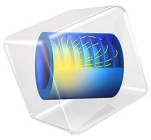
Created in COMSOL Multiphysics 6.2



MOSCAP ID

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.

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 transient study approach (the related model moscap_1d_small_signal shows the small-signal approach). 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 a linear voltage ramp between -2 and 1 V, with a slew rate of 10^{-3} V/s for low frequency and 10^{3} V/s for high frequency. The modeled domain has a thickness of 1 mm. 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. 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 voltage sweep is done using a **Stationary** study step for the initial condition, followed by a **Time Dependent** study step.

Results and Discussion

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

.

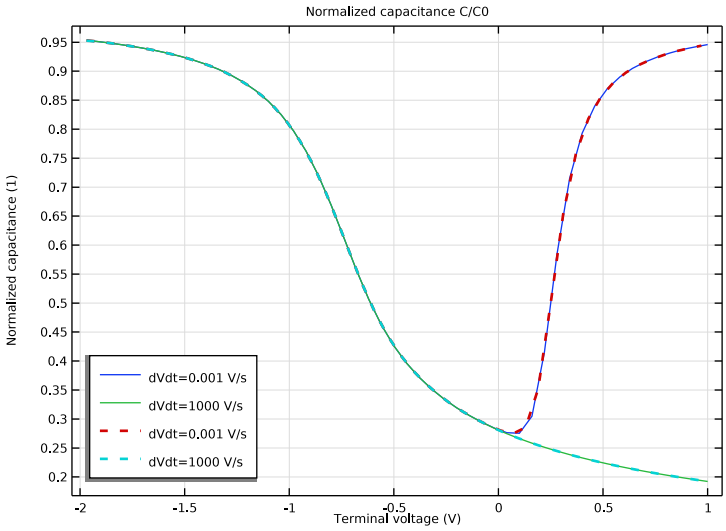


Figure 1: C–V curves for the low-frequency and high-frequency cases.

Reference

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

Application Library path: `Semiconductor_Module/Device_Building_Blocks/moscap_1d`

Modeling Instructions

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

NEW

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

MODEL WIZARD

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

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

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GEOMETRY 1

The Model Wizard starts the COMSOL Desktop with the **Geometry** node selected. 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 1 mm. Add a point for meshing purposes.

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
0.1
1000

Now import some global parameters from a text file. Note the Time parameter `t` which makes the built-in variable for time recognized by the Stationary study step. The Time step `dt` will not only be used in the transient study settings but also be used to compute the rate of change of charges from the metal contact - more on this later.

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.txt`.

Add the built-in silicon material.

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 doping and boundary conditions.

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**.

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 3 only.

Place the gate oxide at the left endpoint, using the dedicated **Thin Insulator Gate** boundary condition. Note the gate voltage will be swept linearly over time with the ramp rate of `dVdt`.

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+dVdt*t`.

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

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

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**.

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 use a global equation to integrate the metal contact current to obtain the charge as a function of time.

To do this, first click on the **Show More Options** button to enable **Equation-Based Contributions**. Then create a global equation with a variable `Q1` for the charge and use its time derivative expression `Q1t` in the equation.

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

5 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Equation-Based Contributions**.

6 Click **OK**.

Global Equations 1

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

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

3 In the table, enter the following settings:

Name	f(u,ut,utt, t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)	Description
Q1	Q1t-semi.IO_1	0	0	

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

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

Dependent variable quantity	Unit
Custom unit	C

6 Click **Define Source Term Unit**.

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

Source term quantity	Unit
Custom unit	A

Set up the mesh in such a way that the resolution is very fine under the silicon/oxide interface.

MESH 1

Edge 1

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

Distribution 1

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

2 Select Domain 1 only.

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

4 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 10.

7 From the **Growth rate** list, choose **Exponential**.

Distribution 2

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

2 Select Domain 2 only.

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

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

5 In the **Number of elements** text field, type 100.

6 In the **Element ratio** text field, type 5000.

7 From the **Growth rate** list, choose **Exponential**.

When setting up the study, use a Parametric Sweep to configure the low-frequency and high-frequency cases.

STUDY 1

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
dVdt (Voltage ramp rate)	`1e-3 1e3`	`V/s`

Remember to disable the global equation in the Stationary study step.

Step 1: Stationary

1 In the **Model Builder** window, click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (comp1)>Semiconductor (semi)>Global Equations 1**.

5 Click **Disable**.

Add a Time Dependent study step to sweep the gate voltage linearly over time.

Step 2: Time Dependent

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

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 In the **Output times** text field, type `range(0,dt,tmax)`.

4 From the **Tolerance** list, choose **User controlled**.

5 In the **Relative tolerance** text field, type `1e-6`.

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

RESULTS

Energy Levels (semi)

Examine the default result graphs. Details near the silicon/oxide interface can be seen easily by using a log scale for the x-axis.

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

Carrier Concentrations (semi)

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

Electric Potential (semi)

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

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, right-click **Net Dopant Concentration (semi)** and choose **Delete**.

Plot the C-V curves for the low-frequency (small voltage ramp rate `dVdt`) and the high-frequency (large `dVdt`) cases. First use the gate terminal charge to compute the differential capacitance.

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**.

5 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (sol3)**.

6 From the **Time selection** list, choose **Manual**.

7 Click  **Range**.

8 In the **Integer Range** dialog box, type `2` in the **Start** text field.

9 In the **Stop** text field, type `51`.

10 Click **Replace**.

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
d(semi.Q0_2,t)/dVdt/C0	1	Normalized capacitance

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

5 In the **Expression** text field, type `semi.V0_2`.

6 Click to expand the **Legends** section. Find the **Include** subsection. Clear the **Description** check box.

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

Then use the alternative method to compute the differential capacitance by calculating the difference quotient of the metal contact terminal charge `Q1` obtained from the global equation. The denominator of the difference quotient is simply the Time step `dt`. The numerator is the charge at the current time minus the charge of the previous time step, given by the expression `Q1-withsol('sol3',Q1,setval(dVdt,dVdt), setval(t,t-dt))`, where the value of the previous time point `t-dt` is used in the `withsol` operator. The voltage value for each difference quotient is set to the average voltage over the time interval, given by the expression `semi.V0_2-0.5*dVdt*dt`. The overall minus sign for the differential capacitance accounts for the fact that the charges on the two terminals are opposite in sign.

8 Right-click **Global 1** and choose **Duplicate**.

Global 2

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

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

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol3)**.

4 From the **Time selection** list, choose **Manual**.

5 In the **Time indices (1-51)** text field, type `range(2,51)`.

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

Expression	Unit	Description
-(Q1-withsol('sol3',Q1, setval(dVdt,dVdt),setval(t, t-dt)))/dt/dVdt/C0	1	Normalized capacitance

7 Locate the **x-Axis Data** section. In the **Expression** text field, type `semi.V0_2-0.5*dVdt*dt`.

8 Select the **Description** check box. In the associated text field, type `Terminal voltage`.

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

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

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