Created in COMSOL Multiphysics 6.0

Thermal Plasma

Introduction

Low pressure discharges are characterized by the fact that the electron temperature is much higher than the neutral gas temperature. As the gas pressure increases, the number of collisions between the electrons and neutrals increases. At high enough pressures the electron temperature becomes equal to the gas temperature. At this point the plasma is in local thermodynamic equilibrium and a much simpler MHD model can be used to model the plasma.

This model simulates a plasma at medium pressure (1 torr), where the gas temperature cannot be assumed to be constant but the plasma is still not in local thermodynamic equilibrium. In Figure 1 the electron (blue) and gas (black) temperatures are plotted as a function of pressure. At low pressures the two temperatures are decoupled, but as the pressure increases the temperatures tend toward the same limit. There are no axes on the plot since the exact temperature and pressure depend strongly on the gas in question.

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

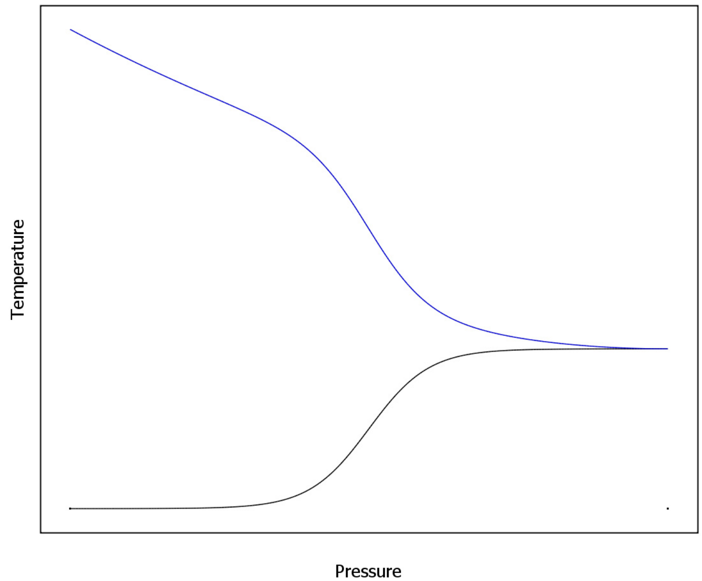

Figure 1: Plot of electron (blue) and gas (black) temperature vs. pressure. At higher pressures the two temperatures become equal.

Model Definition

The electron density and mean electron energy are computed by solving a pair of drift-diffusion equations for the electron density and mean electron energy. For detailed information on electron transport, see *Theory for the Drift Diffusion Interface* in the *Plasma Module User's Guide*.

$$\frac{\partial}{\partial t}(n_e) + \nabla \cdot [-n_e(\mu_e \bullet \mathbf{E}) - \mathbf{D}_e \bullet \nabla n_e] = R_e$$

$$\frac{\partial}{\partial t}(n_\varepsilon) + \nabla \cdot [-n_\varepsilon(\mu_\varepsilon \bullet \mathbf{E}) - \mathbf{D}_\varepsilon \bullet \nabla n_\varepsilon] + \mathbf{E} \cdot \Gamma_e = R_\varepsilon$$

The electron source R_e and the energy loss due to inelastic collisions R_ε are defined later. The electron diffusivity, energy mobility, and energy diffusivity are computed from the electron mobility using:

$$\mathbf{D}_e = \mu_e T_e, \mu_\varepsilon = \left(\frac{5}{3}\right)\mu_e, \mathbf{D}_\varepsilon = \mu_\varepsilon T_e$$

The source coefficients in the above equations are determined by the plasma chemistry using rate coefficients. In the case of rate coefficients, the electron source term is given by:

$$R_e = \sum_{j=1}^{M} x_j k_j N_n n_e$$

where x_j is the mole fraction of the target species for reaction j, k_j is the rate coefficient for reaction j (SI unit: m^3/s), and N_n is the total neutral number density (SI unit: $1/\mathrm{m}^3$). The electron energy loss is obtained by summing the collisional energy loss over all reactions:

$$R_\varepsilon = \sum_{j=1}^{P} x_j k_j N_n n_e \Delta\varepsilon_j$$

where $\Delta\varepsilon_j$ is the energy loss from reaction j (SI unit: V). The rate coefficients can be computed from cross section data by the following integral:

$$k_k = \gamma \int_0^\infty \varepsilon \sigma_k(\varepsilon) f(\varepsilon) d\varepsilon$$

where $\gamma = (2q/m_e)^{1/2}$ (SI unit: $\mathrm{C}^{1/2}/\mathrm{kg}^{1/2}$), m_e is the electron mass (SI unit: kg), ε is energy (SI unit: V), σ_k is the collision cross section (SI unit: m^2), and f is the electron energy distribution function. In this case, a Maxwellian EEDF is assumed.

For nonelectron species, the following equation is solved for the mass fraction of each species. For detailed information on the transport of the nonelectron species, see *Theory for the Heavy Species Transport Interface* in the *Plasma Module User's Guide*.

$$\rho \frac{\partial}{\partial t}(w_k) + \rho(\mathbf{u} \cdot \nabla) w_k = \nabla \cdot \mathbf{j}_k + R_k$$

The electrostatic field is computed using the following equation:

$$-\nabla \cdot \varepsilon_0 \varepsilon_\mathrm{r} \nabla V = \rho$$

The space charge density ρ is automatically computed based on the plasma chemistry specified in the model using the formula:

$$\rho = q\left(\sum_{k=1}^{N} Z_k n_k - n_e\right)$$

For detailed information about electrostatics see *Theory for the Electrostatics Interface* in the *Plasma Module User's Guide*.

PLASMA CHEMISTRY

Argon is one of the simplest mechanisms to implement at low pressures. The electronically excited states can be lumped into a single species which results in a chemical mechanism consisting of only 3 species and 7 reactions (electron impact cross-sections are obtained from Ref. 2):

TABLE 1: TABLE OF COLLISIONS AND REACTIONS MODELED.

REACTION	FORMULA	TYPE	$\Delta\varepsilon(\mathrm{eV})$
1	e+Ar=>e+Ar	Elastic	0
2	e+Ar=>e+Ars	Excitation	11.5
3	e+Ars=>e+Ar	Superelastic	-11.5
4	e+Ar=>2e+Ar+	Ionization	15.8
5	e+Ars=>2e+Ar+	Ionization	4.24
6	Ars+Ars=>e+Ar+Ar+	Penning ionization	-
7	Ars+Ar=>Ar+Ar	Metastable quenching	-

Stepwise ionization (reaction 5) can play an important role in sustaining low pressure argon discharges. Excited argon atoms are consumed via superelastic collisions with electrons, quenching with neutral argon atoms, ionization or Penning ionization where two metastable argon atoms react to form a neutral argon atom, an argon ion and an electron. Reaction number 1, elastic collisions with electrons is primarily responsible for heating of the gas. In addition to volumetric reactions, the following surface reactions are implemented:

TABLE 2: TABLE OF SURFACE REACTIONS.

REACTION	FORMULA	STICKING COEFFICIENT
1	Ars=>Ar	1
2	Ar+=>Ar	1

When a metastable argon atom makes contact with the wall, it reverts to the ground state argon atom with some probability (the sticking coefficient).

ELECTRICAL EXCITATION

The reactor geometry is simply a cylindrical glass tube with a 4 turn coil wrapped around it. Gas flows in from the bottom and exits out of the top. The gas is heated through elastic and inelastic collisions. A fixed power of 700 W is applied to the coil.

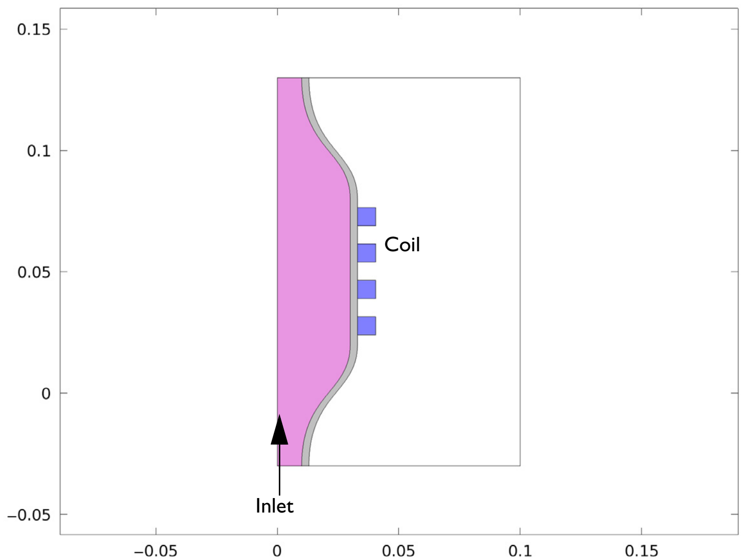

Figure 2: Schematic of the ICP reactor. Flow enters from the base and leaves out the top.

NOTES ON COUPLINGS

In this model, two strategies are used to couple the different physics interfaces: (i) using dedicated Multiphysics coupling features; and (ii) transferring physical quantities between the interfaces by choosing appropriate options available in **Plasma Model** (from the **Plasma** interface), **Fluid Properties** (from the **Laminar Flow** interface), and **Fluid** (from the **Heat Transfer in Fluid** interface).

The Multiphysics coupling features are responsible to heat the electrons in the plasma. The **Plasma Conductivity Coupling** feature computes a plasma conductivity using the cold plasma approximation in the frequency domain. This conductivity is used in the **Magnetic Fields** interface that computes a Joule heating term that is added to the electron energy equation in **Electron Heating Source**.

In the second way of coupling the interfaces, fluid and thermodynamic properties computed in the **Plasma** interface are used in the **Laminar Flow** and **Heat Transfer in Fluid** interfaces. These quantities are the thermal conductivity, density, heat capacity, and dynamic viscosity and the details about the theory can be found in the section **Theory for the Heavy Species Transport Interface** (subsections **Thermodynamic Properties** and **Mixture Transport Properties**) of the **Plasma Module User's Guide**. The **Laminar Flow** and **Heat Transfer in Fluid** interfaces provide the fluid velocity, temperature, and pressure that are set in the **Plasma Model** feature.

The gas heating in this model is set using the **Heat Source** feature that uses an available variable computed in the **Plasma** interface (`plas.Qgas`). This heating term is computed as explained in the section **Theory for the Heavy Species Transport Interface** (subsections **Thermodynamic Properties**) of the **Plasma Module User's Guide**. Briefly, this heating term includes the energy transfer to the heavy species (gain or lost) involved in all reactions. In systems where there are strong DC electric fields the ions can heat the gas considerably. This mechanism is not included in the above described heating mechanism.

STRATEGY TO SETUP A MODEL WITH INLETS AND OUTLETS

In this model, an inlet or argon is added in the system. There is also an outlet responsible to fix the system pressure. After setting the **Inlet** and **Outlet** features in the **Laminar Flow** interface the following reasoning should be used to set up the **Inlet** and **Outflow** sub-features of the species in the **Plasma** interface:

- If a species is set **From mass constraint** it does not need any **Inlet** and **Outflow** sub feature since its mass fraction is computed so that the mass in the system is conserved. There are no dependent variables being solved for the mass constraint species and as such does not need the boundary conditions set by the sub-features.
- If the feed into the system contains another species (the present model does not have) one of these species should be set as mass constraint and the other should use an **Inlet** sub feature that specifies the appropriate proportion. Consider as an example of a feed of Ar/O2 at 60/40 mole fraction. If Ar is set as mass constrain than O2 should have a **Inlet** sub feature with a mole fraction of 0.4.
- If the species in created inside the reactor (as are Ar+ and Ars) no **Inlet** sub-feature should be added.
- If the species flows out of the system (as are Ar+ and Ars) an **Outflow** sub feature should be added.

Results

Results of the different quantities computed in this model are presented below. The electron density reaches high values while the electron temperature is kept very low meaning that electron losses by transport are small. The gas temperature in the system peaks to 1300 K showing that there is a considerable amount of energy transfered from the electrons to the heavy species that is not lost in transport. More that 90% of the heating of the gas is caused by electron impact elastic collisions defined in reaction 1.

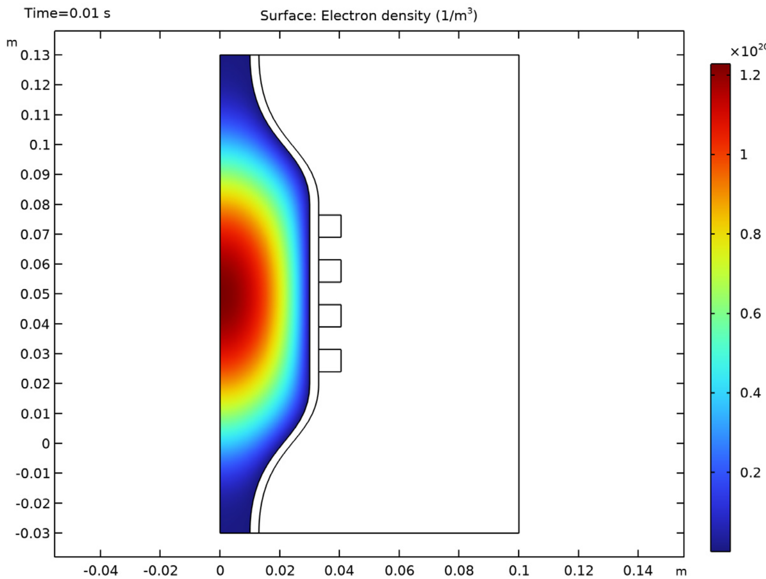

Figure 3: Surface plot of electron density inside the column.

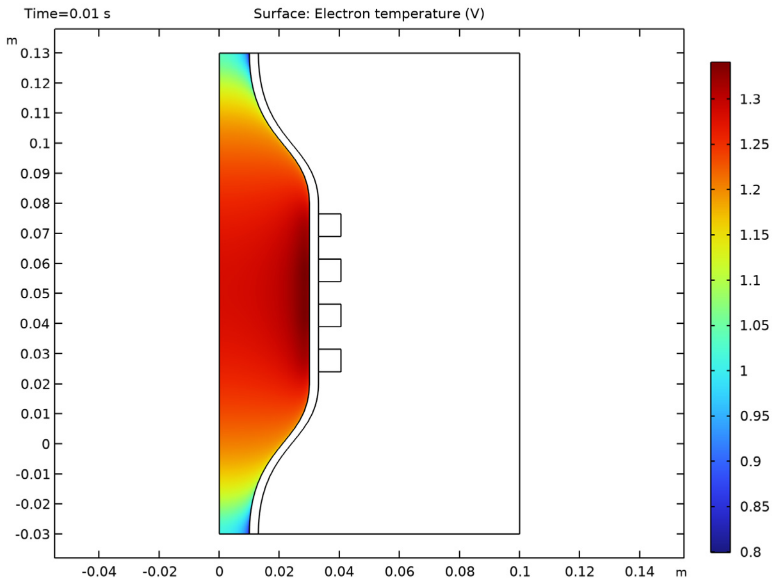

Figure 4: Plot of the electron temperature inside the plasma source.

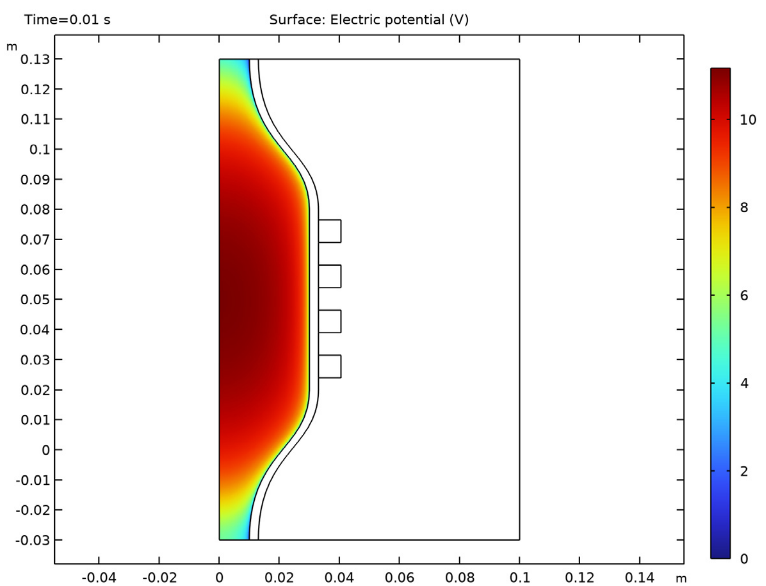

Figure 5: Plot of the plasma potential inside the plasma source.

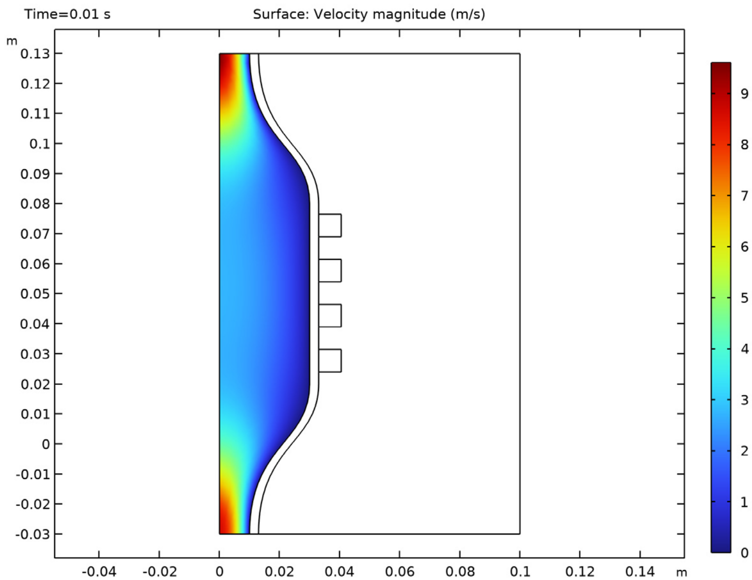

Figure 6: Plot of the velocity field.

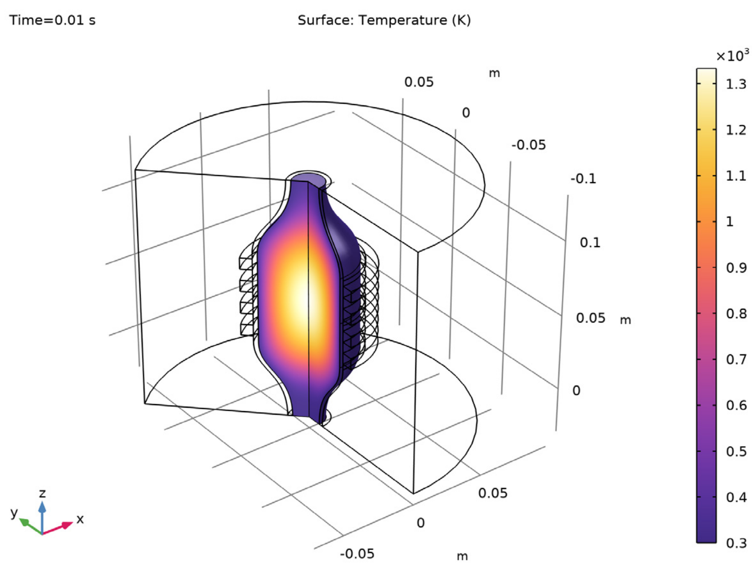

Figure 7: Revolved plot of the temperature inside the plasma source.

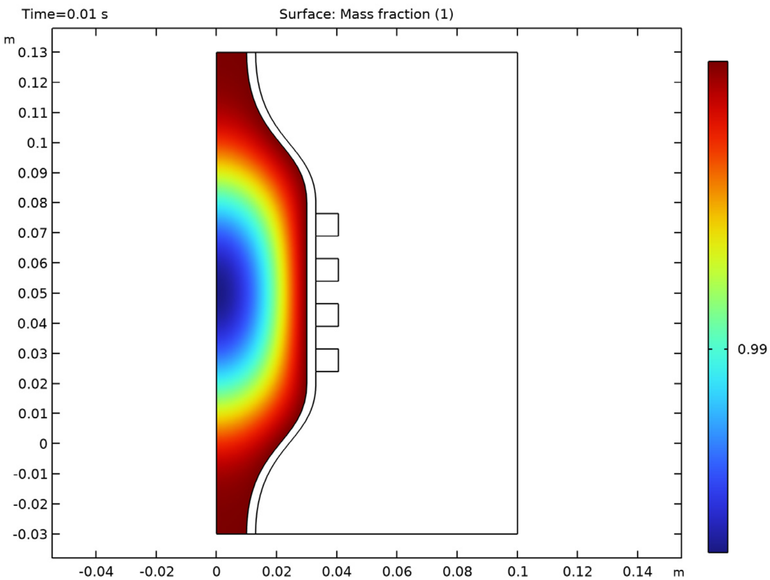

Figure 8: Plot of the mass fraction of ground state argon.

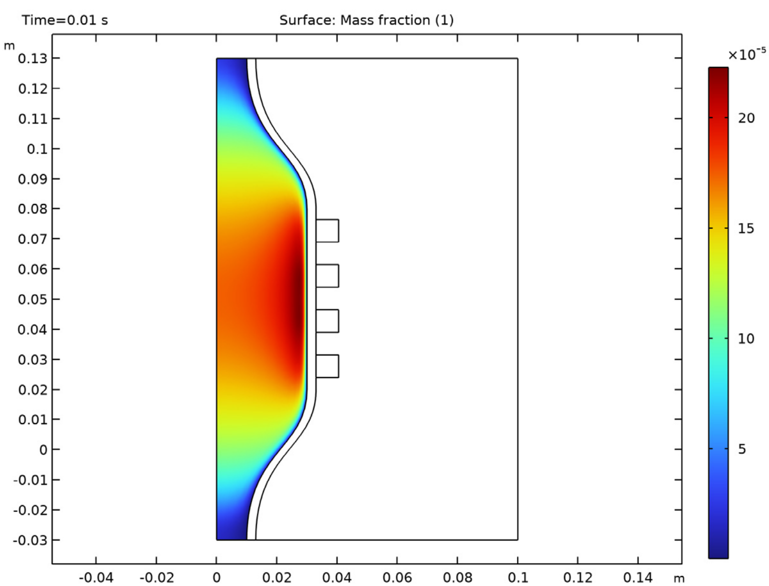

Figure 9: Plot of the mass fraction of electronically excited argon atoms.

Reference

1. M.A. Lieberman and A.J. Lichtenberg, *Principles of Plasma Discharges and Materials Processing*, John Wiley & Sons, 2005.

2. Phelps database, www.lxcat.net, retrieved 2017.

Application Library path: `Plasma_Module/Inductively_Coupled_Plasmas/thermal_plasma`

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>Inductively Coupled Plasma**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Fluid Flow>Single-Phase Flow>Laminar Flow (spf)**.

5 Click **Add**.

6 In the **Select Physics** tree, select **Heat Transfer>Heat Transfer in Fluids (ht)**.

7 Click **Add**.

8 Click **Study**.

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

10 Click **Done**.

GEOMETRY 1

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

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

3 In the tree, select the check box for the node **Physics>Stabilization**.

4 Click **OK**.

Import 1 (imp1)

1 In the **Home** toolbar, click **Import**.

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

3 Click **Browse**.

4 Browse to the model's Application Libraries folder and double-click the file `thermal_plasma.mphbin`.

5 Click **Import**.

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
p0	1[torr]	133.32 Pa	Initial and outlet pressure

DEFINITIONS

Plasma

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

2 Select Domain 1 only.

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

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

5 Click **OK**.

Walls

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

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 9, 10, 34, and 35 only.

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

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

7 Click **OK**.

Outlet

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

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 4 only.

5 Right-click **Explicit 3** and choose **Rename**.

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

7 Click **OK**.

Coil Walls

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

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 12, 13, 15–17, 19, 21, 22, 24–26, and 28–32 only.

5 Right-click **Explicit 4** and choose **Rename**.

6 In the **Rename Explicit** dialog box, type `Coil Walls` in the **New label** text field.

7 Click **OK**.

Start by importing the cross sections for argon and by activating the convection and thermodynamic property evaluation.

PLASMA (PLAS)

1 In the **Settings** window for **Plasma**, locate the **Domain Selection** section.

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

Cross Section Import 1

1 Right-click **Component 1 (comp1)>Plasma (plas)** and choose **Global>Cross Section Import**.

2 In the **Settings** window for **Cross Section Import**, locate the **Cross Section Import** section.

3 Click **Browse**.

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

5 Click **Import**.

6 In the **Model Builder** window, click **Plasma (plas)**.

7 In the **Settings** window for **Plasma**, locate the **Transport Settings** section.

8 Find the **Include** subsection. Select the **Convection** check box.

9 Select the **Calculate thermodynamic properties** check box.

10 Locate the **Plasma Properties** section. Select the **Use reduced electron transport properties** check box.

Reaction 1

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

2 In the **Settings** window for **Reaction**, locate the **Reaction Formula** section.

3 In the **Formula** text field, type `Ars+Ars=>e+Ar+Ar+`.

4 Locate the **Reaction Parameters** section. In the k^{f} text field, type `3.734E8`.

Reaction 2

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

2 In the **Settings** window for **Reaction**, locate the **Reaction Formula** section.

3 In the **Formula** text field, type `Ars+Ar=>Ar+Ar`.

4 Locate the **Reaction Parameters** section. In the k^{f} text field, type `1807`.

Ground 1

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

2 In the **Settings** window for **Ground**, locate the **Boundary Selection** section.

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

Surface Reaction 1

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

2 In the **Settings** window for **Surface Reaction**, locate the **Boundary Selection** section.

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

4 Locate the **Reaction Formula** section. In the **Formula** text field, type `Ar+=>Ar`.

Surface Reaction 2

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

2 In the **Settings** window for **Surface Reaction**, locate the **Boundary Selection** section.

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

4 Locate the **Reaction Formula** section. In the **Formula** text field, type `Ars=>Ar`.

Wall 1

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

2 In the **Settings** window for **Wall**, locate the **Boundary Selection** section.

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

Species: Ars

In the **Model Builder** window, click **Species: Ars**.

Outflow 1

1. In the **Physics** toolbar, click **Attributes** and choose **Outflow**.
2. In the **Settings** window for **Outflow**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Outlet**.

Species: Ar+

In the **Model Builder** window, under **Component 1 (comp1)>Plasma (plas)** click **Species: Ar+**.

Outflow 1

1. In the **Physics** toolbar, click **Attributes** and choose **Outflow**.
2. In the **Settings** window for **Outflow**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Outlet**.

Species: Ar

1. In the **Model Builder** window, under **Component 1 (comp1)>Plasma (plas)** click **Species: Ar**.
2. In the **Settings** window for **Species**, locate the **Species Formula** section.
3. Select the **From mass constraint** check box.
4. Locate the **General Parameters** section. From the **Preset species data** list, choose **Ar**.

Species: Ars

1. In the **Model Builder** window, click **Species: Ars**.
2. In the **Settings** window for **Species**, locate the **General Parameters** section.
3. In the x_0 text field, type `1E-4`.
4. From the **Preset species data** list, choose **Ar**.
5. Click to expand the **Species Thermodynamic Parameters** section. In the Δh text field, type `11.5`.

   The thermodynamic properties for the electronically excited Argon atoms can be the same as for the ground state species plus the threshold energy for the electron impact reaction. In this case this corresponds to an energy of 11.5eV. This is added in the text field **Additional enthalpy contribution**.

Species: Ar+

1. In the **Model Builder** window, click **Species: Ar+**.
2. In the **Settings** window for **Species**, locate the **Species Formula** section.
3. Select the **Initial value from electroneutrality constraint** check box.

4 Locate the **General Parameters** section. From the **Preset species data** list, choose **Ar**.

5 Locate the **Species Thermodynamic Parameters** section. In the Δh text field, type `15.8`.

The thermodynamic properties for the Argon ions can be the same as for the ground state species plus the threshold energy for ionization. In this case this corresponds to an energy of 15.8eV. This is added in the text field **Additional enthalpy contribution**.

You can set the gas temperature and pressure in the plasma model to the computed gas pressure and temperature from other physics interfaces. The velocity field is also set to the velocity field computed from the **Laminar Flow** interface.

Plasma Model 1

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

2 In the **Settings** window for **Plasma Model**, locate the **Model Inputs** section.

3 From the $\mathbf{u}$ list, choose **Velocity field (spf)**.

4 From the T list, choose **Temperature (ht)**.

5 From the p_A list, choose **Absolute pressure (spf)**.

6 Locate the **Electron Density and Energy** section. In the $\mu_e N_n$ text field, type `4E24`.

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 $n_{e,0}$ text field, type `1E15`.

4 In the ε_0 text field, type `3`.

MAGNETIC FIELDS (MF)

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

Coil 1

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

2 Select Domains 4–7 only.

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

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

5 Select the **Coil group** check box.

6 In the P_{coil} text field, type `700[W]`.

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 **Compressible flow (Ma<0.3)**.
4 Click to expand the **Consistent Stabilization** section. Find the **Navier-Stokes equations** subsection. Clear the **Crosswind diffusion** check box.

Define the pressure reference level to be 1 torr.

5 Locate the **Physical Model** section. In the p_{ref} text field, type `p0`.
6 Locate the **Domain Selection** section. Click **Clear Selection**.
7 Select Domain 1 only.
8 Click to expand the **Equation** section. From the **Equation form** list, choose **Stationary**.

Fluid Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Laminar Flow (spf)** click **Fluid Properties 1**.
2 In the **Settings** window for **Fluid Properties**, locate the **Fluid Properties** section.
3 From the ρ list, choose **Density (plas/pes1)**.
4 From the μ list, choose **Dynamic viscosity (plas/pes1)**.

Inlet 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.
2 Select Boundary 2 only.
3 In the **Settings** window for **Inlet**, locate the **Boundary Condition** section.
4 From the list, choose **Mass flow**.
5 Locate the **Mass Flow** section. From the **Mass flow type** list, choose **Standard flow rate (SCCM)**.
6 From the M_n list, choose **Mean molar mass (plas/pes1)**.
7 In the Q_{sccm} text field, type `100*tanh(1E5*t[1/s])`.

Outlet 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Outlet**.
2 Select Boundary 4 only.

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 **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 1 only.

Fluid 1

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

2 In the **Settings** window for **Fluid**, locate the **Heat Convection** section.

3 From the $\mathbf{u}$ list, choose **Velocity field (spf)**.

4 Locate the **Heat Conduction, Fluid** section. From the k list, choose **Thermal conductivity (plas/pes1)**.

5 Locate the **Thermodynamics, Fluid** section. From the **Fluid type** list, choose **Gas/Liquid**.

6 From the ρ list, choose **Density (plas/pes1)**.

7 From the C_p list, choose **Heat capacity at constant pressure (plas/pes1)**.

8 From the γ list, choose **User defined**.

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 300.

Heat Source 1

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

2 Select Domain 1 only.

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

4 From the Q_0 list, choose **Heat source for gas (plas/pes1)**.

Temperature 1

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

2 Select Boundaries 2, 9, 10, 34, and 35 only.

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

4 In the T_0 text field, type 300.

Outflow 1

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

2 Select Boundary 4 only.

MATERIALS

Dielectric

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 Select Domain 2 only.

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

4 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	0	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	4.5	1	Basic

5 Right-click **Material 1 (mat1)** and choose **Rename**.

6 In the **Rename Material** dialog box, type `Dielectric` in the **New label** text field.

7 Click **OK**.

Air

1 In the **Model Builder** window, right-click **Materials** and choose **Blank Material**.

2 Select Domain 3 only.

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

4 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	0	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic

5 Right-click **Material 2 (mat2)** and choose **Rename**.

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

7 Click **OK**.

Copper coil

1 In the **Model Builder** window, right-click **Materials** and choose **Blank Material**.

2 Select Domains 4–7 only.

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

4 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	6E7	S/m	Basic
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	1	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic

5 Right-click **Material 3 (mat3)** and choose **Rename**.

6 In the **Rename Material** dialog box, type `Copper coil` in the **New label** text field.

7 Click **OK**.

A boundary layer mesh is used on the reactor walls so that the region of space charge separation between the ions and electrons can be resolved.

MESH 1

1 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

2 From the **Element size** list, choose **Extra fine**.

Edge 1

1 In the **Mesh** toolbar, click **Edge**.

2 Select Boundary 2 only.

Size 1

1 Right-click **Edge 1** and choose **Size**.

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

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

4 Click to expand the **Element Size Parameters** section. Locate the **Element Size** section. Click the **Custom** button.

5 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

6 In the associated text field, type `0.001`.

Free Triangular 1

1 In the **Mesh** toolbar, click **Free Triangular**.

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

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

4 Select Domain 1 only.

Size 1

1 Right-click **Free Triangular 1** and choose **Size**.

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

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

Boundary Layers 1

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

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

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

4 Select Domain 1 only.

Boundary Layer Properties

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

2 In the **Settings** window for **Boundary Layer Properties**, locate the **Boundary Selection** section.

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

4 Locate the **Layers** section. In the **Number of layers** text field, type `5`.

5 In the **Stretching factor** text field, type `1.5`.

Mapped 1

1 In the **Mesh** toolbar, click **Mapped**.

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

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

4 Select Domains 4–7 only.

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Coil Walls**.

4 Locate the **Distribution** section. From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type `35`.

6 In the **Element ratio** text field, type `8`.

7 From the **Growth rate** list, choose **Exponential**.

8 Select the **Symmetric distribution** check box.

Free Triangular 2

1 In the **Mesh** toolbar, click **Free Triangular**.

2 In the **Settings** window for **Free Triangular**, click **Build All**.

STUDY 1

Step 1: Frequency-Transient

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

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

3 In the **Output times** text field, type `0`.

4 Click **Range**.

5 In the **Range** dialog box, choose **Number of values** from the **Entry method** list.

6 In the **Start** text field, type `-8`.

7 In the **Stop** text field, type `-2`.

8 In the **Number of values** text field, type `21`.

9 From the **Function to apply to all values** list, choose **exp10(x) – Exponential function (base 10)**.

10 Click **Add**.

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

12 In the **Frequency** text field, type `13.56E6`.

13 In the **Home** toolbar, click **Compute**.

RESULTS

Electron Density (plas)

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

Argon Mass Fraction

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

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

Surface 1

1 Right-click **Argon Mass Fraction** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Plasma>Mass fractions>plas.wAr - Mass fraction**.

3 In the **Argon Mass Fraction** toolbar, click **Plot**.

Excited Argon Mass Fraction

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

2 In the **Settings** window for **2D Plot Group**, type `Excited Argon Mass Fraction` in the **Label** text field.

Surface 1

1 Right-click **Excited Argon Mass Fraction** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Plasma>Mass fractions>plas.wArs - Mass fraction**.

3 In the **Excited Argon Mass Fraction** toolbar, click **Plot**.