Created in COMSOL Multiphysics 5.5

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 Inductively Coupled Discharge interface to simulate the plasma generated in an inductively coupled plasma torch.

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

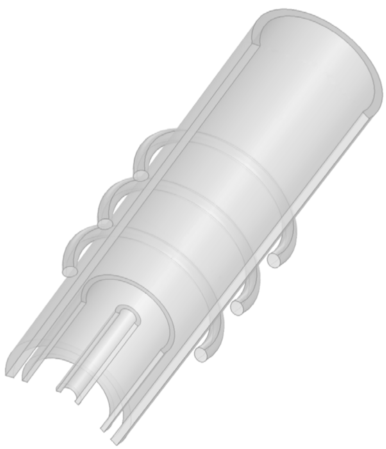

Figure 1: Geometry of an inductively coupled plasma torch. The torch is composed of three concentric quartz tubes in which gas are injected from the bottom and exit from the top the torch. In this model, a fixed power of 11 kW is transferred to the plasma by a three-turn coil operating at 3MHz.

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 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.
- Viscous dissipation and pressure work in the energy equation are neglected.

Figure 2 shows the geometry of the model.

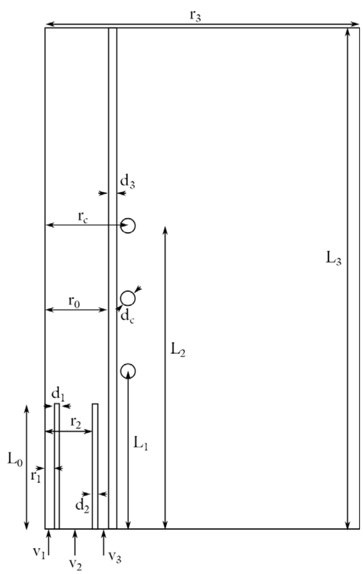

Figure 2: Schematic of the ICP torch. Flow enters from the base (v1, v2 and v3) and leaves out the top. The dimensions of the different part of the model are given in the Modeling Instructions section.

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-transient 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. Steady state is reached when the coil current stabilized to its nominal value.

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 has been used to initiate the plasma. The latter has been set to 1 S/m.

Results and Discussion

Figure 3 and Figure 4, respectively, shows the plasma temperature distribution and velocity magnitude of the argon plasma after 0.3 s. Figure 5 shows the electrical conductivity of the plasma at the same time (0.3 s). Note that, for this figure, the electrical conductivity of the other constituents of the model has been set to 0 for sake of visualization.

Figure 6 displays the magnetic flux norm at steady state (0.3 s). Note that the electrical conductivity of the plasma screens the magnetic flux as a consequence of the skin effect.

Figure 7 shows the coil current as a function of the simulation time. The steady state is reached when the current stabilizes (around t = 0.3 s).

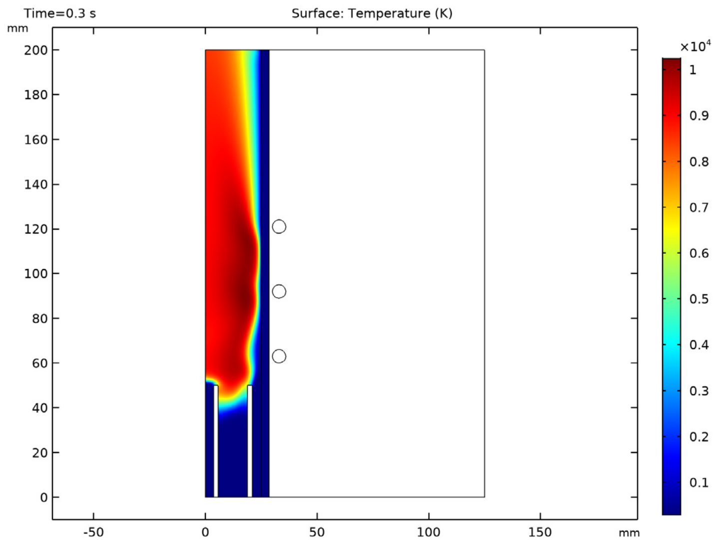

Figure 3: Surface plot of the LTE plasma temperature.

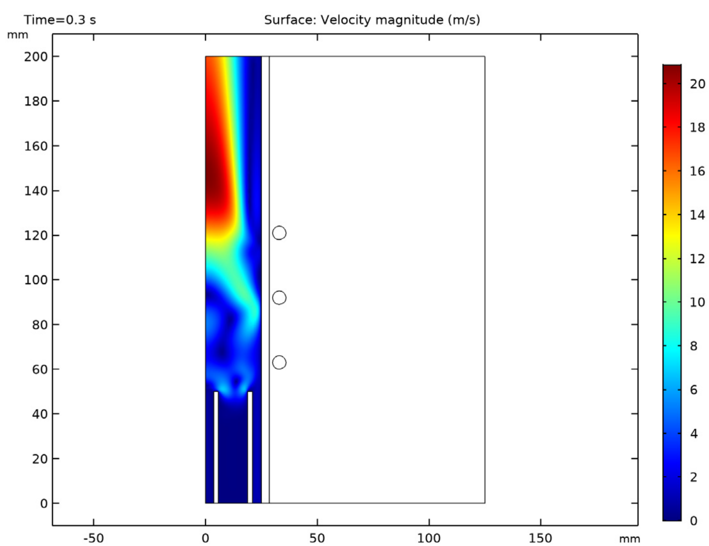

Figure 4: Plot of the velocity magnitude.

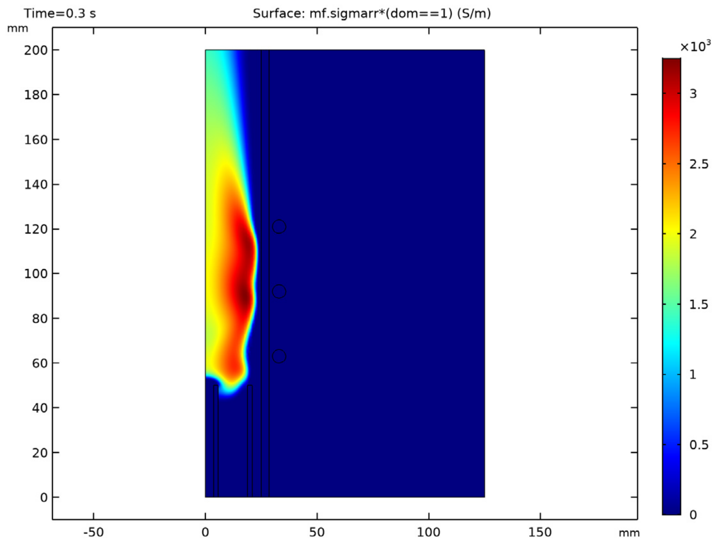

Figure 5: Plot of the plasma electrical conductivity.

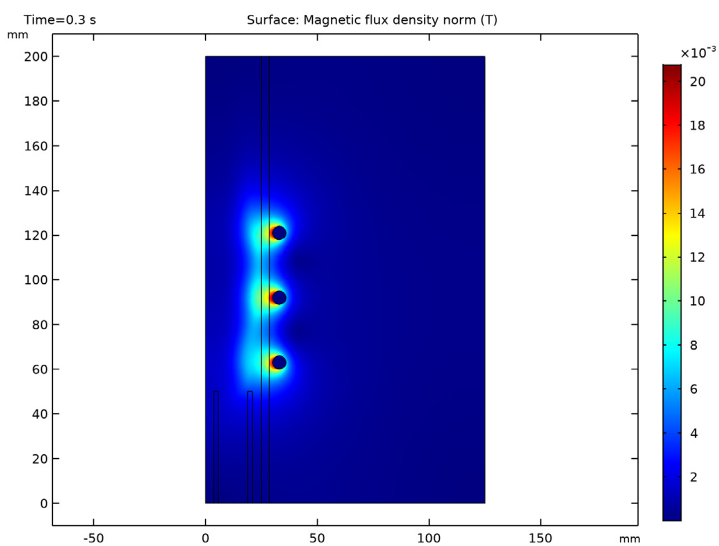

Figure 6: Norm of the magnetic flux. Note the effect of the resistivity on the penetration of the field (skin effect).

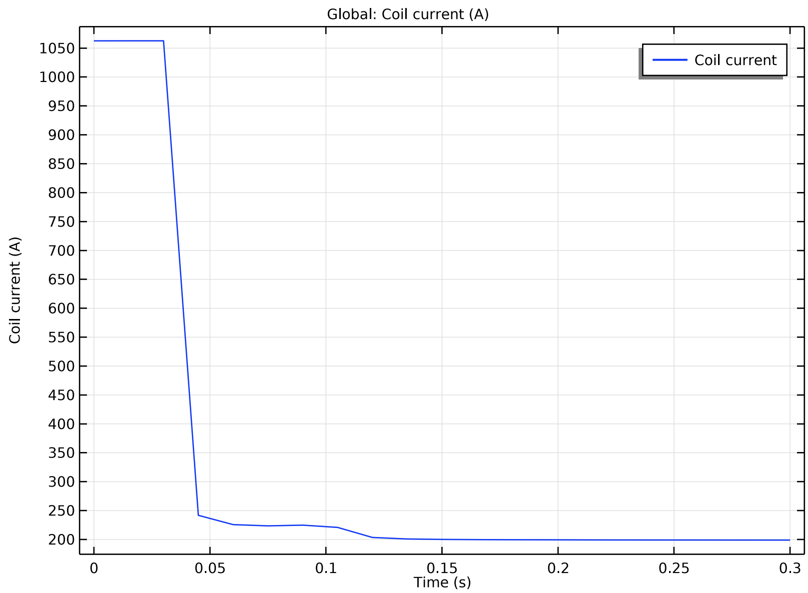

Figure 7: Coil current as a function of time for a fixed excitation power. Note the stabilization of the current density as the system reach the steady state.

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 Inductively Coupled Plasma**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies for Selected Multiphysics>Frequency-Transient**.
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 In the table, enter the following settings:

Name	**Expression**	**Value**	**Description**
T0	300[K]	300 K	Ambient temperature
Pext	11[kW]	11000 W	Coil excitation power
freq	3[MHz]	3E6 Hz	Coil excitation frequency
r_3	125[mm]	0.125 m	Axial length: Computational domain
L_3	200[mm]	0.2 m	Height: Computational domain and sheath tube
d_1	2[mm]	0.002 m	Thickness: Carrier tube
L_0	50[mm]	0.05 m	Height: Carrier tube and central tube
r_1	3.7[mm]	0.0037 m	Inner radius: Carrier tube
d_2	2.2[mm]	0.0022 m	Thickness: Central tube
r_2	18.8[mm]	0.0188 m	Inner radius: Central tube
d_3	3.5[mm]	0.0035 m	Thickness: Sheath tube
r_0	25[mm]	0.025 m	Inner radius: Sheath tube
d_c	6[mm]	0.006 m	Diameter: Coils
r_c	33[mm]	0.033 m	Axial length: Center of the coils
L_1	63[mm]	0.063 m	Height: Center of the lower coil
L_2	121[mm]	0.121 m	Height: Center of the upper coil
Q_1	1[l/min]	1.6667E-5 m^3/s	Gas stream: Carrier tube
Q_2	3[l/min]	5E-5 m^3/s	Gas stream: Central tube
Q_3	31[l/min]	5.1667E-4 m^3/s	Gas stream: Sheath tube
M	0.04[kg/mole]	0.04 kg/mol	Molar mass: Argon

Name	Expression	Value	Description
mv_stp	22.4[l/mole]	0.0224 m^3/mol	Molar volume at stp
mdot1	M*Q_1/mv_stp	2.9762E-5 kg/s	Mass flow rate: Carrier tube
mdot2	M*Q_2/mv_stp	8.9286E-5 kg/s	Mass flow rate: Central tube
mdot3	M*Q_3/mv_stp	9.2262E-4 kg/s	Mass flow rate: Sheath tube
rho_stp	1.91[kg/m^3]	1.91 kg/m^3	Density of argon at stp
A1	pi*(r_1)^2	4.3008E-5 m^2	Cross section: Carrier gas stream
A2	pi*(r_2^2-(r_1+d_1)^2)	0.0010083 m^2	Cross section: Central gas stream
A3	pi*(r_0^2-(r_2+d_2)^2)	5.7805E-4 m^2	Cross section: Sheath gas stream
v1	mdot1/rho_stp/A1	0.3623 m/s	Velocity: Carrier gas stream
v2	mdot2/rho_stp/A2	0.046362 m/s	Velocity: Central gas stream
v3	mdot3/rho_stp/A3	0.83564 m/s	Velocity: Sheath gas stream

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

Explicit 1

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

2 Right-click **Explicit 1** and choose **Rename**.

3 In the **Rename Explicit** dialog box, type `Air` in the **New label** text field.

4 Click **OK**.

5 Select Domain 5 only.

Explicit 2

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

2 In the **Model Builder** window, right-click **Explicit 2** and choose **Rename**.

3 In the **Rename Explicit** dialog box, type `Plasma` in the **New label** text field.

4 Click **OK**.

5 Select Domain 1 only.

Explicit 3

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

2 In the **Model Builder** window, right-click **Explicit 3** and choose **Rename**.

3 In the **Rename Explicit** dialog box, type `Quartz` in the **New label** text field.

4 Click **OK**.

5 Select Domains 2–4 only.

Explicit 4

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

2 In the **Model Builder** window, right-click **Explicit 4** and choose **Rename**.

3 In the **Rename Explicit** dialog box, type `Coils` in the **New label** text field.

4 Click **OK**.

5 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-Turn 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

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 `T0`.

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.

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. Add some isotropic diffusion which is initially very high then ramps down to zero after the plasma has ignited.

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**.

6 Click the **Show More Options** button in the **Model Builder** toolbar.

7 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Stabilization**.

8 Click **OK**.

9 In the **Model Builder** window, click **Laminar Flow (spf)**.

10 In the **Settings** window for **Laminar Flow**, click to expand the **Inconsistent Stabilization** section.

11 Find the **Navier-Stokes equations** subsection. Select the **Isotropic diffusion** check box.

12 In the δ_{id} text field, type `2*(1-tanh(100*(t[1/s]-0.08)))`.

Since the equation form for laminar flow is stationary and the study to resolve is frequency-transient, deactivate the Pseudo-time-stepping in Advanced settings.

13 Click the **Show More Options** button in the **Model Builder** toolbar.

14 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Advanced Physics Options**.

15 Click **OK**.

16 In the **Settings** window for **Laminar Flow**, click to expand the **Advanced Settings** section.

17 Find the **Pseudo time stepping** subsection. From the **Use pseudo time stepping for stationary equation form** list, choose **Off**.

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 **Model Builder** window, right-click **Mesh 1** and choose **More Operations>Edge**.

2 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. Select the **Maximum element size** check box.

5 In the associated text field, type `0.5`.

Size 2

1 In the **Model Builder** window, 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 **Model Builder** window, right-click **Mesh 1** and choose **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 **Boundary Layer Properties** section.

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

5 From the **Thickness of first layer** list, choose **Manual**.

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

7 Click **Build All**.

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)

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

Step 1: Frequency-Transient

1 In the **Model Builder** window, click **Step 1: Frequency-Transient**.

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

3 In the **Times** text field, type `range(0,0.05,1)*0.3`.

4 In the **Frequency** text field, type `freq`.

Solver Configurations

In the **Model Builder** window, expand the **Study 1>Solver Configurations** node.

Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Time-Dependent Solver 1>Segregated 1** node, then click **Velocity u, Pressure p**.

2 In the **Settings** window for **Segregated Step**, click to expand the **Method and Termination** section.

3 From the **Jacobian update** list, choose **On every iteration**.

To improve the robustness of the solver, add temperature to the segregated step of velocity and pressure.

4 Locate the **General** section. Under **Variables**, click **Add**.

5 In the **Add** dialog box, select **Temperature (comp1.T)** in the **Variables** list.

6 Click **OK**.

7 In the **Settings** window for **Segregated Step**, type `Velocity u, Pressure p, Temperature T` in the **Label** text field.

8 In the **Model Builder** window, right-click **Heat transfer** and choose **Disable**.

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

Create some relevant figures.

The temperature.

RESULTS

2D Plot Group 1

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

2 Right-click **2D Plot Group 1** and choose **Rename**.

3 In the **Rename 2D Plot Group** dialog box, type `Temperature` in the **New label** text field.

4 Click **OK**.

Surface 1

1 Right-click **Temperature** 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** toolbar, click **Plot**.

Duplicate the figure to display the fluid velocity magnitude.

Temperature 1

1 In the **Model Builder** window, right-click **Temperature** and choose **Duplicate**.

2 Right-click **Temperature 1** and choose **Rename**.

3 In the **Rename 2D Plot Group** dialog box, type `Velocity` in the **New label** text field.

4 Click **OK**.

Surface 1

1 In the **Model Builder** window, expand the **Results>Velocity** node, then click **Surface 1**.

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**.

Duplicate the figure to display the electrical conductivity.

Velocity 1

1 In the **Model Builder** window, right-click **Velocity** and choose **Duplicate**.

2 Right-click **Velocity 1** and choose **Rename**.

3 In the **Rename 2D Plot Group** dialog box, type `Electrical conductivity` in the **New label** text field.

4 Click **OK**.

Surface 1

1 In the **Model Builder** window, expand the **Results>Electrical conductivity** node, then click **Surface 1**.
2 In the **Settings** window for **Surface**, locate the **Expression** section.
3 In the **Expression** text field, type `mf.sigmarr*(dom==1)`.
4 In the **Electrical conductivity** toolbar, click **Plot**.

Duplicate the figure to display the norm of the magnetic flux. Note the effect of the plasma conductivity on the skin depth.

Electrical conductivity 1

1 In the **Model Builder** window, right-click **Electrical conductivity** and choose **Duplicate**.
2 Right-click **Electrical conductivity 1** and choose **Rename**.
3 In the **Rename 2D Plot Group** dialog box, type `Magnetic flux` in the **New label** text field.
4 Click **OK**.

Surface 1

1 In the **Model Builder** window, expand the **Results>Magnetic flux** node, then click **Surface 1**.
2 In the **Settings** window for **Surface**, locate the **Expression** section.
3 In the **Expression** text field, type `mf.normB`.
4 In the **Magnetic flux** toolbar, click **Plot**.

Display the coil current as a function of time. Note the time it takes to get the steady state (constant current in the coils).

1D Plot Group 5

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2 Right-click **1D Plot Group 5** and choose **Rename**.
3 In the **Rename 1D Plot Group** dialog box, type `Coil current` in the **New label** text field.
4 Click **OK**.

Global 1

1 Right-click **Coil current** 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
mf.ICoil_1	A	Coil current

4 In the **Coil current** toolbar, click **Plot**.

Create a nice 3D plot for the model thumbnail.

Create first a revolution data set.

Revolution 2D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Revolution 2D**.
2 In the **Settings** window for **Revolution 2D**, click to expand the **Revolution Layers** section.
3 In the **Start angle** text field, type `-90`.
4 In the **Revolution angle** text field, type `225`.

Then create the 3D plot.

3D Plot Group 6

1 In the **Results** toolbar, click **3D Plot Group**.
2 Right-click **3D Plot Group 6** and choose **Rename**.
3 In the **Rename 3D Plot Group** dialog box, type `Temperature 3D` in the **New label** text field.
4 Click **OK**.

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 In the **Temperature 3D** toolbar, click **Plot**.
5 Click the **Go to Default View** button in the **Graphics** toolbar.

Set the figure as a model thumbnail by clicking on the root folder in the model builder than expand the model thumbnail section and click on set model thumbnail.