterized by the *specific contact resistivity* ρ_c (voltage drop divided by the current density within an infinitesimal contact area, SI unit: $\Omega \cdot m^2$). Because the distribution of the current density through the contact window is very nonuniform (current crowding effect), it is not straightforward to calculate the *specific contact resistivity* ρ_c from the *Kelvin contact resistance* R_c. Therefore, it is necessary to use numerical simulation to extract the *specific contact resistivity* ρ_c from experimental data.

In the experiment reported in Ref. 1, two series of test structures with various dimensions are used to verify the simulation approach. In the first series, the contact window size l is varied from 5 to 65 μm, while the diffusion tap width w is maintained 5 μm larger than l. In the second series, the diffusion tap width w is varied from 7.5 to 60 μm, while the contact window size l is kept constant at 5 μm. A 2D approximate model of the system (detailed in Ref. 2) is used to generate a family of R_c–l and R_c–w curves with various values of ρ_c. These curves are then compared with experimental data to obtain the optimal value of ρ_c for the contact (Fig. 2 and 3 in Ref. 1).

In this tutorial model, the geometry is parameterized, and **Parametric Sweep** is used in two separate studies for the two series of test structures discussed above. To save computation time and file size, only the optimal value of ρ_c is computed in this example. However, different values of ρ_c can easily be added by using **Auxiliary sweep**. The test structure is modeled both in 3D using the **Semiconductor** interface with its built-in Contact resistance functionality, and in 2D using the **General Form Boundary PDE** interface to implement the approximate model described in Ref. 2. The figure below shows the mesh.



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 shows the simulated R_c–area curve for the first series, to be compared with Fig. 2 in Ref. 1. The two models both agree well with the reference paper.

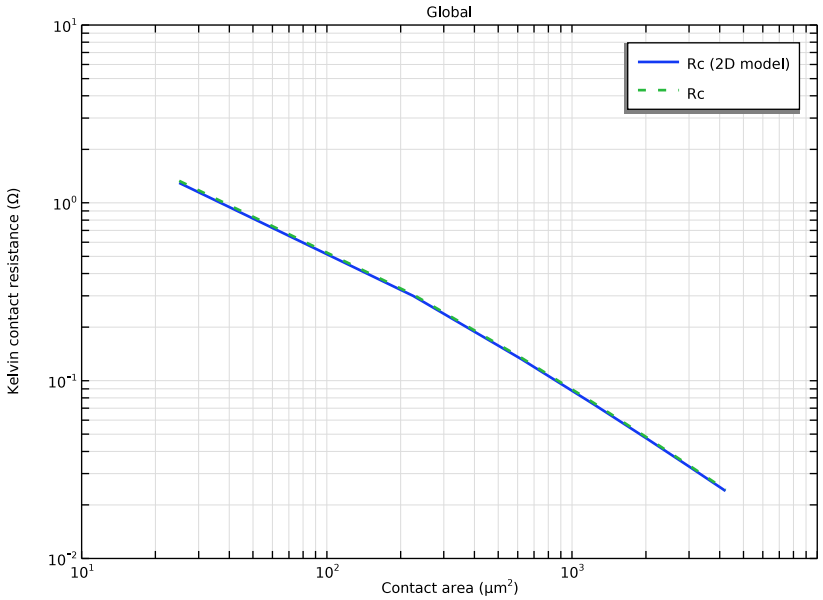


Figure 1: R_c–area curve for the first test structure series.

Figure 2 plots the R_c–w curve for the second series, to be compared with Fig. 3 in Ref. 1. The two models both agree well with the reference paper.

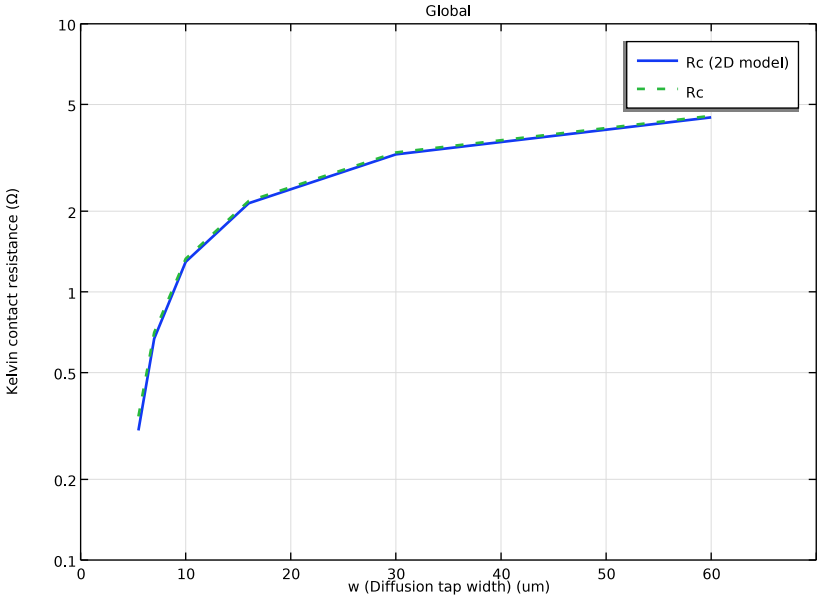


Figure 2: R_c–w curve for the second test structure series.

References

1. W.M. Loh, S.E. Swirhun, E. Crabbe, K. Saraswat, and R. M. Swanson, "An accurate method to extract specific contact resistivity using cross-bridge Kelvin resistors," *IEEE Electron Device Letters*, vol. 6, no. 9, pp. 441–443, 1985, doi: 10.1109/EDL.1985.26185.

2. W. M. Loh, S.E. Swirhun, T.A. Schreyer, R.M. Swanson, and K.C. Saraswat, "Modeling and measurement of contact resistances," *IEEE Transactions on Electron Devices*, vol. 34, no. 3, pp. 512–524, 1987, doi: 10.1109/T-ED.1987.22957.

Application Library path: `Semiconductor_Module/Device_Building_Blocks/cross_bridge_kelvin_resistor_contact_resistivity`

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

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, Use 3D geometry to model the cross-bridge Kelvin resistor.

2 click **3D**.

Add the Semiconductor physics (to be solved in 3D) and a math interface for the 2D model (to be solved on the top surface of the 3D geometry). For the latter, set the dependent variable to V2D with the unit of voltage and also set the unit of the source term accordingly.

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

4 Click **Add**.

5 In the **Select Physics** tree, select **Mathematics>PDE Interfaces>Lower Dimensions> General Form Boundary PDE (gb)**.

6 Click **Add**.

7 In the **Field name (1)** text field, type V2D.

8 In the **Dependent variables (1)** table, enter the following settings:

V2D

9 Click **Define Dependent Variable Unit**.

10 In the **Dependent variable quantity** table, enter the following settings:

Dependent variable quantity	Unit
Custom unit	V

11 Click **Define Source Term Unit**.

12 In the **Source term quantity** table, enter the following settings:

Source term quantity	Unit
Custom unit	V*m^-2

Add a Stationary study.

13 Click **Study**.

14 In the **Select Study** tree, select **General Studies>Stationary**.

15 Click **Done**.

GEOMETRY 1

Set the length unit to µm for convenience.

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**.

Add model parameters.

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
l	5[um]	5E-6 m	Contact window size
w	l+5[um]	1E-5 m	Diffusion tap width
lb	w	1E-5 m	Length of bridge
d	5000[angstrom]	5E-7 m	Thickness of Si
Rs	11.0[ohm]	11 Ω	Sheet resistance
mun0	1450[cm^2/(V*s)]	0.145 $m^2/(V·s)$	Electron mobility
Nd0	1/(d*e_const*mun0*Rs)	7.8263E24 $1/m^3$	Doping concentration from Rs
rho_c	4.5e-8[ohm*cm^2]	4.5E-12 $Ω·m^2$	Specific contact resistivity
lt	sqrt(rho_c/Rs)	6.396E-7 m	Transfer length
V0	100[uV]	1E-4 V	Applied voltage

Create the geometry for the 5000 angstrom thick n+ silicon layer, using a work plane to first draw the 2D shape of the bridge and then extrude in the thickness direction to 3D. The contacts will be modeled as boundary conditions with the metal layer assumed to be a perfect conductor.

GEOMETRY 1

Work Plane 1 (wp1)

In the **Geometry** toolbar, click **Work Plane**.

Work Plane 1 (wp1)>Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Square 1 - Contact window

1. In the **Work Plane** toolbar, click **Square**.
2. In the **Settings** window for **Square**, type `Square 1 - Contact window` in the **Label** text field.
3. Locate the **Size** section. In the **Side length** text field, type `l`.
4. Locate the **Position** section. From the **Base** list, choose **Center**.

Polygon 1 - Bridge

1. In the **Work Plane** toolbar, click **Polygon**.
2. In the **Settings** window for **Polygon**, type `Polygon 1 - Bridge` in the **Label** text field.
3. Locate the **Coordinates** section. In the table, enter the following settings:

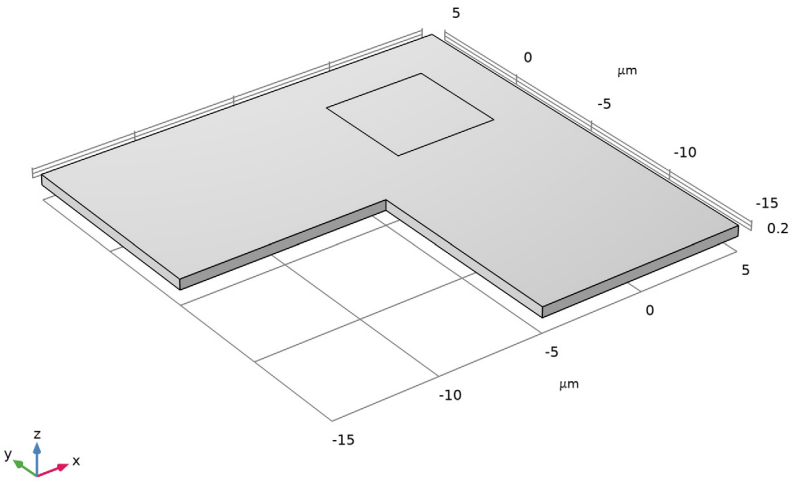
xw (µm)	yw (µm)
-w/2-lb	-w/2
-w/2-lb	w/2
w/2	w/2
w/2	-w/2-lb
-w/2	-w/2-lb
-w/2	-w/2
-w/2-lb	-w/2

Extrude 1 (ext1)

1. In the **Model Builder** window, right-click **Geometry 1** and choose **Extrude**.
2. In the **Settings** window for **Extrude**, locate the **Distances** section.
3. In the table, enter the following settings:

Distances (µm)
d

4 Click **Build All Objects**.



Add silicon material properties from the library. Use the electron mobility parameter `mun0` defined earlier.

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.

MATERIALS

Si - Silicon (mat1)

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

2 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Electron mobility	mun	`mun0`	$m^2/(V \cdot s)$	Semiconductor material

Set up physics conditions. Since the silicon is heavily n-doped, we only need to solve the majority carriers (electrons). Specify the uniform doping and three metal contacts (for the input current, bridge voltmeter, and contact window to the grounded upper metal layer).

SEMICONDUCTOR (SEMI)

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

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

3 From the **Solution** list, choose **Majority carriers only**.

Analytic Doping Model 1

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

2 In the **Settings** window for **Analytic Doping Model**, locate the **Domain Selection** section.

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

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

5 In the N_{D0} text field, type `Nd0`.

Metal Contact 1 - Input current

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

For the input current, it is easier to solve by using a voltage-driven metal contact with a small bias voltage parameter `V0` (defined earlier), and then read off the terminal current after solving.

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

3 Select Boundary 1 only.

4 Locate the **Terminal** section. In the V_0 text field, type `V0`.

Metal Contact 2 - Bridge voltmeter

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

For the contact to the bridge voltmeter, assume infinite impedance (zero current). Use the same voltage parameter `V0` as an offset to facilitate convergence.

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

3 Select Boundary 7 only.

4 Locate the **Terminal** section. From the **Terminal type** list, choose **Current**.

5 In the V_{init} text field, type `V0`.

Metal Contact 3 - Contact window

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

Use the built-in contact resistance option for the contact resistance at the contact window, which is connected to a grounded upper metal layer (zero volt). Define a selection of the contact window boundary for later use.

2 In the **Settings** window for **Metal Contact**, type `Metal Contact 3 - Contact window` in the **Label** text field.

3 Select Boundary 11 only.

4 Locate the **Boundary Selection** section. Click **Create Selection**.

5 In the **Create Selection** dialog box, type `Contact window` in the **Selection name** text field.

6 Click **OK**.

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

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

9 In the ρ_c text field, type `rho_c`.

Next set up the 2D model (Eq. 17 in the reference paper) using the math interface **General Form Boundary PDE**. Select the top surface of the bridge structure. The units of the dependent variable and source term have already been specified earlier.

GENERAL FORM BOUNDARY PDE (GB)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **General Form Boundary PDE (gb)**.

2 In the **Settings** window for **General Form Boundary PDE**, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 Select Boundaries 4 and 11 only.

Use the default domain condition for the left-hand side of Eq. 17 in the reference paper. Expand the **Equation** section of the Settings window to see how to adjust the inputs to match the equation in the paper. In this case, remove the minus signs of the flux components and zero out the source term and damping coefficient.

General Form PDE 1 - L.H.S. of Eq. 17 in ref. paper

1 In the **Model Builder** window, under **Component 1 (comp1)>** **General Form Boundary PDE (gb)** click **General Form PDE 1**.

2 In the **Settings** window for **General Form PDE**, type `General Form PDE 1 - L.H.S. of Eq. 17 in ref. paper` in the **Label** text field.

3 Click to expand the **Equation** section. Locate the **Conservative Flux** section. Specify the Γ vector as

V2DTx	x
V2DTy	y
V2DTz	z

4 Locate the **Source Term** section. In the *f* text field, type `0`.

5 Locate the **Damping or Mass Coefficient** section. In the d_a text field, type `0`.

Add a source term for the right-hand side of Eq. 17 in the reference paper. Select only the contact window - the source term is zero in other regions (Eq. 19 in the reference paper). The selection can be made easily by using the selection defined earlier, now appears in the **Selection:** drop-down menu.

Source 1 - R.H.S. of Eq. 17 in ref. paper

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

2 In the **Settings** window for **Source**, type `Source 1 - R.H.S. of Eq. 17 in ref. paper` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Contact window**.

4 Locate the **Source Term** section. In the *f* text field, type `V2D/1t^2`.

For the input current contact, apply the same bias voltage parameter `V0` at the corresponding boundary (also create a selection for later use). To obtain an accurate result for the terminal current, use the weak constraint, which becomes visible in the GUI after turning on **Advanced Physics Options**.

Dirichlet Boundary Condition 1 - Input current

1 In the **Physics** toolbar, click **Edges** and choose **Dirichlet Boundary Condition**.

2 In the **Settings** window for **Dirichlet Boundary Condition**, type `Dirichlet Boundary Condition 1 - Input current` in the **Label** text field.

3 Select Edge 4 only.

4 Locate the **Dirichlet Boundary Condition** section. In the *r* text field, type `V0`.

5 Locate the **Edge Selection** section. Click **Create Selection**.

6 In the **Create Selection** dialog box, type `Input current (2D model)` in the **Selection name** text field.

7 Click **OK**.

8 Click the **Show More Options** button in the **Model Builder** toolbar.

9 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Advanced Physics Options**.

10 Click **OK**.

11 In the **Settings** window for **Dirichlet Boundary Condition**, click to expand the **Constraint Settings** section.

12 Select the **Weak constraints** check box.

Define integration operators and variables to evaluate the terminal currents using the Lagrange multipliers from the weak constraints (using Eq. 21a from the reference paper) for the input current and the voltmeter current, and using Eq. 21b from the reference paper for the contact window current.

DEFINITIONS

Integration 1 - Input current

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

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

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

4 From the **Selection** list, choose **Input current (2D model)**.

Integration 2 - Bridge voltmeter

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `Integration 2 - Bridge voltmeter` in the **Label** text field.

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

4 Select Edge 13 only.

5 Click **Create Selection**.

6 In the **Create Selection** dialog box, type `Bridge voltmeter (2D model)` in the **Selection name** text field.

7 Click **OK**.

Integration 3 - Contact window

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `Integration 3 - Contact window` in the **Label** text field.

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

4 From the **Selection** list, choose **Contact window**.

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
I2D_1	intop1(V2D_lm[V/m]/Rs)	A	Input current (2D model)
I2D_2	intop2(V2D_lm[V/m]/Rs)	A	Voltmeter current (2D model)
I2D_3	intop3(V2D/rho_c)	A	Contact window current (2D model)

For the bridge voltmeter contact, apply an unknown bias voltage V2D_ode at the corresponding boundary (a selection has been defined earlier when defining the integration operator). Then solve for the unknown voltage using a global equation requiring that the voltmeter current I2D_2 is zero (assuming infinite impedance).

GENERAL FORM BOUNDARY PDE (GB)

Dirichlet Boundary Condition 2 - Bridge voltmeter

1 In the **Physics** toolbar, click **Edges** and choose **Dirichlet Boundary Condition**.

2 In the **Settings** window for **Dirichlet Boundary Condition**, type Dirichlet Boundary Condition 2 - Bridge voltmeter in the **Label** text field.

3 Locate the **Edge Selection** section. From the **Selection** list, choose **Bridge voltmeter (2D model)**.

4 Locate the **Dirichlet Boundary Condition** section. In the *r* text field, type V2D_ode.

5 Click to expand the **Constraint Settings** section. Select the **Weak constraints** check box.

Global Equations 1 - Bridge voltmeter

1 In the **Physics** toolbar, click **Global** and choose **Global Equations**.

2 In the **Settings** window for **Global Equations**, type Global Equations 1 - Bridge voltmeter in the **Label** text field.

3 Locate the **Units** section. Click **Define Dependent Variable Unit**.

4 In the **Dependent variable quantity** table, enter the following settings:

Dependent variable quantity	Unit
Custom unit	V

5 Click **Define Source Term Unit**.

6 In the **Source term quantity** table, enter the following settings:

Source term quantity	Unit
Custom unit	A

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

Name	f(u,ut,utt, t) (A)	Initial value (u_0) (V)	Initial value (u_t0) (V/s)	Description
V2D_ode	I2D_2	0	0	

Create a Swept mesh which is required by the default finite volume discretization. Make sure the mesh is fine enough around the perimeter of the contact window where large gradients exist by parameterizing the mesh sizes with geometric dimensions.

MESH 1

Edge 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Edge**.

2 Select Edges 18, 19, 22, and 25 only.

Distribution 1

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

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

3 In the **Number of elements** text field, type `20*1/5[um]`.

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 `1.2`.

5 In the **Maximum element growth rate** text field, type `1.3`.

Free Triangular 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Free Triangular**.

2 Select Boundaries 4 and 11 only.

Swept 1

In the **Mesh** toolbar, click **Swept**.

Distribution 1

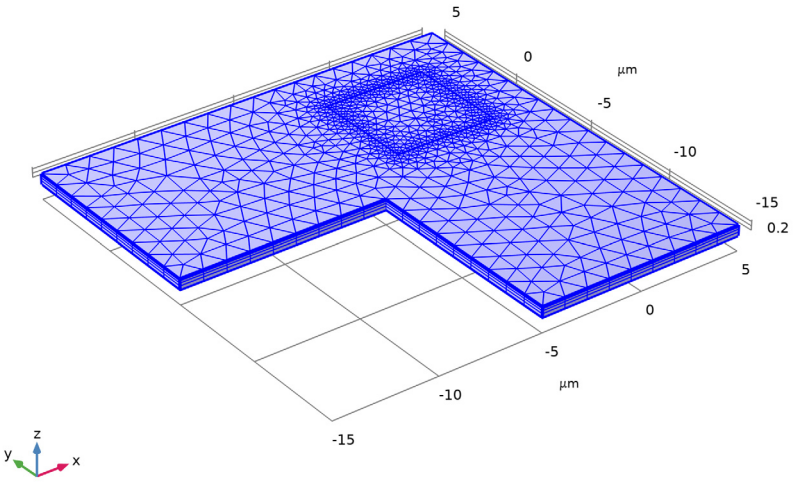
1 Right-click **Swept 1** and choose **Distribution**.

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 `5`.

5 Click **Build All**.



Add an outer **Parametric Sweep** to sweep the contact window size *l* from 5 to 65 um, to be compared with the result shown in Fig. 2 of the reference paper.

STUDY 1 - RC VS. L

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

2 In the **Settings** window for **Study**, type `Study 1 - Rc vs. l` in the **Label** text field.

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
l (Contact window size)	`range(5,10,65)`	`um`

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

Evaluate and compare the bridge voltage and the terminal currents from the 3D semiconductor model and 2D simplified model. There is a good correlation between the results from the 2D and 3D models. Within each model, the input current agrees with the contact window current, and the bridge voltmeter current is essentially zero, both as expected.

RESULTS

Evaluation Group 1 - Rc vs. l

1 In the **Results** toolbar, click **Evaluation Group**.

2 In the **Settings** window for **Evaluation Group**, type `Evaluation Group 1 - Rc vs. l` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1 - Rc vs. l/ Parametric Solutions 1 (sol2)**.

Global Evaluation 1

1 Right-click **Evaluation Group 1 - Rc vs. l** and choose **Global Evaluation**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`V2D_ode`	`V`	`Bridge voltmeter (2D model)`
`semi.V0_2`	`V`	`Bridge voltmeter`
`I2D_1`	`A`	`Input current (2D model)`
`I2D_3`	`A`	`Contact window current (2D model)`
`semi.I0_1`	`A`	`Input current`
`semi.I0_3`	`A`	`Contact window current`

Expression	Unit	Description
I2D_2	A	Voltmeter current (2D model)
semi.IO_2	A	Voltmeter current

4 In the **Evaluation Group 1 - Rc vs. l** toolbar, click **Evaluate**.

Create a plot of the simulated Kelvin contact resistance R_c as a function of the contact window area, to be compared with Fig. 2 of the reference paper. The 2D and 3D models give very similar results and agree with the figure in the paper.

Rc vs. l

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

2 In the **Settings** window for **1D Plot Group**, type `Rc vs. l` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1 - Rc vs. l/ Parametric Solutions 1 (sol2)**.

4 Locate the **Plot Settings** section.

5 Select the **y-axis label** check box. In the associated text field, type `Kelvin contact resistance ([Omega])`.

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

7 In the **x minimum** text field, type `10`.

8 In the **x maximum** text field, type `1e4`.

9 In the **y minimum** text field, type `0.01`.

10 In the **y maximum** text field, type `10`.

11 Select the **x-axis log scale** check box.

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

Global 1

1 Right-click **Rc vs. l** 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
-V2D_ode/I2D_1	Ω	Rc (2D model)
semi.VO_2/semi.IO_1	Ω	Rc

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

5 In the **Expression** text field, type `l^2`.

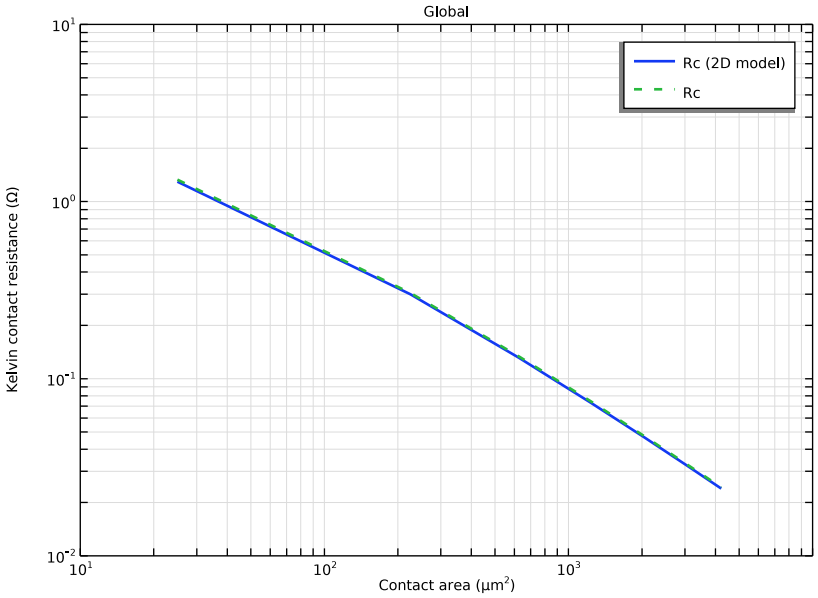
6 From the **Unit** list, choose **µm^2**.

7 Select the **Description** check box. In the associated text field, type `Contact area`.

8 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Cycle**.

9 From the **Width** list, choose **2**.

10 In the **Rc vs. I** toolbar, click **Plot**.



Repeat the steps to create a study to sweep the diffusion tap width w, while keeping the contact window size l at 5 um, to be compared with the result shown in Fig. 3 of the reference paper.

ADD STUDY

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

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

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

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

STUDY 2 - RC VS. W

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

2 In the **Settings** window for **Study**, type `Study 2 - Rc vs. w` in the **Label** text field.

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
w (Diffusion tap width)	`5.5 7 10 16 30 60`	`um`

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

Again evaluate and compare the bridge voltage and the terminal currents from the 3D semiconductor model and 2D simplified model. There is a good correlation between the results from the 2D and 3D models. Within each model, the input current agrees with the contact window current, and the bridge voltmeter current is essentially zero, both as expected.

RESULTS

Evaluation Group 1 - Rc vs. I

In the **Model Builder** window, under **Results** right-click **Evaluation Group 1 - Rc vs. I** and choose **Duplicate**.

Evaluation Group 2 - Rc vs. w

1 In the **Model Builder** window, under **Results** click **Evaluation Group 1 - Rc vs. I 1**.

2 In the **Settings** window for **Evaluation Group**, type `Evaluation Group 2 - Rc vs. w` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - Rc vs. w/ Parametric Solutions 2 (sol11)**.

4 In the **Evaluation Group 2 - Rc vs. w** toolbar, click **Evaluate**.

Duplicate the plot of the simulated Kelvin contact resistance R_c and make it a function of the diffusion tap width w, to be compared with Fig. 3 of the reference paper. The 2D and 3D models give very similar results and agree with the figure in the paper.

Rc vs. I

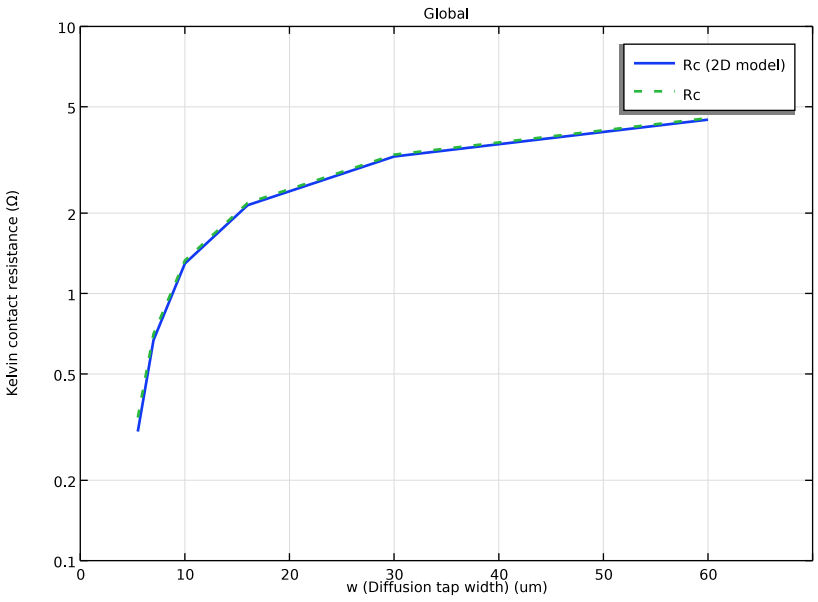
In the **Model Builder** window, right-click **Rc vs. I** and choose **Duplicate**.

Rc vs. w

1. In the **Model Builder** window, under **Results** click **Rc vs. I 1**.
2. In the **Settings** window for **1D Plot Group**, type `Rc vs. w` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Study 2 - Rc vs. w/ Parametric Solutions 2 (sol11)**.
4. Locate the **Axis** section. In the **x minimum** text field, type `0`.
5. In the **x maximum** text field, type `70`.
6. In the **y minimum** text field, type `0.1`.
7. Clear the **x-axis log scale** check box.

Global 1

1. In the **Model Builder** window, expand the **Rc vs. w** node, then click **Global 1**.
2. In the **Settings** window for **Global**, locate the **x-Axis Data** section.
3. From the **Parameter** list, choose **Parameter value**.
4. In the **Rc vs. w** toolbar, click **Plot**.



Create a 3D plot for the model thumbnail.

Electric Potential (semi)

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

Model Thumbnail

1 In the **Model Builder** window, under **Results** click **Electric Potential (semi) 2**.

2 In the **Settings** window for **3D Plot Group**, type `Model Thumbnail` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter value (l (um))** list, choose **5**.

4 Locate the **Plot Settings** section. From the **View** list, choose **New view**.

5 Locate the **Color Legend** section. Clear the **Show legends** check box.

6 In the **Model Thumbnail** toolbar, click **Plot**.

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

Camera

1 In the **Model Builder** window, expand the **View 3D 3** node, then click **Camera**.

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

3 From the **View scale** list, choose **Manual**.

4 In the **z scale** text field, type `5`.

5 Click **Update**.

Volume 1

1 In the **Model Builder** window, expand the **Results>Model Thumbnail** node, then click **Volume 1**.

2 In the **Settings** window for **Volume**, locate the **Coloring and Style** section.

3 Click **Change Color Table**.

4 In the **Color Table** dialog box, select **Aurora>JupiterAuroraBorealis** in the tree.

5 Click **OK**.

Transparency 1

Right-click **Volume 1** and choose **Transparency**.

Streamline 1

1 In the **Model Builder** window, right-click **Model Thumbnail** and choose **Streamline**.

2 In the **Settings** window for **Streamline**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Semiconductor>Currents and charge>semi.JX,...,semi.JZ - Total current density, nodal value**.

3 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Starting-point controlled**.

4 From the **Entry method** list, choose **Coordinates**.

5 In the **X** text field, type `-15 -15 -15 -15 -15 -15 -15 -15 -15 -15 -15 -15 -15 -15 -15 -15 -15 -15`.

6 In the **Y** text field, type `-4.17 -2.5 -0.83 0.83 2.5 4.17 -4.17 -2.5 -0.83 0.83 2.5 4.17 -4.17 -2.5 -0.83 0.83 2.5 4.17`.

7 In the **Z** text field, type `0.083 0.083 0.083 0.083 0.083 0.083 0.25 0.25 0.25 0.25 0.25 0.25 0.417 0.417 0.417 0.417 0.417 0.417`.

8 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Type** list, choose **Arrow**.

Arrow Surface 1

1 Right-click **Model Thumbnail** and choose **Arrow Surface**.

2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Semiconductor>Currents and charge>semi.JX,...,semi.JZ - Total current density, nodal value**.

3 Locate the **Arrow Positioning** section. From the **Placement** list, choose **Uniform anisotropic**.

4 In the **Number of arrows** text field, type `18`.

5 In the **Z weight** text field, type `10`.

6 Locate the **Coloring and Style** section. From the **Arrow base** list, choose **Head**.

7 Select the **Scale factor** check box. In the associated text field, type `1e-6`.

Selection 1

1 Right-click **Arrow Surface 1** and choose **Selection**.

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

3 Click  **Paste Selection**.

4 In the **Paste Selection** dialog box, type `1` in the **Selection** text field.

5 Click **OK**.

Arrow Surface 2

1 In the **Model Builder** window, right-click **Model Thumbnail** and choose **Arrow Surface**.

2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>**

Semiconductor>Currents and charge>semi.JX,...,semi.JZ - Total current density, nodal value.

3 Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type `30`.

4 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Logarithmic**.

5 Select the **Scale factor** check box. In the associated text field, type `1.5e-6`.

Selection 1

1 Right-click **Arrow Surface 2** and choose **Selection**.

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

3 From the **Selection** list, choose **Contact window**.

4 In the **Model Thumbnail** toolbar, click **Plot**.