Model created in COMSOL Multiphysics 6.4

Positive Streamer Propagation in Transformer Oil

Introduction

This example simulates the propagation of a positive streamer in transformer oil under a lightning impulse voltage. The space charge density and the electric field are obtained. The simulated streamer radius agrees well with the measured values (Ref. 1).

Model Definition

The Electric Discharge interface is used to simulate the streamer discharge in transformer oil. The numerical model is described as follows:

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

where

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

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

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

$$R_\mathrm{e} = S_\mathrm{F} - \frac{n_\mathrm{e}}{\tau_a} - \beta_\mathrm{ep} n_\mathrm{e} n_\mathrm{p}$$

$$R_\mathrm{p} = S_\mathrm{F} - \beta_\mathrm{ep} n_\mathrm{e} n_\mathrm{p} - \beta_\mathrm{pn} n_\mathrm{p} n_\mathrm{n}$$

$$R_\mathrm{n} = \frac{n_\mathrm{e}}{\tau_a} - \beta_\mathrm{pn} n_\mathrm{p} n_\mathrm{n}$$

$$S_\mathrm{F} = \frac{e n_\mathrm{ioni} a |\mathbf{E}|}{h} \exp\left[-\frac{\pi^2 m^* a e}{h^2}\left(\frac{\varphi_\Delta}{\sqrt{|\mathbf{E}|}} - \frac{\varphi_\gamma}{\sqrt{10^6 \mathrm{V/cm}}}\right)^2\right]$$

where

- e, p, and n denote electrons, positive ions, and negative ions
- n_i is the number density of the charge carrier (SI unit: $1/\mathrm{m}^3$)
- $\mathbf{E}$ 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: $\mathrm{m}^2/(\mathrm{V \cdot s})$)
- $\mathbf{w}_i$ 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: $1/(\mathrm{m}^3 \cdot \mathrm{s})$)
- τ_a is the attachment time constant (SI unit: s)

- 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)
- S_F is the field ionization (SI unit: $1/(m^3 \cdot s)$)
- e is the electric charge (SI unit: C)
- n_{ioni} is the number density of ionizable species (SI unit: $1/m^3$)
- a denotes the molecular separation distance (SI unit: m)
- m^* denotes the effective electron mass (SI unit: kg)
- φ_Δ and φ_γ are ionization potential parameters (SI unit: V)

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

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

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

Results and Discussion

Figure 1 shows the discharge current as a function of time. Figure 2 plots the distribution of different charge carriers density before and during a current pulse. Figure 3 shows the distribution of space charge density and electric field at $t = 90$ ns.

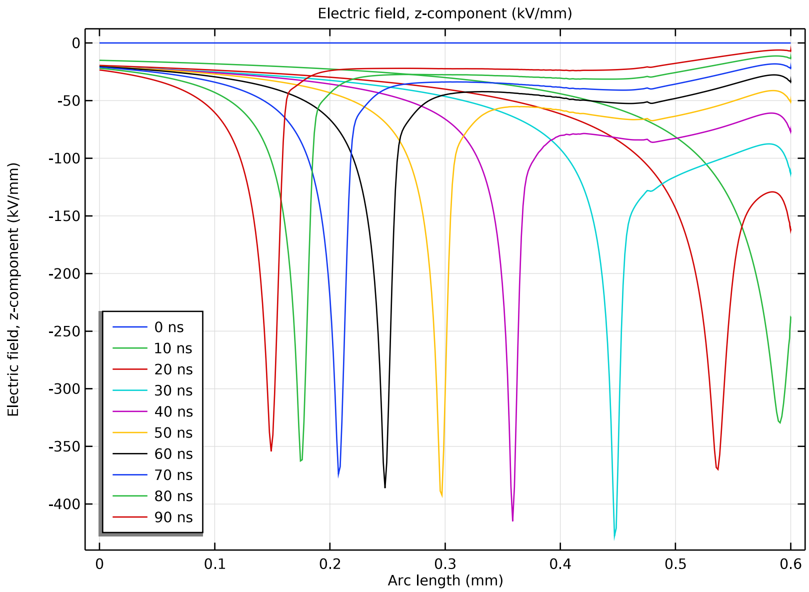


Figure 1: The axial electric field at t = 0, 10, 20, ..., 90 ns.

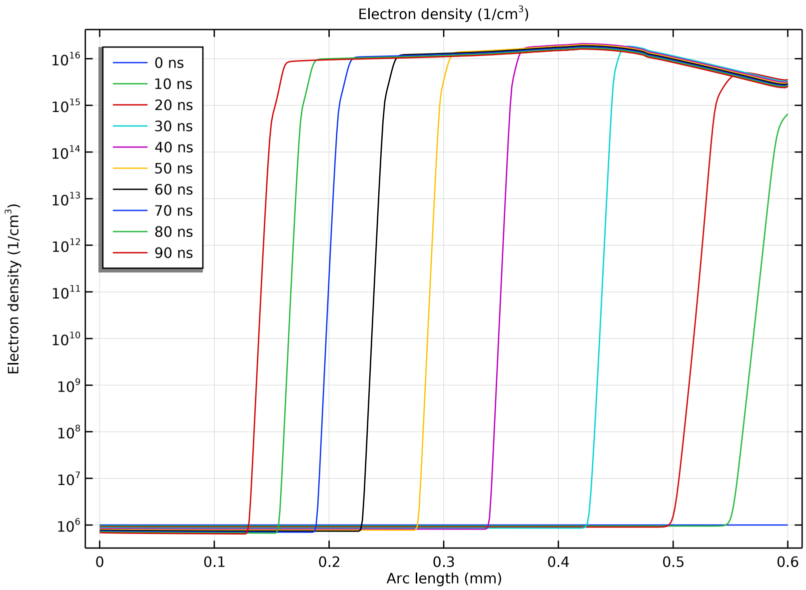


Figure 2: The electron number density at t = 0, 10, 20, ..., 90 ns.

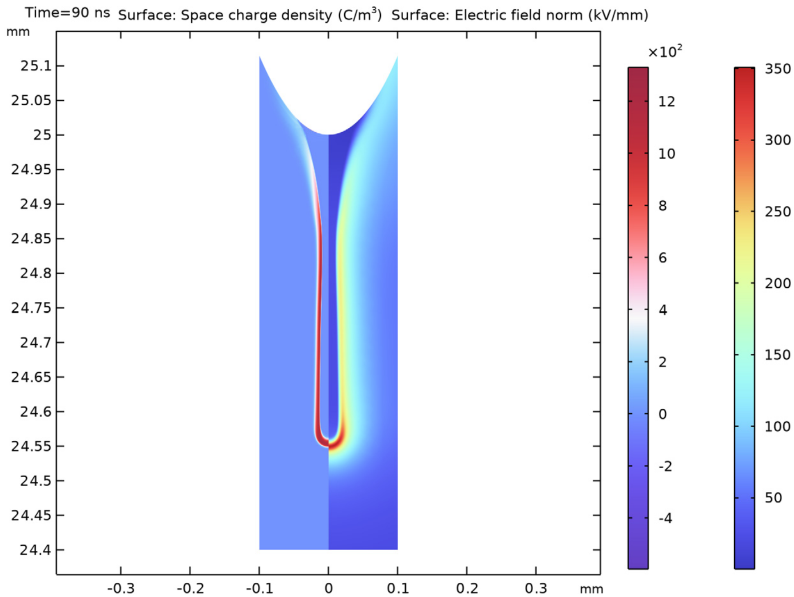


Figure 3: The distribution of space charge density and electric field at t = 90 ns.

References

1. J. Jadidian and others, “Stochastic and deterministic causes of streamer branching in liquid dielectrics,” *J. Appl. Phys.*, vol. 114, no. 6, pp. 63301-1–10, 2013.

Application Library path: `Electric_Discharge_Module/Liquid_Dielectrics/streamer_in_transformer_oil`

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 Axisymmetric**.

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 **General Studies > Time Dependent**.

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**.

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 30.

5 Click to expand the **Layers** section. Select the **Layers to the left** checkbox.

6 Clear the **Layers on bottom** checkbox.

7 In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	0.1

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 0.3.

4 In the **Height** text field, type 3.

5 Locate the **Position** section. In the **z** text field, type 27.

Ellipse 1 (e1)

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

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

3 In the **a-semiaxis** text field, type `0.3`.

4 In the **b-semiaxis** text field, type `2`.

5 Locate the **Position** section. In the **z** text field, type `27`.

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 `0.1`.

4 In the **Height** text field, type `0.6-1/1000`.

5 Locate the **Position** section. In the **z** text field, type `24.4`.

6 Locate the **Layers** section. Clear the **Layers on bottom** checkbox.

7 Select the **Layers to the left** checkbox.

8 In the table, enter the following settings:

Layer name	Thickness (mm)
`Layer 1`	`0.03`

Difference 1 (dif1)

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

2 Select the object **r1** only.

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

4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

5 Select the objects **e1** and **r2** only.

6 Click **Build All Objects**.

DEFINITIONS

Analytic 1 (an1)

1 In the **Definitions** toolbar, click **Analytic**.

2 In the **Settings** window for **Analytic**, type `Vapp` in the **Function name** text field.

3 Locate the **Definition** section. In the **Expression** text field, type `130*(exp(-t/100)-exp(-t/10))`.

4 In the **Arguments** text field, type `t`.

5 Locate the **Units** section. In the **Function** text field, type `kV`.

6 In the table, enter the following settings:

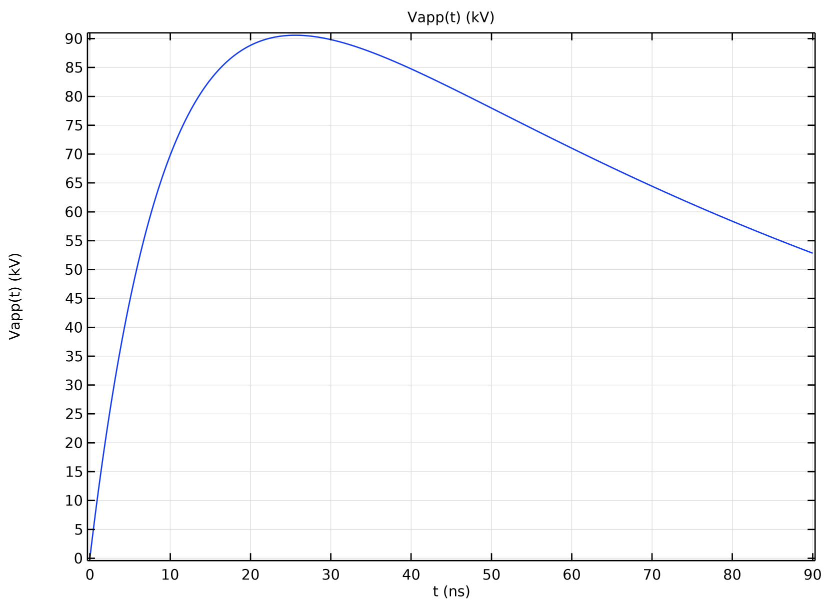
Argument	Unit
t	ns

7 Locate the **Plot Parameters** section. In the table, enter the following settings:

Plot	Argument	Lower limit	Upper limit	Fixed value	Unit
√	t	0	90[ns]	0	s

8 Click **Plot**.

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



ELECTRIC DISCHARGE (EDIS)

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

2 In the **Show More Options** dialog, select **Physics** > **Stabilization** in the tree.

3 In the tree, select the checkbox for the node **Physics** > **Stabilization**.

4 Click **OK**.

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

6 In the **Settings** window for **Electric Discharge**, locate the **Physical Model** section.

7 Clear the **Gas** checkbox.

8 Select the **Liquid** checkbox.

9 Click to expand the **Inconsistent Stabilization** section. Select the **Isotropic diffusion** checkbox.

ADD MATERIAL

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

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

3 In the tree, select **Electric Discharge** > **Liquids** > **Transformer Oil**.

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

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

ELECTRIC DISCHARGE (EDIS)

Electrode 1

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

2 Select Boundaries 14, 17, and 18 only.

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

4 In the V_0 text field, type `Vapp(t)`.

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

Liquid 1

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

Electrode 2

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

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

3 Select Boundaries 2 and 11 only.

4 In the **Settings** window for **Electrode**, locate the **Charge Transport** section.

5 From the **Boundary condition for electrons** list, choose **Number density**.

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

MESH 1

Mapped 1

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

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

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

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

5 Select Domain 2 only.

Distribution 1

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

2 Select Boundaries 4 and 6 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 Select the **Reverse direction** checkbox.

7 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 3 and 7 only.

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

4 In the **Number of elements** text field, type 400.

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 3 only.

Size 1

1 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 size** checkbox. In the associated text field, type 1/300.

Free Triangular 2

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

2 In the **Settings** window for **Free Triangular**, click **Build All**.

Boundary Layers 1

In the **Mesh** toolbar, click **Boundary Layers**.

Boundary Layer Properties

1 In the **Model Builder** window, click **Boundary Layer Properties**.

2 Select Boundary 17 only.

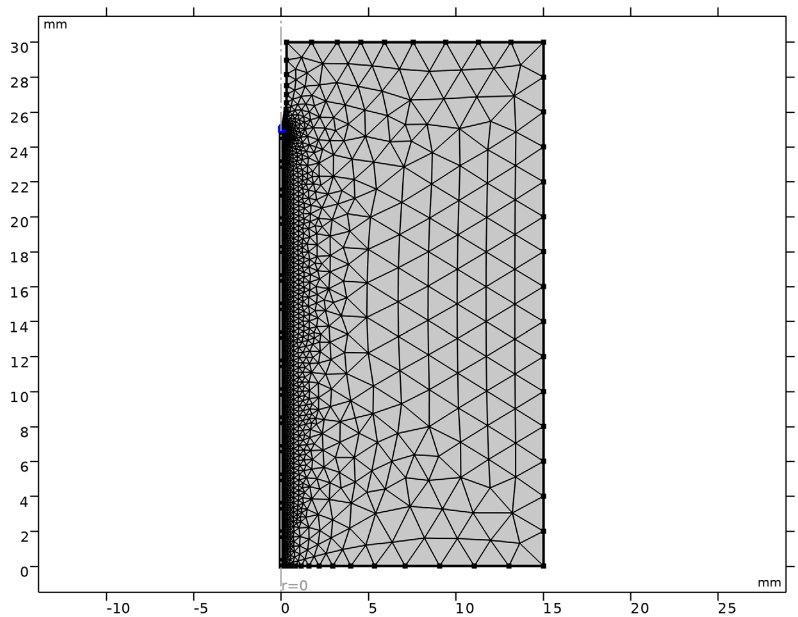
3 In the **Settings** window for **Boundary Layer Properties**, locate the **Layers** section.

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

5 In the **Stretching factor** text field, type `1.5`.

6 Click **Build All**.

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



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

STUDY 1

Step 1: Time Dependent

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

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

3 From the **Time unit** list, choose **ns**.

4 In the **Output times** text field, type `range(0,10,90)`.

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

6 In the **Settings** window for **Study**, locate the **Study Settings** section.

7 Clear the **Generate default plots** checkbox.

Solution 1 (sol1)

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

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

Setting a maximum timestep variable based on the built-in dielectric relaxation time can help improve both efficiency and accuracy.

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 **Maximum step constraint** list, choose **Expression**.

6 In the **Maximum step** text field, type `min(0.1[ns],comp1.edis.minDRT*0.8)`.

The PARDISO direct solver is usually a bit faster and leaner on memory than the default direct solver (MUMPS) on this type of model.

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

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

9 From the **Solver** list, choose **PARDISO**.

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

RESULTS

In the **Model Builder** window, expand the **Results** node.

Study 1/Solution 1 (sol1)

In the **Model Builder** window, expand the **Results** > **Datasets** node, then click **Study 1/ Solution 1 (sol1)**.

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 2–4 only.

Mirror 2D 1

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

2D Plot Group 1

In the **Results** toolbar, click **2D Plot Group**.

Surface 1

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

2D Plot Group 1

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

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

Surface 1

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

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

3 From the **Dataset** list, choose **Mirror 2D 1**.

Surface 2

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

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

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

4 In the **Unit** field, type `kV/mm`.

5 In the **2D Plot Group 1** toolbar, click **Plot**.

Surface 1

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

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

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

Surface 2

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

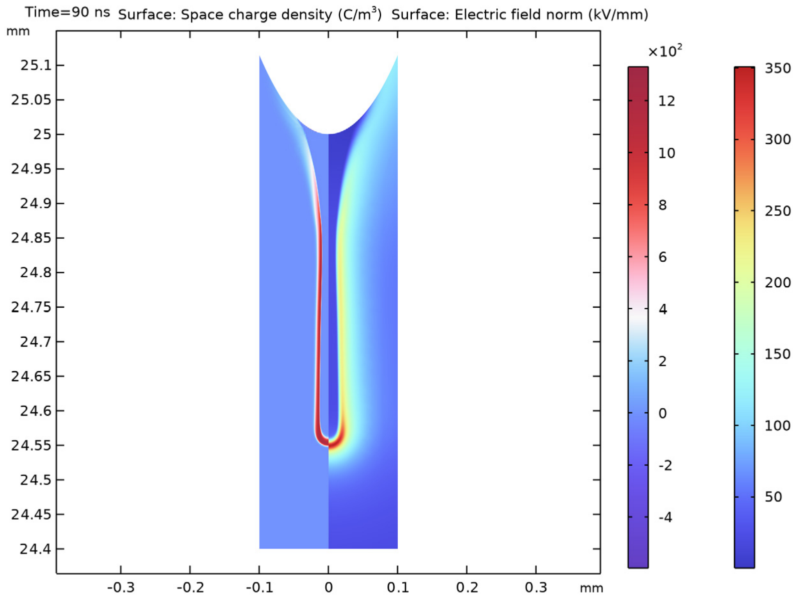
2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

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

2D Plot Group 1

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

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



1D Plot Group 2

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

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

3 From the **Position** list, choose **Lower left**.

Line Graph 1

1 Right-click **1D Plot Group 2** and choose **Line Graph**.

2 Select Boundaries 3 and 5 only.

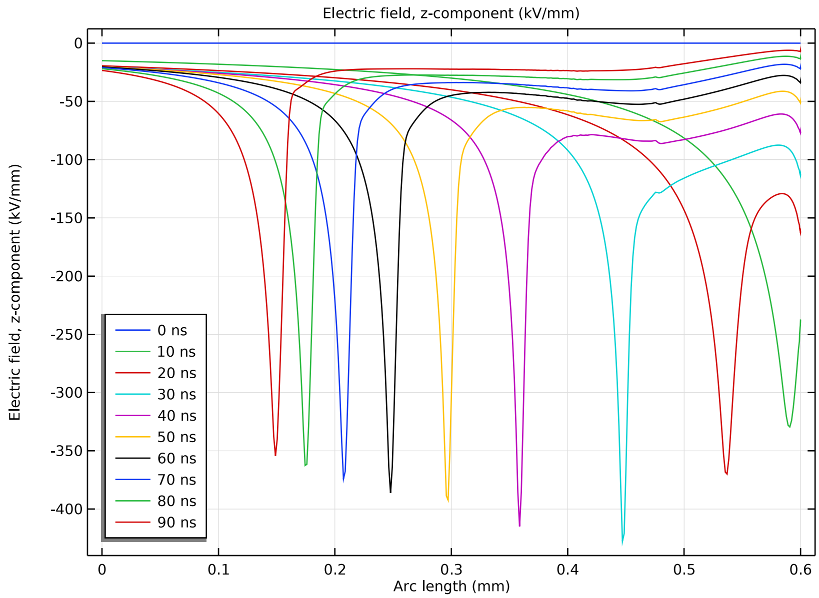
3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `edis.Ez`.

5 In the **Unit** field, type `kV/mm`.

6 Click to expand the **Legends** section. Select the **Show legends** checkbox.

7 In the **1D Plot Group 2** toolbar, click **Plot**.



1D Plot Group 3

1 In the **Model Builder** window, under **Results** right-click **1D Plot Group 2** and choose **Duplicate**.

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

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

4 From the **Position** list, choose **Upper left**.

Line Graph 1

1 In the **Model Builder** window, click **Line Graph 1**.

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

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

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

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

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