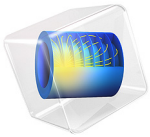

3D ICP Reactor, Argon Chemistry

Introduction

Although not recommended, 3-dimensional plasma modeling is possible to do in COMSOL. A square coil is placed on top of a dielectric window and it electrically excited at 13.56 MHz. A plasma is formed in the chamber beneath the dielectric window, which contains argon gas at low pressure (20 mtorr). The gas flows into the process chamber from two 2 inch ports and the gas is extracted through a single 4 inch port. The plasma is sustained via electromagnetic induction where power is transferred from the electromagnetic fields to the electrons.

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

Model Definition

Inductively coupled discharges typically operate at low pressures (<10 Pa) and high charge density (>10^{17} m^{-3}). High density plasma sources are popular because low pressure ion bombardment can provide a greater degree of anisotropy on the surface of the wafer.

DOMAIN EQUATIONS

The Inductively Coupled Plasma (ICP) reactor interface solves a system of coupled partial differential equations for:

- The electron density.
- The mean electron energy.
- The mass fraction of each of the non-electron species.
- The electrostatic potential.
- The electric field due to induction currents.

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 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. In this model, the electron diffusivity, energy mobility and energy diffusivity are calculated from the electron mobility, μ_e 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 and are written using either rate coefficients. Suppose that there are M reactions which contribute to the growth or decay of electron density and P electron-neutral collisions. In general $P >> M$. 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 (m^3/s), and N_n is the total neutral number density ($1/\mathrm{m}^3$). 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 (V). The electron source and inelastic energy loss are automatically computed by the **Inductively Coupled Plasma** interface. The rate coefficients may 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 $\gamma = (2q/m_e)^{1/2}$ ($\mathrm{C}^{1/2}/\mathrm{kg}^{1/2}$), m_e is the electron mass (kg), ε is energy (V), σ_k is the collision cross section (m^2) and f is the electron energy distribution function (EEDF). In this model 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 *Theory for the Electrostatics Interface* in the *Plasma Module User's Guide*.

For a nonmagnetized, nonpolarized plasma, the induction currents are computed in the frequency domain using the following equation:

$$(j\omega\sigma - \omega^2\varepsilon_0)\mathbf{A} + \nabla \times (\mu_0^{-1}\nabla \times \mathbf{A}) = \mathbf{J}^e$$

The plasma conductivity is automatically defined using the cold plasma approximation:

$$\sigma = \frac{n_e q^2}{m_e(\nu_e + j\omega)}$$

where n_e is the electron density, q is the electron charge, m_e is the electron mass, ν_e is the collision frequency and ω is the angular frequency.

PLASMA CHEMISTRY

Since the physics occurring in an inductively coupled plasma is rather complex, it is always best to start a modeling project with a simple chemical mechanism. 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-section are obtained from Ref. 3):

TABLE 1: TABLE OF COLLISIONS AND REACTIONS MODELED.

REACTION	FORMULA	TYPE	Δε(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

TABLE 1: TABLE OF COLLISIONS AND REACTIONS MODELED.

REACTION	FORMULA	TYPE	$\Delta\varepsilon$(eV)
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. 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

From an electrical point of view, the GEC reactor behaves as a transformer. A current is applied to the driving coil (the primary) and this induces a current in the plasma (the secondary). The plasma then induces an opposing current back in the coil, increasing its impedance. The current flowing in the plasma depends on the current applied to the coil and the reaction kinetics. The total plasma current may vary from no current (plasma not sustained) to the same current as the primary which corresponds to perfect coupling between the coil and the plasma. Since this is a 3D model it is not practical to resolve the skin depth in the driving coil. Instead an impedance boundary condition is used which only computes the coil current density in the tangential direction of the coil surface. This means the mesh does not have to resolve the skin depth in the coil, which at 13.56MHz for copper is only 17 microns.

Results and Discussion

The peak electron density occurs at the center of the reactor, underneath the RF coil. The electron density in this case is high enough to cause some shielding of the azimuthal electric field.

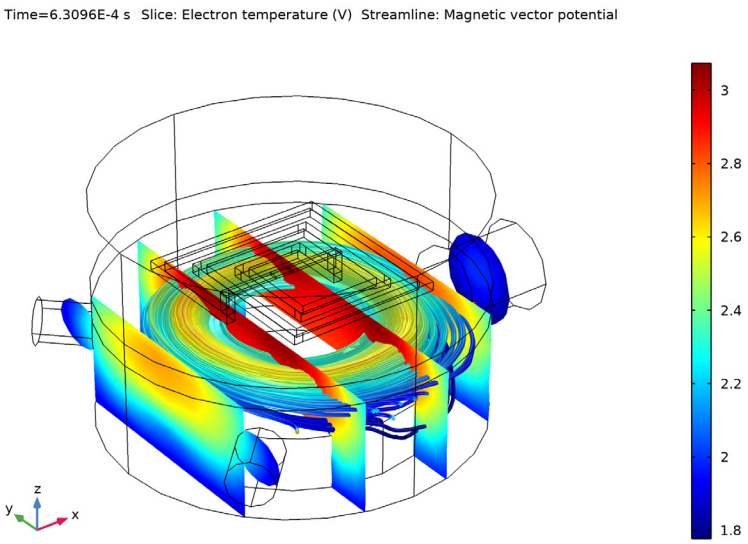

Figure 1: Plot of electron temperature (slice), magnetic vector potential (streamlines) and electric field norm (streamline color).

The electron "temperature" is highest directly underneath the coil, which is where the bulk of the power deposition occurs. The streamlines in Figure 1 show the norm of the electric field. The streamlines aren't perfectly circular since the coil is asymmetric.

References

1. G.J.M. Hagelaar and L.C. Pitchford, "Solving the Boltzmann Equation to Obtain Electron Transport Coefficients and Rate Coefficients for Fluid Models," *Plasma Sources Sci. Technol*, vol. 14, pp. 722–733, 2005.

2. D.P. Lymberopolous and D.J. Economou, "Two-Dimensional Self-Consistent Radio Frequency Plasma Simulations Relevant to the Gaseous Electronics Conference RF Reference Cell," *J. Res. Natl. Inst. Stand. Technol.*, vol. 100, p. 473, 1995.

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

Application Library path: `Plasma_Module/Inductively_Coupled_Plasmas/argon_3d_icp`

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 **3D**.

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

3 Click **Add**.

4 In the **Select Physics** tree, select **Plasma>Inductively Coupled Plasma**.

5 Click **Add**.

6 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 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 `argon_3d_icp.mphbin`.

5 Click **Import**.

DEFINITIONS

Coil boundaries

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

2 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

3 Select Domains 4–6, 8, and 9 only.

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

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

6 Click **OK**.

7 In the **Settings** window for **Explicit**, locate the **Output Entities** section.

8 From the **Output entities** list, choose **Adjacent boundaries**.

Current density

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

2 Select Domain 7 only.

3 In the **Settings** window for **Explicit**, locate the **Output Entities** section.

4 From the **Output entities** list, choose **Adjacent boundaries**.

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

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

7 Click **OK**.

Plasma

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

2 Select Domain 1 only.

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

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

5 Click **OK**.

Inlets

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 1 and 4 only.

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

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

7 Click **OK**.

Outlet

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

2 Select Domain 1 only.

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

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

5 Select Boundary 100 only.

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

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

8 Click **OK**.

9 Click the **Go to Default View** button in the **Graphics** toolbar.

ICP domains

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

2 Select Domains 1–3 only.

3 Right-click **Explicit 6** and choose **Rename**.

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

5 Click **OK**.

Plasma 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 2, 3, 5–9, 12, 17–23, 70, 73–75, and 96–99 only.

5 Right-click **Explicit 7** and choose **Rename**.

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

7 Click **OK**.

View 1

1 In the **Settings** window for **View**, locate the **View** section.

2 Clear the **Show grid** check box.

3 Click the **Go to Default View** button in the **Graphics** toolbar.

Variables 1

1 In the **Definitions** toolbar, click **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
Mw	0.04[kg/mol]	kg/mol	Molecular weight
T	300[K]	K	Model temperature
p0	0.02[Torr]	Pa	Outlet pressure
rho_stp	101325[Pa]*Mw/ R_const/273.15[K]	kg/m³	Density at standard pressure and temperature
Qin	0.5E-2[m^3/s]	m³/s	Volumetric flowrate
Js0	1E3[A/m]*tanh(1E7* t[1/s])	A/m	Surface current density

PLASMA (PLAS)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Plasma (plas)**.

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

3 Click **Clear Selection**.

4 Select Domain 1 only.

MAGNETIC FIELDS (MF)

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

2 Select Domains 1–3 only.

LAMINAR FLOW (SPF)

Since the density variation is not small, the flow cannot be regarded as incompressible. Therefore set the flow to be 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 Locate the **Domain Selection** section. Click **Clear Selection**.

5 Select Domain 1 only.

Define the pressure reference level in the interface properties.

6 Locate the **Physical Model** section. In the p_{ref} text field, type 0.

PLASMA (PLAS)

In the **Model Builder** window, under **Component 1 (comp1)** click **Plasma (plas)**.

Cross Section Import 1

1 In the **Physics** toolbar, click **Global** and choose **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.

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 **From mass constraint** check box.

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.

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`.

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 `1E16`.

4 In the ε_0 text field, type `4`.

Plasma Model 1

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

2 In the **Settings** window for **Plasma Model**, locate the **Electron Density and Energy** section.

3 In the μ_e text field, type `4E24[1/(m*V*s)]/plas.Nn`.

4 Locate the **Model Inputs** section. From the $\mathbf{u}$ list, choose **Velocity field (spf)**.

5 In the T text field, type `T`.

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

MAGNETIC FIELDS (MF)

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

Impedance Boundary Condition 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Impedance Boundary Condition**.

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

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

Surface Current Density 1

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

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

3 From the **Selection** list, choose **Current density**.

4 Locate the **Surface Current Density** section. Specify the $\mathbf{J}_{s0}$ vector as

Js0	x
0	y
0	z

PLASMA (PLAS)

In the **Model Builder** window, under **Component 1 (comp1)** click **Plasma (plas)**.

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 **Plasma walls**.

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 **Plasma 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 **Plasma 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 **Plasma walls**.

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

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, 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**.

LAMINAR FLOW (SPF)

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 In the p_{ref} text field, type `p0`.

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

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

4 Locate the **Boundary Condition** section. From the list, choose **Fully developed flow**.

5 Locate the **Fully Developed Flow** section. Click the **Flow rate** button.

6 In the V_0 text field, type `Qin`.

Outlet 1

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

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

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

MATERIALS

Material 1 (mat1)

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

2 Select Domains 1–3 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

Material 2 (mat2)

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

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

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

4 From the **Selection** list, choose **Coil boundaries**.

5 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	1	Basic
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	6e7	S/m	Basic

MESH 1

Free Tetrahedral 1

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

2 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

MESH 2

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

2 In the **Mesh** toolbar, click  **Free Tetrahedral**.

Free Tetrahedral 1

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

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

3 Select Domains 2 and 3 only.

Boundary Layers 1

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

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

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

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

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 **Plasma walls**.

4 In the **Model Builder** window, right-click **Mesh 2** and choose **Build All**.

ADD STUDY

1 In the **Home** toolbar, click  **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Empty Study**.

4 Click **Add Study** in the window toolbar.

5 In the **Home** toolbar, click  **Add Study** to close the **Add Study** window.

STUDY 1

Stationary

1 In the **Study** toolbar, click  **Study Steps** and choose **Stationary>Stationary**.

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

3 In the table, clear the **Solve for** check boxes for **Plasma (plas)** and **Magnetic Fields (mf)**.

Frequency-Transient

1 In the **Study** toolbar, click  **Study Steps** and choose **Time Dependent>Frequency-Transient**.

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

3 In the **Output times** text field, type `0 10^{range(-8,6/10,-3)}`.

4 In the **Frequency** text field, type `13.56[MHz]`.

5 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for **Laminar Flow (spf)**.

6 Click to expand the **Mesh Selection** section. Click to expand the **Values of Dependent Variables** section. In the **Study** toolbar, click  **Compute**.

RESULTS

Study 1/Solution Store 1 (3) (sol2)

1 In the **Model Builder** window, expand the **Results>Datasets** node.

2 Right-click **Results>Datasets>Study 1/Solution Store 1 (sol2)** and choose **Duplicate**.

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

4 From the **Solution** list, choose **Solution 1 (sol1)**.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

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

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

4 Select Domain 1 only.

Streamline 1

1 In the **Model Builder** window, right-click **Electron Temperature (plas)** and choose **Streamline**.

2 In the **Settings** window for **Streamline**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Magnetic Fields>Magnetic>Ax,Ay,Az - Magnetic vector potential**.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/Solution 1 (1) (sol1)**.

4 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Starting-point controlled**.

5 From the **Entry method** list, choose **Coordinates**.

6 In the **x** text field, type `range(-0.12,0.02,0.12)`.

7 In the **y** text field, type `-0.12`.

8 In the **z** text field, type `-0.05`.

9 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Type** list, choose **Tube**.

Color Expression 1

1 Right-click **Streamline 1** and choose **Color Expression**.

2 In the **Settings** window for **Color Expression**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Magnetic Fields>Electric>mf.normE - Electric field norm - V/m**.

3 Locate the **Coloring and Style** section. Clear the **Color legend** check box.

4 In the **Electron Temperature (plas)** toolbar, click **Plot**.