Model created in COMSOL Multiphysics 6.4

Ionic Wind

Introduction

Ionic wind, created by the movement of charged particles in an electric field, can be harnessed for various applications, such as cooling electronic components and enhancing heat dissipation in high-performance devices. The ability to control and optimize ionic wind has significant potential in developing efficient cooling solutions for modern electronics, where managing heat is crucial for maintaining performance and reliability.

This numerical model starts from the library model Current-Voltage Characteristics of a Wire-to-Wire Corona Discharge and adds a fluid flow analysis.

Model Definition

A sequential coupling approach is employed by applying the electrohydrodynamic force generated by the corona discharge as a load on the fluid flow. The built-in variable for the electrohydrodynamic force, `edis.F`, can be directly accessed from the **Volume Force** feature in the **Laminar Flow** interface.

Results and Discussion

Figure 1 shows the ionic wind velocity driven by the corona discharge.

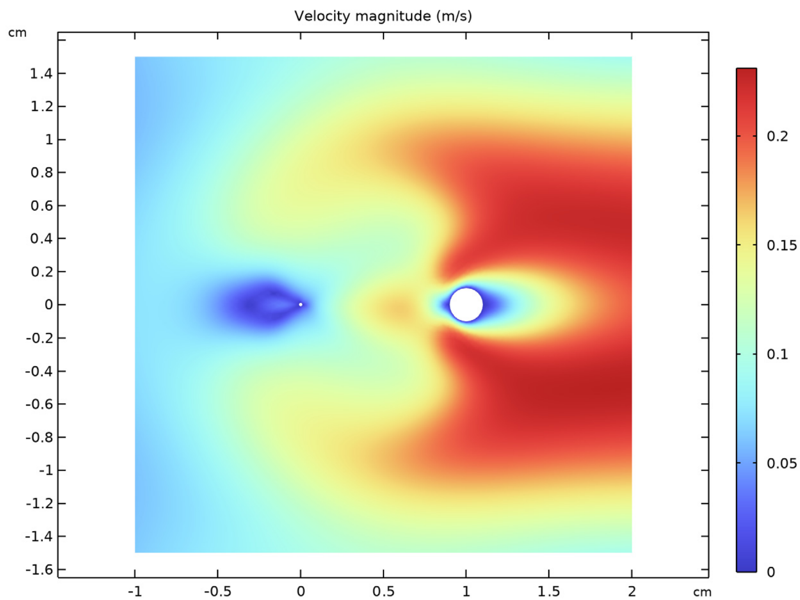

Figure 1: The ionic wind velocity.

Application Library path: `Electric_Discharge_Module/Discharge-Induced_Effects/ionic_wind`

Modeling Instructions

ROOT

Open the corona_discharge_iv_curve model.

APPLICATION LIBRARIES

1. From the **File** menu, choose **Application Libraries**.
2. In the **Application Libraries** window, select **Electric Discharge Module** > **Corona Discharges** > **corona_discharge_iv_curve** in the tree.
3. Click **Open**.

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** > **Single-Phase Flow** > **Laminar Flow (spf)**.
4 Click the **Add to Component 1** button in the window toolbar.
5 In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

LAMINAR FLOW (SPF)

Fluid Properties 1

1 In the **Settings** window for **Fluid Properties**, locate the **Fluid Properties** section.
2 From the ρ list, choose **User defined**. In the associated text field, type `1.225[kg/m^3]`.
3 From the μ list, choose **User defined**. In the associated text field, type `1e-4[Pa*s]`.

Volume Force 1

1 In the **Physics** toolbar, click **Domains** and choose **Volume Force**.
2 In the **Settings** window for **Volume Force**, locate the **Volume Force** section.
3 From the **F** list, choose **Electrohydrodynamic force (edis/gas1)**.
4 Click in the **Graphics** window and then press Ctrl+A to select both domains.

Pressure Point Constraint 1

1 In the **Physics** toolbar, click **Points** and choose **Pressure Point Constraint**.
2 Select Points 15 and 16 only.

STUDY 1

Step 2: Stationary

1 In the **Model Builder** window, expand the **Study 1** node, then 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)**.

ADD STUDY

1 In the **Study** 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 the **Add Study** button in the window toolbar.

5 In the **Study** 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 In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Electric Discharge (edis)**.

3 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**.

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

5 From the **Study** list, choose **Study 1, Stationary**.

6 From the **Parameter value (V0 (V))** list, choose **Last**.

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

RESULTS

Surface

1 In the **Model Builder** window, expand the **Velocity (spf)** node, then click **Surface**.

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

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

Selection 1

1 Right-click **Surface** and choose **Selection**.

2 Select Domain 2 only.

Velocity (spf)

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

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

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

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

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