Model created in COMSOL Multiphysics 6.4

Electrohydrodynamic Flow in Dielectric Liquids

This model is licensed under the COMSOL Software License Agreement 6.4.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

Dielectric liquids are widely used in various technological applications due to their electrical insulating properties and ability to support high electric fields without breakdown. Unlike aqueous electrolytes, dielectric liquids lack free ions under normal conditions, making them ideal for applications such as electrophoretic displays, EHD pumps, electronic cooling, and high-voltage equipment. Their relatively low permittivity and conductivity, arising from impurity dissociation or ion injection under strong electric fields, allow controlled manipulation of charge carriers without the drawbacks of electrolysis and electrode degradation seen in conductive fluids.

Modeling electrohydrodynamic (EHD) flow in dielectric liquids is crucial for optimizing their performance in these applications. EHD flow in such media is governed not only by surface effects near electrodes but also by bulk space charge generated through ion dissociation. Accurately capturing the interplay between ion transport, charge generation, and fluid motion requires a comprehensive model that considers both electrostatic and hydrodynamic forces. Incorporating the field-enhanced ion dissociation, known as the Onsager effect, is particularly important for describing variations in local conductivity and the resulting Coulomb-driven flows.

The numerical model developed in this work simulates EHD flow of a dielectric liquid around a wire electrode placed between two parallel plates. It solves the coupled Poisson–Nernst–Planck equations for ion transport and the Navier–Stokes equations for fluid dynamics. The Onsager effect is included to account for field-dependent ion generation, enabling more accurate predictions of space charge distribution and flow patterns. Simulation results show strong agreement with experimental data (Ref. 1).

Model Definition

The Electric Discharge interface is used to model the EHD flow in a dielectric liquid. 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

$i = \mathrm{p, n}$

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

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

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

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

where

- p, and n denote 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: $1/(\mathrm{m}^3{\cdot}\mathrm{s})$)
- 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:

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

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

The dissociation process is modeled as:

$$S_{\mathrm{d}} = \beta_{\mathrm{pn}} n_0^2 F(b)$$

where

$$n_0 = \frac{\sigma_0}{e(\mu_{\mathrm{p}} + \mu_{\mathrm{n}})}$$

$$F(b) = \frac{I_1(2b)}{b}$$

$$b = \frac{l_B}{l_O}$$

$$l_B = \frac{e^2}{4\pi\varepsilon k_B T}$$

$$l_O = \sqrt{\frac{e}{4\pi\varepsilon|\mathbf{E}|}}$$

where

- $\mathrm{S_d}$ denotes dissociation rate (generating free ions, SI unit: $1/(\mathrm{m}^3 \cdot \mathrm{s})$)
- β_{pn} is the ion–ion recombination coefficient (SI unit: m^3/s)
- n_0 is the zero-field number density (SI unit: $1/\mathrm{m}^3$)
- σ_0 is the zero-field electric conductivity (SI unit: S/m)
- μ_{p} is the positive ion mobility (SI unit: $\mathrm{m}^2/(\mathrm{V}\cdot\mathrm{s})$)
- μ_{n} is the positive ion mobility (SI unit: $\mathrm{m}^2/(\mathrm{V}\cdot\mathrm{s})$)
- F is the Onsager function
- I_1 is the modified Bessel function of the first kind and order 1
- l_{B} is the Bjerrum length (SI unit: m)
- l_{O} is the Onsager length (SI unit: m)

Results and Discussion

Figure 1 shows the profiles of positive and negative ions density close to the wire electrode. It is noted that the dissociation layer is as thin as 5 μm. Therefore, an extremely fine mesh is applied in the region close to the wire electrode. Figure 2 plots the flow field and the streamline of the flow with the applied voltage V_0 = 1 kV. Figure 3 shows the vertical component of the flow velocity along a horizontal line at y = 0.75 mm, with V_0 = 1, 1.5, and 2 kV.

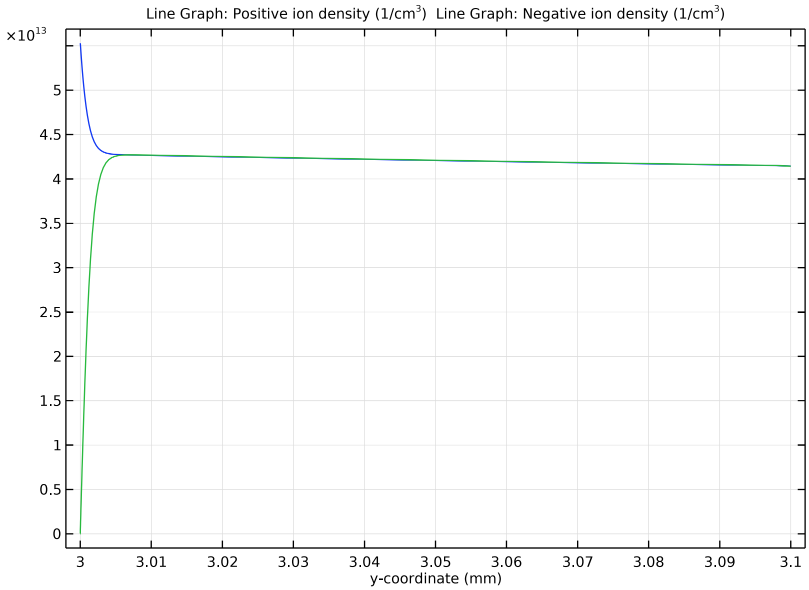


Figure 1: The profiles of positive and negative ions close to the wire electrode.

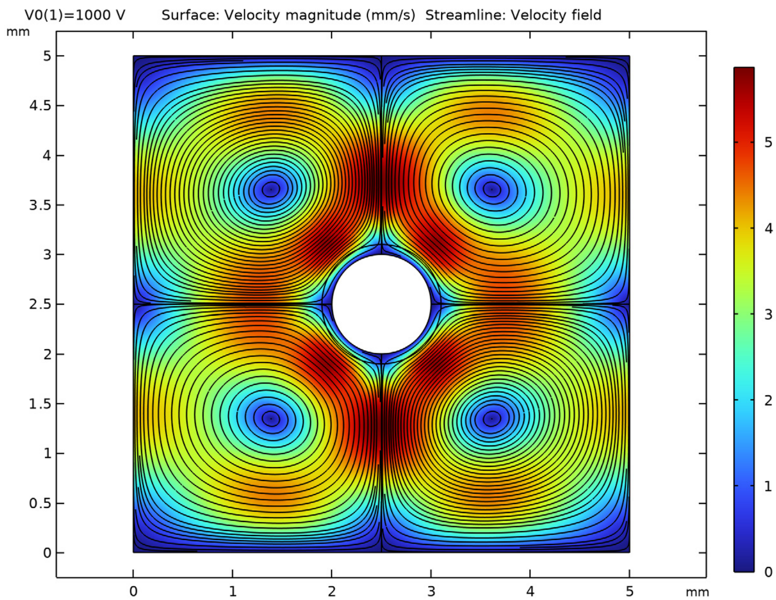


Figure 2: The velocity magnitude and streamlines of the flow field.

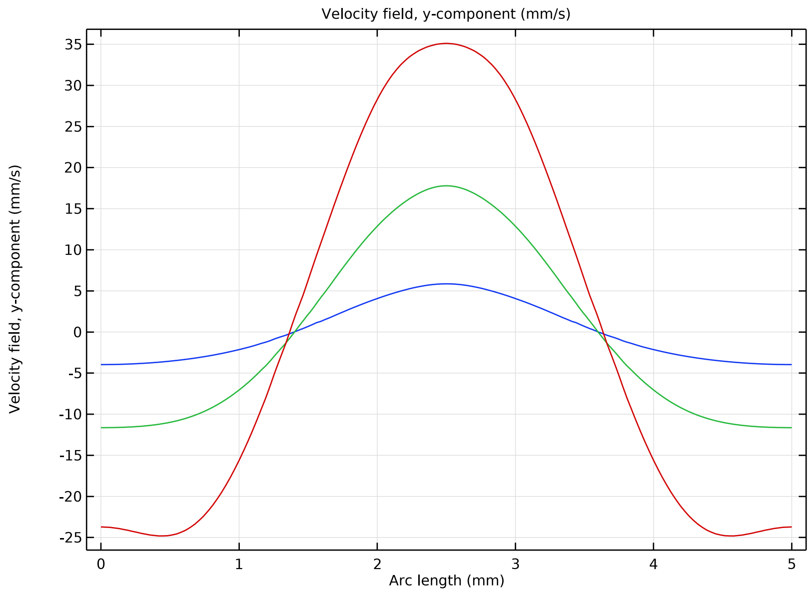


Figure 3: The distribution of space charge density and electric field at t = 90 ns.

Reference

1. D.V. Fernandes, D.S. Cho, and Y.K. Suh, "Electrohydrodynamic flow of dielectric liquid around a wire electrode—effect of truncation of Onsager function," *IEEE Trans. Dielectr. Electr. Insul.*, vol. 21, no. 1, pp. 194–200, 2014.

Application Library path: `Electric_Discharge_Module/Liquid_Dielectrics/ehd_flow_dielectric_liquid`

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 In the **Select Physics** tree, select **Fluid Flow** > **Single-Phase Flow** > **Laminar Flow (spf)**.
5 Click **Add**.
6 Click **Study**.
7 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces** > **Electric Discharge** > **Stationary with Initialization**.
8 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**.

Square 1 (sq1)

1 In the **Geometry** toolbar, click **Square**.
2 In the **Settings** window for **Square**, locate the **Size** section.
3 In the **Side length** text field, type 5.
4 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	2.5

5 Click **Build All Objects**.
6 Select the **Layers to the left** checkbox.

Circle 1 (c1)

1 In the **Geometry** toolbar, click **Circle**.
2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
3 In the **Radius** text field, type 0.5+0.1.
4 Locate the **Position** section. In the **x** text field, type 2.5.
5 In the **y** text field, type 2.5.

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

Layer name	Thickness (mm)
Layer 1	0.1

Union 1 (uni1)

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

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

Delete Entities 1 (del1)

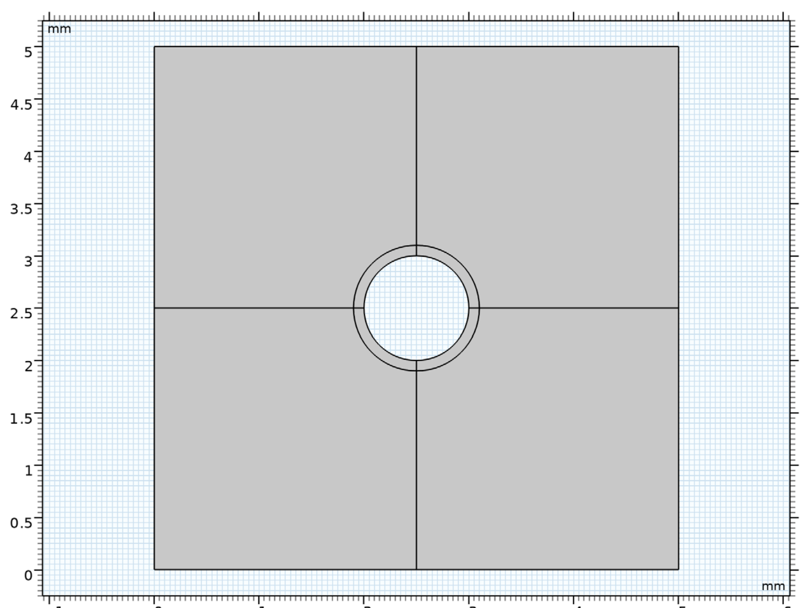
1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.

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 **uni1**, select Domains 5, 6, 9, and 10 only.

5 Click **Build All Objects**.



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
eps_r	2	2	Relative permittivity
sigma0	3E-8[S/m]	3E-8 S/m	Zero-field electric conductivity
rho_0	750[kg/m^3]	750 kg/m³	Liquid density
eta	1.34e-3[Pa*s]	0.00134 Pa·s	Liquid viscosity
mu_i	2.8E-9[m^2/(V*s)]	2.8E-9 m²/(V·s)	Ions mobility
alpha	2*mu_i*e_const/(epsilon0_const*eps_r)	5.0666E-17 m³/s	Recombination rate
V0	1000[V]	1000 V	Applied voltage
n0	sigma0/2/mu_i/e_const	3.3437E19 1/m³	Zero-field ions number density

ELECTRIC DISCHARGE (EDIS)

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

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

3 Clear the **Gas** checkbox.

4 Select the **Liquid** checkbox.

5 In the d_z text field, type 5[mm].

Liquid 1

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

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

3 From the **Charge carriers** list, choose **Positive and negative ions**.

4 Locate the **Transport Properties** section. Find the **Transport mechanisms** subsection. Select the **Convection** checkbox.

5 Select the **Diffusion** checkbox.

6 Find the **Convection** subsection. From the **u** list, choose **Velocity field (spf)**.

Electrode 1

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

2 Select Boundaries 19, 20, 22, and 23 only.

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

4 From the **Boundary condition for negative 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 Select Boundaries 2, 5, 8, and 12 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 positive ions** list, choose **Number density**.

Liquid 1

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

Dissociation 1

In the **Physics** toolbar, click **Attributes** and choose **Dissociation**.

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_p text field, type `n0`.

4 In the n_n text field, type `n0`.

MATERIALS

Dielectric Liquid

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Dielectric Liquid` in the **Label** text field.

3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	eps_r	1	Basic
Positive ion mobility	mu_p_iso ; mu_pii = mu_p_iso, mu_pij = 0	mu_i	$m^2/(V\cdot s)$	Charge transport in liquids
Negative ion mobility	mu_n_iso ; mu_nii = mu_n_iso, mu_nij = 0	mu_i	$m^2/(V\cdot s)$	Charge transport in liquids
Ion-ion recombination coefficient	beta_pn	alpha	m^3/s	Charge transport in liquids
Electric conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	sigma0	S/m	Basic
Density	rho	rho_0	kg/m^3	Basic
Dynamic viscosity	mu	eta	Pa·s	Basic

LAMINAR FLOW (SPF)

Volume Force 1

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

2 In the **Settings** window for **Volume Force**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **All domains**.

4 Locate the **Volume Force** section. From the **F** list, choose **Electrohydrodynamic force (edis/liquid1)**.

Pressure Point Constraint 1

1 In the **Physics** toolbar, click **Points** and choose **Pressure Point Constraint**.

2 Select Point 16 only.

Symmetry 1

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

2 Select Boundaries 1, 3, 15, and 16 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**.

Corner Refinement 1

In the **Model Builder** window, under **Component 1 (comp1)** > **Mesh 1** right-click **Corner Refinement 1** and choose **Delete**.

Mapped 1

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

2 Drag and drop below **Size 1**.

Size 1

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

Mapped 1

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

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

3 Select Domain 7 only.

Distribution 1

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

2 Select Boundary 23 only.

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

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

Distribution 2

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

2 Select Boundaries 10 and 13 only.

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

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

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

6 In the **Element ratio** text field, type 200.

7 Select the **Reverse direction** checkbox.

Copy Domain 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Copying Operations** > **Copy Domain**.

2 Select Domain 7 only.

3 In the **Settings** window for **Copy Domain**, locate the **Destination Domains** section.

4 Click to select the **Activate Selection** toggle button.

5 Select Domains 3, 4, and 6 only.

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 **Domain**.

4 Select Domain 8 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 `0.04`.

Boundary Layers 1

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

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

3 Click **Clear Selection**.

4 Select Domain 8 only.

Boundary Layer Properties 1

1 In the **Model Builder** window, expand the **Boundary Layers 1** node, then click **Boundary Layer Properties 1**.

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

3 Click **Clear Selection**.

4 Select Boundary 12 only.

5 Locate the **Layers** section. In the **Number of layers** text field, type `30`.

6 From the **Thickness specification** list, choose **First layer**.

7 In the **Thickness** text field, type `1e-5`.

Copy Domain 2

In the **Model Builder** window, right-click **Mesh 1** and choose **Copying Operations** > **Copy Domain**.

Free Triangular 1

Drag and drop below **Copy Domain 1**.

Copy Domain 2

1 In the **Model Builder** window, click **Copy Domain 2**.

2 In the **Settings** window for **Copy Domain**, locate the **Source Domains** section.

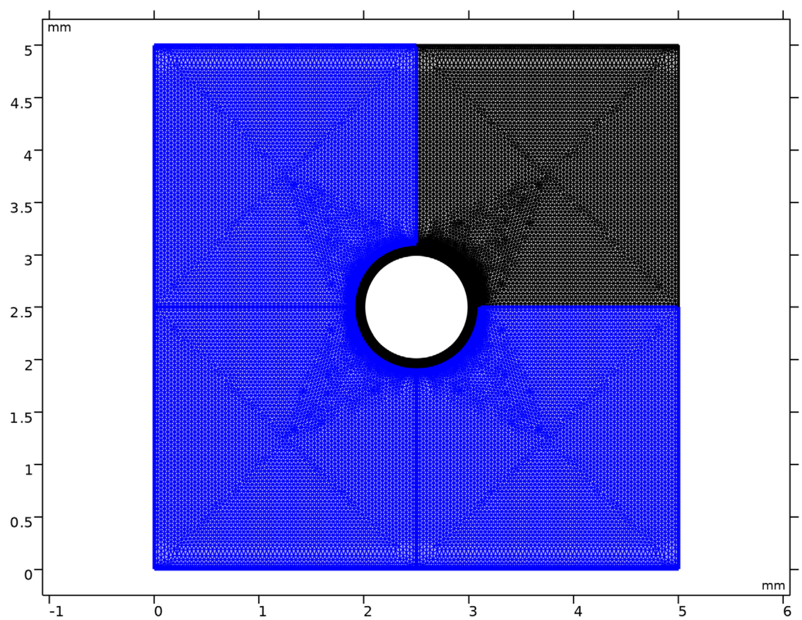
3 Click to select the **Activate Selection** toggle button.

4 Select Domain 8 only.

5 Locate the **Destination Domains** section. Click to select the **Activate Selection** toggle button.

6 Select Domains 1, 2, and 5 only.

7 Click **Build All**.



Electric Discharge Only

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

2 In the **Settings** window for **Study**, type `Electric Discharge Only` in the **Label** text field.

Step 2: Stationary

1 In the **Model Builder** window, under **Electric Discharge Only** click **Step 2: Stationary**.

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

3 In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Laminar Flow (spf)**.

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

Results

Dissociation Layer

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

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

Line Graph 1

1 Right-click **Dissociation Layer** and choose **Line Graph**.

2 Select Boundary 10 only.

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

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

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

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

7 In the **Expression** text field, type `y`.

Line Graph 2

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

Line Graph 1

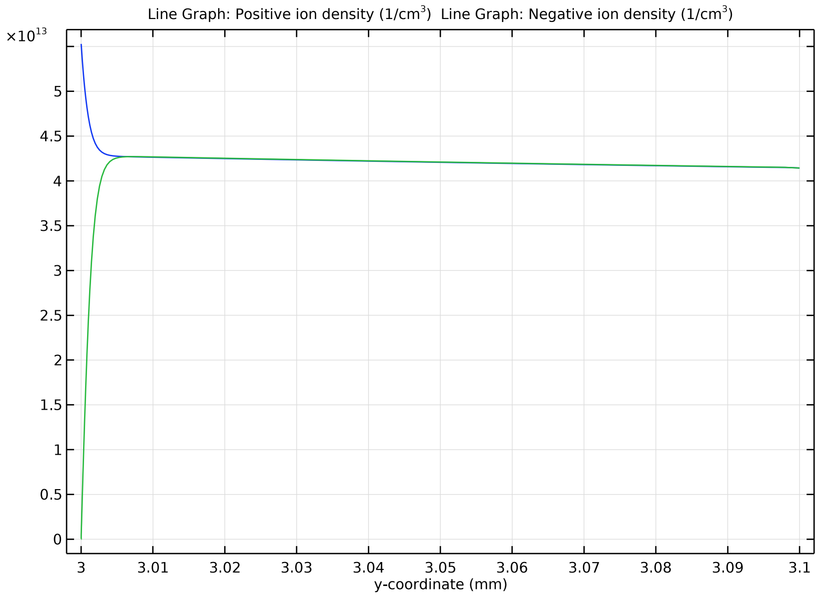
Locate the **y-Axis Data** section. In the **Expression** text field, type `edis.n_p`.

Dissociation Layer

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

2 In the **Model Builder** window, click **Dissociation Layer**.

3 In the **Dissociation Layer** toolbar, click **Plot**.



ADD STUDY

1 In the **Home** toolbar, click **Windows** and choose **Add Study**.

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

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

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

FULLY COUPLED

In the **Settings** window for **Study**, type `Fully Coupled` in the **Label** text field.

Solution 3 (sol3)

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

Step 1: Stationary

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

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

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

4 In the **Relative tolerance** text field, type `1.0E-3`.

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 **Electric Discharge Only, Stationary**.

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

9 Click + **Add**.

10 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
V0 (Applied voltage)	1000 1500 2000	V

Solution 3 (sol3)

1 In the **Model Builder** window, expand the **Fully Coupled** > **Solver Configurations** > **Solution 3 (sol3)** > **Stationary Solver 1** node.

2 Right-click **Fully Coupled** > **Solver Configurations** > **Solution 3 (sol3)** > **Stationary Solver 1** and choose **Segregated**.

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

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

5 In the **Model Builder** window, expand the **Fully Coupled** > **Solver Configurations** > **Solution 3 (sol3)** > **Stationary Solver 1** > **Segregated 1** node, then click **Segregated Step**.

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

7 In the **Variables** list, choose **Pressure (comp1.p)** and **Velocity Field (comp1.u)**.

8 Under **Variables**, click **Delete**.

9 In the **Model Builder** window, under **Fully Coupled** > **Solver Configurations** > **Solution 3 (sol3)** > **Stationary Solver 1** right-click **Segregated 1** and choose **Segregated Step**.

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

11 Under **Variables**, click + **Add**.

12 In the **Add** dialog, in the **Variables** list, choose **Pressure (comp1.p)** and **Velocity Field (comp1.u)**.

13 Click **OK**.

14 In the **Model Builder** window, under **Fully Coupled** > **Solver Configurations** > **Solution 3 (sol3)** > **Stationary Solver 1** click **Direct (Merged)**.

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

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

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

RESULTS

Velocity (spf)

1 In the **Model Builder** window, under **Results** click **Velocity (spf)**.

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

3 From the **Parameter value (V0 (V))** list, choose **1000**.

4 In the **Velocity (spf)** toolbar, click **Plot**.

Streamline 1

1 Right-click **Velocity (spf)** and choose **Streamline**.

2 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.

3 From the **Positioning** list, choose **Uniform density**.

4 In the **Density level** text field, type 10.

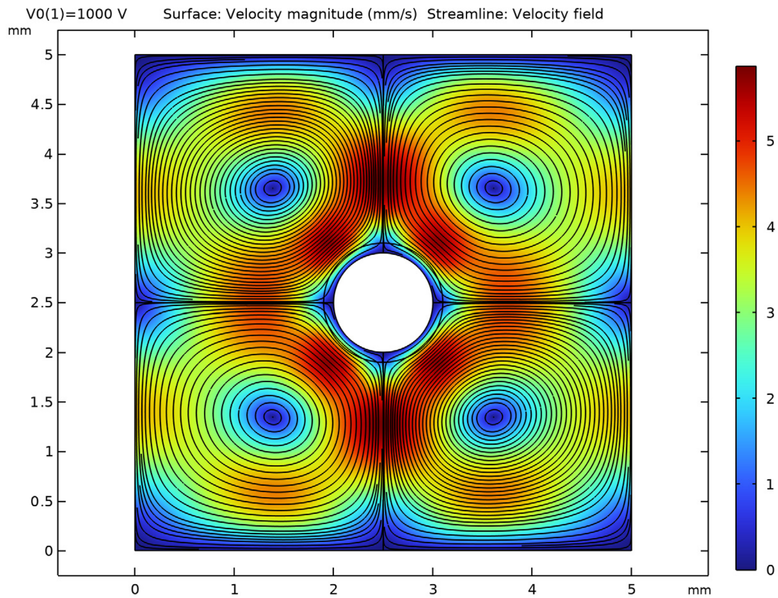
Surface

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

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

3 From the **Unit** list, choose **mm/s**.

4 In the **Velocity (spf)** toolbar, click **Plot**.



1D Plot Group 6

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

Line Graph 1

1 Right-click **1D Plot Group 6** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, click **Define Cut Line**.

3 Locate the **Data** section. Click **Go to Source**.

Cut Line 2D 1

1 In the **Model Builder** window, under **Results** > **Datasets** click **Cut Line 2D 1**.

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

3 In row **Point 1**, set **x** to `0`.

4 In row **Point 1**, set **y** to `3.75`.

5 In row **Point 2**, set **x** to `5`.

6 In row **Point 2**, set **y** to `3.75`.

Line Graph 1

1 In the **Model Builder** window, under **Results** > **1D Plot Group 6** 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 `v`.

4 From the **Unit** list, choose **mm/s**.

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

Cut Line 2D 1

1 In the **Model Builder** window, under **Results** > **Datasets** click **Cut Line 2D 1**.

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

3 From the **Dataset** list, choose **Fully Coupled/Solution 3 (sol3)**.

Velocity Profile

1 In the **Model Builder** window, under **Results** click **1D Plot Group 6**.

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

3 In the **Velocity Profile** toolbar, click **Plot**.