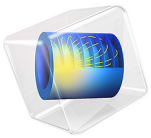
Created in COMSOL Multiphysics 6.2



MOSCAP ID Small Signal

This tutorial uses a simple 1D model of a metal–oxide–silicon capacitor (MOSCAP) to illustrate the basic steps to set up and perform a semiconductor simulation with the COMSOL Semiconductor Module. The model employs the quasi-Fermi level formulation and the Semiconductor Equilibrium study step.

Introduction

The metal–silicon–oxide (MOS) structure is the fundamental building block for many silicon planar devices. Its capacitance measurements provide a wealth of insight into the working principles of such devices. This tutorial constructs a simple 1D model of a MOS capacitor (MOSCAP). Both the low- and high-frequency C–V curves are computed using the approach of small-signal analysis (the related model moscap_1d shows the transient approach). The model employs the quasi-Fermi level formulation and the Semiconductor Equilibrium study step. Two alternative methods to compute the differential capacitance are demonstrated, using either the gate terminal or the metal contact terminal, respectively.

Model Definition

This model simulates the behavior of the MOSCAP under an array of DC bias voltages between -2 and 1 V, plus a small-signal harmonic perturbation of 1 mV with frequencies of 10^{-2} Hz and 10^{4} Hz for the low- and high- frequency cases, respectively. The modeled domain has a thickness of 10 um. The built-in Si material data is used.

The device is grounded at the right endpoint and the oxide/silicon interface is placed at the left endpoint, using the dedicated Thin Insulator Gate boundary condition with the Harmonic Perturbation subnode. A uniform doping and Shockley–Read–Hall recombination is applied to the entire modeling domain.

A user-controlled mesh is used to refine the mesh under the oxide/silicon interface. The small-signal analysis is done using a Semiconductor Equilibrium study step for the DC operating points, followed by a Frequency Domain Perturbation study step for the AC perturbation.

Results and Discussion

Figure 1shows the C–V curves for the low- and high-frequency cases. They exhibit the typical behavior seen in textbooks such as in Ref. 1.

.

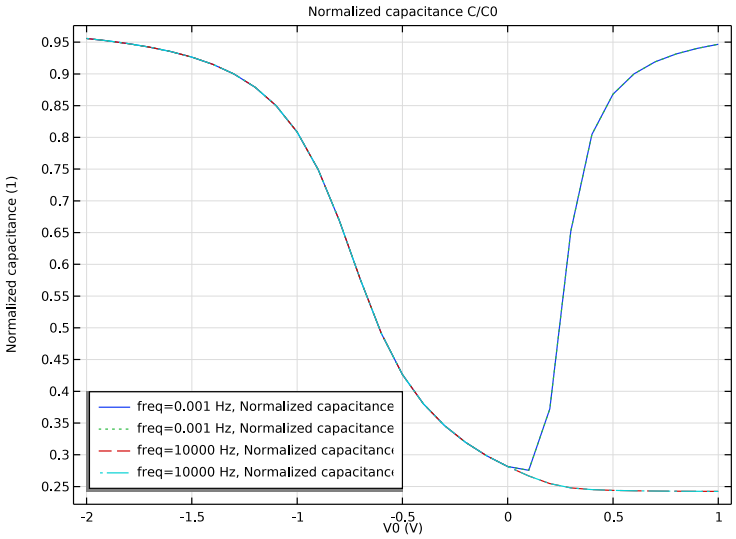


Figure 1: C–V curves for the low- and high-frequency cases.

Reference

1. Sze and Ng, *Physics of Semiconductor Devices*, 3rd ed., John Wiley & Sons, Chap. 4.

Application Library path: `Semiconductor_Module/Device_Building_Blocks/moscap_1d_small_signal`

Modeling Instructions

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

NEW

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

MODEL WIZARD

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

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

3 Click **Add**.

In this tutorial we use the approach of small signal analysis to compute the values of the differential capacitance at an array of DC bias points. For these DC operating points, since there is no current flowing, it is best to use the **Semiconductor Equilibrium** study step.

4 Click **Study**.

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

6 Click **Done**.

GEOMETRY 1

The Model Wizard starts the COMSOL Desktop with the **Geometry** node selected. We can take the opportunity to set the length unit to a more convenient one.

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**.

The geometry is a simple line interval of length 10 um.

Interval 1 (i1)

1 Right-click **Component 1 (comp1)>Geometry 1** and choose **Interval**.

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

3 In the table, enter the following settings:

Coordinates (µm)
0
10

Now import some global parameters from a text file.

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 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `moscap_1d_small_signal.txt`.

Add built-in silicon material and close the Add Material pane when done.

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 physics. We will demonstrate the quasi-Fermi level formulation.

SEMICONDUCTOR (SEMI)

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

2 In the **Settings** window for **Semiconductor**, locate the **Cross-Section Area** section.

3 In the A text field, type `area`.

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

5 Click to expand the **Discretization** section. From the **Formulation** list, choose **Finite element quasi Fermi level (quadratic shape function)**.

Ground the device at the right endpoint.

Metal Contact 1

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

2 Click the **Zoom Extents** button in the **Graphics** toolbar.

3 Select Boundary 2 only.

Place the oxide at the left endpoint, using the dedicated **Thin Insulator Gate** boundary condition.

Thin Insulator Gate 1

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

2 Select Boundary 1 only.

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

4 In the V_0 text field, type `V0`.

5 Locate the **Gate Contact** section. In the ε_{ins} text field, type `epsr0x`.

6 In the d_{ins} text field, type `d0x`.

Use the **Harmonic Perturbation** subnode to set up the small signal analysis.

Harmonic Perturbation 1

1 In the **Physics** toolbar, click **Attributes** and choose **Harmonic Perturbation**.

2 In the **Settings** window for **Harmonic Perturbation**, locate the **Terminal** section.

3 In the V_0 text field, type `1[mV]`.

Add doping and SRH recombination.

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. In the N_{A0} text field, type `1e15[1/cm^3]`.

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 `10[ns]`.

5 From the τ_p list, choose **User defined**. In the associated text field, type `10[ns]`.

Set up a user-defined mesh for a better compromise between discretization error and roundoff error.

MESH 1

Edge 1

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

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 `100`.

Set up study: use **Auxiliary sweep** to set up the array of DC bias points.

STUDY 1

Step 1: Semiconductor Equilibrium

1 In the **Model Builder** window, under **Study 1** click **Step 1: Semiconductor Equilibrium**.

2 In the **Settings** window for **Semiconductor Equilibrium**, 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
V0 (DC bias voltage)	`range(-2,0.1,1)`	V

Add a **Frequency Domain Perturbation** study step to perform the small signal analysis. Specify two values of frequency for the low- and high-frequency limiting behaviors. Use **Auxiliary sweep** to set up the same array of DC bias points as in the previous study step.

Step 2: Frequency-Domain Perturbation

1 In the **Study** toolbar, click **Study Steps** and choose **Frequency Domain>Frequency-Domain Perturbation**.

2 In the **Settings** window for **Frequency-Domain Perturbation**, locate the **Study Settings** section.

3 In the **Frequencies** text field, type `0.001 1e4`.

4 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.

5 Click + **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
V0 (DC bias voltage)	`range(-2,0.1,1)`	V

Adjust the default solver settings for better solution accuracy.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Stationary Solver 1**.

3 In the **Settings** window for **Stationary Solver**, locate the **General** section.

4 In the **Relative tolerance** text field, type `1.0E-9`.

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

RESULTS

Energy Levels (semi)

The plots by default show the harmonic perturbation part of the solution. We can change them to show the DC operating points instead. Details near the silicon/oxide interface can be seen easily by setting the x-axis to log scale.

Conduction Band Energy Level

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

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

3 From the **Expression evaluated for** list, choose **Static solution**.

Electron Quasi-Fermi Energy Level

1 In the **Model Builder** window, click **Electron Quasi-Fermi Energy Level**.

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

3 From the **Expression evaluated for** list, choose **Static solution**.

Hole Quasi-Fermi Energy Level

1 In the **Model Builder** window, click **Hole Quasi-Fermi Energy Level**.

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

3 From the **Expression evaluated for** list, choose **Static solution**.

Valence Band Energy Level

1 In the **Model Builder** window, click **Valence Band Energy Level**.

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

3 From the **Expression evaluated for** list, choose **Static solution**.

Energy Levels (semi)

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

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

3 Clear the **Show legends** check box.

4 Click the **x-Axis Log Scale** button in the **Graphics** toolbar.

5 In the **Energy Levels (semi)** toolbar, click **Plot**.

Carrier Concentrations (semi)

1 In the **Model Builder** window, click **Carrier Concentrations (semi)**.

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

3 Clear the **Show legends** check box.

Electron Concentration

1 In the **Model Builder** window, expand the **Carrier Concentrations (semi)** node, then click **Electron Concentration**.

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

3 From the **Expression evaluated for** list, choose **Static solution**.

Hole Concentration

1 In the **Model Builder** window, click **Hole Concentration**.

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

3 From the **Expression evaluated for** list, choose **Static solution**.

4 Click the  **x-Axis Log Scale** button in the **Graphics** toolbar.

5 In the **Carrier Concentrations (semi)** toolbar, click  **Plot**.

Line Graph 1

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

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

3 From the **Expression evaluated for** list, choose **Static solution**.

4 Click the  **x-Axis Log Scale** button in the **Graphics** toolbar.

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

Net Dopant Concentration (semi)

The model has a uniform p-doping therefore, we remove the generated default plot, Net Dopant Concentration.

1 In the **Model Builder** window, under **Results** right-click **Net Dopant Concentration (semi)** and choose **Delete**.

Plot the C-V curves for the low- and high-frequency cases. In this model we show two alternative methods to compute the differential capacitance, using either the gate terminal or the metal contact terminal, respectively. The charge on the gate terminal is readily available as a built-in variable. There is no such variable on metal contacts, since usually there are conduction currents flowing through them. In situations where conduction is minimal and the main contribution to the terminal current is capacitive charging, such as in this model, we can integrate the metal contact current to obtain the charge. In frequency domain this is simply `semi.IO_1/semi.iomega`. There should also be an

overall minus sign to account for the fact that the charges on the two terminals are opposite in sign.

Normalized capacitance C/C0

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

2 In the **Settings** window for **1D Plot Group**, type `Normalized capacitance C/C0` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

4 Locate the **Legend** section. From the **Position** list, choose **Lower left**.

Global 1

1 Right-click **Normalized capacitance C/C0** 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
`semi.Q0_2/1[mV]/C0`	1	`Normalized capacitance`
`-semi.I0_1/semi.iomega/ 1[mV]/C0`	1	`Normalized capacitance`

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

When evaluating the result from a **Frequency Domain Perturbation** study step, it is important that the **Compute differential** check box is selected, so that COMSOL differentiates the solution at the linearization point when evaluating the expression, in order to correctly account for the nonlinearities in the equation system.

5 Locate the **y-Axis Data** section. Select the **Compute differential** check box.

6 In the **Normalized capacitance C/C0** toolbar, click **Plot**.