Model created in COMSOL Multiphysics 6.4

Surface Dielectric Barrier Discharge

Introduction

Surface dielectric barrier discharges (SDBDs) are a class of non-thermal plasmas generated when a high voltage is applied across electrodes separated by a dielectric material, with at least one electrode exposed to the surrounding gas. They are widely studied for applications in flow control, ozone generation, and plasma-assisted combustion.

This model simulates SDBDs using the Electric Discharge interface. The formulation captures the key physical mechanisms governing the discharge, including charge transport, impact ionization, electron attachment, recombination, and surface charge accumulation, all self-consistently coupled through the Poisson equation.

The simulation outputs include the spatial distribution of the electric field and the time-dependent number densities of electrons, positive ions, and negative ions, providing detailed insight into discharge behavior near dielectric surfaces.

Model Definition

The Electric Discharge interface is used to simulate the SDBDs. The built-in charge transport model is used:

$$\frac{\partial n_i}{\partial t} + \nabla \cdot (\mathbf{w}_i n_i - D_i \nabla n_i) = R_i$$

where

$$i = \text{e, p, n}$$

$$z_{\text{e, p, n}} = -1, +1, -1$$

$$\mathbf{w}_i = z_i \mu_i \mathbf{E}$$

$$R_\text{e} = \alpha|\mathbf{w}_\text{e}|n_\text{e} - \eta|\mathbf{w}_\text{e}|n_\text{e} - \beta_\text{ep} n_\text{e} n_\text{p}$$

$$R_\text{p} = \alpha|\mathbf{w}_\text{e}|n_\text{e} - \beta_\text{ep} n_\text{e} n_\text{p} - \beta_\text{pn} n_\text{p} n_\text{n}$$

$$R_\text{n} = \eta|\mathbf{w}_\text{e}|n_\text{e} - \beta_\text{pn} n_\text{p} n_\text{n}$$

where

- e, p, n denote electrons, positive ions, and negative ions
- n_i is the number density of the charge carrier (SI unit: $1/\text{m}^3$)
- $\mathbf{E}$ is the electric field (SI unit: V/m)
- z_i denotes the carrier charge (SI unit: 1)
- μ_i denotes the carrier mobility (SI unit: $\text{m}^2/(\text{V}\cdot\text{s})$)
- $\mathbf{w}_i$ is the drift velocity in the electric field (SI unit: m/s)
- D_i is the diffusion coefficient (SI unit: m^2/s)
- R_i is the reaction rate (SI unit: $1/(\text{m}^3\cdot\text{s})$)
- α is the ionization coefficient (SI unit: 1/m)
- η is the attachment coefficient (SI unit: 1/m)
- β_ep is the electron–ion recombination coefficient (SI unit: m^3/s)
- β_pn is the ion–ion recombination coefficient (SI unit: m^3/s)

The above transport equations are fully coupled with Poisson's equation through the electric field and the space charge:

$$\nabla \cdot (\varepsilon_r \varepsilon_0 \mathbf{E}) = \rho$$

$$\rho = e \sum_i z_i n_i$$

where e is the elementary charge.

The Gas–Solid interface is modeled with the dedicated *Dielectric Interface*, *Surface Transport* feature described in the *Electric Discharge Module User's Guide*.

Results and Discussion

Figure 1 shows the space charge density at several time instants. Figure 2 shows the isosurface plot of positive ions density along dielectric surface in 3D.

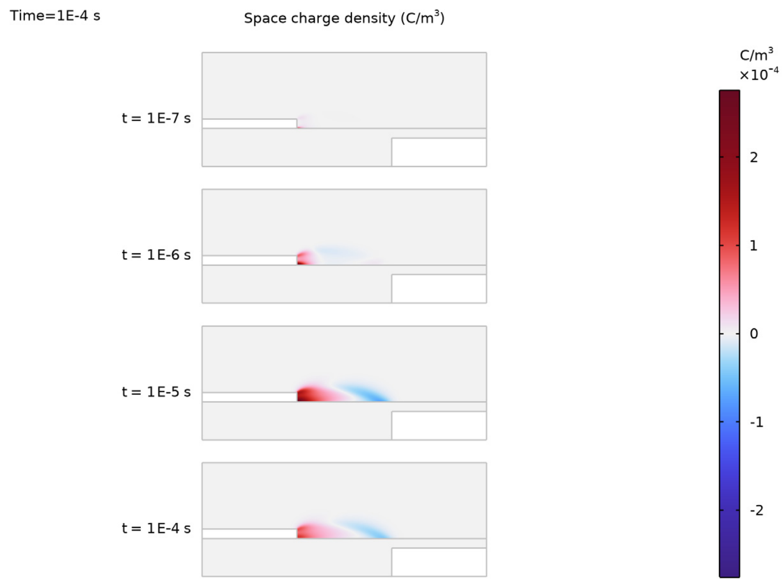


Figure 1: The distribution of space charge density at several instants.

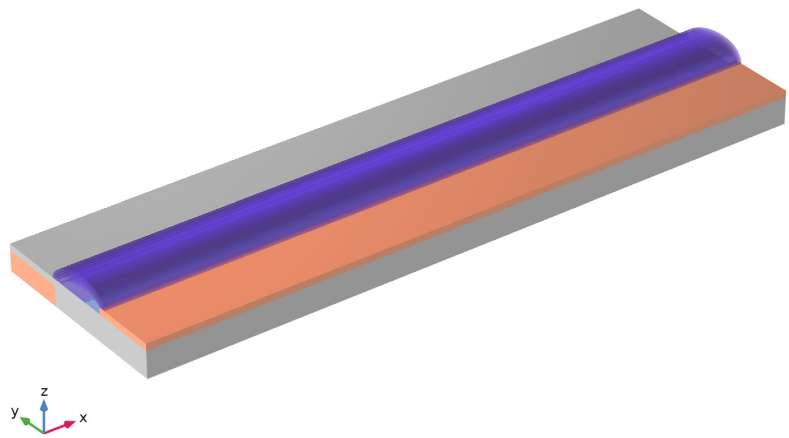


Figure 2: The distribution of positive ions in 3D.

Application Library path: `Electric_Discharge_Module/Dielectric_Barrier_Discharges/sdbd`

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 **Electric Discharge** > **Electric Discharge (edis)**.
3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces** > **Time Dependent with Initialization**.

6 Click **Done**.

GEOMETRY 1

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

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

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
V0	-15[kV]	-15000 V	Applied voltage
z0	50[mm]	0.05 m	Out-of-plane thickness

GEOMETRY 1

Rectangle 1 (r1)

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

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type 15.

4 In the **Height** text field, type 6.

5 Locate the **Position** section. In the **y** text field, type -2.

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

Layer name	Thickness (mm)
Layer 1	2
Layer 2	0.5

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

Rectangle 2 (r2)

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type `5`.
4 In the **Height** text field, type `0.5`.

Rectangle 3 (r3)

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type `5`.
4 In the **Height** text field, type `1.5`.
5 Locate the **Position** section. In the **x** text field, type `10`.
6 In the **y** text field, type `-2`.

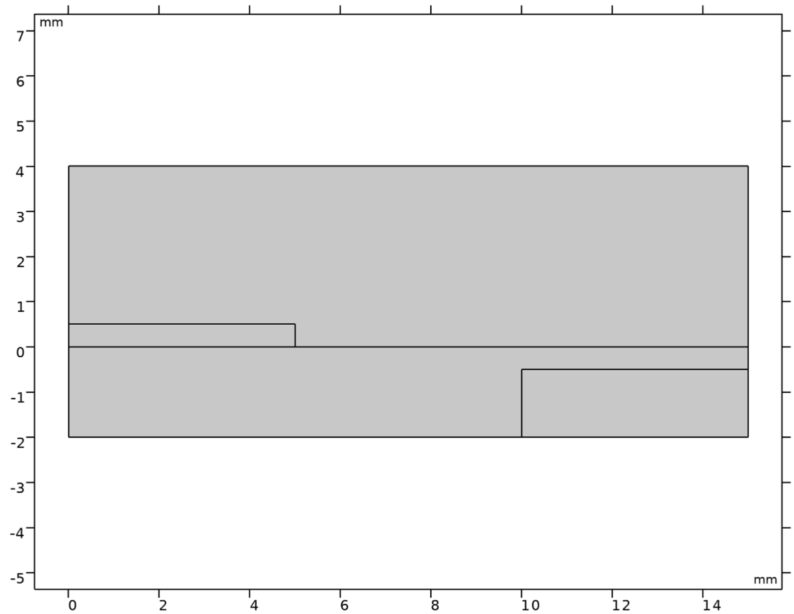
Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.
2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.
3 From the **Specify** list, choose **Coordinates**.
4 In the **x** text field, type `10`.
5 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.
6 In the **x** text field, type `10`.
7 In the **y** text field, type `0.5`.

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 10, 14, and 16 only.
3 In the **Geometry** toolbar, click **Build All**.

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



ELECTRIC DISCHARGE (EDIS)

Exclude the electrodes from the computation domains (with modeling the electrode boundaries only).

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Electric Discharge (edis)**.
2. Select Domains 1 and 3 only.
3. In the **Settings** window for **Electric Discharge**, locate the **Physical Model** section.
4. In the d_z text field, type `z0`.

   Enable the isotropic diffusion stabilization.

5. Click the **Show More Options** button in the **Model Builder** toolbar.
6. In the **Show More Options** dialog, select **Physics** > **Stabilization** in the tree.
7. In the tree, select the checkbox for the node **Physics** > **Stabilization**.
8. Click **OK**.
9. In the **Model Builder** window, click **Electric Discharge (edis)**.
10. In the **Settings** window for **Electric Discharge**, click to expand the **Inconsistent Stabilization** section.

11 Select the **Isotropic diffusion** checkbox.

12 Locate the **Physical Model** section. Select the **Solid** checkbox.

Solid 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Electric Discharge (edis)** click **Solid 1**.

2 Select Domain 1 only.

ADD MATERIAL FROM LIBRARY

In the **Home** toolbar, click **Windows** and choose **Add Material from Library**.

ADD MATERIAL

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

2 In the tree, select **Electric Discharge** > **Gases** > **Air** > **Air [Kang et al. 2003]**.

3 Right-click and choose **Add to Component 1 (comp1)**.

MATERIALS

Air [Kang et al. 2003] (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Materials** click **Air [Kang et al. 2003] (mat1)**.

2 Select Domain 3 only.

ELECTRIC DISCHARGE (EDIS)

Solid 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Electric Discharge (edis)** click **Solid 1**.

2 In the **Settings** window for **Solid**, locate the **Model Formulation** section.

3 From the **Material model** list, choose **Insulator**.

4 Locate the **Constitutive Relation D-E** section. From the ε_r list, choose **User defined**. In the associated text field, type `3.2`.

Gas 1

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

Electrode 1

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

2 Select Boundaries 6 and 8 only.

3 In the **Settings** window for **Electrode**, locate the **Terminal** section.

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

5 Locate the **Charge Transport** section. From the **Boundary condition for electrons** list, choose **Number density**.

6 From the **Boundary condition for negative ions** list, choose **Number density**.

Solid 1

In the **Model Builder** window, under **Component 1 (comp1)** > **Electric Discharge (edis)** click **Solid 1**.

Electrode 1

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

2 Select Boundary 4 only.

3 In the **Settings** window for **Electrode**, locate the **Terminal** section.

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

Solid 1

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

Electrode 2

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

2 Select Boundaries 10 and 12 only.

MESH 1

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

2 In the **Settings** window for **Mesh**, locate the **Sequence Type** section.

3 From the list, choose **User-controlled mesh**.

Mapped 1

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

2 Drag and drop below **Size**.

3 In the **Settings** window for **Mapped**, locate the **Domain Selection** section.

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

5 Select Domain 6 only.

Distribution 1

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

2 Select Boundaries 9 and 18 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 `400`.
6. In the **Element ratio** text field, type `5`.

Distribution 2

1. In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.
2. Select Boundaries 8 and 19 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 `40`.
6. In the **Element ratio** text field, type `5`.
7. Click **Build Selected**.

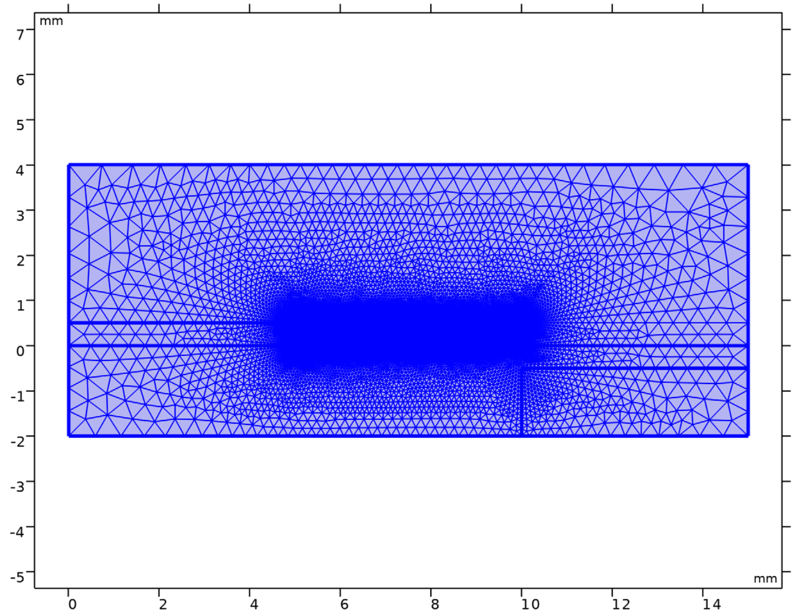
Size 1

1. In the **Model Builder** window, right-click **Free Triangular 1** and choose **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.
5. Select the **Maximum element growth rate** checkbox. In the associated text field, type `1.1`.
6. Click the **Zoom Box** button in the **Graphics** toolbar.

Free Triangular 1

1. In the **Model Builder** window, click **Free Triangular 1**.
2. In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.
3. From the **Geometric entity level** list, choose **Remaining**.
4. Click to expand the **Control Entities** section. From the **Smooth across removed control entities** list, choose **Off**.
5. Click **Build All**.

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



STUDY 1

Step 2: Time Dependent

1 In the **Model Builder** window, under **Study 1** click **Step 2: Time Dependent**.

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

3 In the **Output times** text field, type `0 10^{range(log10(1.0e-9),1/10, log10(1.0e-4))}`.

Solution 1 (sol1)

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

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

Use lower of BDF solver to increase stability.

3 In the **Model Builder** window, under **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** click **Time-Dependent Solver 1**.

4 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

5 From the **Minimum BDF order** list, choose **1**.

6 From the **Maximum BDF order** list, choose **2**.

Use Anderson acceleration and split the solution of ions to speed up computation.

7 In the **Model Builder** window, under **Study 1 > Solver Configurations > Solution 1 (sol1) > Time-Dependent Solver 1** click **Segregated 1**.

8 In the **Settings** window for **Segregated**, locate the **General** section.

9 From the **Stabilization and acceleration** list, choose **Anderson acceleration**.

10 Right-click **Study 1 > Solver Configurations > Solution 1 (sol1) > Time-Dependent Solver 1 > Segregated 1** and choose **Segregated Step**.

11 Drag and drop Study 1 > **Solver Configurations > Solution 1 (sol1) > Time-Dependent Solver 1 > Segregated 1 > Segregated Step 3below Segregated Step 1**.

12 In the **Model Builder** window, under **Study 1 > Solver Configurations > Solution 1 (sol1) > Time-Dependent Solver 1 > Segregated 1** click **Segregated Step 1**.

13 In the **Settings** window for **Segregated Step**, locate the **General** section.

14 In the **Variables** list, choose **Natural Logarithm of the Number Density Multiplied by 1[cm^3] (comp1.edis.logn_p)** and **Natural Logarithm of the Number Density Multiplied by 1[cm^3] (comp1.edis.logn_n)**.

15 Under **Variables**, click **Delete**.

16 In the **Model Builder** window, under **Study 1 > Solver Configurations > Solution 1 (sol1) > Time-Dependent Solver 1 > Segregated 1** click **Segregated Step 3**.

17 In the **Settings** window for **Segregated Step**, locate the **General** section.

18 Under **Variables**, click **Add**.

19 In the **Add** dialog, in the **Variables** list, choose **Natural Logarithm of the Number Density Multiplied by 1[cm^3] (comp1.edis.logn_n)** and **Natural Logarithm of the Number Density Multiplied by 1[cm^3] (comp1.edis.logn_p)**.

20 Click **OK**.

21 In the **Settings** window for **Segregated Step**, locate the **Method and Termination** section.

22 From the **Jacobian update** list, choose **Once per time step**.

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

RESULTS

Surface 1

1 In the **Model Builder** window, expand the **2D Plot Group 1** node, then click **Surface 1**.

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

3 From the **Color table** list, choose **Wave**.

4 From the **Scale** list, choose **Linear symmetric**.

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

Space Charge Density

1 In the **Model Builder** window, under **Results** click **2D Plot Group 1**.

2 In the **Settings** window for **2D Plot Group**, type `Space Charge Density` in the **Label** text field.

3 Click to expand the **Plot Array** section. From the **Array type** list, choose **Linear**.

4 From the **Array axis** list, choose **y**.

5 In the **Relative padding** text field, type `-2.2`.

Solution Array 1

1 In the **Model Builder** window, right-click **Surface 1** and choose **Solution Array**.

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

3 From the **Time selection** list, choose **Interpolated**.

4 In the **Times (s)** text field, type `10^{range(log10(1.0e-7),1,log10(1e-4))}`.

Annotation 1

1 In the **Model Builder** window, right-click **Space Charge Density** and choose **Annotation**.

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

3 In the **Text** text field, type `t = eval(t,s,3) s`.

4 In the **Space Charge Density** toolbar, click **Plot**.

Solution Array 1

In the **Model Builder** window, under **Results** > **Space Charge Density** > **Surface 1** right-click **Solution Array 1** and choose **Copy**.

Solution Array 1

In the **Model Builder** window, right-click **Annotation 1** and choose **Paste Solution Array**.

Annotation 1

1 In the **Settings** window for **Annotation**, locate the **Annotation** section.

2 In the **Text** text field, type `t = eval(t,s,3) s`.

3 Locate the **Coloring and Style** section. Clear the **Show point** checkbox.

4 From the **Anchor point** list, choose **Lower right**.

5 Click to expand the **Plot Array** section. Select the **Manual indexing** checkbox.

Space Charge Density

1 In the **Model Builder** window, click **Space Charge Density**.

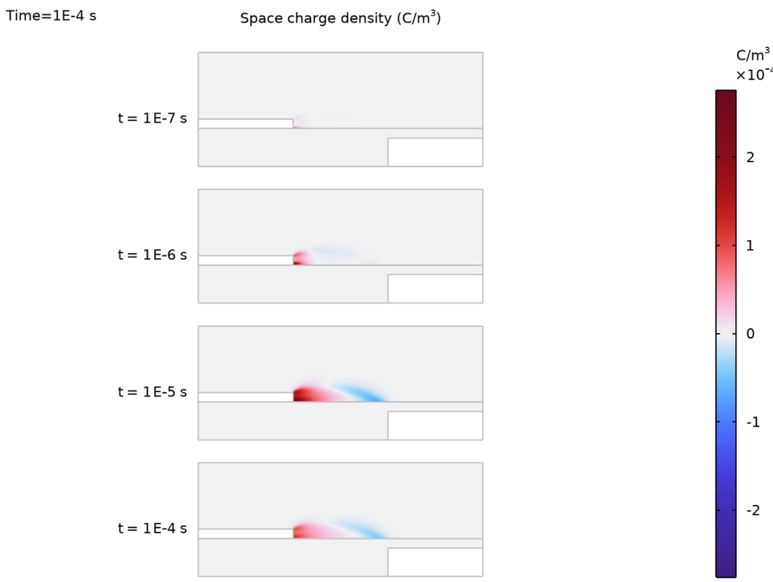
2 In the **Settings** window for **2D Plot Group**, locate the **Color Legend** section.

3 Select the **Show units** checkbox.

4 Locate the **Plot Settings** section. From the **Color** list, choose **Gray**.

5 In the **Space Charge Density** toolbar, click **Plot**.

6 Click the **Show Grid** button in the **Graphics** toolbar.



Extrusion 2D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Extrusion 2D**.

2 In the **Settings** window for **Extrusion 2D**, locate the **Extrusion** section.

3 Find the **Embedding** subsection. From the **Map plane to** list, choose **yz-plane**.

4 In the **z maximum** text field, type z0.

Study 1/Solution 1 (3) (sol1)

In the **Model Builder** window, under **Results > Datasets** right-click **Study 1/Solution 1 (sol1)** and choose **Duplicate**.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

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

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

4 Select Domains 2 and 4 only.

5 Click the **Zoom Box** button in the **Graphics** toolbar.

Extrusion 2D 2

1 In the **Results** toolbar, click **More Datasets** and choose **Extrusion 2D**.

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

3 From the **Dataset** list, choose **Study 1/Solution 1 (3) (sol1)**.

4 Locate the **Extrusion** section. In the **z maximum** text field, type `z0`.

5 Find the **Embedding** subsection. From the **Map plane to** list, choose **yz-plane**.

Study 1/Solution 1 (4) (sol1)

In the **Model Builder** window, under **Results** > **Datasets** right-click **Study 1/ Solution 1 (1) (sol1)** and choose **Duplicate**.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

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

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

4 Select Domain 1 only.

Extrusion 2D 3

1 In the **Results** toolbar, click **More Datasets** and choose **Extrusion 2D**.

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

3 From the **Dataset** list, choose **Study 1/Solution 1 (4) (sol1)**.

4 Locate the **Extrusion** section. In the **z maximum** text field, type `z0`.

5 Find the **Embedding** subsection. From the **Map plane to** list, choose **yz-plane**.

3D Plot Group 2

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

3 Clear the **Plot dataset edges** checkbox.

Surface 1

1 Right-click **3D Plot Group 2** and choose **Surface**.

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

3 From the **Dataset** list, choose **Extrusion 2D 3**.

4 Locate the **Expression** section. In the **Expression** text field, type 1.

5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

6 From the **Color** list, choose **Custom**.

7 On Windows, click the colored bar underneath, or — if you are running the cross-platform desktop — the **Color** button.

8 Click **Define custom colors**.

9 Set the RGB values to 192, 192, and 192, respectively.

10 Click **Add to custom colors**.

11 Click **Show color palette only** or **OK** on the cross-platform desktop.

12 Click the **Show Grid** button in the **Graphics** toolbar.

Surface 2

1 In the **Model Builder** window, right-click **3D Plot Group 2** and choose **Surface**.

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

3 From the **Dataset** list, choose **Extrusion 2D 2**.

4 Locate the **Expression** section. In the **Expression** text field, type 1.

5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

6 From the **Color** list, choose **Custom**.

7 On Windows, click the colored bar underneath, or — if you are running the cross-platform desktop — the **Color** button.

8 Click **Define custom colors**.

9 Set the RGB values to 255, 160, and 122, respectively.

10 Click **Add to custom colors**.

11 Click **Show color palette only** or **OK** on the cross-platform desktop.

3D Plot Group 2

1 In the **Model Builder** window, click **3D Plot Group 2**.

2 In the **Settings** window for **3D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

Isosurface 1

1 Right-click **3D Plot Group 2** and choose **Isosurface**.

2 In the **Settings** window for **Isosurface**, locate the **Expression** section.

3 In the **Expression** text field, type `edis.n_p`.

4 In the **Unit** field, type `1/cm^3`.

5 Locate the **Levels** section. From the **Entry method** list, choose **Levels**.

6 In the **Levels** text field, type `10^{range(log10(1e6),1/2,log10(1.0e9))}`.

7 Locate the **Coloring and Style** section. From the **Isosurface type** list, choose **Filled**.

8 Clear the **Fill volume outside of isosurface levels** checkbox.

9 From the **Color table** list, choose **WaveLight**.

Transparency 1

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

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

3 Find the **Fresnel transmittance** subsection. Set the **Fresnel transmittance** value to **0.5**.

3D Plot Group 2

1 In the **Model Builder** window, under **Results** click **3D Plot Group 2**.

2 In the **Settings** window for **3D Plot Group**, locate the **Color Legend** section.

3 Clear the **Show legends** checkbox.

4 In the **3D Plot Group 2** toolbar, click **Plot**.

5 Click the **Zoom Box** button in the **Graphics** toolbar.