Created in COMSOL Multiphysics 6.1

# Inductively Coupled Plasma (ICP) Torch

## 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, Out-of-Plane Currents interface (available in 2D and 2D axisymmetric) to simulate the plasma generated in an inductively coupled plasma torch.

Figure 1 displays the geometry of the to-be-modeled inductive plasma torch.

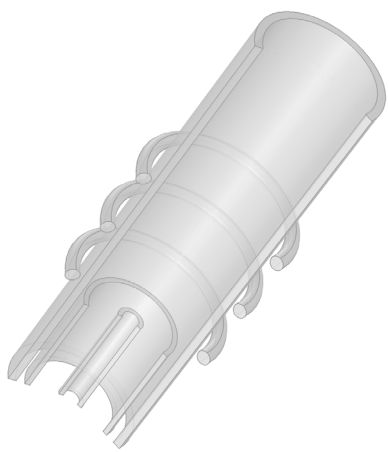

*Figure 1: Geometry of an inductively coupled plasma torch. The torch is composed of three concentric quartz tubes in which gas are injected from the bottom and exit from the top the torch. In this model, a fixed power of 11 kW is transferred to the plasma by a three-turn coil operating at 3MHz.*

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

## Model Definition

This model is based on the work presented in Ref. 1 and uses the following assumptions:

- The plasma torch is modeled by a fully axisymmetric configuration.

- The current inside the plasma is dominated by the induced current which is out-of-plane, that is, in the azimuth direction.
- The coil consists of parallel current carrying rings with a circle cross section, 6 mm in diameter. This implies neglecting the axial component of the coil current.
- Steady state, laminar pure argon plasma flow at atmospheric pressure.
- Optically thin plasma under local thermodynamic equilibrium (LTE) conditions.

Figure 2 shows the geometry of the model.

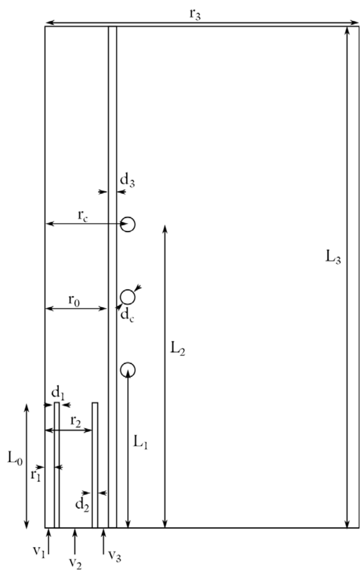


*Figure 2: Schematic of the ICP torch. Flow enters from the base (v1, v2 and v3) and leaves out the top. The dimensions of the different part of the model are given in the Modeling Instructions section.*

In this model excitation is provided to a three turns coil at 3 MHz. The gas flowing in the sheath tube (plasma confinement tube) is then ionized by Joule heating.

The model is solved using a frequency-stationary study in combination with a single turn coil feature which set a fixed power to the system (11 kW). By fixing the power, the current and electric potential can vary in the coil as the plasma electrical conductivity builds up.

In this model the three different gas stream velocities (v1 for the carrier tube, v2 for the central tube and v3 for the sheath tube) are composed of pure argon. The temperature-dependent physical properties of argon are loaded from the material library under Equilibrium Discharge. Note that the temperature range of the physical properties span from 500 K to 25,000K. Note also that a minimum electrical conductivity of 1 S/m is used for numerical stability reasons.

If the initial temperature is too low chances are that the solution found corresponds to a flat profile of the minimum electrical conductivity (the default is 1 S/m). This is obviously a solution without interest and in fact it is the easiest solution to obtain. To avoid this start at an higher temperature closer to the experimental value as it is done in the present example. Always make sure to plot the conductivity to see if it is set to the minimum electrical conductivity.

## *Results and Discussion*

Figure 3 and Figure 4, respectively, shows the plasma temperature distribution and velocity magnitude of the argon plasma. The temperature peaks near the coils to a value of 10,000 K. The plasma conductivity increases with the temperature and it has a maximum in the regions of maximum temperature as shown in Figure 5 where the electrical conductivity of the plasma is plotted. Figure 6 displays the magnetic flux norm. Note that the electrical conductivity of the plasma screens the magnetic flux as a consequence of the skin effect.

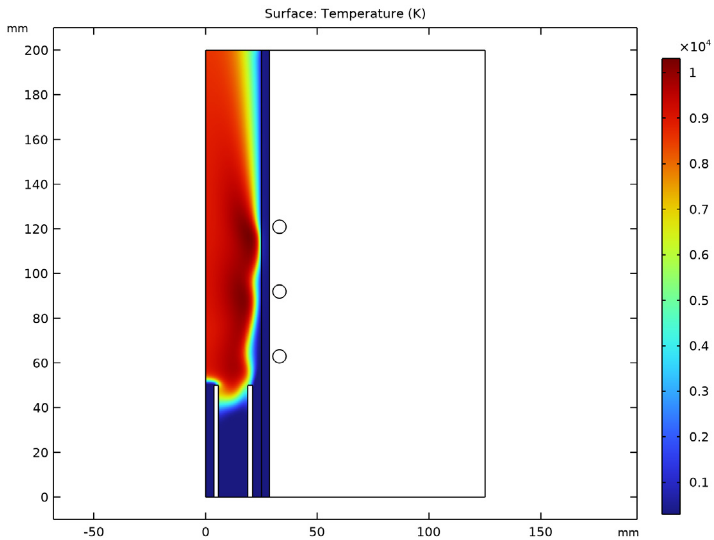


*Figure 3: Surface plot of the LTE plasma temperature.*

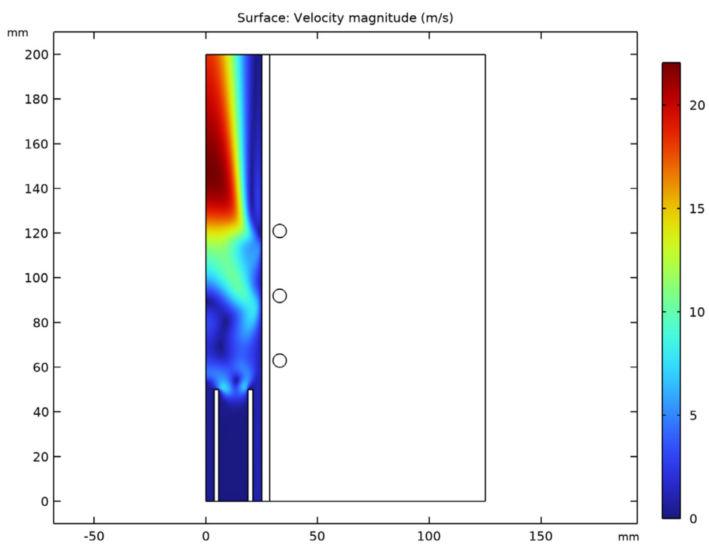


*Figure 4: Plot of the velocity magnitude.*

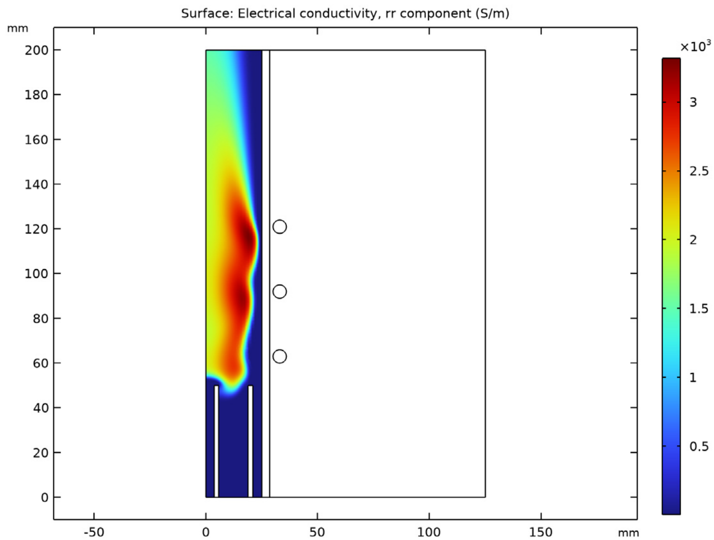


*Figure 5: Plot of the plasma electrical conductivity.*

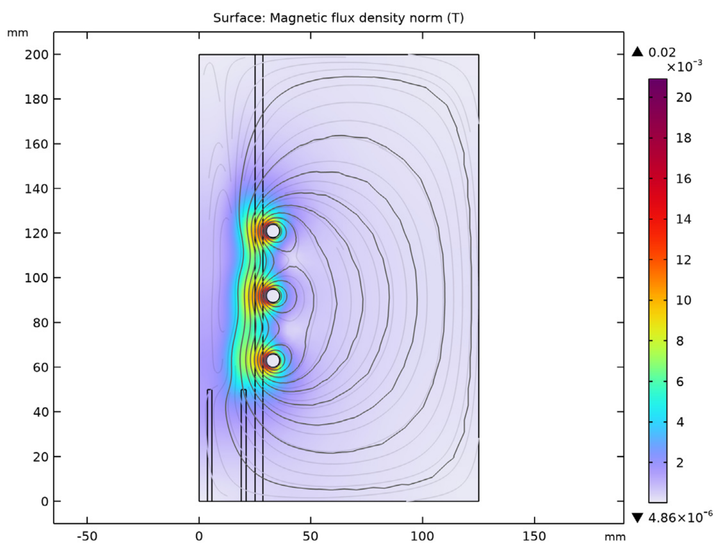


*Figure 6: Norm of the magnetic flux. Note the effect of the resistivity on the penetration of the field (skin effect).*

## *Reference*

1. S. Xue, P. Proulx, and M.I. Boulos, "Extended-field electromagnetic model for inductively coupled plasma," *J. Phys. D.* 34, 1897, 2001.

**Application Library path:** `Plasma_Module/Equilibrium_Discharges/icp_torch`

## *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, Out-of-Plane Currents**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Multiphysics>Frequency-Stationary**.

6 Click **Done**.

### ROOT

Select the mm units.

### GEOMETRY 1

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**.

### 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** Click **Load from File**.

**4** Browse to the model's Application Libraries folder and double-click the file `icp_torch_parameters.txt`.

Define the computational domain.

## 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 `r_3`.

**4** In the **Height** text field, type `L_3`.

Define the carrier tube.

*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 `d_1`.

**4** In the **Height** text field, type `L_0`.

**5** Locate the **Position** section. In the **r** text field, type `r_1`.

Define the central tube.

*Rectangle 3 (r3)*

**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 `d_2`.

**4** In the **Height** text field, type `L_0`.

**5** Locate the **Position** section. In the **r** text field, type `r_2`.

Define the tube.

*Rectangle 4 (r4)*

**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 `d_3`.

**4** In the **Height** text field, type `L_3`.

**5** Locate the **Position** section. In the **r** text field, type `r_0`.

Define the coils.

*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 `d_c/2`.

**4** Locate the **Position** section. In the **r** text field, type `r_c`.

**5** In the **z** text field, type `L_1`.

*Circle 2 (c2)*

**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 `d_c/2`.

**4** Locate the **Position** section. In the **r** text field, type `r_c`.

**5** In the **z** text field, type `(L_1+L_2)/2`.

*Circle 3 (c3)*

**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 `d_c/2`.

**4** Locate the **Position** section. In the **r** text field, type `r_c`.

**5** In the **z** text field, type `L_2`.

**6** Click **Build All Objects**.

Define the different domain type for easy selection.

## DEFINITIONS

*Air*

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

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

**3** Select Domain 5 only.

*Plasma*

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

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

3 Select Domain 1 only.

*Quartz*

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

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

3 Select Domains 2–4 only.

*Coils*

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

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

3 Select Domains 6–8 only.

Add the different materials used in the model using the material library.

## 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 tree, select **AC/DC>Copper**.

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

7 In the tree, select **AC/DC>Quartz**.

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

9 In the tree, select **Equilibrium Discharge>Argon**.

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

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

## MATERIALS

*Air (mat1)*

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Air (mat1)**.

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

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

*Copper (mat2)*

1 In the **Model Builder** window, click **Copper (mat2)**.

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

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

*Quartz (mat3)*

1 In the **Model Builder** window, click **Quartz (mat3)**.

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

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

*Argon (mat4)*

1 In the **Model Builder** window, click **Argon (mat4)**.

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

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

Adjust the selection and features of each physics composing the model.

The magnetic field interface is used over the whole computational domain. The Single conductor Coil feature is used here to transfer the excitation power to the plasma.

### MAGNETIC FIELDS (MF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Magnetic Fields (mf)**.

2 In the **Settings** window for **Magnetic Fields**, click to expand the **Discretization** section.

3 From the **Magnetic vector potential** list, choose **Linear**.

*Coil 1*

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

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

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

4 Locate the **Coil** section. Select the **Coil group** check box.

5 From the **Coil excitation** list, choose **Power**.

6 In the $P_{coil}$ text field, type `Pext`.

The heat transfer in the air is neglected in this model.

### HEAT TRANSFER IN FLUIDS (HT)

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

2 Select Domains 1 and 4 only.

*Solid 1*

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

2 Select Domain 4 only.

*Initial Values 1*

It is important to start with an high temperature.

1 In the **Model Builder** window, click **Initial Values 1**.

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

3 In the $T$ text field, type `6000`.

Add a heat transfer in solids feature for the solid part of the heat transfer model (tubes and coils).

*Temperature 1*

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

2 Select Boundaries 2, 8, 13, 15, and 17 only.

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

4 In the $T_0$ text field, type `T0`.

*Outflow 1*

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

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

3 Select Boundary 3 only.

The single phase flow is only applied to the plasma region.

## LAMINAR FLOW (SPF)

Since the density variation is not small, the flow cannot be regarded as incompressible. Therefore set the flow to be weakly compressible.

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. From the **Selection** list, choose **Plasma**.

5 Click to expand the **Equation** section. From the **Equation form** list, choose **Stationary**.

*Inlet 1*

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

Add the inlets with their proper velocities.

2 Select Boundary 2 only.

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

4 In the $U_0$ text field, type `v1`.

*Inlet 2*

1 In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.
2 Select Boundary 8 only.
3 In the **Settings** window for **Inlet**, locate the **Velocity** section.
4 In the $U_0$ text field, type `v2`.

*Inlet 3*

1 In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.
2 Select Boundary 13 only.
3 In the **Settings** window for **Inlet**, locate the **Velocity** section.
4 In the $U_0$ text field, type `v3`.

*Outlet 1*

1 In the **Physics** toolbar, click **Boundaries** and choose **Outlet**.
2 Select Boundary 3 only.
3 In the **Settings** window for **Outlet**, locate the **Pressure Conditions** section.
4 Clear the **Suppress backflow** check box.

## MESH 1

*Size*

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

*Size 1*

1 In the **Settings** window for **Size**, locate the **Element Size** section.
2 From the **Predefined** list, choose **Extra fine**.

*Edge 1*

1 In the **Mesh** toolbar, click **Edge**.
2 Drag and drop below **Size**.
3 Select Boundaries 2, 8, and 13 only.

*Size 1*

1 Right-click **Edge 1** and choose **Size**.
2 In the **Settings** window for **Size**, locate the **Element Size** section.
3 Click the **Custom** button.

4 Click to collapse the **Element Size Parameters** section. Click to expand the **Element Size Parameters** section.

5 Select the **Maximum element size** check box. In the associated text field, type `0.5`.

*Size 2*

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size 2**.

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

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

*Boundary Layers 2*

1 In the **Mesh** toolbar, click **Boundary Layers**.

2 Right-click **Boundary Layers 2** and choose **Move Up**.

3 In the **Settings** window for **Boundary Layers**, locate the **Domain Selection** section.

4 From the **Geometric entity level** list, choose **Domain**.

5 Select Domains 6–8 only.

*Boundary Layer Properties*

1 In the **Model Builder** window, click **Boundary Layer Properties**.

2 Select Boundaries 21–32 only.

3 In the **Settings** window for **Boundary Layer Properties**, locate the **Layers** section.

4 In the **Number of layers** text field, type `4`.

5 From the **Thickness specification** list, choose **First layer**.

6 In the **Thickness** text field, type `8[um]`.

7 Click **Build All**.

For this case it is better to solve the equation fully coupled. Some settings in the solver need to be changed to increase stability.

## STUDY 1

*Step 1: Frequency-Stationary*

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

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

3 In the **Frequency** text field, type `f0`.

*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 `1.2`.

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 **Study** toolbar, click **Compute**.

## RESULTS

*Temperature, 2D*

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, 2D` in the **Label** text field.

*Surface 1*

1 Right-click **Temperature, 2D** and choose **Surface**.

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

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

4 In the **Temperature, 2D** toolbar, click **Plot**.

*Temperature, 2D*

In the **Model Builder** window, right-click **Temperature, 2D** and choose **Move Up**.

*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 `Conductivity` in the **Label** text field.

*Surface 1*

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

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

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

*Selection 1*

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

**2** Select Domain 1 only.

**3** In the **Conductivity** toolbar, click **Plot**.

*Conductivity*

In the **Model Builder** window, under **Results** right-click **Conductivity** and choose **Move Up**.