Model created in COMSOL Multiphysics 6.4

Simulation of a Free Burning Argon Arc

Introduction

Electric arcs have nowadays a large range of industrial applications including cutting, welding, spraying, waste destruction, and surface treatment. Arc discharges are assumed to be under partial to complete local thermodynamic equilibrium (LTE) conditions. Under LTE, the plasma can be considered a conductive fluid mixture and therefore be modeled using the magnetohydrodynamics (MHD) equations. This model shows how to use the Arc Discharge interface to simulate the discharge generated in a DC arc.

Model Definition

This model is based on the work presented in Ref. 1, where the authors develop a complex model that includes the description of the weld pool under the action of a pulsed arc. In this work, only the plasma and the transfer of heat and currents in the metals are simulated, neglecting the weld pool, and a DC excitation is used. These simplifications make it possible to set up a fast-solving model that can be used to understand basic physical effects and provide initial conditions for a time-dependent model.

The model is solved using a stationary study. A current of 100 A is set at the cathode and the bottom plate is grounded. In the 5 mm gap between the electrodes, an argon plasma arc that heats the metal electrodes and the surrounding gas is created. A shielding flow is added along the cathode.

The temperature-dependent physical properties of argon are loaded from the material library under Equilibrium Discharge. The temperature range of the physical properties span from 500 K to 25,000 K. A minimum electric conductivity of 1 S/m is used for numerical stability reasons. Another important aspect to keep in mind is that the model used is not valid to describe the plasma sheath region since in this region there is charge separation and deviations from equilibrium. From a practical point of view, having a fine resolution in the plasma–electrode region causes numerical instabilities (and does not bring a better description of the physics). To make the model more stable, use a mesh that is coarse enough so that the plasma sheath is averaged out.

In this model, the initial conditions need special attention. It is very difficult to start from a flat temperature profile. Instead, it is necessary to add an initial high-temperature region between the electrodes. The model uses the continuation solver to obtain the results for the input currents 200 A and 300 A having solved the model for the 100 A case.

Results and Discussion

Figure 1, Figure 2, and Figure 3 show the temperatures in the solids and gas, fluid velocity, and electric conductivity for the 300 A arc current. The temperature reaches a maximum of 24,500 K at the anode region. The fluid velocity in the electrodes gap is much larger that the inlet as a consequence of the pressure and Lorentz forces. Note also that the conductivity in the gap is of the order or 10 kS/m, thus creating an effective conducting channel.

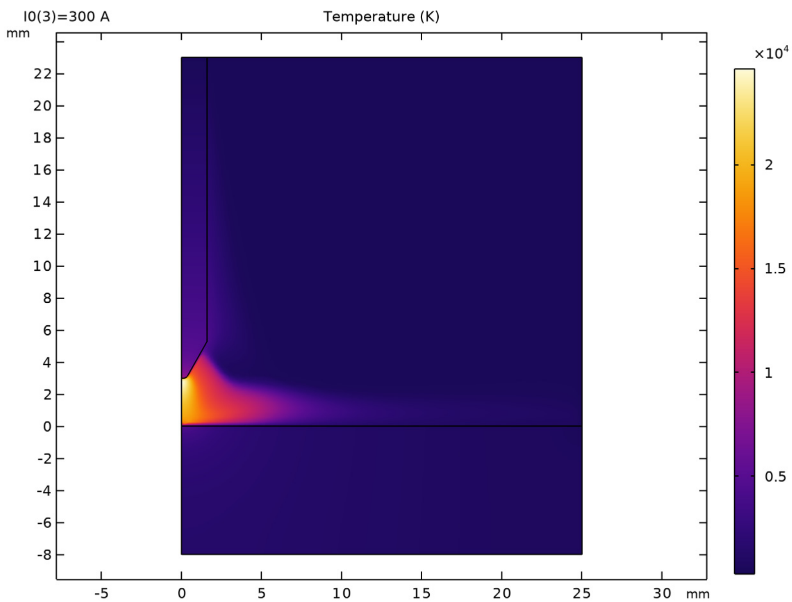


Figure 1: LTE plasma temperature.

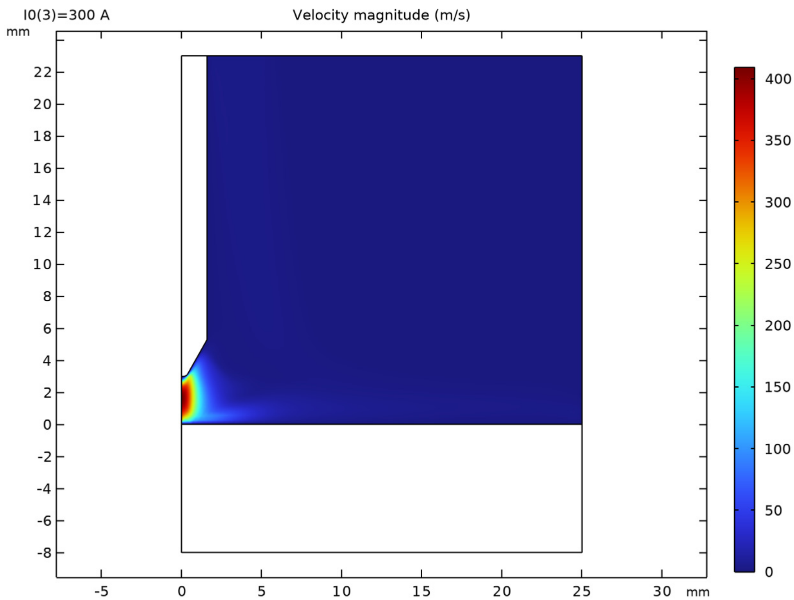


Figure 2: Fluid velocity magnitude.

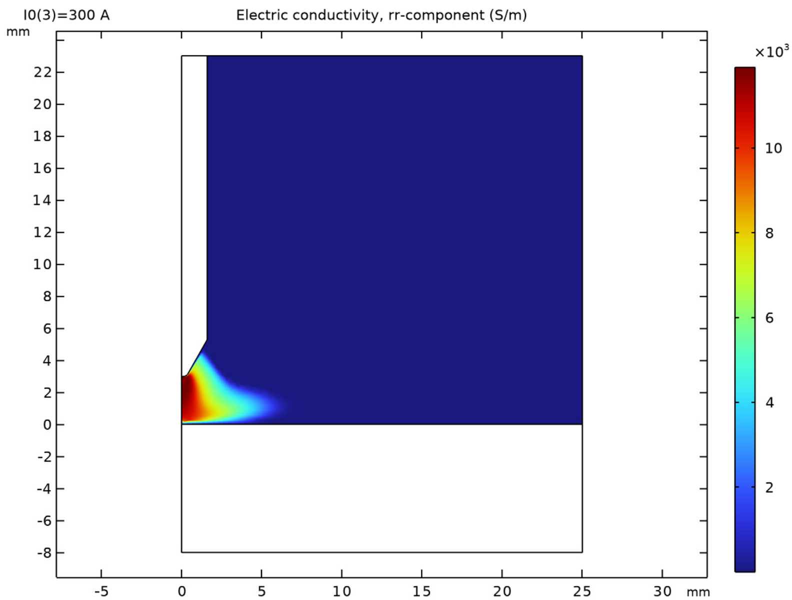


Figure 3: Electric conductivity.

Reference

1. A. Traidia, F. Roger, A. Chidley, J. Schroeder, and T. Marlaud "Effect of Helium-Argon Mixtures on the Heat Transfer and Fluid Flow in Gas Tungsten Arc Welding," *Int. J. Mech. Mechatron.*, vol. 5, no. 1, pp. 223–228, 2011.

Application Library path: `Electric_Discharge_Module/Arc_Discharges/free_burning_argon_arc`

Modeling Instructions

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

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, click **2D Axisymmetric**.

2 In the **Select Physics** tree, select **Electric Discharge > Arc Discharge**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GEOMETRY 1

Select mm units and create the geometry for the arc model.

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**.

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 `25`.

4 In the **Height** text field, type `23`.

Rectangle 2 (r2)

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

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

3 In the **Width** text field, type `25`.

4 In the **Height** text field, type `8`.

5 Locate the **Position** section. In the **z** text field, type `-8`.

Polygon 1 (pol1)

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

2 In the **Settings** window for **Polygon**, locate the **Object Type** section.

3 From the **Type** list, choose **Open curve**.

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

r (mm)	z (mm)
0	3
0.3	3
1.6	5.3
1.6	23

5 Click **Build All Objects**.

Fillet 1 (fil1)

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

2 On the object **pol1**, select Point 2 only.

3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type `0.3`.

5 Click **Build All Objects**.

Point 1 (pt1)

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

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

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

4 In the **z** text field, type `23`.

5 Click **Build All Objects**.

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

Add materials to the model. Note that the plasma properties are introduced via the argon from the Equilibrium Discharge Library.

GEOMETRY 1

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

ADD MATERIAL

1 In the **Materials** toolbar, click **Add Material** to open the **Add Material** window.

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

3 In the tree, select **Equilibrium Discharge** > **Argon (1[atm])**.

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

MATERIALS

Argon (1[atm]) (mat1)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 Click **Clear Selection**.

3 Select Domain 2 only.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-in** > **Steel AISI 4340**.

3 Click the **Add to Component** button in the window toolbar.

MATERIALS

Steel AISI 4340 (mat2)

Select Domain 1 only.

Define parameters for the input current and inlet velocity.

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
I0	100[A]	100 A	Current
U0	3[m/s]	3 m/s	Inlet velocity

MAGNETIC AND ELECTRIC FIELDS (MEF)

Magnetic Insulation 1

In the **Model Builder** window, under **Component 1 (comp1)** > **Magnetic and Electric Fields (mef)** click **Magnetic Insulation 1**.

Electric Insulation 1

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

2 Select Boundaries 10, 11, and 13 only.

Magnetic Insulation 1

In the **Model Builder** window, click **Magnetic Insulation 1**.

Boundary Terminal 1

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

2 Select Boundary 7 only.

3 In the **Settings** window for **Boundary Terminal**, locate the **Terminal** section.

4 In the I_0 text field, type `I0`.

Gauge Fixing for A-Field 1

In the **Physics** toolbar, click **Domains** and choose **Gauge Fixing for A-Field**.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-in** > **Tungsten**.

3 Click the **Add to Component** button in the window toolbar.

4 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Tungsten (mat3)

1 Select Domain 3 only.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	1	Basic
Electric conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	`20e6[S/m]`	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic

In the Heat Transfer in Fluids you need to define the regions that correspond to solid materials, some boundary conditions, and a special initial condition.

HEAT TRANSFER IN FLUIDS (HT)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Fluids (ht)**.

2 In the **Settings** window for **Heat Transfer in Fluids**, locate the **Physical Model** section.

3 In the T_{ref} text field, type `300[K]`.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Heat Transfer in Fluids (ht)** click **Initial Values 1**.

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

3 In the T text field, type `300[K]`.

Solid 1

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

2 In the **Settings** window for **Solid**, locate the **Model Input** section.

3 From the T_{ref} list, choose **User defined**. In the associated text field, type `300[K]`.

4 Select Domains 1 and 3 only.

Temperature 1

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

2 Select Boundaries 7, 10, 11, and 13 only.

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

4 In the T_0 text field, type `300[K]`.

Heat Flux 1

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

2 Select Boundaries 2 and 12 only.

3 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.

4 From the **Flux type** list, choose **Convective heat flux**.

5 In the h text field, type `200`.

6 In the T_{ext} text field, type `300[K]`.

Surface-to-Ambient Radiation 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Surface-to-Ambient Radiation**.

2 Select Boundaries 4, 6, 8, 9, and 14 only.

3 In the **Settings** window for **Surface-to-Ambient Radiation**, locate the **Surface-to-Ambient Radiation** section.

4 From the ε list, choose **User defined**. In the associated text field, type `0.4`.

This problem needs some special initial conditions. It is important to start with a high temperature, though not everywhere; define a temperature profile.

Initial Values 2

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

2 Select Domain 2 only.

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

4 In the T text field, type `15e3*exp(-(r/1[mm])^2)+300`.

LAMINAR FLOW (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

2 In the **Settings** window for **Laminar Flow**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 2 only.

Open Boundary 1

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

2 Select Boundaries 11 and 13 only.

In the Laminar Flow interface you need to add an inlet and an open boundary condition.

Inlet 1

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

2 Select Boundary 10 only.

3 In the **Settings** window for **Inlet**, locate the **Velocity** section.

4 In the U_0 text field, type `U0`.

The model does not describe the physics of the non-LTE layer near the electrodes. This leads to unphysically low temperature and electric conductivity at the electrodes that can cause numerical issues. Making the mesh at the electrode coarse enough makes things easier.

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**.

Size 1

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

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

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

Size 2

1 In the **Model Builder** window, click **Size 2**.

2 Select Boundaries 3, 4, 6, 8, 9, and 14 only.

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

4 From the **Predefined** list, choose **Extra fine**.

Boundary Layers 1

1 In the **Model Builder** window, right-click **Boundary Layers 1** and choose **Disable**.

2 In the **Settings** window for **Boundary Layers**, click **Build All**.

This type of problems solve better with a Fully Coupled solver. This option needs to be added manually. A few options in the solver are also adjusted for the present problem.

STUDY 1

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

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

3 Clear the **Generate default plots** checkbox.

Solution 1 (sol1)

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

The continuation solver is suitable for running a parametric sweep for this type of model.

Step 1: Stationary

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

2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 Select the **Auxiliary sweep** checkbox.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
I0 (Current)	100 200 300	A

Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

2 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 1** node, then click **Fully Coupled 1**.

3 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

4 In the **Initial damping factor** text field, type `1E-4`.

5 In the **Restriction for step-size update** text field, type `1.5`.

6 In the **Recovery damping factor** text field, type `0.1`.

7 In the **Maximum number of iterations** text field, type `200`.

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

Create some plots.

RESULTS

Temperature

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

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

Surface 1

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

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

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

4 Locate the **Expression** section. In the **Expression** text field, type `T`.

5 In the **Temperature** toolbar, click **Plot**.

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

Velocity

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

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

Surface 1

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

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

3 In the **Expression** text field, type `spf.U`.

4 In the **Velocity** toolbar, click **Plot**.

Electric conductivity

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

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

Surface 1

1 Right-click **Electric conductivity** and choose **Surface**.

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

3 In the **Expression** text field, type `mef.sigmarr`.

4 In the **Electric conductivity** toolbar, click **Plot**.

Selection 1

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

2 Select Domain 2 only.

3 In the **Electric conductivity** toolbar, click **Plot**.

Magnetic Flux

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

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

Surface 1

1 Right-click **Magnetic Flux** and choose **Surface**.

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

3 In the **Expression** text field, type `log10(mef.normB)`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **JupiterAuroraBorealis**.

5 In the **Magnetic Flux** toolbar, click **Plot**.

6 From the **Color table transformation** list, choose **Reverse**.

Revolution 2D 1

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

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

3 In the **Settings** window for **Revolution 2D**, click to expand the **Revolution Layers** section.

4 In the **Start angle** text field, type `-90`.

5 In the **Revolution angle** text field, type `225`.

Temperature 3D

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Temperature 3D` in the **Label** text field.

Volume 1

1 Right-click **Temperature 3D** and choose **Volume**.

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

3 In the **Expression** text field, type `T`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **HeatCamera**.

5 In the **Temperature 3D** toolbar, click **Plot**.

VI Curve

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

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

Global 1

1 Right-click **VI Curve** and choose **Global**.

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

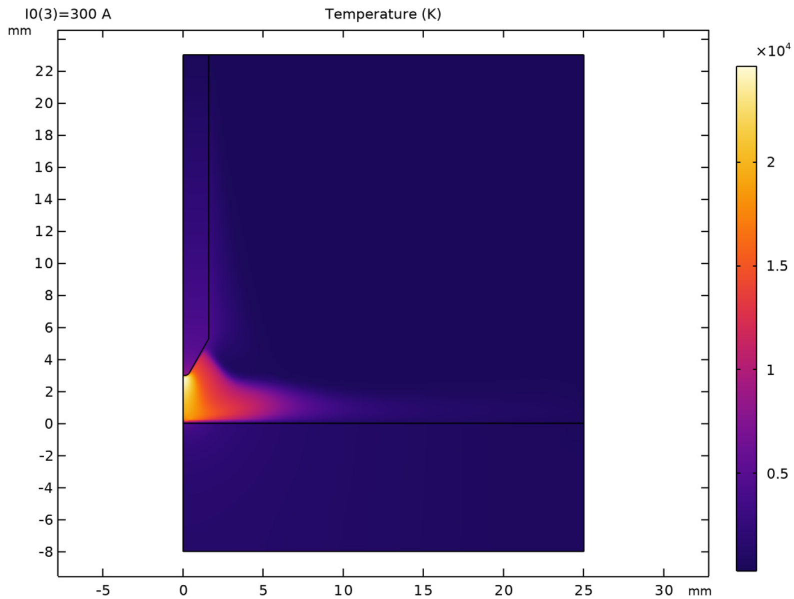
3 In the table, enter the following settings:

Expression	Unit	Description
`mef.V0_1`	`V`	`Terminal voltage`

4 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

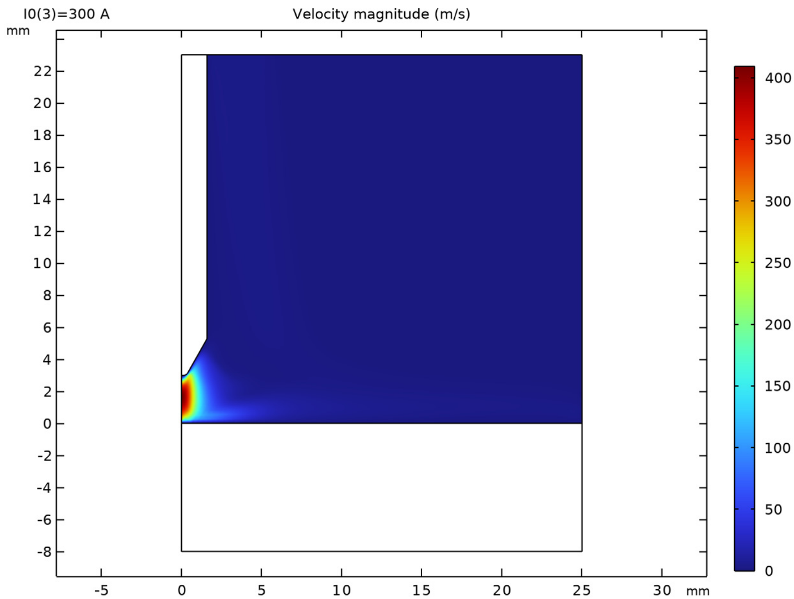
Temperature

In the **Model Builder** window, under **Results** click **Temperature**.



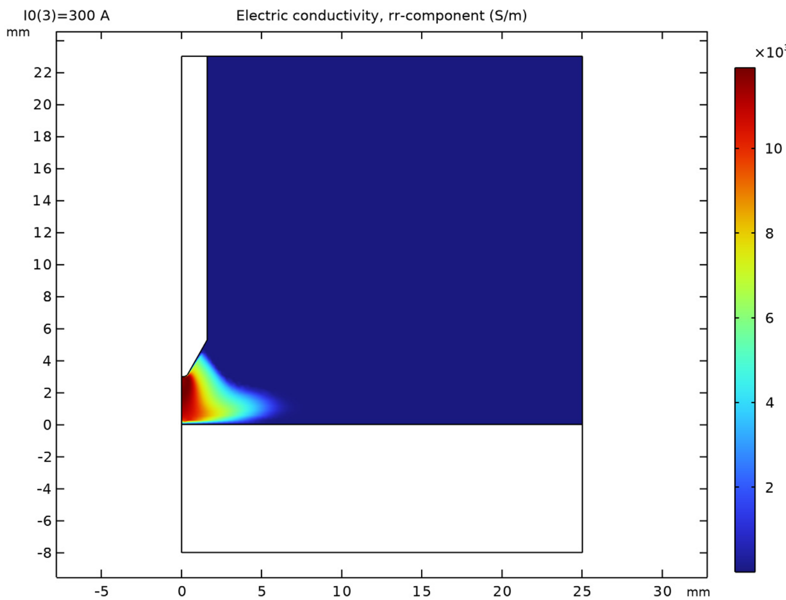
Velocity

In the **Model Builder** window, click **Velocity**.



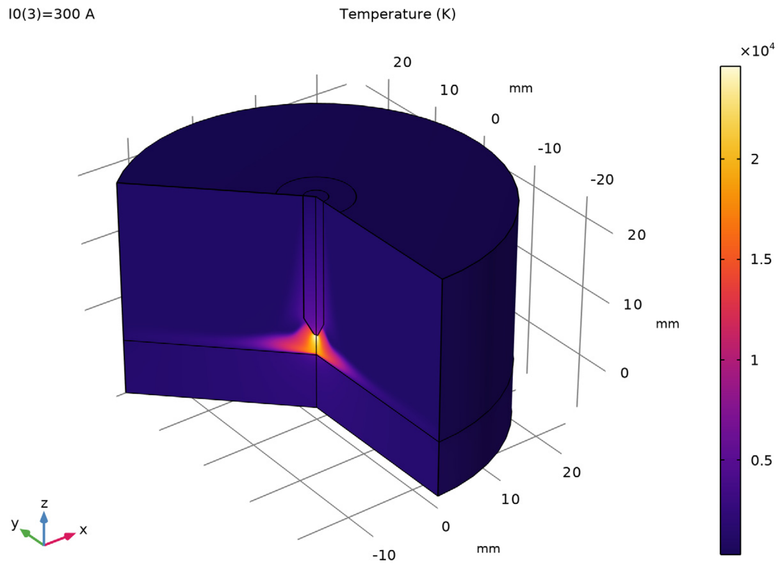
Electric conductivity

In the **Model Builder** window, click **Electric conductivity**.



Temperature 3D

In the **Model Builder** window, click **Temperature 3D**.



VI Curve

In the **Model Builder** window, click **VI Curve**.