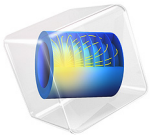

Model created in COMSOL Multiphysics 6.4

Microwave Plasma Torch

Introduction

This tutorial demonstrates how to set up a 3D model of a microwave plasma torch operating with argon at moderate pressures. The model is based on a generic configuration in which the plasma is generated inside a dielectric tube inserted into a rectangular waveguide. A useful reference for this type of plasma reactor is Ref. 1. Microwave power at 2.45 GHz is introduced into the waveguide in the TE_{01} mode, with the opposite end of the waveguide short-circuited. The dielectric tube is positioned at the location of maximum electromagnetic field intensity. The simulation solves a fully self-consistent set of coupled equations, including plasma transport and heating, Maxwell's equations, fluid flow, and heat transfer

Note: The model requires the Plasma Module and RF Module.

Model Definition

Electron transport is modeled by solving the continuity equation, the momentum equation under the drift-diffusion approximation, and the mean electron energy equation (for detailed information on electron transport, see *Theory for the Drift Diffusion Interface* in the *Plasma Module User's Guide*)

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

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

The source coefficients in the above equations are determined by the plasma chemistry. The electron rate expression is defined as

$$R_e = N_A \sum_j \nu_{e,j} r_j$$

where $\nu_{e,j}$ is the stoichiometric coefficient, and the reaction rate is defined as

$$r_j = k_j^f \prod_{i \in \text{ react}} c_i^{-\nu_{i,j}} + k_j^r \prod_{i \in \text{ prod}} c_k^{\nu_{i,j}}$$

where k_j^f is the forward rate constant and k_j^r is the reversed rate constant. Both the Electron Impact Reaction feature and Reaction feature can contribute to the electron rate expression. However, when using the Reaction feature it is important to note that the associated electron energy gain or loss is not included in the source term of the electron mean energy equation.

The rate constants can be computed from electron impact cross-section data

$$k^f = N_A \gamma \int_0^\infty \varepsilon \sigma(\varepsilon) f(\varepsilon) d\varepsilon$$

where $\gamma = (2q/m_e)^{1/2}$ (SI unit: $C^{1/2}/kg^{1/2}$), m_e is the electron mass (SI unit: kg), ε is the electron energy (SI unit: V), σ is the electron impact collision cross section (SI unit: m^2), and f is the electron energy distribution function.

When *Townsend coefficients* are used, the reaction rate is defined as

$$r_j = \frac{\alpha_j}{N_n} |\Gamma_e| \prod_{i \neq e \in \text{ react}} c_i^{-\nu_{i,j}}$$

where α_j/N_n is the reduced Townsend coefficient for reaction j (SI unit: m^2) and Γ_e is the electron flux as defined above (SI unit: $1/(m^2 \cdot s)$). Townsend coefficients can increase the stability of the numerical scheme when the electron flux is field driven as is the case with DC discharges.

The total electron energy loss or gained is calculated by summing the collisional energy changes from all reactions defined with the Electron Impact Reaction feature as

$$S_{\text{en}} = -\sum_j r_j \Delta\varepsilon_j F$$

where $\Delta\varepsilon_j$ is the energy loss from reaction j (SI unit: V) and F is the Faraday constant (SI unit: C/mol). For excitation and ionization collisions $\Delta\varepsilon_j$ corresponds to the energy of the excited state being excited/deexcited or ionized, for attachment $\Delta\varepsilon_j$ is set to zero, and for elastic collisions

$$\Delta\varepsilon = 2\frac{m_e}{m_k}\frac{3}{2}\left[T_e(eV) - \frac{k_B}{e}T_{\text{gas}}(K)\right]$$

where m_e and m_k are the electron and heavy species mass in kg, T_e is the electron temperature in eV, and T_{gas} is the gas temperature in K.

For heavy 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:

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

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*.

In a microwave reactor the high frequency electric field is computed in the frequency domain using the following equation:

$$\nabla\times(\mu_r^{-1}\nabla\times\mathbf{E}) - k_0^2\left(\varepsilon_r - \frac{j\sigma}{\omega\varepsilon_0}\right)\mathbf{E} = 0$$

The electromagnetic wave "sees" a plasma defined by the plasma conductivity in the cold plasma approximation that is set in the **Plasma Conductivity Coupling** multiphysics feature:

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

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. The Joule heating term that is responsible to heat the electrons is set in the **Electron Heat Source** multiphysics feature.

PLASMA CHEMISTRY

Argon plasmas feature one of the simplest reaction schemes, with electronically excited states often lumped into single effective levels. At high pressures, three-body reactions become significant, leading to the formation of dimers. The model employs a simplified plasma chemistry similar to that described in Ref. 2, comprising nine volumetric reactions

involving electrons (with electron impact cross sections obtained from Ref. 2), atomic and molecular ions, and a lumped level representing the argon 4s excited state.

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	e+Ar2+=>Ars+Ar	Excitation	-2.5
7	Ars+Ars=>e+Ar+Ar+	Penning ionization	-
8	Ars=>Ar	Spontaneous emission	-
9	2Ar+Ar+=>Ar+Ar2+	3-body Ar2+ creation	-

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
3	Ar2+=>2Ar	1

Results and Discussion

The figures in this section present the simulation results for an argon plasma sustained at 20 Torr with 50 W of input power. Figure 1 displays the spatial distribution of the electric field norm, where the maximum field intensity is observed at the interface between the waveguide and the dielectric tube, proximal to the microwave power injection port. This localized field enhancement corresponds to a region of elevated ionization, where the electron density reaches peak values of the order of 10^{18} m^{-3}, as shown in Figure 2. At such high plasma densities, energy transfer to neutral argon atoms via electron impact elastic collisions becomes significant, resulting in a neutral gas temperature increase up to 1000 K, as illustrated in Figure 3. Due to the gas inflow at the bottom of the dielectric tube, the gas temperature and number density, and consequently the charged species densities, exhibit asymmetry along the axial (z) direction.

Figure 4 presents the electron density isosurfaces along with the electric field norm in a logarithmic scale. The results clearly demonstrates that the presence of the plasma

significantly alters the electromagnetic field distribution, with the electric field amplitude attenuating rapidly upon entering the plasma region due to plasma shielding. Additionally, the electron density exhibits localized maxima near the waveguide-dielectric tube junction, forming an annular distribution pattern. This structure is indicative of enhance ionization near the periphery of the discharge channel.

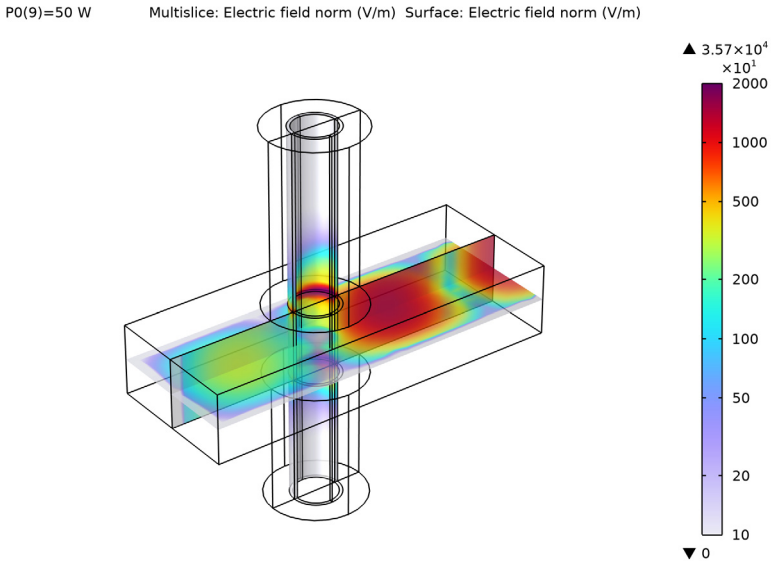

Figure 1: Electric field norm in the waveguide and at the surface of the dielectric tube for an input power of 50 W.

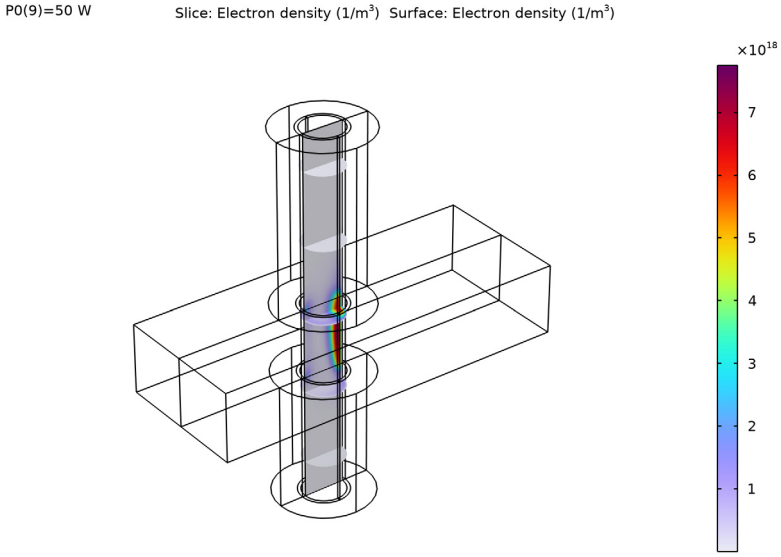

Figure 2: Electron number density for an input power of 50 W.

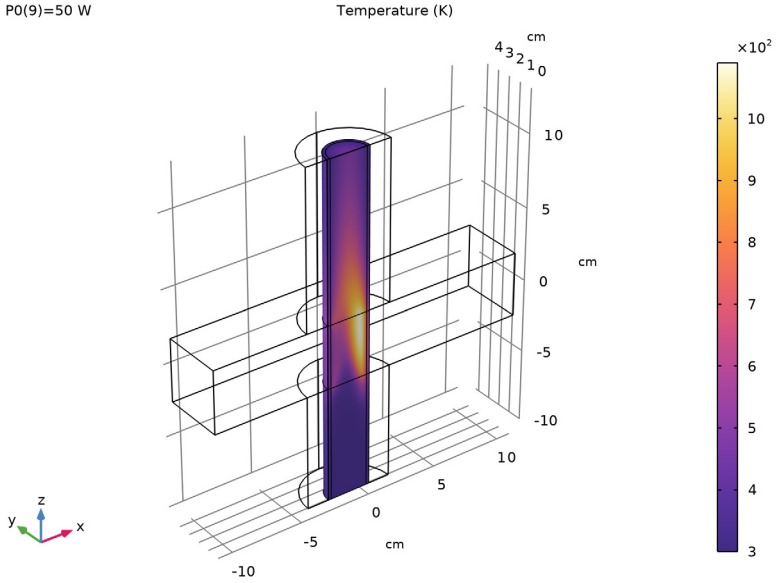

Figure 3: Gas temperature for an input power of 50 W.

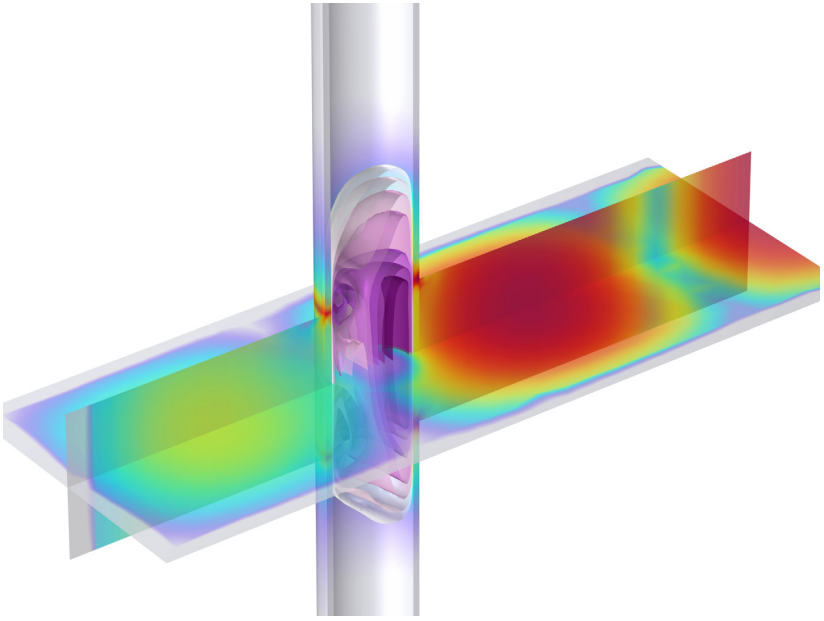

Figure 4: Electric field norm and electron number density isosurfaces on a logarithmic scale for an input power of 50 W. The electron number density isosurface levels are: 0.05, 0.11, 0.23, 0.5, 1.1, 2.3, and 50 in units of 10^{18} m^{-3}. The electric field norm has the same scale as in Figure 1.

References

1. M. Baeva, F. Hempel, H. Baiel, T. Trautvetter, R. Foest, and D. Loffhagen, "Two- and three-dimensional simulation analysis of a microwave excited plasma for deposition application: operation with argon at atmospheric pressure," *J. Phys. D: Appl. Phys.*, vol. 51, p. 385202, 2018.

2. N. Balcon, G.J.M. Hagelaar, and J.P. Boeuf, "Numerical Model of an Argon Atmospheric Pressure RF Discharge," *IEEE Trans. Plasma Sci.*, vol. 36, p. 2782, 2008.

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

Application Library path: `Plasma_Module/Wave-Heated_Discharges/microwave_plasma_torch`

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 **Plasma** > **Nonisothermal Plasma Flow** > **Microwave Plasma**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GEOMETRY 1

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

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **cm**.

GLOBAL DEFINITIONS

Parameters 1

Import parameters to use in the model.

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

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

3 Click **Load from File**.

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

Next, create a geometry for a microwave plasma torch.

GEOMETRY 1

Block 1 (blk1)

1 In the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Width** text field, type `WaveguideLength`.

4 In the **Depth** text field, type `WaveguideWidth`.

5 In the **Height** text field, type `WaveguideHeight`.

6 Locate the **Position** section. From the **Base** list, choose **Center**.

Cylinder 1 (cyl1)

1 In the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `RadiusOut`.

4 In the **Height** text field, type `CylinderHeight*(1+TopCylinderFactor)`.

5 Locate the **Position** section. In the **x** text field, type `CylincerCenter`.

6 In the **z** text field, type `-CylinderHeight/2`.

7 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (cm)
`Layer 1`	`RadiusOut-RadiusIn`

8 Click **Build Selected**.

Cylinder 2 (cyl2)

1 In the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `RadiusOut*2`.

4 In the **Height** text field, type `(CylinderHeight*(1+2*TopCylinderFactor))/2-WaveguideHeight/2`.

5 Locate the **Position** section. In the **x** text field, type `CylincerCenter`.

6 In the **z** text field, type `WaveguideHeight/2`.

Cylinder 3 (cyl3)

1 Right-click **Cylinder 2 (cyl2)** and choose **Duplicate**.

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Height** text field, type `CylinderHeight/2-WaveguideHeight/2`.

4 Locate the **Position** section. In the **z** text field, type `-CylinderHeight/2`.

5 Click **Build All Objects**.

Work Plane 1 (wp1)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **zx-plane**.

Partition Objects 1 (par1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Objects**.

2 Click in the **Graphics** window and then press Ctrl+A to select all objects.

3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

4 From the **Partition with** list, choose **Work plane**.

Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

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

4 On the object **par1(1)**, select Domain 1 only.

5 On the object **par1(2)**, select Domains 1, 3, and 5 only.

6 On the object **par1(3)**, select Domain 1 only.

7 On the object **par1(4)**, select Domain 1 only.

8 Click **Build All Objects**.

Create explicit selections for later use.

GEOMETRY 1

In the **Model Builder** window, collapse the **Component 1 (comp1)** > **Geometry 1** node.

DEFINITIONS

Tube

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

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

3 Select Domains 4–6 and 10–12 only.

Walls

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

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

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

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

5 Select Boundaries 24, 27, 30, 35, 38, and 41 only.

Plasma

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

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

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

4 Select Domains 7–9 only.

Specify material properties for use by the different physics interfaces.

ADD MATERIAL

1 In the **Materials** toolbar, click **Add Material** to open the **Add Material** window.

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

3 In the tree, select **Built-in** > **Air**.

4 Click the **Add to Component** button in the window toolbar.

5 In the tree, select **Built-in** > **Glass (quartz)**.

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

7 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Glass (quartz) (mat2)

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

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

In the **Plasma** interface, prepare a simple plasma chemistry for argon and apply boundary conditions for all species being solved. Poisson's equation is only solved in the plasma domain, and for simplicity the electric potential is set to zero at the dielectric.

On the symmetry plane, make sure that the default boundary conditions — **Insulation** and **Zero Charge** — are used, and that no surface reaction is applied.

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**.

Electron Impact Reaction 6

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

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

3 In the **Formula** text field, type `e+Ar2+=>Ars+Ar`.

4 Locate the **Collision Type** section. From the **Collision type** list, choose **Excitation**.

5 In the Δε text field, type `-2.5[V]`.

6 Locate the **Reaction Parameters** section. In the k^f text field, type `7e-13[m^3/s]*N_A_const*(300[K]/(plas.Te*11600[K/V]))^0.5`.

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 `1.2e-15[m^3/s]*N_A_const`.

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`.

4 Locate the **Reaction Parameters** section. In the k^f text field, type `5e5[1/s]`.

Reaction 3

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 `Ar+Ar+Ar+=>Ar2++Ar`.

4 Locate the **Reaction Parameters** section. In the k^f text field, type `2.5e-43[m^6/s]*N_A_const^2`.

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** checkbox.

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

4 Click to expand the **Species Thermodynamic Parameters** section. In the Δh text field, type `11.50`.

Species: Ar+

1 In the **Model Builder** window, click **Species: Ar+**.

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

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

4 Locate the **Species Thermodynamic Parameters** section. In the Δh text field, type `15.80`.

Species: Ar2+

1 In the **Model Builder** window, click **Species: Ar2+**.

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

3 In the M_w text field, type `0.08[kg/mol]`.

4 In the σ text field, type `3.330[angstrom]`.

5 In the ε/k_b text field, type `136.500[K]`.

6 Locate the **Species Thermodynamic Parameters** section. In the $a_{low,1}$ text field, type `0.02500000E+02`.

7 In the $a_{low,6}$ text field, type `-0.07453750E+04`.

8 In the $a_{low,7}$ text field, type `0.04366000E+02`.

9 In the $a_{hi,1}$ text field, type `0.02500000E+02`.

10 In the $a_{hi,6}$ text field, type `-0.07453750E+04`.

11 In the $a_{hi,7}$ text field, type `0.04366000E+02`.

12 In the Δh text field, type `14.5`.

Surface Reaction 1

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

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

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

4 Locate the **Boundary Selection** section. From the **Selection** list, choose **Walls**.

Surface Reaction 2

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

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

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

4 Locate the **Boundary Selection** section. From the **Selection** list, choose **Walls**.

Surface Reaction 3

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

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

3 In the **Formula** text field, type `Ar2+=>2Ar`.

4 Locate the **Boundary Selection** section. From the **Selection** list, choose **Walls**.

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 From the **Electron transport properties** list, choose **From electron impact reactions**.

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[1/m^3]`.

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**.

Outflow 1

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

2 Select Boundary 33 only.

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**.

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

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

6 Click **Clear Selection**.

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

In the **Laminar Flow** interface, set the domain where the physics is to be solved, add an inlet and outlet to the system, and set the symmetry plane.

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 **Domain Selection** section.

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

4 Locate the **Physical Model** section. In the p_{ref} text field, type `p0`.

Inlet 1

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

2 Select Boundary 26 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**.

6 In the Q_{sv} text field, type `Qsv`.

Outlet 1

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

2 Select Boundary 33 only.

Symmetry 1

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

2 Select Boundaries 25, 28, and 31 only.

LAMINAR FLOW (SPF)

In the **Model Builder** window, collapse the **Component 1 (comp1)** > **Laminar Flow (spf)** node.

Specify the domains where the **Heat Transfer in Fluids** interface is to be solved, set a constant temperature at the inlet, use a heat flux boundary conditions in the dielectric, and set the symmetry plane.

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 **Physical Model** section.

3 In the T_{ref} text field, type `T0`.

4 Locate the **Domain Selection** section. From the **Selection** list, choose **Plasma**.

5 Select Domains 4–12 only.

Solid 1

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

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

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

Outflow 1

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

2 Select Boundary 33 only.

Temperature 1

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

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

3 In the T_0 text field, type `T0`.

4 Select Boundary 26 only.

Heat Flux 1

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

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

3 From the **Flux type** list, choose **Convective heat flux**.

4 In the h text field, type `htc`.

5 In the T_{ext} text field, type `T0`.

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

7 Select Boundaries 15, 18, 21, and 44–46 only.

Symmetry 1

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

2 Select Boundaries 25, 28, and 31 only.

Initial Values 1

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

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

3 In the T text field, type `T0`.

In the **Electromagnetic Waves, Frequency Domain** interface, define a port and set the input power.

The **Scattering Boundary Condition** is used at some distance from the dielectric tube to correctly model the radiation fields.

HEAT TRANSFER IN FLUIDS (HT)

In the **Model Builder** window, collapse the **Component 1 (comp1)** > **Heat Transfer in Fluids (ht)** node.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Frequency Domain (emw)**.

Port 1

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

2 Select Boundary 55 only.

3 In the **Settings** window for **Port**, locate the **Port Properties** section.

4 From the **Type of port** list, choose **Rectangular**.

5 In the P_{in} text field, type `P0`.

Scattering Boundary Condition 1

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

2 Select Boundaries 7, 8, 11, 13, 16, 23, 26, 33, 36, 43, 47, and 48 only.

Perfect Electric Conductor 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Perfect Electric Conductor**.

2 Select Boundaries 9 and 12 only.

Symmetry Plane 1

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

2 Select Boundaries 2, 6, 10, 14, 17, 20, 25, 28, 31, and 49–54 only.

Proceed to create a mesh fine enough to resolve the wavelength and the plasma inside the dielectric tube.

MESH 1

Size 1

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

Free Triangular 1

1 In the **Mesh** toolbar, click  **More Generators** and choose **Free Triangular**.

2 Select Boundary 33 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 **Extremely fine**.

4 Click the **Custom** button.

5 Locate the **Element Size Parameters** section.

6 Select the **Maximum element size** checkbox. In the associated text field, type `0.18`.

Mapped 1

1 In the **Mesh** toolbar, click  **More Generators** and choose **Mapped**.

2 Select Boundaries 23 and 43 only.

Distribution 1

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

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

3 In the **Number of elements** text field, type `2`.

4 Select Edges 28, 50, and 71 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 **Boundary**.

4 Select Boundary 33 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 **Layers** section.

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

4 In the **Stretching factor** text field, type 1.25.

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

6 In the **Thickness** text field, type 100[um].

7 Select Edges 39 and 51 only.

8 Click **Build Selected**.

Swept 1

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

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

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

4 Select Domains 4–12 only.

Distribution 1

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

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

3 Click **Clear Selection**.

4 Select Domains 4, 7, and 10 only.

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

6 In the **Number of elements** text field, type 15.

7 In the **Element ratio** text field, type 4.

Distribution 2

1 In the **Model Builder** window, right-click **Swept 1** and choose **Distribution**.

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

3 Click **Clear Selection**.

4 Select Domains 5, 8, and 11 only.

5 Locate the **Distribution** section. In the **Number of elements** text field, type 20.

Distribution 3

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

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

3 Click **Clear Selection**.

4 Select Domains 6, 9, and 12 only.

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

6 In the **Number of elements** text field, type 15.

7 In the **Element ratio** text field, type 4.

8 Select the **Reverse direction** checkbox.

Free Triangular 2

1 In the **Mesh** toolbar, click **More Generators** and choose **Free Triangular**.

2 Select Boundaries 4 and 12 only.

Size 1

1 Right-click **Free Triangular 2** and choose **Size**.

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

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

Swept 2

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

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

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

4 Select Domains 1–3 only.

5 Click **Build All**.

The first study is used to provide initial conditions to a second study. With that goal in mind, the **Laminar Flow** interface is not solved for and the simulation stops at 0.1 ms before reaching a steady state.

The solver settings need to be modified for better convergence. The main change is to use a **Fully Coupled** solver.

INITIALIZATION

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

2 In the **Settings** window for **Study**, type `Initialization` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.

Step 1: Frequency–Transient

1 In the **Model Builder** window, under **Initialization** click **Step 1: Frequency–Transient**.

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

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

4 Click **Range**.

5 In the **Range** dialog, choose **Logarithmic** from the **Entry method** list.

6 In the **Start** text field, type `1e-10`.

7 In the **Stop** text field, type `1e-4`.

8 In the **Steps per decade** text field, type `1`.

9 Click **Replace**.

10 In the **Settings** window for **Frequency–Transient**, locate the **Physics and Variables Selection** section.

11 In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Laminar Flow (spf)**.

12 In the **Study** toolbar, click **Get Initial Value**.

INITIALIZATION

Solver Configurations

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

Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Initialization** > **Solver Configurations** > **Solution 1 (sol1)** > **Time-Dependent Solver 1** node.

2 Right-click **Time-Dependent Solver 1** and choose **Fully Coupled**.

3 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

4 In the **Damping factor** text field, type `0.9`.

5 From the **Jacobian update** list, choose **Once per time step**.

6 In the **Maximum number of iterations** text field, type `10`.

7 In the **Tolerance factor** text field, type `0.1`.

RESULTS

3D Plot Group 2

In the **Results** toolbar, click **3D Plot Group**.

Surface 1

Right-click **3D Plot Group 2** and choose **Surface**.

Selection 1

1 In the **Model Builder** window, right-click **Surface 1** and choose **Selection**.

2 Select Boundaries 25, 28, and 31 only.

Electron Density Initialization

1 In the **Model Builder** window, under **Results** click **3D Plot Group 2**.

2 In the **Settings** window for **3D Plot Group**, type `Electron Density Initialization` in the **Label** text field.

INITIALIZATION

Step 1: Frequency–Transient

1 In the **Model Builder** window, under **Initialization** click **Step 1: Frequency–Transient**.

2 In the **Settings** window for **Frequency–Transient**, click to expand the **Results While Solving** section.

3 Select the **Plot** checkbox.

4 In the table, enter the following settings:

Plot group	Plot window
Electron Density Initialization	Graphics

5 From the **Probes** list, choose **None**.

Add a second study to do a power sweep for the fully coupled problem. This study uses a Frequency–Stationary study step and takes as initial conditions the solutions from the previous study.

Again, the solver settings need to be modified for better convergence. A **Fully Coupled** solver is used and adjustments are made. The critical factors are the Initial damping factor and the Restriction to step-size update. The latter controls how fast the damping factor increases and needs to be low enough to ensure smooth convergence but not so low that it takes an unnecessarily long time to reach convergence.

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

INITIALIZATION

In the **Model Builder** window, collapse the **Initialization** node.

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 **Preset Studies for Selected Multiphysics** > **Frequency–Stationary**.

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

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

STUDY 2

Step 1: Frequency–Stationary

1 In the **Settings** window for **Frequency–Stationary**, locate the **Study Settings** section.

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

3 Click to expand the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.

4 From the **Method** list, choose **Solution**.

5 From the **Study** list, choose **Initialization, Frequency–Transient**.

6 From the **Time (s)** list, choose **Last**.

7 In the **Model Builder** window, click **Study 2**.

8 In the **Settings** window for **Study**, type `Power Sweep` in the **Label** text field.

9 In the **Study** toolbar, click **Get Initial Value**.

RESULTS

Mirror 3D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 3D**.

2 In the **Settings** window for **Mirror 3D**, locate the **Data** section.

3 From the **Dataset** list, choose **Power Sweep/Solution 2 (sol2)**.

4 Locate the **Plane Data** section. From the **Plane** list, choose **xz-planes**.

Electron Density (plas)

1 In the **Model Builder** window, under **Results** click **Electron Density (plas)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Mirror 3D 1**.

Slice 1

1 In the **Model Builder** window, expand the **Electron Density (plas)** node, then click **Slice 1**.

2 In the **Settings** window for **Slice**, locate the **Plane Data** section.

3 From the **Plane** list, choose **xy-planes**.

Surface 1

In the **Model Builder** window, right-click **Electron Density (plas)** and choose **Surface**.

Selection 1

1 In the **Model Builder** window, right-click **Surface 1** and choose **Selection**.

2 Select Boundaries 25, 28, and 31 only.

Surface 1

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

2 In the **Settings** window for **Surface**, click to expand the **Inherit Style** section.

3 From the **Plot** list, choose **Slice 1**.

POWER SWEEP

Step 1: Frequency–Stationary

1 In the **Model Builder** window, under **Power Sweep** click **Step 1: Frequency–Stationary**.

2 In the **Settings** window for **Frequency–Stationary**, click to expand the **Study Extensions** section.

3 Select the **Auxiliary sweep** checkbox.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
P0 (Input power)		W

6 In the table, click to select the cell at row number 1 and column number 2.

7 Click **Range**.

8 In the **Range** dialog, type 10 in the **Start** text field.

9 In the **Step** text field, type 5.

10 In the **Stop** text field, type 50.

11 Click **Replace**.

12 In the **Settings** window for **Frequency–Stationary**, locate the **Study Extensions** section.

13 From the **Run continuation for** list, choose **No parameter**.

14 From the **Reuse solution from previous step** list, choose **Yes**.

Solver Configurations

In the **Model Builder** window, expand the **Power Sweep** > **Solver Configurations** node.

Solution 2 (sol2)

1 In the **Model Builder** window, expand the **Power Sweep** > **Solver Configurations** > **Solution 2 (sol2)** > **Stationary Solver 1** node.

2 Right-click **Stationary Solver 1** and choose **Fully Coupled**.

3 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

4 In the **Initial damping factor** text field, type `0.5`.

5 In the **Minimum damping factor** text field, type `1.0E-8`.

6 In the **Restriction for step-size update** text field, type `1.5`.

7 In the **Recovery damping factor** text field, type `0.1`.

8 In the **Maximum number of iterations** text field, type `200`.

9 Click to expand the **Results While Solving** section. Select the **Plot** checkbox.

10 In the table, enter the following settings:

Plot group	Plot window
Electron Density (plas)	Graphics

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

RESULTS

Electron Density (plas)

Next, create figures that show the electromagnetic fields and plasma quantities of interest, such as electron and ion densities, electron temperature, and power absorbed by the electrons.

Slice 1

1 In the **Model Builder** window, expand the **Results** > **Electron Temperature (plas)** node, then click **Slice 1**.

2 In the **Settings** window for **Slice**, locate the **Plane Data** section.

3 From the **Plane** list, choose **xy-planes**.

Surface 1

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

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

3 In the **Expression** text field, type `plas.Te`.

Selection 1

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

2 Select Boundaries 25, 28, and 31 only.

Surface 1

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

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

3 From the **Plot** list, choose **Slice 1**.

Electron Temperature (plas)

1 In the **Model Builder** window, click **Electron Temperature (plas)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Mirror 3D 1**.

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

Electric Potential (plas)

1 In the **Model Builder** window, click **Electric Potential (plas)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Mirror 3D 1**.

Slice 1

1 In the **Model Builder** window, expand the **Electric Potential (plas)** node, then click **Slice 1**.

2 In the **Settings** window for **Slice**, locate the **Plane Data** section.

3 From the **Plane** list, choose **xy-planes**.

Surface 1

1 In the **Model Builder** window, right-click **Electric Potential (plas)** and choose **Surface**.

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

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

4 Locate the **Inherit Style** section. From the **Plot** list, choose **Slice 1**.

Selection 1

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

2 Select Boundaries 25, 28, and 31 only.

3 In the **Electric Potential (plas)** toolbar, click **Plot**.

Velocity (spf)

1 In the **Model Builder** window, under **Results** click **Velocity (spf)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Mirror 3D 1**.

Multislice 1

1 In the **Model Builder** window, expand the **Velocity (spf)** node, then click **Multislice 1**.

2 In the **Settings** window for **Multislice**, locate the **Multiplane Data** section.

3 Find the **y-planes** subsection. From the **Entry method** list, choose **Coordinates**.

4 In the **Coordinates** text field, type `0`.

5 Find the **z-planes** subsection. In the **Planes** text field, type `5`.

6 In the **Velocity (spf)** toolbar, click  **Plot**.

Streamline 1

1 In the **Model Builder** window, right-click **Velocity (spf)** and choose **Streamline**.

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

3 In the **x-component** text field, type `u`.

4 In the **y-component** text field, type `v`.

5 In the **z-component** text field, type `w`.

6 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Uniform density**.

7 In the **Density level** text field, type `9.5`.

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

9 Select the **Radius scale factor** checkbox.

10 In the **Tube radius expression** text field, type `0.02`.

11 Find the **Point style** subsection. From the **Type** list, choose **Arrow**.

12 From the **Arrow length** list, choose **Proportional**.

13 Select the **Scale factor** checkbox. In the associated text field, type `0.2`.

Color Expression 1

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

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

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

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Plasma**.

Pressure (spf)

1 In the **Model Builder** window, under **Results** click **Pressure (spf)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Mirror 3D 1**.

4 In the **Pressure (spf)** toolbar, click **Plot**.

Temperature (ht)

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

Ar+ Number Density

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

2 In the **Settings** window for **3D Plot Group**, type `Ar+ Number Density` in the **Label** text field.

Slice 1

1 In the **Model Builder** window, expand the **Ar+ Number Density** node, then click **Slice 1**.

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

3 In the **Expression** text field, type `plas.n_wAr_1p`.

Surface 1

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

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

3 In the **Expression** text field, type `plas.n_wAr_1p`.

4 In the **Ar+ Number Density** toolbar, click **Plot**.

Ar2+ Number Density

1 In the **Model Builder** window, right-click **Ar+ Number Density** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Ar2+ Number Density` in the **Label** text field.

Slice 1

1 In the **Model Builder** window, expand the **Ar2+ Number Density** node, then click **Slice 1**.

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

3 In the **Expression** text field, type `plas.n_wAr2_1p`.

Surface 1

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

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

3 In the **Expression** text field, type `plas.n_wAr2_1p`.

4 In the **Ar2+ Number Density** toolbar, click **Plot**.

Ars Number Density

1 In the **Model Builder** window, right-click **Ar2+ Number Density** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Ars Number Density` in the **Label** text field.

Slice 1

1 In the **Model Builder** window, expand the **Ars Number Density** node, then click **Slice 1**.

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

3 In the **Expression** text field, type `plas.n_wArs`.

Surface 1

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

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

3 In the **Expression** text field, type `plas.n_wArs`.

4 In the **Ars Number Density** toolbar, click **Plot**.

Electric Conductivity

1 In the **Model Builder** window, right-click **Ars Number Density** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Electric Conductivity` in the **Label** text field.

Slice 1

1 In the **Model Builder** window, expand the **Electric Conductivity** node, then click **Slice 1**.

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

3 In the **Expression** text field, type `emw.sigmaxx`.

Selection 1

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

2 Select Domains 7–9 only.

Surface 1

1 In the **Model Builder** window, under **Results** > **Electric Conductivity** click **Surface 1**.

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

3 In the **Expression** text field, type `emw.sigmaxx`.

4 In the **Electric Conductivity** toolbar, click **Plot**.

Power Absorbed by Electrons

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

2 In the **Settings** window for **3D Plot Group**, type `Power Absorbed by Electrons` in the **Label** text field.

3 In the **Model Builder** window, expand the **Power Absorbed by Electrons** node.

Slice 1

1 In the **Model Builder** window, expand the **Results** > **Power Absorbed by Electrons** > **Surface 1** node, then click **Results** > **Power Absorbed by Electrons** > **Slice 1**.

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

3 In the **Expression** text field, type `emw.Qrh`.

Surface 1

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

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

3 In the **Expression** text field, type `emw.Qrh`.

4 In the **Power Absorbed by Electrons** toolbar, click **Plot**.

Slice 1

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

2 In the **Settings** window for **Slice**, locate the **Coloring and Style** section.

3 From the **Color table** list, choose **HeatCamera**.

4 From the **Color table transformation** list, choose **Reverse**.

5 From the **Scale** list, choose **Logarithmic**.

6 Click to expand the **Range** section. Select the **Manual color range** checkbox.

7 In the **Minimum** text field, type `1e4`.

8 In the **Maximum** text field, type `1e7`.

9 In the **Power Absorbed by Electrons** toolbar, click **Plot**.

Electric Field (emw)

1 In the **Model Builder** window, under **Results** click **Electric Field (emw)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Mirror 3D 1**.

Multislice 1

1 In the **Model Builder** window, expand the **Electric Field (emw)** node, then click **Multislice 1**.

2 In the **Settings** window for **Multislice**, locate the **Multiplane Data** section.

3 Find the **x-planes** subsection. In the **Planes** text field, type `0`.

4 Find the **z-planes** subsection. From the **Entry method** list, choose **Coordinates**.

5 In the **Coordinates** text field, type `0`.

6 Click to expand the **Range** section. Select the **Manual color range** checkbox.

7 In the **Minimum** text field, type `1e2`.

8 In the **Maximum** text field, type `20e3`.

9 Locate the **Coloring and Style** section. From the **Color table** list, choose **Prism**.

10 From the **Color table transformation** list, choose **Nonlinear**.

11 In the **Color calibration parameter** text field, type `0.5`.

12 From the **Scale** list, choose **Logarithmic**.

Transparency 1

1 Right-click **Multislice 1** and choose **Transparency**.

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

3 Find the **Transparency** subsection. In the **Transparency** text field, type `0.3`.

Surface 1

1 In the **Model Builder** window, under **Results** > **Electric Field (emw)** click **Surface 1**.

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

3 From the **Dataset** list, choose **Power Sweep/Solution 2 (sol2)**.

Selection 1

1 In the **Model Builder** window, expand the **Surface 1** node, then click **Selection 1**.

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

3 Click to select the  **Activate Selection** toggle button.

4 Select Boundaries 2, 14, 15, 17, 18, 20, 21, 23, 24, 27, 30, 35, 38, 41, 43–46, 49–51, and 53 only.

Transparency 1

In the **Model Builder** window, right-click **Surface 1** and choose **Transparency**.

Electric Field and Electron Density

1 In the **Model Builder** window, right-click **Electric Field (emw)** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Electric Field and Electron Density` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **None**.

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

5 Locate the **Color Legend** section. Clear the **Show legends** checkbox.

Isosurface 1

1 Right-click **Electric Field and Electron Density** and choose **Isosurface**.

2 In the **Settings** window for **Isosurface**, locate the **Data** section.

3 From the **Dataset** list, choose **Power Sweep/Solution 2 (sol2)**.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Amethyst**.

5 Locate the **Levels** section. From the **Entry method** list, choose **Levels**.

6 Click  **Range**.

7 In the **Range** dialog, choose **Logarithmic** from the **Entry method** list.

8 In the **Start** text field, type `5e16`.

9 In the **Stop** text field, type `1e19`.

10 In the **Steps per decade** text field, type `3`.

11 Click **Replace**.

12 In the **Settings** window for **Isosurface**, locate the **Coloring and Style** section.

13 From the **Scale** list, choose **Logarithmic**.

Transparency 1

1 Right-click **Isosurface 1** and choose **Transparency**.

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

3 Find the **Transparency** subsection. In the **Transparency** text field, type `0.15`.

4 Find the **Fresnel transmittance** subsection. In the **Fresnel transmittance** text field, type `0.75`.

Electric Field and Electron Density

1 Click the  **Show Grid** button in the **Graphics** toolbar.

2 Click the  **Show Axis Orientation** button in the **Graphics** toolbar.