Created in COMSOL Multiphysics 5.6

Electrostatic Precipitator

Introduction

In this tutorial are modeled several aspects of an electrostatic precipitator. First, a simplified model for corona discharges coupled with the Laminar Flow interface is used to compute the fluid velocity, electric field, and space charge density, which are necessary to compute the particle charging and relevant forces acting on particles. After, the Particle Tracing for Fluid Flow interface is used to compute the particle collection efficiency as a function of the particle radius.

Model Definition

Figure 1 shows the simulation domain, which consists of a cross section of a rectangular electrostatic precipitator in a wire-to-plane configuration. The DC high voltage source is applied to the inner electrodes and the walls are grounded. The particles are released from the inlet at the left and are transported with the fluid. Particles accumulate charge along their trajectory and become susceptible to electric forces that deflect their trajectories in the direction of the collecting plates. The operation conditions of the electrostatic precipitator are presented in Table 1.

TABLE 1: OPERATION CONDITIONS OF THE ELECTROSTATIC PRECIPITATOR.

Inner electrode radius	0.5 mm
Inner electrode separation	15 cm
Inner electrode distance to wall	5 cm
Applied voltage V_0	20 kV
Inlet fluid velocity	1m/s
Temperature	293.15 K
Pressure	1 atm

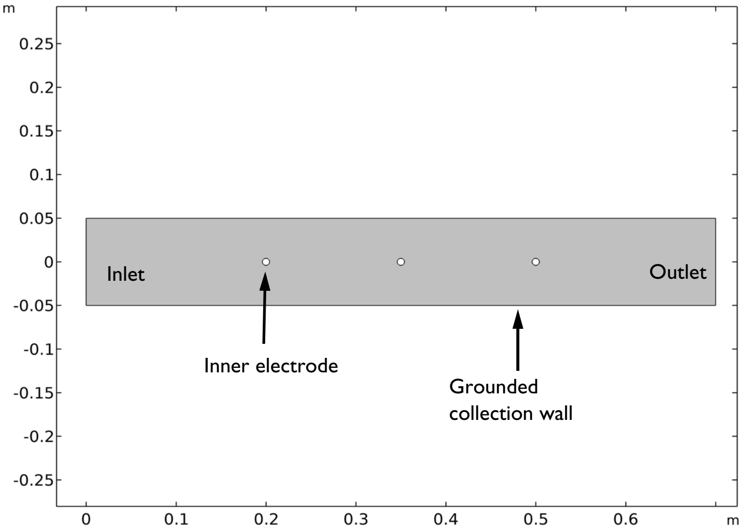

Figure 1: Simulation domain of the electrostatic precipitator (the inner electrodes are not to scale).

Corona Model

The simplified corona model is based on the conservation of current transported by the charged carriers. It should be emphasized that the model is not self-consistent in the sense that the both potential and the electric field need to be given at the corona electrode. In other words, the electric field necessary to sustain the discharge is not obtained from first principles: electron and ion transport, electrons gaining energy from the electric field, and electrons losing energy in collisions with the background gas.

DOMAIN EQUATIONS

The simplified model for the corona solves the transport of a charge carrier using the charge conservation equation coupled with Poisson's equation. The transport of the charge carriers includes drift in the electric field and convection. Without source terms the domain equations are

$$\nabla \bullet \mathbf{J} = 0 \tag{1}$$

$$\mathbf{J} = z_q \mu \rho_q \mathbf{E} + \rho \mathbf{u} \tag{2}$$

$$\varepsilon_0 \nabla^2 V = -\rho_q \tag{3}$$

where $\mathbf{J}$ (SI unit: A/m^2)is the current density, z_q is the charge number, μ (SI unit: m^2/V·s) is the mobility, ρ_q (SI unit: C/m^3) is the space charge number density, $\mathbf{E}$ is the electric field, $\mathbf{u}$ is the fluid velocity (SI unit: m/s), V is electric potential, and ε_0 is the vacuum permittivity. This set of equations can be manipulated to obtain the following transport equation

$$\mu\left(\frac{\rho_q^{\ 2}}{\varepsilon_0} - \nabla V \bullet \nabla \rho_q\right) + \nabla \rho_q \bullet \mathbf{u} = 0 \tag{4}$$

where it is assumed that the mobility is constant. It is interesting to note that the domain equations do not contain any information related to plasma creation and maintenance. All plasma physics is condensed in the boundary conditions for the inner electrodes.

BOUNDARY CONDITIONS

The normal component of the electric field at the corona electrode is used as a boundary condition for Poisson's equation

$$\mathbf{n} \bullet \mathbf{E} = E_0 . \tag{5}$$

The other boundary conditions for Poisson's equation are $V = 0$ at the collection plates and zero charge at the inlet and outlet. The boundary condition for Equation 4 involves in finding the space charge density ρ_q at the corona electrode, using a Lagrange multiplier, so that the imposed potential V_0 is verified

$$V - V_0 = 0 . \tag{6}$$

In this model both potential and electric field are imposed at the corona electrode. To obtain predictive physical results the value of the electric field at the wire needs to be close enough to the real one. Here, it is used Peek's law

$$E_0 = 3\times 10^6 \delta\left(1 + \frac{0.03}{\sqrt{\delta r_i}}\right) \tag{7}$$

where E_0 (SI unit: V/m) is the breakdown electric field, δ is the gas number density normalized to the gas density at 760 torr and 293.15 K, r_i is the radius of the corona electrode.

Laminar Flow Model

The Laminar Flow interface is used to solve for the fluid velocity and pressure

$$\begin{aligned} \rho(\mathbf{u}\cdot\nabla)\mathbf{u} &= \nabla\cdot[-p\mathbf{I}+\mu(\nabla\mathbf{u}+(\nabla\mathbf{u})^{\mathrm{T}})]+\mathbf{F}_{EHD} \\ \nabla\cdot\mathbf{u} &= 0 \end{aligned} \tag{8}$$

where μ is the dynamic viscosity (SI unit: kg/(m·s)), ρ is the fluid density (SI unit: kg/m^3), p if the pressure (SI unit: Pa), and $\mathbf{F}_{EHD}$ is the electrohydrodynamic force define as

$$\mathbf{F}_{EHD} = \rho_q\mathbf{E} \tag{9}$$

Particle Tracing Model

The particle positions are computed by solving second-order equations of motion for the particle position vector components, following Newton's second law,

$$\begin{aligned} \frac{d\mathbf{q}}{dt} &= \mathbf{v} \\ \frac{d}{dt}(m_p\mathbf{v}) &= \mathbf{F}_t \end{aligned} \tag{10}$$

where $\mathbf{q}$ is the particle position (SI unit: m), $\mathbf{v}$ is the particle velocity (SI unit: m/s), m_p is the particle mass (SI unit: kg), and $\mathbf{F}_t$ is the total force (SI unit: N) acting on the particle. In this example the forces acting on the particles are the drag force and the electric force. Rarefaction effects need to be included in the drag force because the particle radius become very small. Here, the drag force $\mathbf{F}_D$ (SI unit: N) is described with the Cunningham-Millikan-Davis model

$$\mathbf{F}_D = \frac{1}{\tau_p S}m_p(\mathbf{u}-\mathbf{v}) \tag{11}$$

where τ_p is the particle velocity response time (SI unit: s) define as

$$\tau_p = \frac{4\rho_p d_p^2}{3\mu C_D \mathrm{Re}_r} \tag{12}$$

where ρ_p is the density of the particles (SI unit: kg/m^3), d_p is the particle diameter (SI unit: m), C_{D} is the drag coefficient, and Re_{r} is the relative Reynolds number given by the expression

$$\mathrm{Re}_r = \frac{\rho\|\mathbf{u}-\mathbf{v}\|d_p}{\mu}, \tag{13}$$

and S is the drag correction coefficient defined as

$$S = 1 + \mathrm{Kn}\left(C_1 + C_2 \exp\left(-\frac{C_3}{\mathrm{Kn}}\right)\right) \tag{14}$$

where are dimensionless coefficients.

The electric force $\mathbf{F}_e$ (SI unit: N) acting on the particles is defined as

$$\mathbf{F}_e = eZ\mathbf{E} \tag{15}$$

where e (SI unit: C) is the elementary charge, and Z is the accumulated charge number on each particle.

The charge accumulated on the particles is computed with the Lawless model

$$\tau_c \frac{dZ}{dt} = \begin{cases} R_f + f_a & (|v_e| \le |v_s|) \\ R_d f_a & (|v_e| > |v_s|) \end{cases} \tag{16}$$

where τ_c is the characteristic charging time

$$\tau_c = \frac{e^2}{4\pi\rho_q\mu \mathrm{k}_B T_i} \tag{17}$$

where k_B is the Boltzmann constant, and T_i is the ion temperature. R_f and R_d are the dimensionless charging rates due to field and diffusion transport, respectively, defined as

$$R_f = \frac{v_s}{4\varepsilon_0}\left(1 - \frac{v_e}{v_s}\right)^2 \tag{18}$$

$$R_d = \frac{v_e - v_s}{\exp(v_e - v_s) - 1} \tag{19}$$

where

$$v_e = \frac{Ze^2}{4\pi\varepsilon_0 r_p \mathrm{k}_B T_i} \tag{20}$$

$$v_s = 3w_e\frac{\varepsilon_{r,p}}{\varepsilon_{r,p+2}} \tag{21}$$

$$w_e = \frac{er_p|E|}{\mathrm{k}_B T_i}, \tag{22}$$

where $\varepsilon_{r,p}$ is the relative permittivity of the particles. f_a is a function used to join the diffusion and field charging rates defines as

$$f_a = \begin{cases} \dfrac{1}{(w_e + 0.475)^{0.575}} & (w_e \geq 0.525) \\ 1 & (w_e < 0.525) \end{cases} . \tag{23}$$

Results and Discussion

Figure 2, Figure 3, and Figure 4 show the fluid velocity, the electrostatic potential, and the space charge density obtained with the corona model coupled with the Laminar Flow interface. It is with this information that the particles trajectories are computed.

In the present, the corona and the fluid model are fully coupled. However, model results show that the fluid velocity is practically not influence by the electrohydrodynamic force, and the drift velocity is always much larger than the fluid velocity in the regions of interest. In future works in similar operation conditions it could be of interest to uncouple the two models since the computation times become considerable shorter.

The space charge density is more intense near the inner electrodes, as expected, where a corona discharge is luminous. It is in the regions near the inner electrodes that particles accumulate charge at a faster rate due to the combined effect of large space charge densities and intense electric fields.

Figure 5, Figure 6, Figure 7, and Figure 8 show particle trajectories and charge accumulated in the particle expressed in color for particles of several radius. Particles are released at the left and are transported in the fluid flow toward the right outlet. The particles become progressively charged along their trajectory resulting in electric forces that deflects their trajectory in the wall direction. The particle radius influences the balance of the drag and electric force felt by particles and consequently influences the particle trajectory and the electrostatic precipitator collection efficiency.

Figure 9 and Figure 10 present the particle collection efficiency and the average particle charge at the last time step as a function of the particle radius. The collection efficiency is larger at the extremes of the particle dimensions. Larger particles are collected more efficiently because they attain greater electric charge, while smaller particles are collected more efficiently because are subjected to less drag force. Between this two extremes, the drag force influences the most the particle trajectories resulting in almost straight lines parallel to the collection plates (see Figure 6) that result in low collection efficiency.

At the small particle branch partial charging becomes important. A model that correctly describes partial charging should capture the random nature in which a particle can have a charge of 1 or 0. In the present model, particles can be charged with a number between 0 and 1, which might result in inaccurate collection efficiency results for small particles.

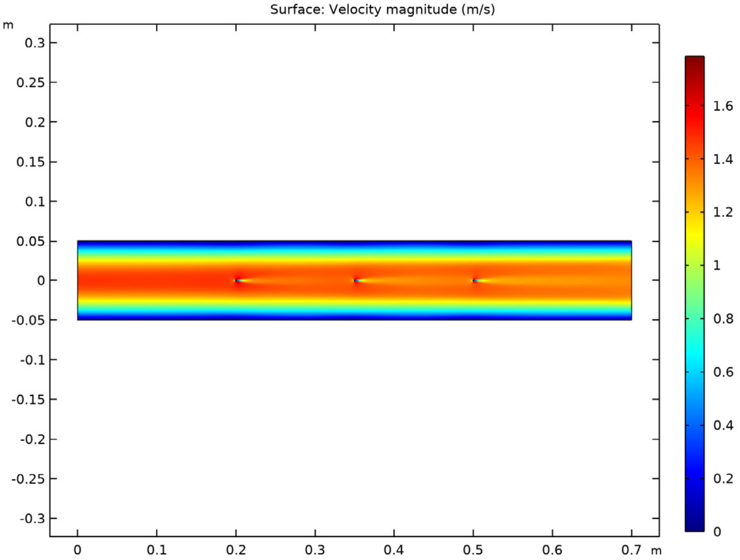

Figure 2: Velocity magnitude of the flow in the electrostatic precipitator.

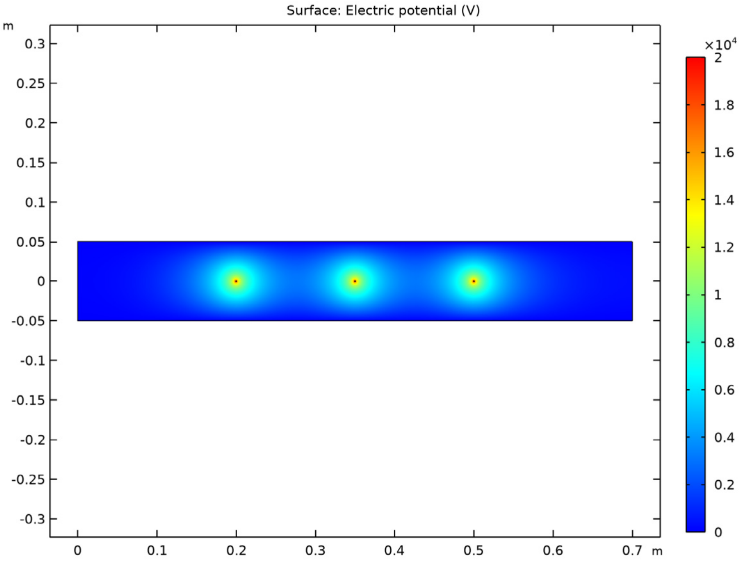

Figure 3: Electric potential.

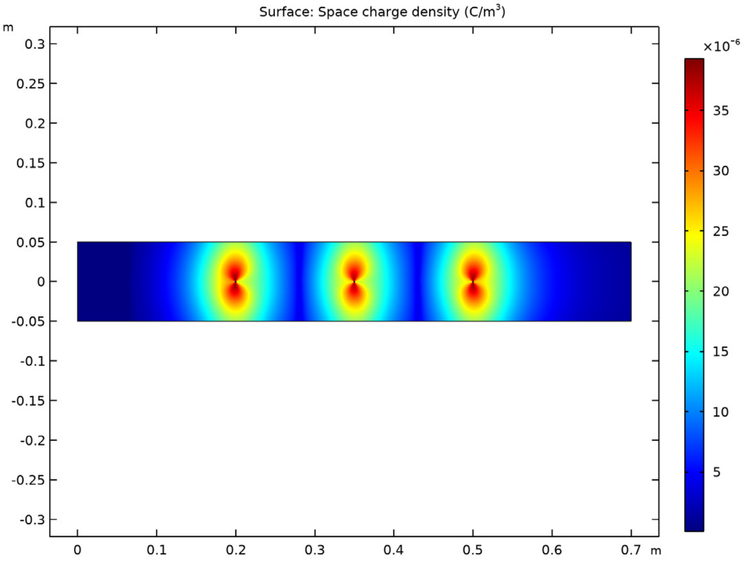

Figure 4: Space charge density.

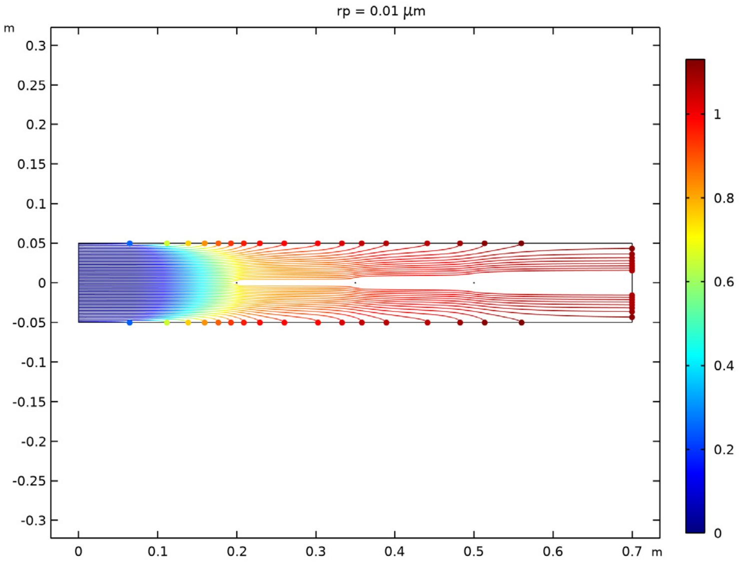

Figure 5: Particle trajectories with the charge number along the trajectory expressed in color for particles with a radius of 0.01 µ*m.*

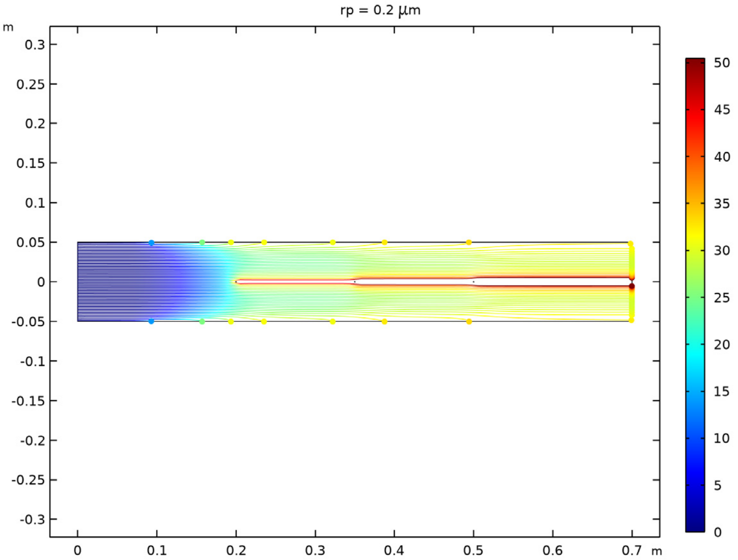

Figure 6: Same as in Figure 5 *for a particle radius of 0.2* µ*m.*

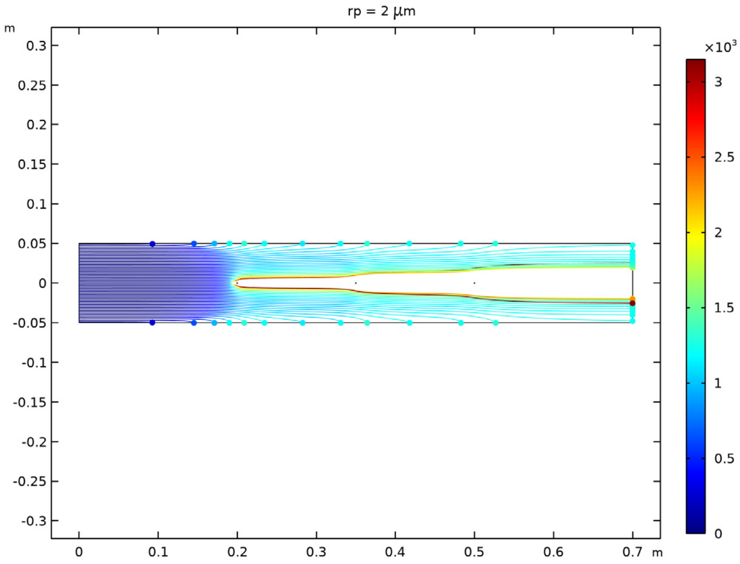

Figure 7: same as in Figure 5 *for a particle radius of 2 µm.*

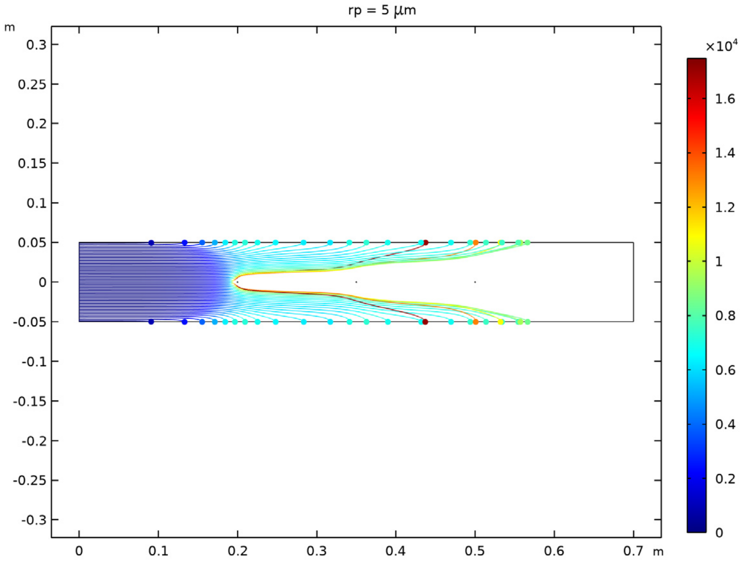

Figure 8: same as in Figure 5 *for a particle radius of 5 µm.*

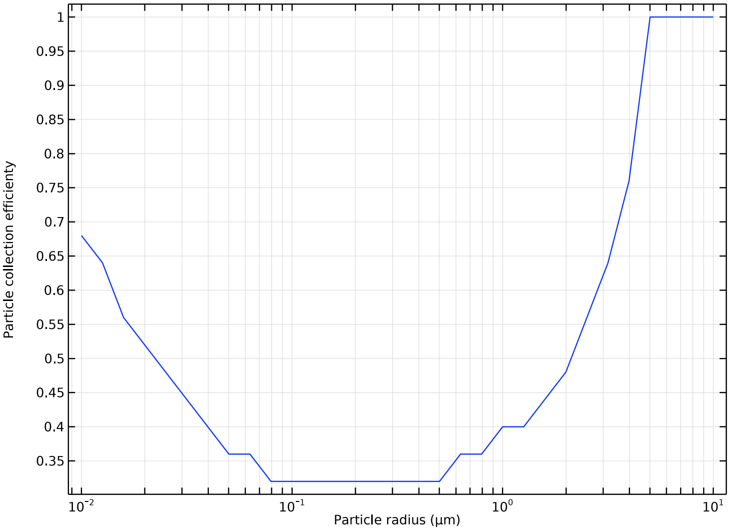

Figure 9: Particle collection efficiency as a function of the particle radius.

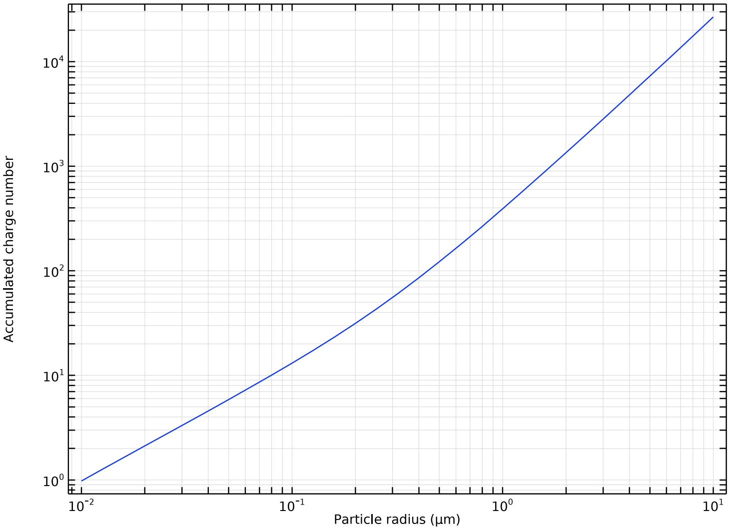

Figure 10: Average charge accumulated per particle at the last time step as a function of the particle radius.

Application Library path: `Plasma_Module/Corona_Discharges/electrostatic_precipitator`

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.

3. LXCAT, see http://fr.lxcat.net for Phelps database (2016).

Modeling Instructions

The following instructions show how to create a 2D model of an electrostatic precipitator and how to obtain the particle collection efficiency as a function of the particle radius. Two studies are needed:

- A **Stationary** study that couples the **Laminar Flow (spf)**, **Electrostatics (es)** and **Charge Transport (ct)** interfaces.
- A **Time Dependent** study that solves for the particle trajectories using the **Particle Tracing for Fluid Flow (fpt)** interface to obtain the particle collection efficiency.

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1. In the **Model Wizard** window, Select the **Laminar Flow (spf)** interface and **Corona Discharge** to compute the fluid velocity, the electric field, and the space charge density that are necessary for the **Particle Tracing for Fluid Flow (fpt)** interface to compute the particle charging and trajectories.
2. click **2D**.
3. In the **Select Physics** tree, select **Fluid Flow>Single-Phase Flow>Laminar Flow (spf)**.
4. Click **Add**.

5. In the **Select Physics** tree, select **Plasma>Corona Discharge**.
6. Click **Add**.
7. Click **Study**.
8. In the **Select Study** tree, select **General Studies>Stationary**.
9. Click **Done**.

Add some parameters for the precipitator dimensions, the corona electrode configuration, and the ion reduced mobility.

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
`H`	`0.1[m]`	0.1 m	`Height`
`rin`	`0.5[mm]`	5E-4 m	`Electrode radius`
`W`	`0.7[m]`	0.7 m	`Width`
`sp`	`15[cm]`	0.15 m	`Electrode separation`
`muN`	`3e21[1/(V*m*s)]`	3E21 1/(V·m·s)	`Reduced ion mobility`

GEOMETRY 1

Rectangle 1 (r1)

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

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 `rin`.
4. Locate the **Position** section. In the **x** text field, type `W/2-sp`.

Array 1 (arr1)

1. In the **Geometry** toolbar, click **Transforms** and choose **Array**.
2. Select the object **c1** only.
3. In the **Settings** window for **Array**, locate the **Size** section.
4. In the **x size** text field, type `3`.
5. Locate the **Displacement** section. In the **x** text field, type `sp`.

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. Find the **Objects to subtract** subsection. Select the **Activate Selection** toggle button.
5. Select the objects **arr1(1,1)**, **arr1(2,1)**, and **arr1(3,1)** only.
6. Click **Build All Objects**.

ADD MATERIAL

1. In the **Home** toolbar, click **Add Material** to open the **Add Material** window.
2. Go to the **Add Material** window.
3. In the tree, select **Built-in>Air**.
4. Click **Add to Component** in the window toolbar.
5. In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

Add **inlet**, **outlet** and **Volume Force** features. The **Volume Force** feature is used to introduce the electrohydrodynamic force computed in the **Charge Transport (ct)** interface into the **Laminar Flow (spf)** interface.

LAMINAR FLOW (SPF)

Inlet 1

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Laminar Flow (spf)** and choose **Inlet**.
2. Select Boundary 1 only.
3. In the **Settings** window for **Inlet**, locate the **Boundary Condition** section.
4. From the list, choose **Fully developed flow**.
5. Locate the **Fully Developed Flow** section. In the U_{av} text field, type `1[m/s]`.

Outlet 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Outlet**.
2. Select Boundary 4 only.

Volume Force 1

1. In the **Physics** toolbar, click **Domains** and choose **Volume Force**.
2. Select Domain 1 only.
3. In the **Settings** window for **Volume Force**, locate the **Volume Force** section.
4. Specify the **F** vector as

`ct.Fehdx`	x
`ct.Fehdy`	y

In the **Electrostatics (es)** interface only the ground needs to be defined. The applied voltage is defined in the **Electrode** feature in the **Multiphysics** node.

ELECTROSTATICS (ES)

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

Ground 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Ground**.
2. Select Boundaries 2 and 3 only.

In the **Charge Transport (ct)** feature the electric potential for the charged species migration comes automatically from the **Electrostatics (es)** interface. The ion mobility and the charge number of the ion need to be set.

Add a second coupling mechanism between the **Charge Transport (ct)** and the **Laminar Flow (spf)** interfaces by adding convection to the transport mechanisms.

CHARGE TRANSPORT (CT)

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Charge Transport (ct)**.
2. In the **Settings** window for **Charge Transport**, locate the **Transport Mechanisms** section.
3. Select the **Convection** check box.

Transport Properties 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Charge Transport (ct)** click **Transport Properties 1**.
2. In the **Settings** window for **Transport Properties**, locate the **Migration in Electric Field** section.

3 In the $\mu_i N$ text field, type `muN`.

4 Locate the **Convection** section. From the **u** list, choose **Velocity field (spf)**.

Create an explicit selection of the corona electrodes to be used below.

DEFINITIONS

Electrodes

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

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

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

4 Select the **Group by continuous tangent** check box.

5 Select Boundaries 5–16 only.

The voltage and the electric field at the corona electrodes are defined in the **Electrode** feature.

MULTIPHYSICS

Electrode 1 (el1)

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Boundary>Electrode**.

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

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

4 Locate the **Electric Potential** section. In the V_0 text field, type `20[kV]`.

5 Locate the **Electric Field** section. In the r_c text field, type `rin`.

MESH 1

Size

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Edit Physics-Induced Sequence**.

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

3 From the **Predefined** list, choose **Normal**.

Edge 1

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

2 Right-click **Edge 1** and choose **Move Up**.

3 Right-click **Edge 1** and choose **Move Up**.

4 Right-click **Edge 1** and choose **Move Up**.

5 In the **Settings** window for **Edge**, locate the **Boundary Selection** section.

6 From the **Selection** list, choose **Electrodes**.

Distribution 1

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

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

3 In the **Number of elements** text field, type `50`.

4 Click **Build All**.

CORONA AND LAMINAR FLOW

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

2 In the **Settings** window for **Study**, type `Corona and laminar flow` in the **Label** text field.

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

Add a **Particle Tracing for Fluid Flow** interface and a **Time Dependent** study to compute the particles trajectory.

ADD PHYSICS

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

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

3 In the tree, select **Fluid Flow>Particle Tracing>Particle Tracing for Fluid Flow (fpt)**.

4 Click **Add to Component 1** in the window toolbar.

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

PARTICLE TRACING FOR FLUID FLOW (FPT)

1 In the **Settings** window for **Particle Tracing for Fluid Flow**, locate the **Additional Variables** section.

2 From the **Particle size distribution** list, choose **Specify particle diameter**.

The smallest particle radius is of 10 nm. So, it is necessary to introduce rarefaction effects that significantly reduce the drag force on the smallest particles.

3 Locate the **Particle Release and Propagation** section. Select the **Include rarefaction effects** check box.

4 Locate the **Additional Variables** section. Select the **Store particle status data** check box.

Particles are released on the left boundary and are collected at the walls. A particle counter is added at the outlet on the right in order to compute the particle collection efficiency.

Outlet 1

1. Right-click **Component 1 (comp1)>Particle Tracing for Fluid Flow (fpt)** and choose **Outlet**.
2. Select Boundary 4 only.

Particle Counter 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Particle Counter**.
2. Select Boundary 4 only.

Add drag and electric forces to the particles.

Drag Force 1

1. In the **Physics** toolbar, click **Domains** and choose **Drag Force**.
2. Select Domain 1 only.
3. In the **Settings** window for **Drag Force**, locate the **Drag Force** section.
4. From the **Drag law** list, choose **Standard drag correlations**.
5. From the $\mathbf{u}$ list, choose **Velocity field (spf)**.

Electric Force 1

1. In the **Physics** toolbar, click **Domains** and choose **Electric Force**.
2. Select Domain 1 only.
3. In the **Settings** window for **Electric Force**, locate the **Electric Force** section.
4. From the **Specify force using** list, choose **Electric potential**.
5. From the V list, choose **Electric potential (es)**.

To model the charge acquired by particles while being transported in the presence of space charge add a **Charge Accumulation** feature.

Charge Accumulation 1

1. In the **Physics** toolbar, click **Domains** and choose **Charge Accumulation**.
2. Select Domain 1 only.
3. In the **Settings** window for **Charge Accumulation**, locate the **Ion Properties** section.
4. From the ρ_q list, choose **Space charge density (ct)**.
5. In the $\mu_i N$ text field, type `muN`.
6. Locate the **Electric Field** section. From the V list, choose **Electric potential (es)**.

Particle Properties 1

1 In the **Model Builder** window, click **Particle Properties 1**.

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

3 From the ρ_p list, choose **User defined**. Locate the **Charge Number** section. From the **Charge specification** list, choose **Charge Accumulation 1**.

4 Locate the **Additional Material Properties** section. From the $\varepsilon_{r,p}$ list, choose **User defined**. In the associated text field, type `5`.

Release from Grid 1

1 In the **Physics** toolbar, click **Global** and choose **Release from Grid**.

2 In the **Settings** window for **Release from Grid**, locate the **Initial Coordinates** section.

3 Click **Y Range**.

4 In the **Range** dialog box, choose **Number of values** from the **Entry method** list.

5 In the **Start** text field, type `-0.049`.

6 In the **Stop** text field, type `0.049`.

7 In the **Number of values** text field, type `50`.

8 Click **Replace**.

9 In the **Settings** window for **Release from Grid**, locate the **Initial Velocity** section.

10 Specify the $\mathbf{v}_0$ vector as

u	x
v	y

11 Locate the **Initial Particle Diameter** section. From the **Distribution function** list, choose **List of values**.

12 Click **Range**.

13 In the **Range** dialog box, choose **Logarithmic** from the **Entry method** list.

14 In the **Start** text field, type `2e-8`.

15 In the **Stop** text field, type `2e-5`.

16 In the **Steps per decade** text field, type `10`.

17 Click **Replace**.

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 **Preset Studies for Some Physics Interfaces>Time Dependent**.

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: Time Dependent

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

2 In the table, clear the **Solve for** check boxes for **Laminar Flow (spf)**, **Electrostatics (es)**, and **Charge Transport (ct)**.

3 In the table, clear the **Solve for** check boxes for **Space Charge Density Coupling 1 (scdc1)**, **Potential Coupling 1 (pc1)**, and **Electrode 1 (el1)**.

4 Locate the **Study Settings** section. In the **Output times** text field, type `range(0,0.01,1.7)`.

5 Click to expand the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

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

7 From the **Study** list, choose **Corona and laminar flow, Stationary**.

8 In the **Model Builder** window, click **Study 2**.

9 In the **Settings** window for **Study**, type `Particle tracing` in the **Label** text field.

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

Plot the particles trajectories for several particles radius and represent the charge number of the particles as a color expression along the trajectories.

RESULTS

Particle Trajectories rp = 1e-8 m

1 In the **Settings** window for **2D Plot Group**, type `Particle Trajectories rp = 1e-8 m` in the **Label** text field.

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

3 In the **Title** text area, type `rp = 0.01 \mu m`.

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

Particle Trajectories 1

1. In the **Model Builder** window, expand the **Particle Trajectories rp = 1e-8 m** node, then click **Particle Trajectories 1**.
2. In the **Settings** window for **Particle Trajectories**, locate the **Coloring and Style** section.
3. Find the **Line style** subsection. From the **Type** list, choose **Line**.
4. From the **Interpolation** list, choose **Uniform**.
5. In the **Number of interpolated times** text field, type `1000`.

Color Expression 1

1. In the **Model Builder** window, expand the **Particle Trajectories 1** node, then click **Color Expression 1**.
2. In the **Settings** window for **Color Expression**, locate the **Expression** section.
3. In the **Expression** text field, type `fpt.Z`.

Filter 1

1. In the **Model Builder** window, right-click **Particle Trajectories 1** and choose **Filter**.
2. In the **Settings** window for **Filter**, locate the **Particle Selection** section.
3. From the **Particles to include** list, choose **Logical expression**.
4. In the **Logical expression for inclusion** text field, type `abs(fpt.rp-10^-8)<1e-20`.
5. In the **Particle Trajectories rp = 1e-8 m** toolbar, click **Plot**.

Particle Trajectories rp = 2e-7 m

1. In the **Model Builder** window, right-click **Particle Trajectories rp = 1e-8 m** and choose **Duplicate**.
2. In the **Settings** window for **2D Plot Group**, type `Particle Trajectories  rp = 2e-7 m` in the **Label** text field.
3. Locate the **Title** section. In the **Title** text area, type `rp = 0.2 \mu m`.
4. In the **Model Builder** window, expand the **Particle Trajectories rp = 2e-7 m** node.

Filter 1

1. In the **Model Builder** window, expand the **Results>Particle Trajectories rp = 2e-7 m>Particle Trajectories 1** node, then click **Filter 1**.
2. In the **Settings** window for **Filter**, locate the **Particle Selection** section.
3. In the **Logical expression for inclusion** text field, type `abs(fpt.rp-10^-6.7)<1e-10`.
4. In the **Particle Trajectories rp = 2e-7 m** toolbar, click **Plot**.

Particle Trajectories rp = 2e-6 m

1 In the **Model Builder** window, right-click **Particle Trajectories rp = 2e-7 m** and choose **Duplicate**.

2 In the **Settings** window for **2D Plot Group**, type `Particle Trajectories rp = 2e-6 m` in the **Label** text field.

3 Locate the **Title** section. In the **Title** text area, type `rp = 2 \mu m`.

4 In the **Model Builder** window, expand the **Particle Trajectories rp = 2e-6 m** node.

Filter 1

1 In the **Model Builder** window, expand the **Results>Particle Trajectories rp = 2e-6 m>Particle Trajectories 1** node, then click **Filter 1**.

2 In the **Settings** window for **Filter**, locate the **Particle Selection** section.

3 In the **Logical expression for inclusion** text field, type `abs(fpt.rp-10^-5.7)<1e-10`.

4 In the **Particle Trajectories rp = 2e-6 m** toolbar, click **Plot**.

Particle Trajectories rp = 5e-6 m

1 In the **Model Builder** window, right-click **Particle Trajectories rp = 2e-6 m** and choose **Duplicate**.

2 In the **Settings** window for **2D Plot Group**, type `Particle Trajectories rp = 5e-6 m` in the **Label** text field.

3 Locate the **Title** section. In the **Title** text area, type `rp = 5 \mu m`.

4 In the **Model Builder** window, expand the **Particle Trajectories rp = 5e-6 m** node.

Filter 1

1 In the **Model Builder** window, expand the **Results>Particle Trajectories rp = 5e-6 m>Particle Trajectories 1** node, then click **Filter 1**.

2 In the **Settings** window for **Filter**, locate the **Particle Selection** section.

3 In the **Logical expression for inclusion** text field, type `abs(fpt.rp-10^-5.3)<1e-10`.

4 In the **Particle Trajectories rp = 5e-6 m** toolbar, click **Plot**.

Create a **Particle Bin** dataset to be used to plot the particle collection efficiency and the accumulated charge as a function of the particle radius.

Particle Bin 1

1 In the **Results** toolbar, click **More Datasets** and choose **Particle Bin**.

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

3 In the **Expression** text field, type `fpt.rp`.

4. Locate the **Bins** section. From the **Entry method** list, choose **Tolerance**.
5. In the **Tolerance** text field, type `1e-9`.

Efficiency vs. Particle Radius

1. In the **Results** toolbar, click **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `Efficiency vs. Particle Radius` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Particle Bin 1**.
4. From the **Time selection** list, choose **Last**.
5. Click to expand the **Title** section. From the **Title type** list, choose **None**.
6. Locate the **Plot Settings** section. Select the **x-axis label** check box.
7. In the associated text field, type `Particle radius (µm)`.
8. Select the **y-axis label** check box.
9. In the associated text field, type `Particle collection efficienty`.
10. Locate the **Axis** section. Select the **x-axis log scale** check box.

Particle 1

1. In the **Efficiency vs. Particle Radius** toolbar, click **More Plots** and choose **Particle**.

   To compute the efficiency use the variable fpt.pcnt1.rL that is created by the **Particle Counter** feature.
2. In the **Settings** window for **Particle**, locate the **y-Axis Data** section.
3. In the **Expression** text field, type `1-fpt.pcnt1.rL`.
4. Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
5. In the **Expression** text field, type `fpt.rp`.
6. From the **Unit** list, choose **µm**.
7. In the **Efficiency vs. Particle Radius** toolbar, click **Plot**.

Accumulated Charge Number vs. Particle Radius

1. In the **Model Builder** window, right-click **Efficiency vs. Particle Radius** and choose **Duplicate**.
2. In the **Settings** window for **1D Plot Group**, type `Accumulated Charge Number vs. Particle Radius` in the **Label** text field.
3. Locate the **Plot Settings** section. In the **y-axis label** text field, type `Accumulated charge number`.
4. Locate the **Axis** section. Select the **y-axis log scale** check box.

Particle 1

1. In the **Model Builder** window, expand the **Accumulated Charge Number vs. Particle Radius** node, then click **Particle 1**.
2. In the **Settings** window for **Particle**, locate the **y-Axis Data** section.
3. In the **Expression** text field, type `fpt.cacc1.Za`.
4. In the **Accumulated Charge Number vs. Particle Radius** toolbar, click **Plot**.