Created in COMSOL Multiphysics 6.0

Plasma Pulsed Arc

This model is licensed under the COMSOL Software License Agreement 6.0.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

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 Combined Inductively/DC Discharge interface to simulate the plasma generated in a pulsed 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 it is only simulated the plasma and the transfer of heat and currents in the metals neglecting the weld pool. This model starts by opening a model of a DC arc which solution are going to be used as initial conditions to the time dependent problem.

The applied current consists of a pulse with a frequency of 1 Hz, a peak current of 160 A, a floor current of 80 A, and a duty cycle of 0.5. The current source 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 coarse enough so that the plasma sheath is averaged out.

Results and Discussion

Figure 1 and Figure 2 show the fluid velocity and temperature for instants in the current peak and floor of 160 A and 80 A at 5 s. As expected, at the peak value the fluid velocity and temperature are higher. In the results shown the temperature of the subtract still hasn't reached a steady state that is reached at about 50 s. The final time of computation was set to 5s for practical reasons. The weld pool (not solved here) also has significant changes during the pulse different phases as shown in Ref. 1.

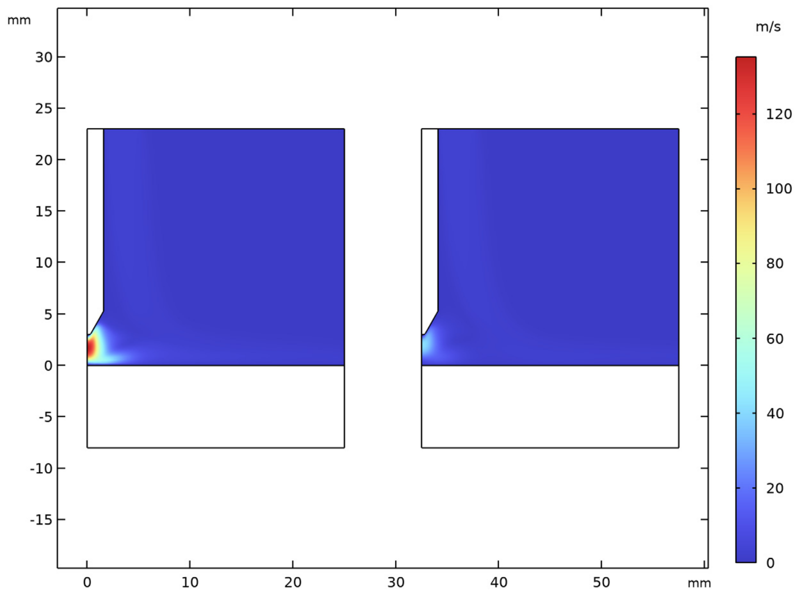

Figure 1: Plot of the velocity magnitude of the fluid at the current peak of 160 A (left) and current floor of 80 A (right).

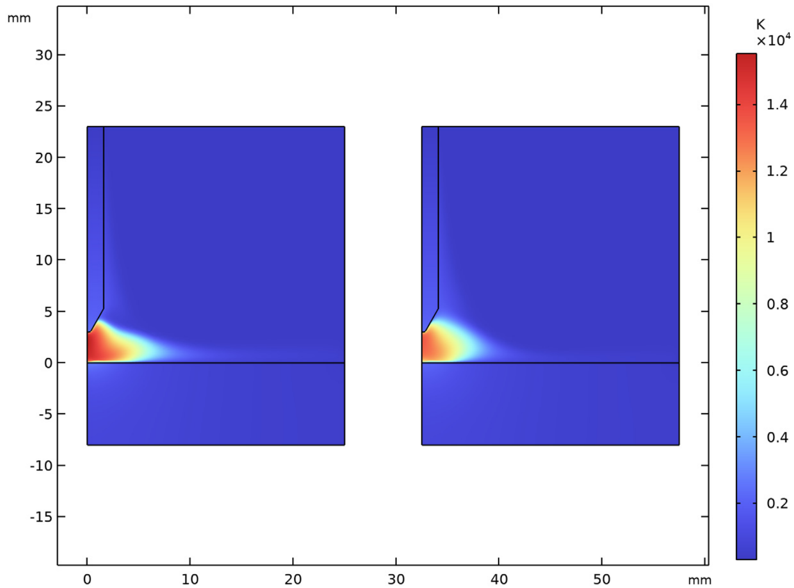

Figure 2: Plot of the plasma temperature at the current peak of 160 A (left) and current floor of 80 A (right).

Reference

1. A. Traidia, *Multiphysics Modeling and Numerical Simlation of GTA Weld Pools*, Ecole Polytechique PhD Thesis, Paris France, 2011.

Application Library path: `Plasma_Module/Equilibrium_Discharges/plasma_pulsed_arc`

Modeling Instructions

This model uses an existing model from a dc arc and sets an ac excitation instead. The solution of the dc arc model are used as initial conditions to the time dependent model.

ROOT

Open the plasma_dc_arc model from the Application Libraries.

1 In the **Home** toolbar, click **Windows** and choose **Application Libraries**.

APPLICATION LIBRARIES

1 In the **Model Builder** window, click the root node.

2 In the **Application Libraries** window, select **Plasma Module>Equilibrium Discharges>plasma_dc_arc** in the tree.

3 Click **Open**.

RESULTS

Temperature 3D

Add a parameter to define the floor of the current pulse.

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
delta	80[A]	80 A	

Add a square wave function to define the pulsed excitation.

COMPONENT 1 (COMP1)

In the **Model Builder** window, expand the **Component 1 (comp1)** node.

DEFINITIONS

Waveform 1 (wv1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

2 Right-click **Definitions** and choose **Functions>Waveform**.

3 In the **Settings** window for **Waveform**, type `current` in the **Function name** text field.

4 Locate the **Parameters** section. From the **Type** list, choose **Square**.

5 In the **Size of transition zone** text field, type `0.05`.

6 In the **Period** text field, type `1`.

7 In the **Phase** text field, type `180`.

8 In the **Amplitude** text field, type `delta/2`.

Duplicate the existent Normal Current Density feature and set a pulsed current. The Normal Current Density feature are disabled at the study level when needed.

ELECTRIC CURRENTS (EC)

Normal Current Density 2

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Electric Currents (ec)** node.

2 Right-click **Component 1 (comp1)>Electric Currents (ec)>Normal Current Density 1** and choose **Duplicate**.

3 In the **Settings** window for **Normal Current Density**, locate the **Normal Current Density** section.

4 In the J_n text field, type `-(I0+delta/2+current(t[1/s]))/(pi*(1.6[mm])^2)`.

For the stationary study disable the pulsed excitation. This way, if this study needs to be solved later the correct current feature is used.

STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Stationary**.

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

3 Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (Comp1)>Electric Currents (Ec)> Normal Current Density 2**.

5 Click **Disable**.

RESULTS

Electrical conductivity, Magnetic flux, Temperature, Temperature 3D, Velocity

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Temperature**, **Velocity**, **Electrical conductivity**, **Magnetic flux**, and **Temperature 3D**.

2 Right-click and choose **Group**.

DC arc

In the **Settings** window for **Group**, type `DC arc` in the **Label** text field.

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 **General Studies> 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

Prepare the time dependent study. Here, a few changes are necessary: disable the DC excitation, change the relative tolerance to 0.005, use previous solutions as initial conditions, use a fully coupled solver, uncheck the initial step option, and uncheck the reuse sparsity patter option.

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

2 Select the **Modify model configuration for study step** check box.

3 In the tree, select **Component 1 (Comp1)>Electric Currents (Ec)> Normal Current Density 1**.

4 Click **Disable**.

5 Locate the **Study Settings** section. In the **Output times** text field, type `range(0,0.1,5)`.

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

7 In the **Relative tolerance** text field, type `0.005`.

8 In the **Study** toolbar, click **Get Initial Value**.

RESULTS

Current Density Norm (ec), Electric Potential (ec), Electric Potential, Revolved Geometry (ec), Isothermal Contours (ht), Magnetic Flux Density Norm (mf), Magnetic Flux Density Norm, Revolved Geometry (mf), Pressure (spf), Temperature, 3D (ht), Velocity (spf)

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Electric Potential (ec)**, **Electric Potential, Revolved Geometry (ec)**, **Current Density Norm (ec)**, **Magnetic Flux Density Norm (mf)**, **Magnetic Flux Density Norm, Revolved Geometry (mf)**, **Temperature, 3D (ht)**, **Isothermal Contours (ht)**, **Velocity (spf)**, and **Pressure (spf)**.

2 Right-click and choose **Group**.

Pulsed arc

In the **Settings** window for **Group**, type `Pulsed arc` in the **Label** text field.

Temperature pulsed

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 pulsed` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

Surface 1

1 Right-click **Temperature pulsed** and choose **Surface**.
2 In the **Settings** window for **Surface**, 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 **Thermal**.

STUDY 2

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 2** click **Step 1: Time Dependent**.
2 In the **Settings** window for **Time Dependent**, click to expand the **Values of Dependent Variables** section.
3 Find the **Initial values of variables 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**.

Solver Configurations

In the **Model Builder** window, expand the **Study 2>Solver Configurations** node.

Solution 2 (sol2)

1 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 2 (sol2)** node, then click **Time-Dependent Solver 1**.
2 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.
3 Clear the **Initial step** check box.
4 Right-click **Time-Dependent Solver 1** and choose **Fully Coupled**.
5 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

6 From the **Nonlinear method** list, choose **Automatic (Newton)**.

7 In the **Model Builder** window, click **Advanced**.

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

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

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 2** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, click to expand the **Results While Solving** section.

3 Select the **Plot** check box.

4 From the **Plot group** list, choose **Temperature pulsed**.

5 From the **Update at** list, choose **Time steps taken by solver**.

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

RESULTS

Electric Potential (ec)

Prepare plots to show the velocity and the temperature for the peak and floor values of the current side-by-side.

Velocity (spf)

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

2 In the **Settings** window for **2D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

4 Click to expand the **Plot Array** section. Select the **Enable** check box.

Surface

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

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

3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 From the **Time (s)** list, choose **4.6**.

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **RainbowLight**.

Surface 2

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

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

3 From the **Time (s)** list, choose **5**.

4 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface**.

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

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

Velocity (spf)

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

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

3 Select the **Show units** check box.

Temperature pulsed

1 In the **Model Builder** window, click **Temperature pulsed**.

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

3 From the **Title type** list, choose **None**.

4 Locate the **Color Legend** section. Select the **Show units** check box.

5 Locate the **Plot Array** section. Select the **Enable** check box.

Surface 1

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

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

3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 From the **Time (s)** list, choose **4.6**.

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **RainbowLight**.

Surface 2

1 Right-click **Results>Pulsed arc>Temperature pulsed>Surface 1** and choose **Duplicate**.

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

3 From the **Time (s)** list, choose **5**.

4 Locate the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

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

Temperature pulsed

Plot the temporal evolution of the applied current and the temperature of the bottom plate.

Current vs. Time

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

2 In the **Settings** window for **1D Plot Group**, type `Current vs. Time` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Point Graph 1

1 Right-click **Current vs. Time** and choose **Point Graph**.

2 Select Point 4 only.

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

4 In the **Expression** text field, type `-ec.ncd2.nJ*(pi*(1.6[mm])^2)`.

5 In the **Current vs. Time** toolbar, click **Plot**.

Temperature vs. Time

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

2 In the **Settings** window for **1D Plot Group**, type `Temperature vs. Time` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 Locate the **Title** section. From the **Title type** list, choose **None**.

Point Graph 1

1 Right-click **Temperature vs. Time** and choose **Point Graph**.

2 Select Point 1 only.

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

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

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