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 (2 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 towards the same limit. There are no axes on the plot since the exact temperature and pressure depends strongly on the gas in question.

Molecular gases tend to heat more easily than atomic gases for to the following reasons:

- Continual dissociation and recombination of the molecule. An electron impact reaction may dissociate a molecule into its atomic components. The energy lost by the electron when this reaction occurs is given back to the gas in the form of thermal energy during recombination.
- Vibrational excitation and relaxation. The threshold energy for vibrational excitation is much smaller than the threshold energy for dissociation. The continuous vibrational excitation and relaxation of the molecules can cause the gas temperature to increase.

©2012 COMSOL

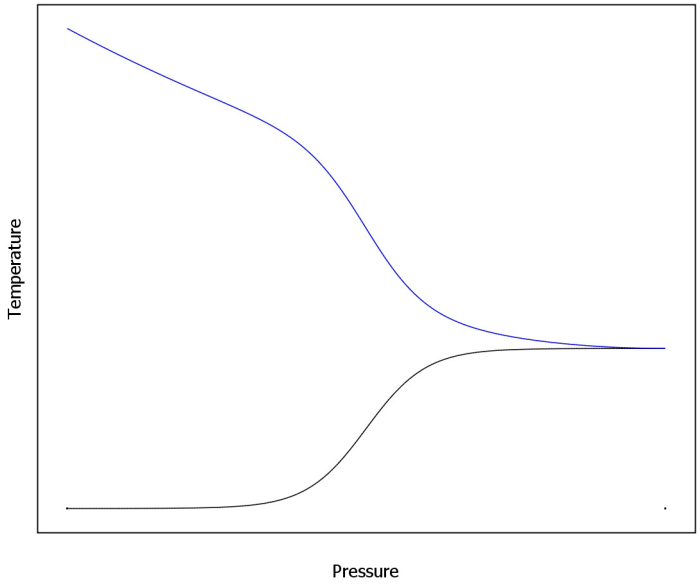

Figure 1: Plot of electron (blue) and gas (black) temperature v.s. pressure. At higher pressures the two temperatures become equal.

Model Definition

DOMAIN EQUATIONS

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. Convection of electrons due to fluid motion is neglected. For detailed information on electron transport see Theory for the Drift Diffusion Interface in the *Plasma Module User's Guide*.

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

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

 ©2012 COMSOL

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:

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

The source coefficients in the above equations are determined by the plasma chemistry using rate coefficients. Suppose that there are M reactions that contribute to the growth or decay of electron density and P inelastic electron-neutral collisions. In general $P >> M$. 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 non-electron species, the following equation is solved for the mass fraction of each species. For detailed information on the transport of the non-electron 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:

$$-\nabla \cdot \varepsilon_0 \varepsilon_r \nabla V = \rho$$

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 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:

TABLE 1: TABLE OF COLLISIONS AND REACTIONS MODELED

REACTION	FORMULA	TYPE	$\Delta\varepsilon$(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 7 is responsible for heating of the gas. The 11.5eV of energy which was consumed in creating the electronically excited argon atom is

©2012 COMSOL

returns to the gas as thermal energy when the excited metastable quenches. 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 will revert 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. The inelastic collisions are responsible for the bulk of the gas heating. A current of 15 Amps is applied to each turn of the coil. Due to the high pressure.

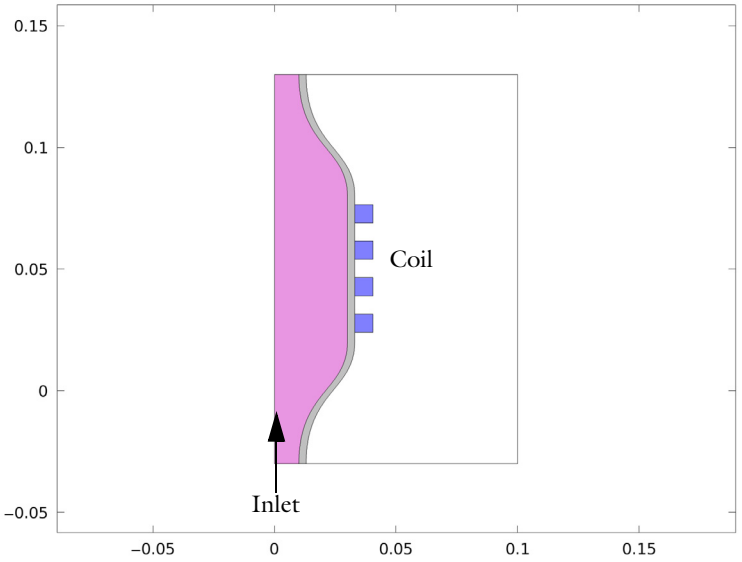

Figure 2: Schematic of the ICP reactor. Flow enters from the base and leaves out the top.

Results and Discussion

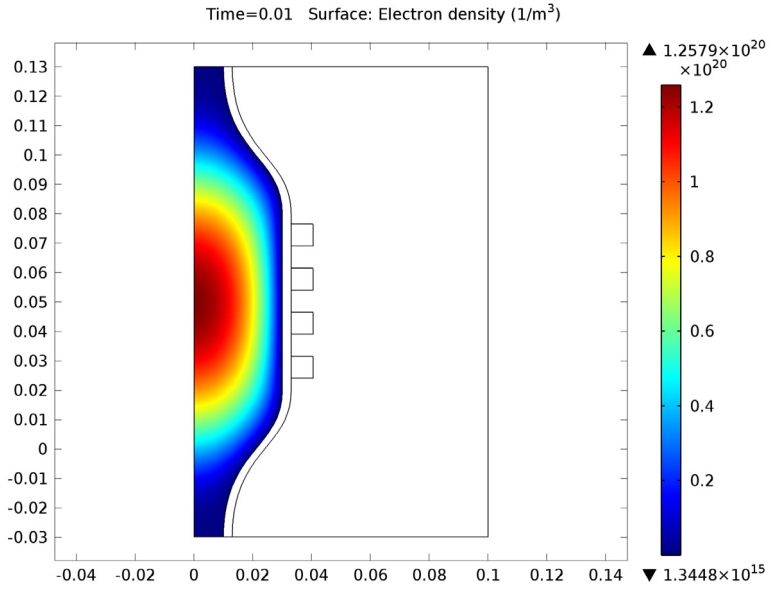

Figure 3: Surface plot of electron density inside the column.

©2012 COMSOL

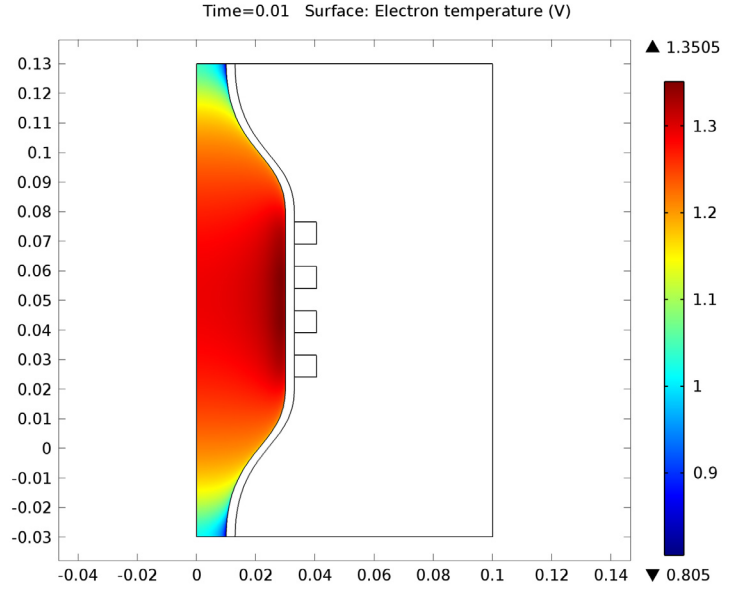

Figure 4: Plot of the electron temperature inside the plasma source.

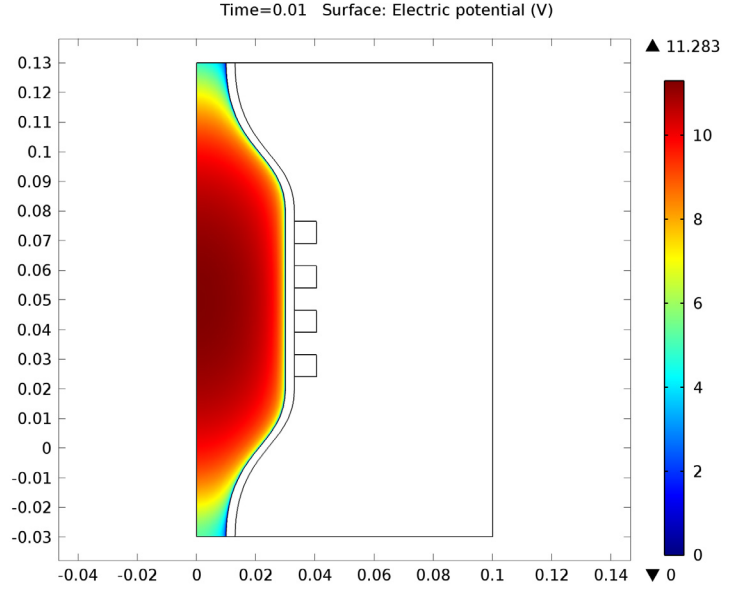

Figure 5: Plot of the plasma potential inside the plasma source.

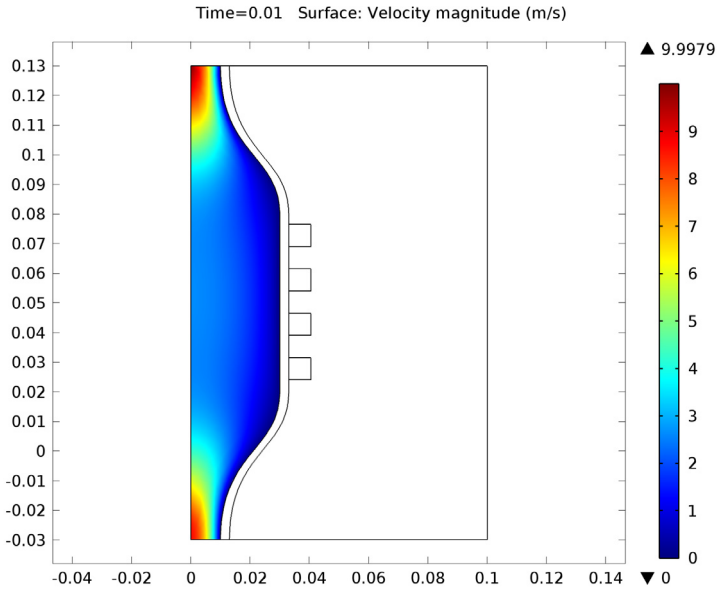

Figure 6: Plot of the velocity field.

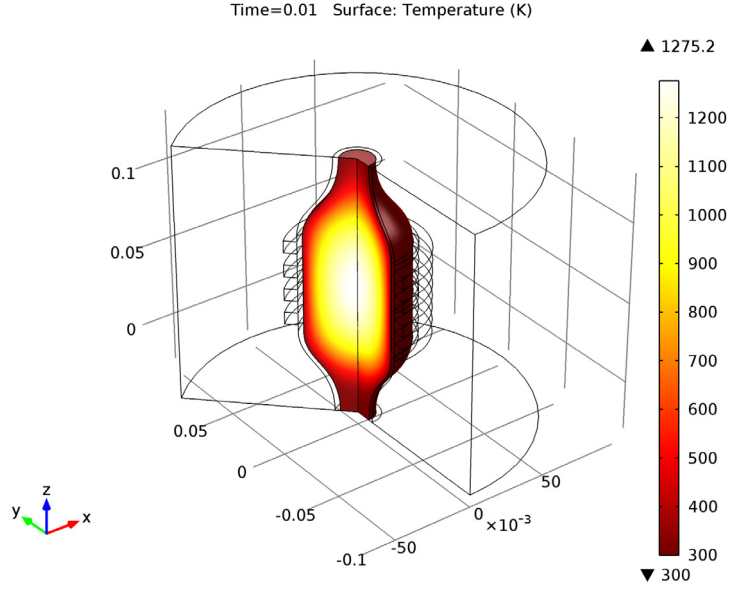

Figure 7: Revolved plot of the temperature inside the plasma source.

©2012 COMSOL

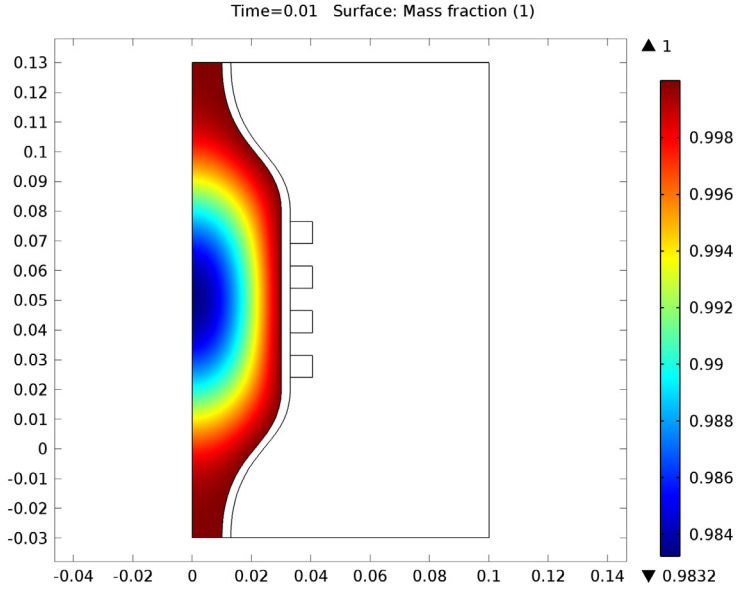

Figure 8: Plot of the mass fraction of ground state argon.

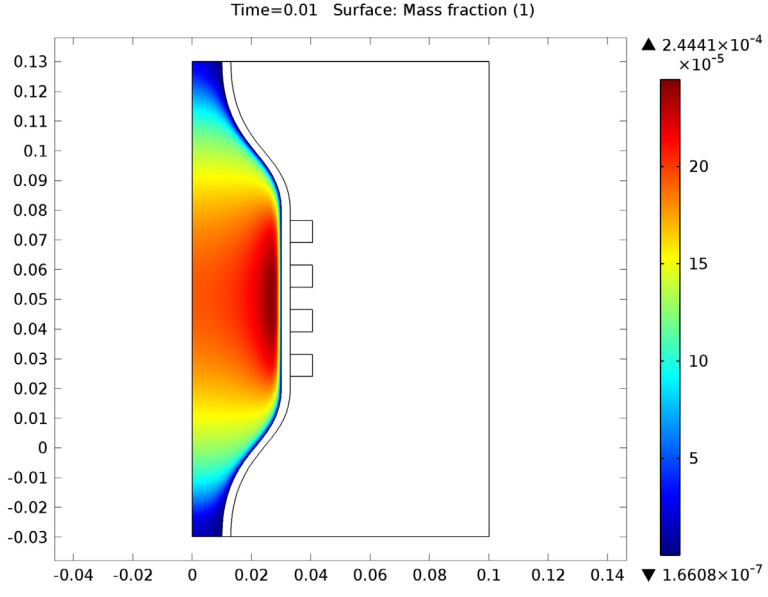

Figure 9: Plot of the mass fraction of electronically excited argon atoms.

©2012 COMSOL

Reference

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

Model Library path: `Plasma_Module/Inductively_Coupled_Plasmas/thermal_plasma`

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click the **2D axisymmetric** button.

3 Click **Next**.

4 In the **Add physics** tree, select **Plasma>Inductively Coupled Plasma (icp)**.

5 Click **Add Selected**.

6 In the **Add physics** tree, select **Fluid Flow>Single-Phase Flow>Laminar Flow (spf)**.

7 Click **Add Selected**.

8 In the **Add physics** tree, select **Heat Transfer>Heat Transfer in Fluids (ht)**.

9 Click **Add Selected**.

10 Click **Next**.

11 Find the **Studies** subsection. In the tree, select **Preset Studies for Selected Physics>Frequency-Transient**.

12 Click **Finish**.

13 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.

14 In the **Model Builder** window's toolbar, click the **Show** button and select **Stabilization** in the menu.

GEOMETRY 1

Import 1

1 In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Import**.

 ©2012 COMSOL

2 In the **Import** settings window, locate the **Import** section.

3 Click the **Browse** button.

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

5 Click the **Import** button.

DEFINITIONS

Variables 1

1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Variables**.

2 In the **Variables** settings window, locate the **Variables** section.

3 In the table, enter the following settings:

NAME	EXPRESSION	DESCRIPTION
p0	1[torr]	Outlet pressure

Explicit 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Selections>Explicit**.

2 Select Domain 1 only.

3 Right-click **Model 1>Definitions>Explicit 1** and choose **Rename**.

4 Go to the **Rename Explicit** dialog box and type `Plasma` in the **New name** edit field.

5 Click **OK**.

Explicit 2

1 Right-click **Definitions** and choose **Selections>Explicit**.

2 In the **Explicit** settings window, 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 **Model 1>Definitions>Explicit 2** and choose **Rename**.

6 Go to the **Rename Explicit** dialog box and type `Walls` in the **New name** edit field.

7 Click **OK**.

Explicit 3

1 Right-click **Definitions** and choose **Selections>Explicit**.

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

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

©2012 COMSOL

4 Select Boundary 4 only.

5 Right-click **Model 1>Definitions>Explicit 3** and choose **Rename**.

6 Go to the **Rename Explicit** dialog box and type `Outlet` in the **New name** edit field.

7 Click **OK**.

Explicit 4

1 Right-click **Definitions** and choose **Selections>Explicit**.

2 In the **Explicit** settings window, 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 **Model 1>Definitions>Explicit 4** and choose **Rename**.

6 Go to the **Rename Explicit** dialog box and type `Coil Walls` in the **New name** edit field.

7 Click **OK**.

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

INDUCTIVELY COUPLED PLASMA

In the **Model Builder** window, expand the **Inductively Coupled Plasma** node.

Cross Section Import 1

1 Right-click **Model 1>Inductively Coupled Plasma** and choose the domain setting **Heavy Species Transport>Cross Section Import**.

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

3 Click the **Browse** button.

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

5 In the **Model Builder** window, click **Inductively Coupled Plasma**.

6 In the **Inductively Coupled Plasma** settings window, locate the **Transport Settings** section.

7 Select the **Convection** check box.

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

Reaction 1

1 In the **Model Builder** window, right-click **Inductively Coupled Plasma** and choose the domain setting **Heavy Species Transport>Reaction**.

 ©2012 COMSOL

2 In the **Reaction** settings window, locate the **Reaction Formula** section.

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

4 Locate the **Kinetics Expressions** section. In the k^{f} edit field, type `3.734E8`.

Reaction 2

1 In the **Model Builder** window, right-click **Inductively Coupled Plasma** and choose the domain setting **Heavy Species Transport>Reaction**.

2 In the **Reaction** settings window, locate the **Reaction Formula** section.

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

4 Locate the **Kinetics Expressions** section. In the k^{f} edit field, type `1807`.

Now you add an Amperes law feature to the model. This is applied to all domains where the AC electric and magnetic field exists but the plasma is not present. Since Ampere's law is an exclusive feature, it automatically removes the Plasma Model feature from the selected domains. This means that the AC electric field is computed both inside and outside the plasma. The species and electrostatic field which make up the plasma are only computed in domain 1.

Ampère's Law 1

1 In the **Model Builder** window, right-click **Inductively Coupled Plasma** and choose the domain setting **Magnetic Fields>Ampère's Law**.

2 Select Domains 2–7 only.

Ground 1

1 In the **Model Builder** window, right-click **Inductively Coupled Plasma** and choose the boundary condition **Electrostatics>Ground**.

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

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

Surface Reaction 1

1 In the **Model Builder** window, right-click **Inductively Coupled Plasma** and choose the boundary condition **Heavy Species Transport>Surface Reaction**.

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

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

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

Surface Reaction 2

1 In the **Model Builder** window, right-click **Inductively Coupled Plasma** and choose the boundary condition **Heavy Species Transport>Surface Reaction**.

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

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

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

Wall 1

1 In the **Model Builder** window, right-click **Inductively Coupled Plasma** and choose the boundary condition **Drift Diffusion>Wall**.

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

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

Outflow 1

1 In the **Model Builder** window, right-click **Model 1>Inductively Coupled Plasma>Species: Ars** and choose the boundary condition **Outflow**.

2 In the **Outflow** settings window, locate the **Boundary Selection** section.

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

Outflow 1

1 In the **Model Builder** window, right-click **Model 1>Inductively Coupled Plasma>Species: Ar+** and choose the boundary condition **Outflow**.

2 In the **Outflow** settings window, locate the **Boundary Selection** section.

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

Species: Ar

1 In the **Model Builder** window, under **Model 1>Inductively Coupled Plasma** click **Species: Ar**.

2 In the **Species** settings window, 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, under **Model 1>Inductively Coupled Plasma** click **Species: Ars**.

2 In the **Species** settings window, locate the **General Parameters** section.

3 In the x_0 edit field, type `1E-4`.

4 From the **Preset species data** list, choose **Ar**.

 ©2012 COMSOL

5 Click to expand the **Species Thermodynamic Parameters** section. In the Δh edit 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 11.5 eV. This is added in the edit field Additional enthalpy contribution.

Species: Ar+

1 In the **Model Builder** window, under **Model 1>Inductively Coupled Plasma** click **Species: Ar+**.

2 In the **Species** settings window, 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 Click to expand the **Species Thermodynamic Parameters** section. In the Δh edit 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 15.8 eV. This is added in the edit 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, under **Model 1>Inductively Coupled Plasma** click **Plasma Model 1**.

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

3 From the $\mathbf{u}$ list, choose **Velocity field (spf/fp1)**.

4 From the T list, choose **Temperature (ht/fluid1)**.

5 From the p_A list, choose **Pressure (spf/fp1)**.

6 Clear the **Reference pressure** check box.

7 Locate the **Electron Density and Energy** section. In the μ_e edit field, type `4E24/icp.Nn`.

Now you assign the plasma conductivity. The plasma conductivity is automatically computed as a tensor property. This is needed in cases where the plasma conductivity is dependent on a DC magnetic flux density. You simply select the

automatically computed plasma conductivity from the Electrical conductivity combobox.

8 Locate the **Conduction Current** section. From the σ list, choose **Electrical conductivity (icp/icp)**.

Initial Values 1

1 In the **Model Builder** window, under **Model 1>Inductively Coupled Plasma** click **Initial Values 1**.

2 In the **Initial Values** settings window, locate the **Initial Values** section.

3 In the $n_{e,0}$ edit field, type `1E15`.

4 In the ε_0 edit field, type `3`.

In this example you assume that the coil is driven by a constant power of 700 W.

Coil Group Domain 1

1 In the **Model Builder** window, right-click **Inductively Coupled Plasma** and choose the domain setting **Magnetic Fields>Coil Group Domain**.

2 Select Domains 4–7 only.

3 In the **Coil Group Domain** settings window, locate the **Coil Group Domain** section.

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

5 In the P_{coil} edit field, type `700[W]`.

LAMINAR FLOW

1 In the **Model Builder** window, under **Model 1** click **Laminar Flow**.

2 In the **Laminar Flow** settings window, locate the **Consistent Stabilization** section.

3 Find the **Navier-Stokes equations** subsection. Clear the **Crosswind diffusion** check box.

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

5 Select Domain 1 only.

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

Fluid Properties 1

1 In the **Model Builder** window, expand the **Laminar Flow** node, then click **Fluid Properties 1**.

2 In the **Fluid Properties** settings window, locate the **Fluid Properties** section.

3 From the ρ list, choose **Density (icp/pem1)**.

4 From the μ list, choose **Dynamic viscosity (icp/pem1)**.

 ©2012 COMSOL

Initial Values 1

1 In the **Model Builder** window, under **Model 1>Laminar Flow** click **Initial Values 1**.

2 In the **Initial Values** settings window, locate the **Initial Values** section.

3 In the p edit field, type `p0`.

Inlet 1

1 In the **Model Builder** window, right-click **Laminar Flow** and choose the boundary condition **Inlet**.

2 Select Boundary 2 only.

3 In the **Inlet** settings window, locate the **Boundary Condition** section.

4 From the **Boundary condition** list, choose **Mass flow**.

5 Locate the **Mass Flow Rate** section. From the **Mass flow type** list, choose **Standard flow rate (SCCM)**.

6 From the M_n list, choose **Mean molar mass (icp/pem1)**.

7 In the Q_{sccm} edit field, type `100*tanh(1E5*t[1/s])`.

Outlet 1

1 In the **Model Builder** window, right-click **Laminar Flow** and choose the boundary condition **Outlet**.

2 Select Boundary 4 only.

3 In the **Outlet** settings window, locate the **Boundary Condition** section.

4 In the p_0 edit field, type `p0`.

HEAT TRANSFER IN FLUIDS

1 In the **Model Builder** window, under **Model 1** click **Heat Transfer in Fluids**.

2 In the **Heat Transfer in Fluids** settings window, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 1 only.

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

Heat Transfer in Fluids 1

1 In the **Model Builder** window, expand the **Heat Transfer in Fluids** node, then click **Heat Transfer in Fluids 1**.

2 In the **Heat Transfer in Fluids** settings window, locate the **Model Inputs** section.

3 From the p_A list, choose **Pressure (spf/fp1)**.

4 Clear the **Reference pressure** check box.

5 From the **u** list, choose **Velocity field (spf/fp1)**.

6 Locate the **Heat Conduction** section. From the k list, choose **Thermal conductivity (icp/pem1)**.

7 Locate the **Thermodynamics** section. From the ρ list, choose **Density (icp/pem1)**.

8 From the C_p list, choose **Mass-averaged mixture specific heat (icp/pem1)**.

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

Initial Values 1

1 In the **Model Builder** window, under **Model 1>Heat Transfer in Fluids** click **Initial Values 1**.

2 In the **Initial Values** settings window, locate the **Initial Values** section.

3 In the T edit field, type `300`.

Heat Source 1

1 In the **Model Builder** window, right-click **Heat Transfer in Fluids** and choose the domain setting **Heat Source**.

2 Select Domain 1 only.

3 In the **Heat Source** settings window, locate the **Heat Source** section.

4 In the Q edit field, type `icp.Qgas`.

Temperature 1

1 In the **Model Builder** window, right-click **Heat Transfer in Fluids** and choose the boundary condition **Temperature**.

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

3 In the **Temperature** settings window, locate the **Temperature** section.

4 In the T_0 edit field, type `300`.

Outflow 1

1 In the **Model Builder** window, right-click **Heat Transfer in Fluids** and choose the boundary condition **Outflow**.

2 Select Boundary 4 only.

MATERIALS

Material 1

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

 ©2012 COMSOL

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

3 Click **Clear Selection**.

4 Select Domain 1 only.

5 In the **Material** settings window, locate the **Material Contents** section.

6 In the table, enter the following settings:

PROPERTY	NAME	VALUE
Relative permittivity	epsilonr	1
Relative permeability	mur	1

7 Right-click **Model 1>Materials>Material 1** and choose **Rename**.

8 Go to the **Rename Material** dialog box and type `Plasma` in the **New name** edit field.

9 Click **OK**.

Material 2

1 Right-click **Materials** and choose **Material**.

2 Select Domain 2 only.

3 In the **Material** settings window, locate the **Material Contents** section.

4 In the table, enter the following settings:

PROPERTY	NAME	VALUE
Electric conductivity	sigma	0
Relative permittivity	epsilonr	4.5
Relative permeability	mur	1

5 Right-click **Model 1>Materials>Material 2** and choose **Rename**.

6 Go to the **Rename Material** dialog box and type `Dielectric` in the **New name** edit field.

7 Click **OK**.

Material 3

1 Right-click **Materials** and choose **Material**.

2 Select Domain 3 only.

3 In the **Material** settings window, locate the **Material Contents** section.

©2012 COMSOL

4 In the table, enter the following settings:

PROPERTY	NAME	VALUE
Electric conductivity	sigma	0
Relative permittivity	epsilonr	1
Relative permeability	mur	1

5 Right-click **Model 1>Materials>Material 3** and choose **Rename**.

6 Go to the **Rename Material** dialog box and type `Air` in the **New name** edit field.

7 Click **OK**.

Material 4

1 Right-click **Materials** and choose **Material**.

2 Select Domains 4–7 only.

3 In the **Material** settings window, locate the **Material Contents** section.

4 In the table, enter the following settings:

PROPERTY	NAME	VALUE
Electric conductivity	sigma	6E7
Relative permittivity	epsilonr	1
Relative permeability	mur	1

5 Right-click **Model 1>Materials>Material 4** and choose **Rename**.

6 Go to the **Rename Material** dialog box and type `Copper coil` in the **New name** edit 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 **Mesh** settings window, locate the **Mesh Settings** section.

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

Edge 1

1 Right-click **Model 1>Mesh 1** and choose **Edge**.

2 Select Boundary 2 only.

©2012 COMSOL

Size 1

1 Right-click **Model 1>Mesh 1>Edge 1** and choose **Size**.

2 In the **Size** settings window, 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 edit field, type `0.001`.

Free Triangular 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Free Triangular**.

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

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

4 Select Domain 1 only.

Size 1

1 Right-click **Model 1>Mesh 1>Free Triangular 1** and choose **Size**.

2 In the **Size** settings window, locate the **Element Size** section.

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

Boundary Layers 1

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

2 In the **Boundary Layers** settings window, 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, under **Model 1>Mesh 1>Boundary Layers 1** click **Boundary Layer Properties**.

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

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

4 Locate the **Boundary Layer Properties** section. In the **Number of boundary layers** edit field, type `5`.

5 In the **Boundary layer stretching factor** edit field, type `1.5`.

©2012 COMSOL

Mapped 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Mapped**.

2 In the **Mapped** settings window, 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 **Model 1>Mesh 1>Mapped 1** and choose **Distribution**.

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

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

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

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

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

7 From the **Distribution method** list, choose **Geometric sequence**.

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

Free Triangular 2

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Free Triangular**.

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

STUDY 1

Step 1: Frequency-Transient

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

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

3 In the **Times** edit field, type `0 `.

4 Click the **Range** button.

5 Go to the **Range** dialog box.

6 From the **Entry method** list, choose **Number of values**.

7 In the **Start** edit field, type `-8`.

8 In the **Stop** edit field, type `-2`.

9 In the **Number of values** edit field, type `21`.

10 From the **Function to apply to all values** list, choose **exp10**.

©2012 COMSOL

11 Click the **Add** button.

12 In the **Frequency-Transient** settings window, locate the **Study Settings** section.

13 In the **Frequency** edit field, type `13.56E6`.

14 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

15 Click the **Zoom Extents** button on the Graphics toolbar.

RESULTS

2D Plot Group 9

1 In the **Model Builder** window, right-click **Results** and choose **2D Plot Group**.

2 Right-click **2D Plot Group 9** and choose **Surface**.

3 In the **Surface** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Inductively Coupled Plasma (Heavy Species Transport)>Mass fractions>Mass fraction (icp.wAr)**.

4 Click the **Plot** button.

2D Plot Group 10

1 In the **Model Builder** window, right-click **Results** and choose **2D Plot Group**.

2 Right-click **2D Plot Group 10** and choose **Surface**.

3 In the **Surface** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Inductively Coupled Plasma (Heavy Species Transport)>Mass fractions>Mass fraction (icp.wArs)**.

4 Click the **Plot** button.

©2012 COMSOL

©2012 COMSOL