Created in COMSOL Multiphysics 6.1

Plasma DC Arc

Introduction

Thermal plasmas have nowadays a large range of industrial applications including cutting, welding, spraying, waste destruction, and surface treatment. Thermal plasmas 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 Equilibrium Discharges, In-Plane Currents interface (available in 2D and 2D axisymmetric) interface to simulate the plasma generated in a DC arc.

Note: This application requires the Plasma Module and AC/DC Module.

Model Definition

This model is based on the work presented in Ref. 1. In Ref. 1, 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 have a model that solves fast, that can be used to understand basic physical effects, and that can be used as initial conditions for a time-dependent model.

The model is solved using a stationary study. A current of 80 A is set at the cathode and the bottom plate is grounded. In the 5 mm gap between the electrodes, an argon plasma arc is created that heats the metal electrodes and surrounding gas. 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 electrical 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 regions there is charge separation and deviations from equilibrium. From the 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 condition need special attention. It is very difficult to start from a flat profile of temperature. It is necessary to add an initial high temperature region between the electrodes.

Results and Discussion

Figure 1, Figure 2, and Figure 3 show the temperatures in the solids and gas, fluid velocity, and electrical conductivity. The temperature reaches a maximum of 14,000 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 10k S/m, thus creating an effective conducting channel.

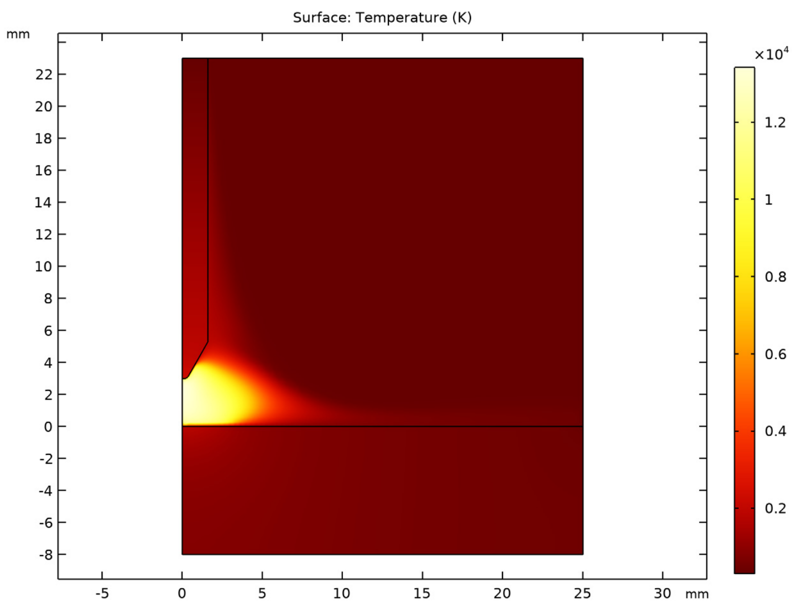

Figure 1: Plot of the LTE plasma temperature.

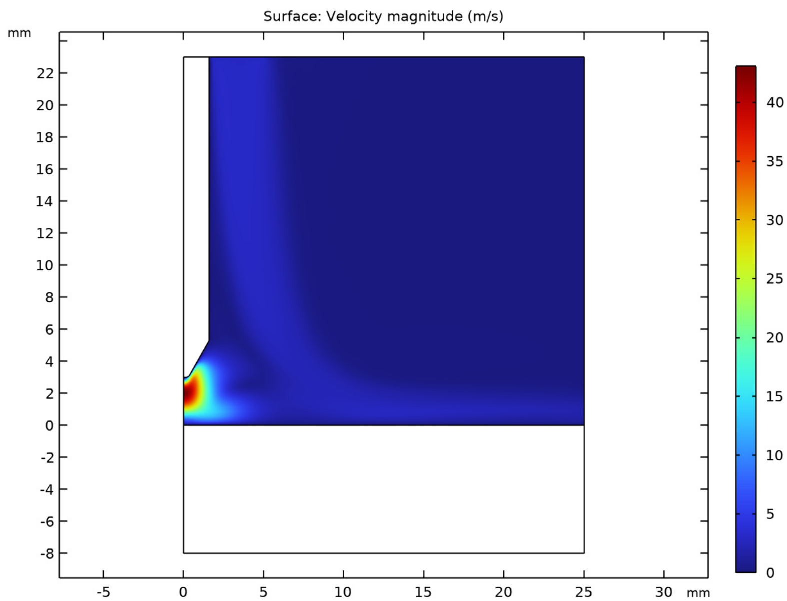

Figure 2: Plot of the velocity magnitude of the fluid.

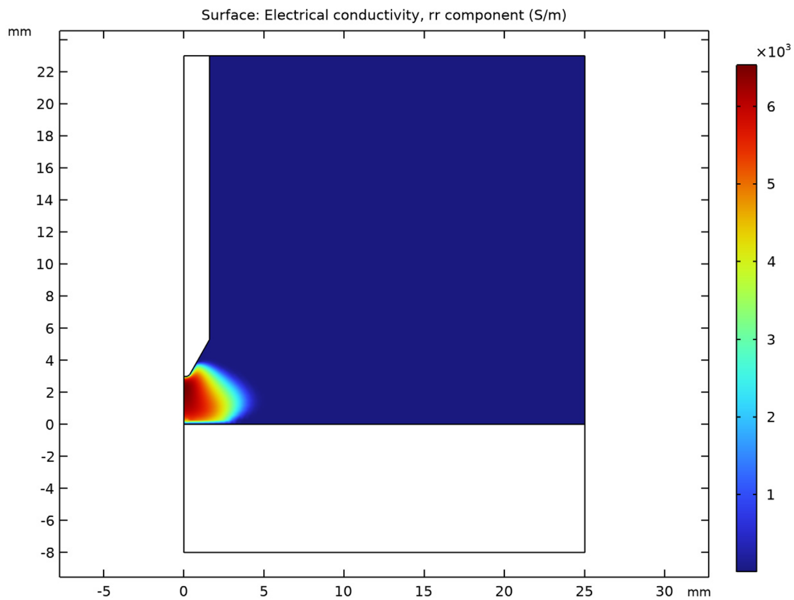

Figure 3: Plot of the electrical conductivity.

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: `Plasma_Module/Equilibrium_Discharges/plasma_dc_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 **Plasma>Equilibrium Discharges>Equilibrium Discharges, In-Plane Currents**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies>Stationary**.
6 Click **Done**.

GEOMETRY 1

Select the 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**.

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 **Home** 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**.

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

MATERIALS

Argon (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 **Add to Component** in the window toolbar.

MATERIALS

Steel AISI 4340 (mat2)

Select Domain 1 only.

ADD MATERIAL

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

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

3 Click **Add to Component** 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
Electrical 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

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`	`80[A]`	80 A	`Current`
`J0`	`-I0/(pi*(1.6[mm])^2)`	-9.9472E6 A/m²	`Normal Current density`
`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**.

Normal Current Density 1

1. In the **Physics** toolbar, click **Attributes** and choose **Normal Current Density**.
2. Select Boundary 7 only.
3. In the **Settings** window for **Normal Current Density**, locate the **Normal Current Density** section.
4. In the J_n text field, type `J0`.

Gauge Fixing for A-field 1

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

   In the Heat Transfer in Fluids you will 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].

Boundary Heat Source 1

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

Add radiation cooling at the electrodes surface. Here it is used an emissivity coefficient of the object of 0.4.

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

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

4 In the Q_b text field, type -0.4*sigma_const*T^4.

This problem needs some special initial conditions. It is important to start with a high temperature but it can't be everywhere. A profile of temperature is defined.

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 **Physical Model** section.

3 From the **Compressibility** list, choose **Weakly compressible flow**.

4 Locate the **Domain Selection** section. Click **Clear Selection**.

5 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 will need to: add an inlet and an open boundary conditions.

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`.

In the Multiphysics coupling features you will have to: set the boundary conditions for the anode and cathode.

MULTIPHYSICS

Equilibrium Discharge Boundary Heat Source 1 (bphs1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Equilibrium Discharge Boundary Heat Source 1 (bphs1)**.

2 In the **Settings** window for **Equilibrium Discharge Boundary Heat Source**, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 Select Boundary 4 only.

Equilibrium Discharge Boundary Heat Source 2 (bphs2)

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Boundary> Equilibrium Discharge Boundary Heat Source**.

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

3 In the **Settings** window for **Equilibrium Discharge Boundary Heat Source**, locate the **Electrode Properties** section.

4 From the **Electrode polarity** list, choose **Cathode**.

Some adjustments to the default mesh are necessary. They consist in some size adjustments and in removing the Boundary Layers.

The present model does not describe the physics of the non-LTE layer near the electrodes. This leads to unphysical low temperature and electrical 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** check box.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.
2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.
3 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** node.
4 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** and choose **Fully Coupled**.
5 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.
6 In the **Initial damping factor** text field, type `1E-4`.

7 In the **Minimum damping factor** text field, type `1.0E-6`.

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

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

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

11 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** click **Advanced**.

12 In the **Settings** window for **Advanced**, click to expand the **Assembly Settings** section.

13 Clear the **Reuse sparsity pattern** check box.

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

Create some plots.

RESULTS

Temperature

1 In the **Home** toolbar, click **Add Plot Group** and choose **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 Click **Change Color Table**.

4 In the **Color Table** dialog box, select **Thermal>Thermal** in the tree.

5 Click **OK**.

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

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

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

Velocity

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

Electrical conductivity

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

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

Surface 1

1 Right-click **Electrical 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 **Electrical conductivity** toolbar, click **Plot**.

Selection 1

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

2 Select Domain 2 only.

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

Magnetic flux

1 In the **Home** toolbar, click **Add Plot Group** and choose **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. Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Aurora>JupiterAuroraBorealis** in the tree.

6 Click **OK**.

7 In the **Magnetic flux** toolbar, click **Plot**.

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

9 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. Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Thermal>Thermal** in the tree.

6 Click **OK**.

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