Model created in COMSOL Multiphysics 6.4

Trichel Pulses

Introduction

Trichel pulses are a well-known phenomenon in the study of corona discharges, first observed by H. Trichel in the early 20th century. These pulses occur during negative corona discharges, which are electrical discharges brought about by a high voltage applied between a pointed electrode and a flat surface, commonly referred to as a point-to-plate configuration. The negative corona discharge is characterized by a series of rapid, repetitive pulses, resulting from the ionization of the surrounding gas and the subsequent movement of charged particles.

This model simulates Trichel pulses produced by a negative corona discharge under a point-to-plate configuration. The simulation captures key parameters such as electron density, electric field distribution, and discharge current. The model effectively demonstrates the pulsating nature of the discharge current, which is a distinctive characteristic of Trichel pulses. Each current pulse exhibits a rise time of approximately 40 nanoseconds and a decay time of about 150 nanoseconds, highlighting the rapid and transient nature of these electrical events. This simulation provides valuable insights into the dynamics of negative corona discharges and the formation of Trichel pulses. The simulated discharge current is in good agreement with those published in Ref. 1.

Model Definition

The Electric Discharge interface is used to simulate the negative corona discharge. 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:

$$\nabla \cdot (\varepsilon_\text{r} \varepsilon_0 \mathbf{E}) = \rho$$

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

where e is the elementary charge.

The secondary electron emission is responsible for generating current pulses. At the cathode, new electrons are generated by the collision of positive ions to the cathode:

$$-\mathbf{n} \cdot \Gamma_e + = \gamma \mathbf{n} \cdot \Gamma_p$$

where γ is the secondary emission coefficient. The Electrode feature has the built-in boundary condition to model the secondary electron emission.

Results and Discussion

Figure 1 shows the discharge current as a function of time. Figure 2 and Figure 3 plot the distribution of electron density and electric field during a current pulse, respectively.

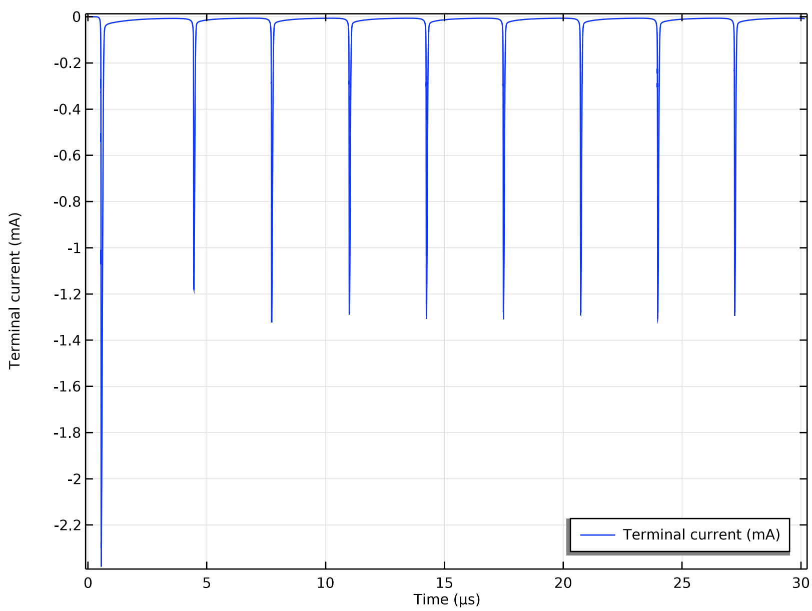


Figure 1: The Trichel pulses.

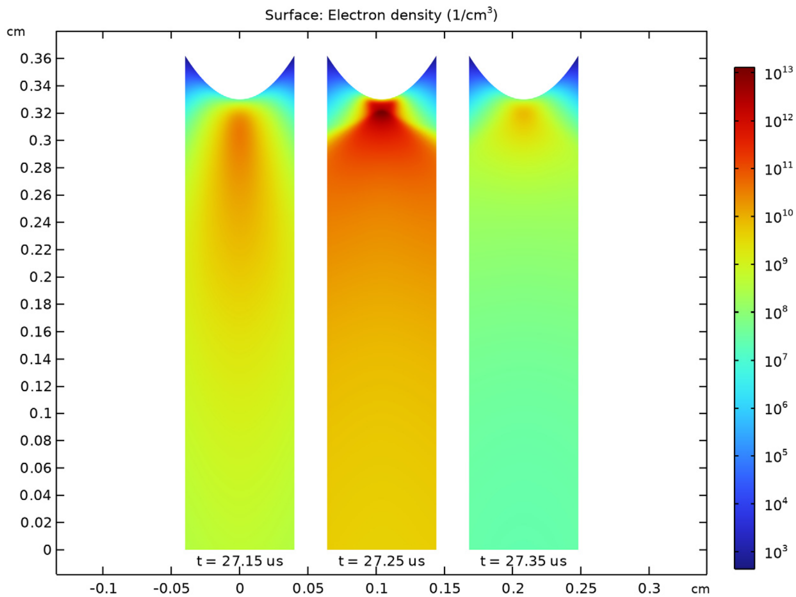


Figure 2: The distributions of electron density during a Trichel pulse.

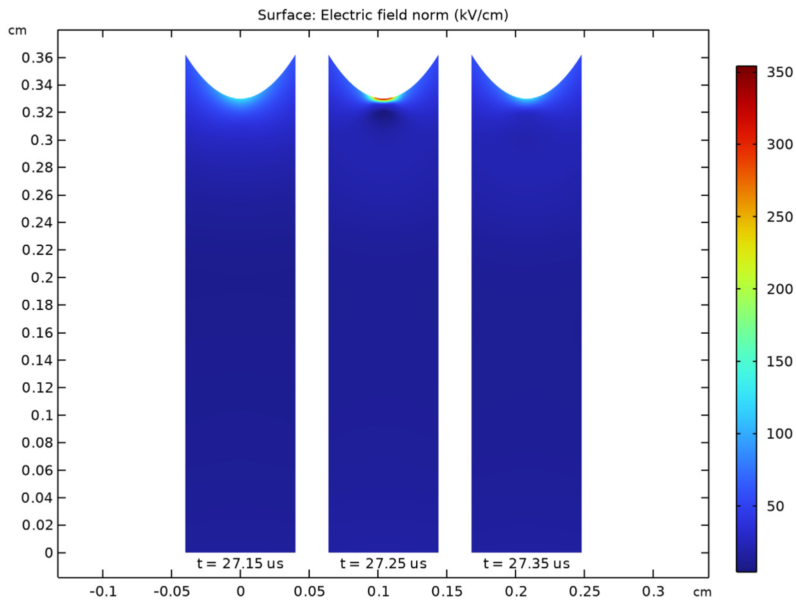


Figure 3: The distributions of the electric field during a Trichel pulse.

References

1. T. Tran and others, "Numerical modelling of negative discharges in air with experimental validation," *J. Phys. D: Appl. Phys.*, vol. 44, no. 1, p. 15203, 2010.

Application Library path: `Electric_Discharge_Module/Corona_Discharges/trichel_pulses`

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 **Preset Studies for Selected Physics Interfaces > Time Dependent with Initialization**.
6 Click **Done**.

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	-5.5[kV]	-5500 V	Applied voltage
Rcurv	250[um]	2.5E-4 m	Radius of curvature
a	1/Rcurv/2*1[cm]	20	Parabola parameter in cm
gap	3.3[mm]	0.0033 m	Gap

DEFINITIONS

Global Variable Probe 1 (var1)

1 In the **Model Builder** window, expand the **Component 1 (comp1) > Definitions** node.
2 Right-click **Definitions** and choose **Probes > Global Variable Probe**.
3 In the **Settings** window for **Global Variable Probe**, type `i0` in the **Variable name** text field.
4 Locate the **Expression** section. In the **Expression** text field, type `edis.I0_0`.
5 In the **Table and plot unit** field, type `mA`.
6 In the **Model Builder** window, collapse the **Definitions** node.

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 **cm**.

Rectangle 1 (r1)

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

2 In the **Settings** window for **Rectangle**, click to expand the **Layers** section.

3 In the table, enter the following settings:

Layer name	Thickness (cm)
`Layer 1`	`0.04`

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

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

6 Click **Build Selected**.

Parametric Curve 1 (pc1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Parametric Curve**.

2 In the **Settings** window for **Parametric Curve**, locate the **Parameter** section.

3 In the **Maximum** text field, type `0.2`.

4 Locate the **Expressions** section. In the **r** text field, type `s`.

5 In the **z** text field, type `a*(s)^2*1[cm]+gap`.

6 Click **Build Selected**.

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 On the object **pc1**, select Point 2 only.

3 In the **Settings** window for **Line Segment**, locate the **Endpoint** section.

4 Click to select the **Activate Selection** toggle button for **End vertex**.

5 On the object **r1**, select Point 2 only.

6 Click **Build Selected**.

Line Segment 2 (ls2)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 On the object **ls1**, select Point 2 only.

3 In the **Settings** window for **Line Segment**, locate the **Endpoint** section.

4 Click to select the **Activate Selection** toggle button for **End vertex**.

5 On the object **pc1**, select Point 1 only.

6 Click to clear the **Activate Selection** toggle button for **End vertex**.

7 Click **Build Selected**.

Convert to Solid 1 (csol1)

1 In the **Geometry** toolbar, click **Conversions** and choose **Convert to Solid**.

2 Select the objects **ls1**, **ls2**, and **pc1** only.

3 In the **Settings** window for **Convert to Solid**, click **Build Selected**.

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 object **csol1** only.

Rectangle 2 (r2)

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

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

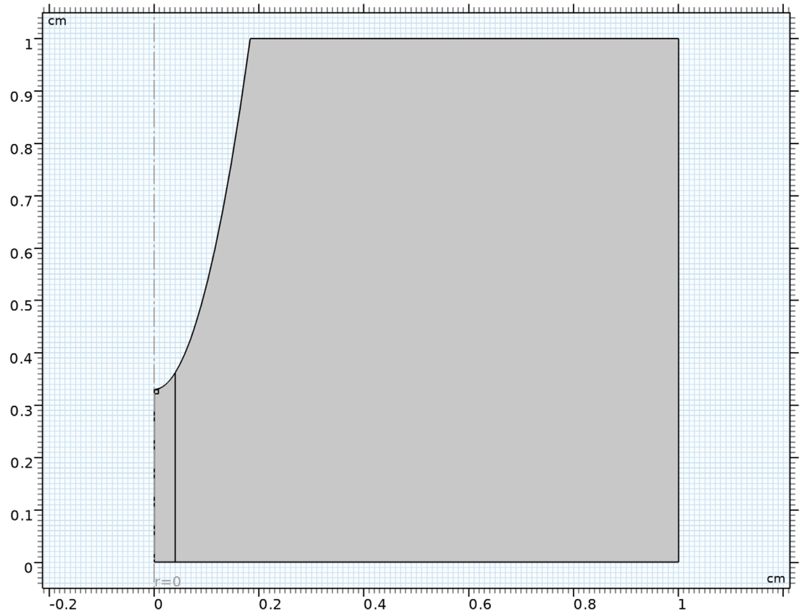
3 In the **Width** text field, type `80[um]`.

4 In the **Height** text field, type `80[um]`.

5 Locate the **Position** section. In the **z** text field, type `gap-85[um]`.

6 Click **Build All Objects**.

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



8 In the **Model Builder** window, collapse the **Geometry 1** node.

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** > **Gases** > **Air** > **Air [Kang et al. 2003]**.

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)

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 **Select All**.

5 Click **OK**.

6 In the **Settings** window for **Electric Discharge**, click to expand the **Consistent Stabilization** section.

7 Clear the **Streamline diffusion** checkbox.

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

Gas 1

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

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

3 Clear the **Include background ionization** checkbox.

4 Locate the **Transport Properties** section. Find the **Diffusion** subsection. From the **Diffusion coefficient** list, choose **User defined**.

5 In the D_e text field, type `0.18`.

6 In the D_p text field, type `0.01`.

7 In the D_n text field, type `0.01`.

Initial Values 1

1 In the **Model Builder** window, click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the n_e text field, type `1E5[1/cm^3]`.

4 In the n_p text field, type `1E5[1/cm^3]`.

5 In the n_n text field, type `1E5[1/cm^3]`.

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 12 and 13 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 **Surface emission**.

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

7 In the $n_{0,n}$ text field, type `1E5[1/cm^3]`.

8 Locate the **Surface Emission** section. Find the **Surface emission mechanisms** subsection. Select the **Secondary electron emission** checkbox.

Gas 1

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

Electrode 2

1 In the **Physics** toolbar, click **Attributes** and choose **Electrode**.
2 Select Boundaries 2 and 9 only.
3 In the **Settings** window for **Electrode**, locate the **Charge Transport** section.
4 From the **Boundary condition for positive ions** list, choose **Number density**.
5 In the $n_{0,p}$ text field, type `1E5[1/cm^3]`.

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 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 `80`.
6 In the **Element ratio** text field, type `5`.
7 Select the **Reverse direction** checkbox.

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 `80`.

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 **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Boundary**.
4 Click the **Zoom Box** button in the **Graphics** toolbar.
5 Click the **Zoom Box** button in the **Graphics** toolbar.
6 Select Boundaries 1, 5, and 12 only.
7 Locate the **Element Size** section. Click the **Custom** button.
8 Locate the **Element Size Parameters** section.
9 Select the **Maximum element size** checkbox. In the associated text field, type 1/1000.

Size 2

1 Right-click **Free Triangular 1** and choose **Size**.
2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 Select Domain 1 only.
5 Locate the **Element Size** section. Click the **Custom** button.
6 Locate the **Element Size Parameters** section.
7 Select the **Maximum element size** checkbox. In the associated text field, type 1/200.
8 Select the **Maximum element growth rate** checkbox. In the associated text field, type 1.1.
9 Click **Build All**.

Boundary Layers 1

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

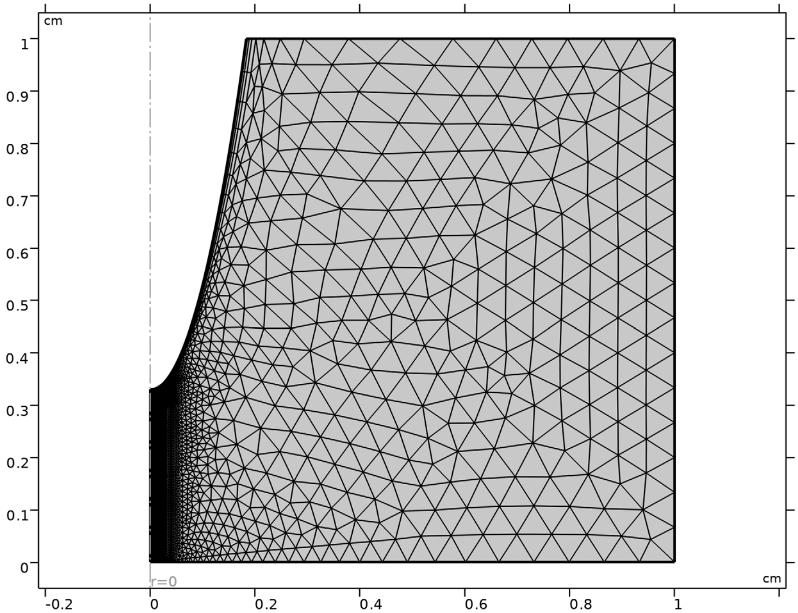
Boundary Layer Properties

1 In the **Model Builder** window, click **Boundary Layer Properties**.
2 In the **Settings** window for **Boundary Layer Properties**, locate the **Boundary Selection** section.
3 Click **Paste Selection**.
4 In the **Paste Selection** dialog, type 12-13 in the **Selection** text field.
5 Click **OK**.
6 In the **Settings** window for **Boundary Layer Properties**, locate the **Layers** section.
7 In the **Number of layers** text field, type 4.

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

9 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

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



11 In the **Model Builder** window, collapse the **Mesh 1** node.

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 From the **Time unit** list, choose **µs**.

4 In the **Output times** text field, type `range(0,3,27) 27.15 27.25 27.35 30`.

5 Click to expand the **Results While Solving** section. 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.

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 Select the **Initial step** checkbox. In the associated text field, type `1e-15[s]`.

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

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

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

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

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

RESULTS

Mirror 2D 1

1 In the **Model Builder** window, expand the **Results** > **Datasets** node.

2 Right-click **Results** > **Datasets** and choose **More 2D Datasets** > **Mirror 2D**.

3 In the **Settings** window for **Mirror 2D**, click to expand the **Advanced** section.

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 1 and 2 only.

Discharge Current

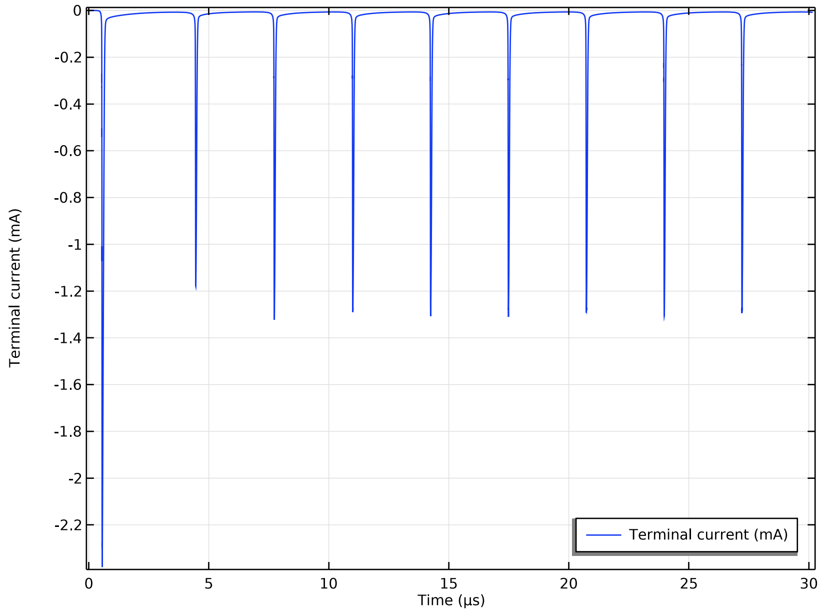
1 In the **Model Builder** window, under **Results** click **Probe Plot Group 1**.

2 In the **Settings** window for **1D Plot Group**, type `Discharge Current` in the **Label** text field.

3 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

4 In the **Discharge Current** toolbar, click **Plot**.

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



Electron Density

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

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

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

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

Surface 1

1 Right-click **Electron Density** and choose **Surface**.

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

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

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

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

Solution Array 1

1 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 **From list**.

4 In the **Times (μs)** list, choose **27.15**, **27.25**, and **27.35**.

Electron Density

1 In the **Model Builder** window, under **Results** click **Electron Density**.

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

3 From the **Title type** list, choose **Custom**.

4 Find the **Solution** subsection. Clear the **Solution** checkbox.

Annotation 1

1 Right-click **Electron 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,us,4) us`.

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

5 From the **Anchor point** list, choose **Upper middle**.

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

Solution Array 1

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

Solution Array 1

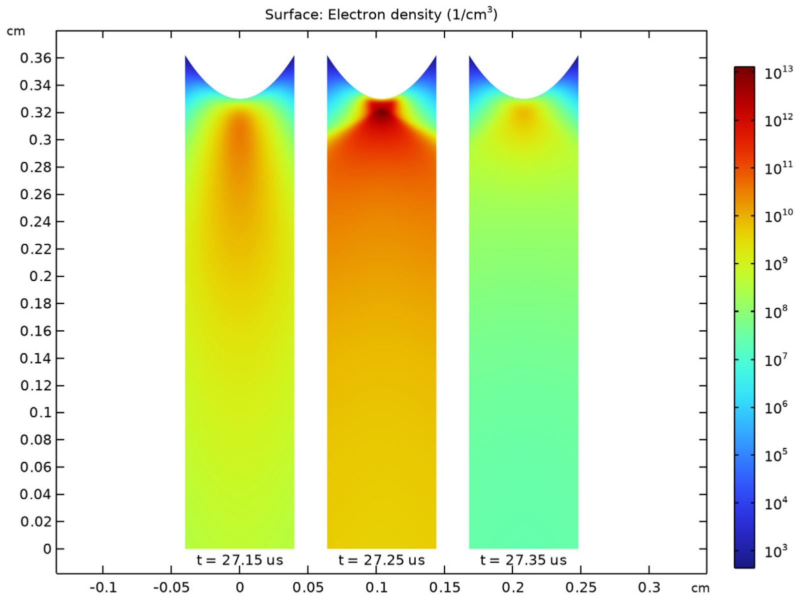
In the **Model Builder** window, right-click **Annotation 1** and choose **Paste Solution Array**.

Electron Density

1 In the **Electron Density** toolbar, click **Plot**.

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

3 In the **Model Builder** window, under **Results** click **Electron Density**.



Electron Field

1 Right-click **Electron Density** and choose **Duplicate**.

2 In the **Settings** window for **2D Plot Group**, type `Electron Field` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Electron Field** node, then click **Surface 1**.

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/cm`.

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

Electron Field

1 In the **Model Builder** window, click **Electron Field**.

2 In the **Electron Field** toolbar, click **Plot**.

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