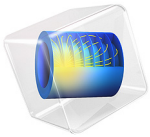

DC Corona Discharge in Air in a Point-to-Plane Configuration

Introduction

This tutorial model presents a study of a negative corona discharge in dry air at atmospheric pressure in a point-to-plane configuration. An elliptical electrode with millimeter dimensions creates a high intensity electric field where the corona discharge occurs. A ground plane is placed 10 cm away from the corona electrode. The voltage is varied between –5 to –50 kV. The creation and transport of charged species is solved self-consistently coupled with Poisson's equation.

Model Definition

The model solves the electron and ions continuity and momentum equations, in the drift-diffusion approximation, self-consistently coupled with Poisson's equation. The local field approximation is used, which means that transport and source coefficients are assumed to be well parameterized through the reduced electric field (E/N). In the local field approximation the fluid equation for the mean electron energy is not solved, which reduces significantly the complexity of the numerical problem.

The local field approximation is valid in a situation where the rate of electron energy gain from the electric field is locally balanced by the energy loss rate. When this condition is met the electrons are said to be in local equilibrium with the electric field and the electron mean properties can be expressed as a function the reduced electric field.

This model uses streamline diffusion stabilization techniques for all charged species. When using the finite element formulation this is needed to avoid instabilities in problems with high Peclet number such as corona discharges and streamers (inconsistent and consistent stabilization are only available for the **Finite element** options and are not available for the **Finite element, Log formulation** options). For the present case, isotropic diffusion is added to the ions. This makes the problem easier to solve but attention should be made to not introduce unphysical behaviors to the system. Isotropic diffusion should be used as little as possible.

When applying a DC voltage to a system like the one studied here, it is possible to obtain current pulses normally referred to as Trichel pulses for certain operating conditions. COMSOL Multiphysics can naturally capture these pulses, and in such cases a steady-state solution is not possible.

Negative coronas are much easier to model than positive coronas. Simulations of positive coronas are still considered difficult and in many cases a solution is not possible to obtain. Photoionization is an electron creation mechanism that is known to be important in

corona discharges. This model was design to be a generic point-to-plane corona discharge and does not include photoionization for simplicity.

DOMAIN EQUATIONS

The electron density is computed by solving the drift-diffusion equation for the electron density:

$$\frac{\partial}{\partial t}(n_e) + \nabla \cdot [-n_e(\mu_e \bullet \mathbf{E}) - \mathbf{D}_e \bullet \nabla n_e] = R_e \tag{1}$$

For more detailed information on electron transport see the section *Theory for the Drift Diffusion Interface* in the *Plasma Module User's Guide*.

The source coefficients in the above equation is determined by the plasma chemistry using rate coefficients. Suppose that there are M reactions that contribute to the growth or decay of electron density and P inelastic electron-neutral collisions. In general, $P \gg M$. The electron source term is given by

$$R_e = \sum_{j=1}^{M} x_j k_j N_n n_e \tag{2}$$

where x_j is the mole fraction of the target species for reaction j, k_j is the rate coefficient for reaction j (SI unit: m^3/s), and N_n is the total neutral number density (SI unit: $1/\mathrm{m}^3$). For DC discharges it is better practice to use Townsend coefficients instead of rate coefficients to define reaction rates. Townsend coefficients provide a better description of what happens in the cathode fall region Ref. 1. When Townsend coefficients are used, the electron source term is given by:

$$R_e = \sum_{j=1}^{M} x_j \alpha_j N_n |\Gamma_e| \tag{3}$$

where α_j is the Townsend coefficient for reaction j (SI unit: m^2), and Γ_e is the electron flux as defined above (SI unit: $1/(\mathrm{m}^2 \cdot \mathrm{s})$). Townsend coefficients can increase the stability of the numerical scheme when the electron flux is field driven as is the case with DC discharges.

For nonelectron species, the following equation is solved for the mass fraction of each species:

$$\rho\frac{\partial}{\partial t}(w_k) + \rho(\mathbf{u}\cdot\nabla)w_k = \nabla\cdot\mathbf{j}_k + R_k \tag{4}$$

For detailed information on the transport of the nonelectron species see the section *Theory for the Heavy Species Transport Interface* in the *Plasma Module User's Guide*.

The electrostatic field is computed using the equation

$$-\nabla\cdot\varepsilon_0\varepsilon_r\nabla V = \rho \tag{5}$$

The space charge density ρ is automatically computed based on the plasma chemistry specified in the model using the formula

$$\rho = q\left(\sum_{k=1}^{N} Z_k n_k - n_e\right) \tag{6}$$

For detailed information about electrostatics see *Theory for the Electrostatics Interface* in the *Plasma Module User's Guide*.

BOUNDARY CONDITIONS

Electrons are lost to the wall due to random motion within a few mean free paths of the wall and gained due to secondary emission effects, resulting in the boundary condition for the electron flux

$$\mathbf{n}\cdot\Gamma_e = \left(\frac{1}{2}\nu_{e,\,\mathrm{th}} n_e\right) - \sum_p \gamma_p(\Gamma_p\cdot\mathbf{n}) \tag{7}$$

The second term on the right-hand side is the gain of electrons due to secondary emission effects, γ_p being the secondary emission coefficient. For the heavy species, ions are lost to the wall due to surface reactions and the fact that the electric field is directed toward the wall:

$$\mathbf{n}\cdot\mathbf{j}_k = M_w R_k + M_w c_k Z\mu_k(\mathbf{E}\cdot\mathbf{n})[Z_k\mu_k(\mathbf{E}\cdot\mathbf{n}) > 0] \tag{8}$$

The discharge is driven by a DC electric potential (V_0) applied to small electrode. To facilitate the beginning of the numerical simulation a step function is used to modulate V_0 with the transient applied potential assuming the form

$$V = V_0\tanh\left(\frac{t}{\tau}\right). \tag{9}$$

This numeric technique does not interfere with the results at steady state, which are the ones of interest in this work.

PLASMA CHEMISTRY

The chemistry of a plasma sustained in air can be very complex and a detailed study of the main excited states could easily have hundreds of reactions. The main goal of this model is to study the charge particle density profiles and currents. With that in mind it is used a simplified set of reactions that describe correctly the creation and destruction of charged species in a background of dry air.

Table 1 lists the chemical reactions considered Ref. 2. In the fluid equations nitrogen and oxygen are not treated separately as in a detailed chemistry. Instead a generic species A is used for the background gas. A can be ionized forming positive ions, and A can attach electrons forming negative ions.

TABLE 1: TABLE OF COLLISIONS AND REACTIONS MODELED.

Reaction	Formula	Type	$\Delta\varepsilon$ (eV)	k_f (m^3/s)
1	e+A=>A++2e	Ionization	-	-
2	e+A=>A-	Attachment	-	-
3	e+2A=>A-+A	Attachment	-	-
4	e+A+=>A	Reaction	-	$5 \cdot 10^{-14}$
5	A-+A+=>2A	Reaction	-	$2 \cdot 10^{-12}$

At steady state, the plasma main charged species are ions. For this reason, the initial conditions have an equal density of positive and negative ions and a small density of electrons. These initial conditions preserve charge neutrality as it is important for numerical reasons.

In addition to the volumetric reactions, the following surface reactions are implemented:

TABLE 2: TABLE OF SURFACE REACTIONS.

Reaction	Formula	Sticking coefficient
1	A+=>A	1
2	A-=>A	1

When the ions reach the wall, they are assumed to change back to neutral atoms.

For this model the source terms and transport parameters are given directly as a function of the reduced electric field. This way, it is not necessary to introduce in the model a relation between the reduced electric field and the electron mean energy that normally is set in the section **Mean Electron Energy Specification** in the **Plasma Model** feature. Since the

mean electron energy uses the default value of 3 V the electron temperature in the results section has a constant value of 2 V.

Results and Discussion

In this section, the results of the plasma self-consistent model for the negative corona are presented. Figure 1 to Figure 4 present the spatial distribution of the charged species for an applied potential of -55 kV. The number densities of all species reaches values of 10^{18} m^{-3} in the vicinity of the point and drop very fast away from it. The dominant species away from the electrode tip are the negative ions that are formed by electron attachment and drift toward the ground closing the current loop. The current as a function of the voltage is presented in Figure 5. The collected current naturally increases with the potential difference but note how small is the current. These low current values are a characteristic of corona discharges.

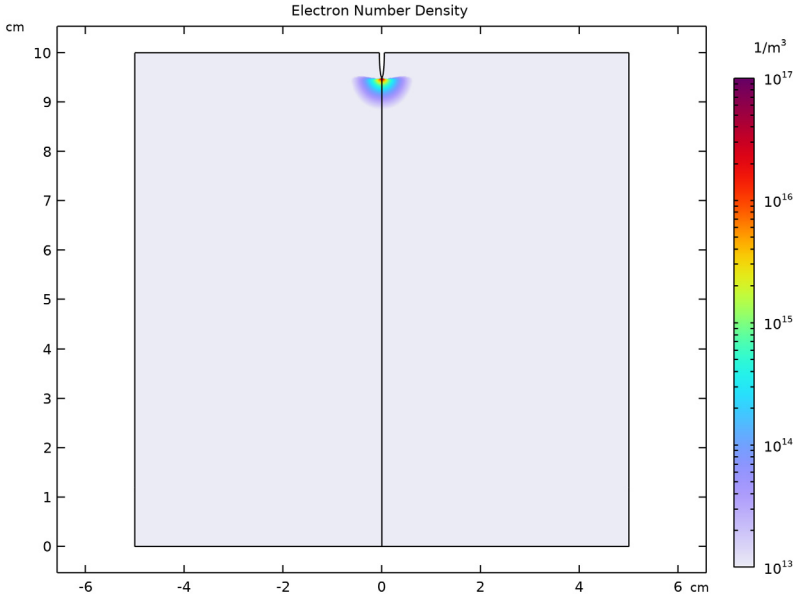

Figure 1: Electron number density for an applied voltage of -50 kV.

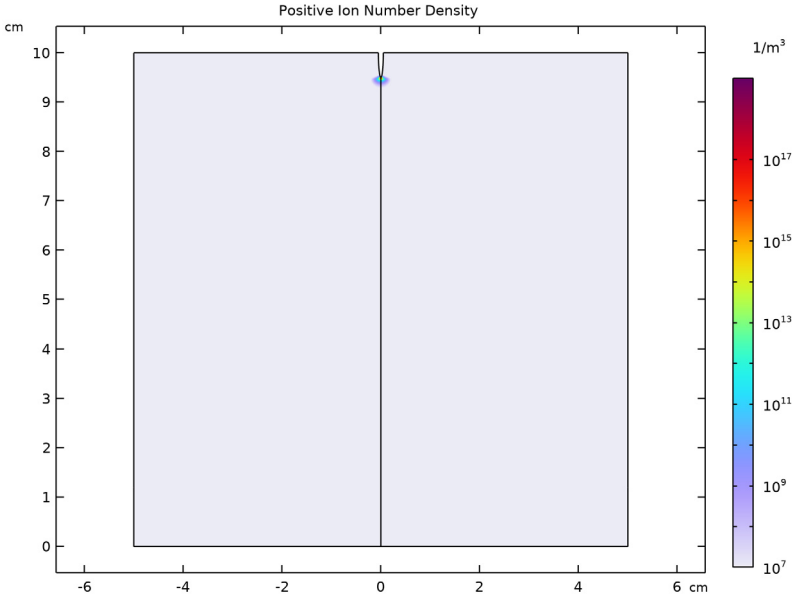

Figure 2: Positive ion number density for an applied voltage of -50 kV.

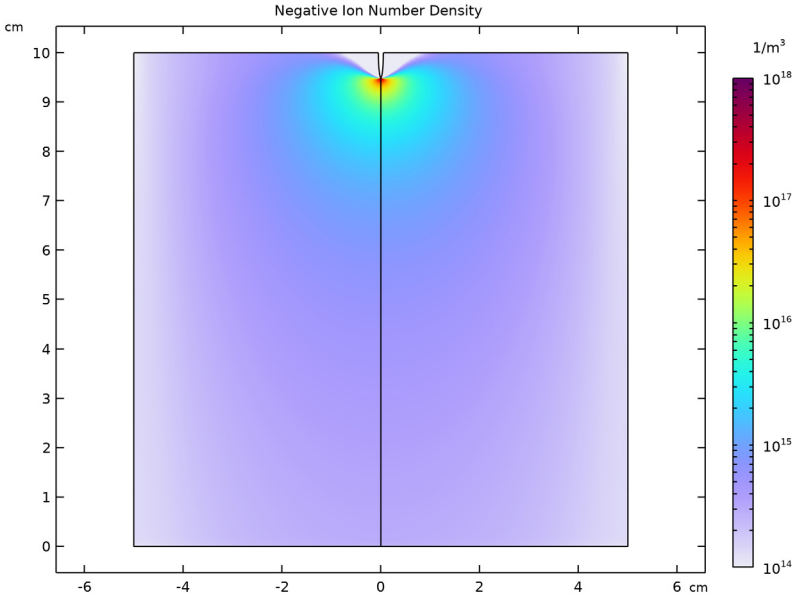

Figure 3: Negative ion number density for an applied voltage of -50 kV.

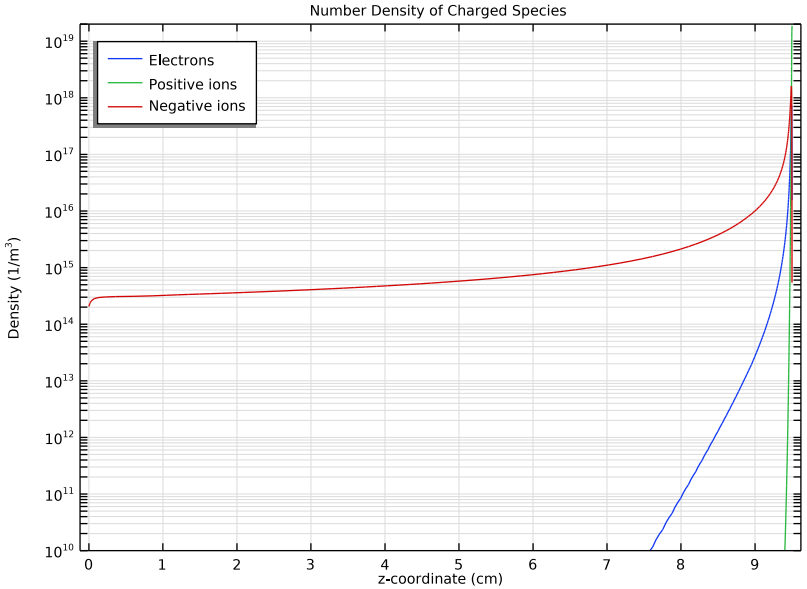

Figure 4: Number density of the charged species along the axis of symmetry for an applied voltage of -50 kV.

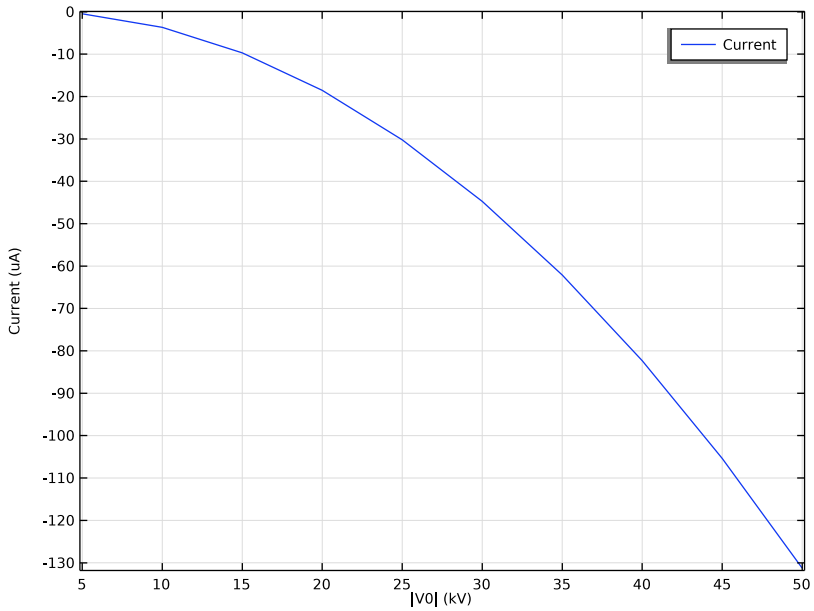

Figure 5: Current at the point as a function of the applied voltage.

References

1. M.A. Lieberman and A.J. Lichtenberg, *Principles of Plasma Discharges and Materials Processing*, John Wiley & Sons, 2005.

2. A.A. Kulikovsky, "Positive streamer between parallel plate electrode in atmospheric pressure air," *J. Phys. D: Appl. Phys.*, vol. 30, pp. 441–450, 1997.

Application Library path: `Plasma_Module/Corona_Discharges/point_to_plane_dc_corona`

Modeling Instructions

In this model a time-dependent study is used to obtain a solution at -5 kV. Even at this voltage it is difficult to start the simulation and a ramp function is used. The solution of this study are given as initial conditions to a stationary study that easily ramp the voltage

to -50 kV. Since the ramp function uses the variable time it cannot be used in the stationary study. That is why two **Metal contact** features are added and disabled as needed at the study level.

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 **Plasma>Plasma (plas)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GEOMETRY 1

Create the simulation geometry that consists of thin needle 10 cm away from a ground plane.

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**, locate the **Size and Shape** section.

3 In the **Width** text field, type `5`.

4 In the **Height** text field, type `10`.

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 `0.5[mm]`.

4 In the **b-semiaxis** text field, type `5[mm]`.

5 In the **Sector angle** text field, type `180`.

6 Locate the **Position** section. In the **z** text field, type 10.

7 Locate the **Rotation Angle** section. In the **Rotation** text field, type 180.

Ellipse 2 (e2)

1 Right-click **Ellipse 1 (e1)** and choose **Duplicate**.

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

3 In the **a-semiaxis** text field, type `3[mm]`.

4 In the **b-semiaxis** text field, type `10.5[mm]`.

5 Click **Build All Objects**.

Difference 1 (dif1)

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

2 Select the objects **e2** and **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 **e1** only.

Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

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

4 On the object **dif1**, select Domain 1 only.

Mesh Control Edges 1 (mce1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Mesh Control Edges**.

2 On the object **fin**, select Boundary 7 only.

3 In the **Geometry** toolbar, click **Build All**.

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

Add the applied voltage as a parameter so that one can ramp the voltage with a stationary solver using the continuation solver.

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[kV]	-5000 V	Applied Voltage

DEFINITIONS

Variables 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.

Import variables to be used in the model.

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

3 Click **Load from File**.

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

Add analytic functions to define Townsend coefficients and electron mobility.

alpha

1 In the **Home** toolbar, click **Functions** and choose **Local>Analytic**.

2 In the **Settings** window for **Analytic**, type `alpha` in the **Label** text field.

3 In the **Function name** text field, type `alpha`.

4 Locate the **Definition** section. In the **Expression** text field, type `1.4e-20*exp(-660/x)`.

5 Locate the **Units** section. In the **Function** text field, type `m^2`.

6 In the table, enter the following settings:

Argument	Unit
x	Td

eta1

1 In the **Home** toolbar, click **Functions** and choose **Local>Analytic**.

2 In the **Settings** window for **Analytic**, type `eta1` in the **Label** text field.

3 In the **Function name** text field, type `eta1`.

4 Locate the **Definition** section. In the **Expression** text field, type `6e-23*exp(-100/x)`.

5 Locate the **Units** section. In the **Function** text field, type `m^2`.

6 In the table, enter the following settings:

Argument	Unit
x	Td

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

Argument	Lower limit	Upper limit	Unit	Plot
x	0	10000[Td]	V*m^2	√

eta2

1 In the **Home** toolbar, click f(x) **Functions** and choose **Local>Analytic**.

2 In the **Settings** window for **Analytic**, type `eta2` in the **Label** text field.

3 In the **Function name** text field, type `eta2`.

4 Locate the **Definition** section. In the **Expression** text field, type `1.6e-37*(x)^-1.1`.

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

6 In the table, enter the following settings:

Argument	Unit
x	Td

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

Argument	Lower limit	Upper limit	Unit	Plot
x	0	10000[Td]	V*m^2	√

mueN

1 In the **Home** toolbar, click f(x) **Functions** and choose **Local>Analytic**.

2 In the **Settings** window for **Analytic**, type `mueN` in the **Label** text field.

3 In the **Function name** text field, type `mueN`.

4 Locate the **Definition** section. In the **Expression** text field, type `3.74e22*(x^-0.25)`.

5 Locate the **Units** section. In the **Function** text field, type `1/V/s/cm`.

6 In the table, enter the following settings:

Argument	Unit
x	Td

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

Argument	Lower limit	Upper limit	Unit	Plot
x	0	10000[Td]	V*m^2	√

Define an explicit selection to make our live easier when setting boundary conditions.

Wall

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

2 In the **Settings** window for **Explicit**, type `Wall` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 2 and 5 only.

Choose to use the local field approximation. Also, choose to use the Finite element (linear shape function) option that adds streamline stabilization by default. Streamline stabilization is not available for the log formulation.

Streamline stabilization is necessary to make problems of this class, with high Peclet numbers, to converge. Some isotropic diffusion is added to ions to avoid to use an excessive number of mesh elements.

DEFINITIONS

In the **Model Builder** window, collapse the **Component 1 (comp1)>Definitions** node.

GEOMETRY 1

In the **Model Builder** window, collapse the **Component 1 (comp1)>Geometry 1** node.

PLASMA (PLAS)

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

2 In the **Settings** window for **Plasma**, locate the **Plasma Properties** section.

3 From the **Mean electron energy** list, choose **Local field approximation**.

4 Select the **Use reduced electron transport properties** check box.

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

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

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

8 In the tree, select the check box for the node **Physics>Stabilization**.

9 Click **OK**.

10 In the **Settings** window for **Plasma**, click to expand the **Inconsistent Stabilization** section.

11 Select the **Isotropic diffusion for ions** check box.

Set the plasma model: define the operation pressure and temperature and define the plasma chemistry.

Plasma Model 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Plasma (plas)** click **Plasma Model 1**.

2 In the **Settings** window for **Plasma Model**, locate the **Model Inputs** section.

3 In the T text field, type `t0`.

4 In the p_A text field, type `p0`.

5 Locate the **Electron Density and Energy** section. From the **Electron transport properties** list, choose **Specify all**.

6 In the $\mu_e N_n$ text field, type `mueN(plas.Erd)`.

7 In the $D_e N_n$ text field, type `DeN`.

Electron Impact Reaction 1

1 In the **Physics** toolbar, click **Domains** and choose **Electron Impact Reaction**.

2 In the **Settings** window for **Electron Impact Reaction**, locate the **Reaction Formula** section.

3 In the **Formula** text field, type `e+A=>A++2e`.

4 Locate the **Collision Type** section. From the **Collision type** list, choose **Ionization**.

5 In the $\Delta\varepsilon$ text field, type `15`.

6 Locate the **Reaction Parameters** section. In the k^f text field, type `Ri`.

Electron Impact Reaction 2

1 In the **Physics** toolbar, click **Domains** and choose **Electron Impact Reaction**.

2 In the **Settings** window for **Electron Impact Reaction**, locate the **Reaction Formula** section.

3 In the **Formula** text field, type `A+e=>A-`.

4 Locate the **Collision Type** section. From the **Collision type** list, choose **Attachment**.

5 Locate the **Reaction Parameters** section. In the k^f text field, type `Ratt1+Ratt2`.

Reaction 1

1 In the **Physics** toolbar, click **Domains** and choose **Reaction**.

2 In the **Settings** window for **Reaction**, locate the **Reaction Formula** section.

3 In the **Formula** text field, type `e+A+=>A`.

4 Locate the **Reaction Parameters** section. In the k^f text field, type `betaep`.

Reaction 2

1 In the **Physics** toolbar, click **Domains** and choose **Reaction**.

2 In the **Settings** window for **Reaction**, locate the **Reaction Formula** section.

3 In the **Formula** text field, type `A-+A+=>A+A`.

4 Locate the **Reaction Parameters** section. In the k^f text field, type `betapn`.

Species: A

1 In the **Model Builder** window, click **Species: A**.

2 In the **Settings** window for **Species**, locate the **General Parameters** section.

3 From the **Preset species data** list, choose **N2**.

4 Locate the **Species Formula** section. Select the **From mass constraint** check box.

Species: A+

1 In the **Model Builder** window, click **Species: A+**.

2 In the **Settings** window for **Species**, locate the **General Parameters** section.

3 In the n_0 text field, type `ni0`.

4 From the **Preset species data** list, choose **N2**.

Species: A-

1 In the **Model Builder** window, click **Species: A-**.

2 In the **Settings** window for **Species**, locate the **General Parameters** section.

3 From the **Preset species data** list, choose **N2**.

4 In the n_0 text field, type `ni0`.

Define the boundary conditions for the heavy species and electrons transport equations, and for Poisson's equation.

Surface Reaction 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Surface Reaction**.

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

3 In the **Formula** text field, type `A+=>A`.

4 Locate the **Boundary Selection** section. From the **Selection** list, choose **Wall**.

5 Locate the **Secondary Emission Parameters** section. In the γ_i text field, type `0.05`.

6 In the ε_i text field, type `4`.

2: A+=>A

1 Right-click **1: A+=>A** and choose **Duplicate**.

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

3 In the **Formula** text field, type `A-=>A`.

4 Locate the **Secondary Emission Parameters** section. In the γ_i text field, type `0`.

5 In the ε_i text field, type `0`.

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 $n_{e,0}$ text field, type `ne0`.

Ground 1

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

2 Select Boundary 2 only.

Two metal contact features are defined here but at the study level only one is going to be used at a time by disabling the other.

For the time dependent study the final voltage is multiplied by a ramp function to make convergence easier. The stationary solver receives the parameter V0 directly.

Metal Contact 1, ramp

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

2 In the **Settings** window for **Metal Contact**, locate the **Terminal** section.

3 In the V_0 text field, type `Vapp`.

4 Select Boundary 5 only.

5 In the **Label** text field, type `Metal Contact 1, ramp`.

Metal Contact 2, constant

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

2 In the **Settings** window for **Metal Contact**, locate the **Terminal** section.

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

4 Select Boundary 5 only.

5 In the **Label** text field, type `Metal Contact 2, constant`.

Wall 1

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

2 In the **Settings** window for **Wall**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Wall**.

Create a mesh that is extremely fine at the point where the electric field is more intense.

MESH 1

Size 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 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 Select Boundaries 1, 5, and 6 only.

5 Locate the **Element Size** section. From the **Calibrate for** list, choose **Plasma**.

6 From the **Predefined** list, choose **Fine**.

Size

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

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Calibrate for** list, choose **Plasma**.

4 From the **Predefined** list, choose **Fine**.

Edge 1

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

2 Select Boundaries 3, 5, and 6 only.

Distribution 1

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

2 In the **Settings** window for **Distribution**, locate the **Boundary Selection** section.

3 Click  **Clear Selection**.

4 Select Boundary 5 only.

5 Locate the **Distribution** section. From the **Distribution type** list, choose **Predefined**.

6 In the **Number of elements** text field, type 150.

7 In the **Element ratio** text field, type 150.

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

Distribution 2

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

2 In the **Settings** window for **Distribution**, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 Select Boundary 6 only.

5 Locate the **Distribution** section. From the **Distribution type** list, choose **Predefined**.

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

7 In the **Element ratio** text field, type 80.

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

9 Select the **Reverse direction** check box.

Distribution 3

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

2 In the **Settings** window for **Distribution**, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 Select Boundary 3 only.

5 Locate the **Distribution** section. From the **Distribution type** list, choose **Predefined**.

6 In the **Number of elements** text field, type 13.

7 In the **Element ratio** text field, type 2.

Free Triangular 1

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

Size 1

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 2 only.

5 Locate the **Element Size** section. From the **Calibrate for** list, choose **Plasma**.

6 From the **Predefined** list, choose **Extremely fine**.

Size 2

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 **Domain**.

4 Select Domain 1 only.

5 Locate the **Element Size** section. From the **Calibrate for** list, choose **Plasma**.

6 From the **Predefined** list, choose **Fine**.

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 **Layers** section.

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

4 Select Boundary 2 only.

5 Click **Build All**.

Set the study to observe the results while solving. And disable the Metal Contact feature that does not have a ramp function.

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 In the **Output times** text field, type `10^{range(log10(1.0e-8),1/10,log10(1))}`.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

5 In the tree, select **Component 1 (comp1)>Plasma (plas)>Metal Contact 2, constant**.

6 Right-click and choose **Disable**.

7 In the **Study** toolbar, click **Get Initial Value**.

RESULTS

Mirror 2D 1

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

Electron Density (plas)

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

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

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

Electric Potential (plas)

1 In the **Model Builder** window, click **Electric Potential (plas)**.

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

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

Electric Potential (plas), Electron Density (plas), Electron Temperature (plas)

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Electron Density (plas)**, **Electron Temperature (plas)**, and **Electric Potential (plas)**.

2 Right-click and choose **Group**.

Time dependent -5 kV

In the **Settings** window for **Group**, type `Time dependent -5 kV` in the **Label** text field.

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**, click to expand the **Results While Solving** section.

3 Select the **Plot** check box.

4 From the **Update at** list, choose **Time steps taken by solver**.

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

Add a stationary solver to ramp the voltage to -50 kV using the continuation solver. Do not forget to disable the Metal Contact feature that uses the ramp function. Use the solutions of the time-dependent study as initial conditions.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

4 Click **Add Study** in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Stationary

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

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

3 In the tree, select **Component 1 (comp1)>Plasma (plas)>Metal Contact 1, ramp**.

4 Click **Disable**.

5 Click to expand the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.

6 From the **Method** list, choose **Solution**.

7 From the **Study** list, choose **Study 1, Time Dependent**.

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

9 Click + **Add**.

10 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
V0 (Applied Voltage)	`range(-5,-5,-50)`	kV

11 In the **Study** toolbar, click **Get Initial Value**.

RESULTS

Mirror 2D 2

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

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

3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

Electron Density (plas) 1

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

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

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

4 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Electron Number Density`.

6 Clear the **Parameter indicator** text field.

7 Locate the **Color Legend** section. Select the **Show units** check box.

Surface 1

1 In the **Model Builder** window, expand the **Electron Density (plas) 1** node, then click **Surface 1**.

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

3 From the **Scale** list, choose **Logarithmic**.

4 Click to expand the **Range** section. Select the **Manual color range** check box.

5 In the **Minimum** text field, type `1e13`.

6 In the **Maximum** text field, type `1e17`.

7 In the **Electron Density (plas) 1** toolbar, click **Plot**.

Electric Potential (plas) 1

1 In the **Model Builder** window, under **Results** click **Electric Potential (plas) 1**.

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

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

Electric Potential (plas) 1, Electron Density (plas) 1, Electron Temperature (plas) 1

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Electron Density (plas) 1**, **Electron Temperature (plas) 1**, and **Electric Potential (plas) 1**.

2 Right-click and choose **Group**.

Stationary, ramping to -50 kV

In the **Settings** window for **Group**, type `Stationary, ramping to -50 kV` in the **Label** text field.

STUDY 2

In the **Home** toolbar, click **Compute**.

RESULTS

Positive Ion Number Density

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

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

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

4 Locate the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Positive Ion Number Density`.

6 Clear the **Parameter indicator** text field.

7 Locate the **Color Legend** section. Select the **Show units** check box.

Surface 1

1 Right-click **Positive Ion Number Density** and choose **Surface**.

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

3 In the **Expression** text field, type `plas.n_wA_1p`.

4 In the **Positive Ion Number Density** toolbar, click **Plot**.

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

6 Click to expand the **Range** section. Select the **Manual color range** check box.

7 In the **Minimum** text field, type `1e7`.

8 In the **Maximum** text field, type `1e19`.

9 In the **Positive Ion Number Density** toolbar, click **Plot**.

Negative Ion Number Density

1 In the **Model Builder** window, right-click **Positive Ion Number Density** and choose **Duplicate**.

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

3 Locate the **Title** section. In the **Title** text area, type `Negative Ion Number Density`.

Surface 1

1 In the **Model Builder** window, expand the **Negative Ion Number Density** node, then click **Surface 1**.

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

3 In the **Expression** text field, type `plas.n_wA_1m`.

4 Locate the **Range** section. In the **Minimum** text field, type `1e14`.

5 In the **Maximum** text field, type `1e18`.

6 In the **Negative Ion Number Density** toolbar, click **Plot**.

Number Density of Charged Species

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

2 In the **Settings** window for **1D Plot Group**, type `Number Density of Charged Species` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 From the **Parameter selection (V0)** list, choose **Last**.

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

6 Locate the **Plot Settings** section.

7 Select the **x-axis label** check box. In the associated text field, type `z-coordinate (cm)`.

8 Select the **y-axis label** check box. In the associated text field, type `Density (1/m<sup>3</sup>)`.

9 Locate the **Axis** section. Select the **Manual axis limits** check box.

10 In the **x minimum** text field, type `-0.12`.

11 In the **x maximum** text field, type `9.62`.

12 In the **y minimum** text field, type `1e10`.

13 In the **y maximum** text field, type `2e19`.

14 Select the **y-axis log scale** check box.

15 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Line Graph 1

1 Right-click **Number Density of Charged Species** and choose **Line Graph**.

2 Select Boundary 1 only.

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

4 In the **Expression** text field, type `plas.ne`.

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

6 In the **Expression** text field, type `z`.

7 Click to expand the **Legends** section. Select the **Show legends** check box.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends
Electrons

10 In the **Number Density of Charged Species** toolbar, click **Plot**.

Line Graph 2

1 Right-click **Line Graph 1** and choose **Duplicate**.

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

3 In the **Expression** text field, type `plas.n_wA_1p`.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
Positive ions

Line Graph 3

1 Right-click **Line Graph 2** and choose **Duplicate**.

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

3 In the **Expression** text field, type `plas.n_wA_1m`.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
Negative ions

5 In the **Number Density of Charged Species** toolbar, click **Plot**.

Voltage vs Current

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 Locate the **Title** section. From the **Title type** list, choose **None**.

Global 1

1 Right-click **Voltage vs Current** 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
`plas.I_2`	uA	`Current`

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

5 In the **Expression** text field, type `-V0`.

6 From the **Unit** list, choose **kV**.

7 Select the **Description** check box. In the associated text field, type `|V0|`.

8 In the **Voltage vs Current** toolbar, click **Plot**.