



Model created in COMSOL Multiphysics 6.4

# Model of an SF<sub>6</sub>/Argon Inductively Coupled Plasma Reactor

## Introduction

---

This tutorial studies the chemistry of a SF<sub>6</sub>/Ar plasma in an inductively coupled plasma reactor with moderate pressures. The main goal is to show how to prepare a model with a mixture of different elements (in this case Ar and SF<sub>6</sub>) in which one of the species can dissociate by electron impact into many fragments (SF<sub>6</sub> dissociates into SF<sub>x</sub>, F, and S) and where multiple negative ions exist.

A simplified plasma chemistry is used to discuss the main aspects of such discharges. It is important to keep in mind that a benchmark is not attempted and the idea is to provide a template that can be used to develop more complex and precise chemistries. In fact, quite probably it will be necessary to modify the data used and add more reactions to achieve experimental verification. The chemistry is based on [Ref. 1](#), [Ref. 2](#), [Ref. 3](#) and [Ref. 4](#).

The plasma model is solved self-consistently with the **Magnetic Fields, Laminar Flow** and **Heat Transfer in Fluids** interfaces. An initial **Frequency-Transient** study step is used to get a solution for the plasma problem alone without solving for fluid flow and heat transfer. In the following studies, the **Plasma, Magnetic Fields, Laminar Flow** and **Heat Transfer in Fluids** interfaces are solved fully coupled. Two parameterization studies are performed with respect to the input power and the argon mole fraction.

## 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*)

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

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

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

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

where  $v_{e,j}$  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

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

where  $\gamma = (2q/m_e)^{1/2}$  (SI unit:  $C^{1/2}/kg^{1/2}$ ),  $m_e$  is the electron mass (SI unit: kg),  $\epsilon$  is the electron energy (SI unit: V),  $\sigma$  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  $\alpha_j/N_n$  is the reduced Townsend coefficient for reaction  $j$  (SI unit:  $m^2$ ) and  $\Gamma_e$  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

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

where  $\Delta \epsilon_j$  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  $\Delta \epsilon_j$  corresponds to the energy of the excited state being excited/deexcited or ionized, for attachment  $\Delta \epsilon_j$  is set to zero, and for elastic collisions

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

where  $m_e$  and  $m_k$  are the electron and heavy species mass in kg,  $T_e$  is the electron temperature in eV, and  $T_{\text{gas}}$  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:

$$-\nabla \cdot \epsilon_0 \epsilon_r \nabla V = \rho$$

The space charge density  $\rho$  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:

$$(j\omega\sigma - \omega^2\epsilon_0)\mathbf{A} + \nabla \times (\mu_0^{-1}\nabla \times \mathbf{A}) = \mathbf{J}^e$$

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,  $\nu_e$  is the collision frequency, and  $\omega$  is the angular frequency. The Joule heating term that is responsible to heat the electrons is set in the **Electron Heat Source** multiphysics feature.

## BOUNDARY CONDITIONS

Electrons are lost to the wall due to random motion within a few mean free paths of the wall and gained due to secondary emission effects, resulting in the following boundary condition for the electron flux:

$$\mathbf{n} \cdot \Gamma_e = \left( \frac{1}{2} v_{e, \text{th}} n_e \right)$$

and the electron energy flux:

$$\mathbf{n} \cdot \Gamma_\varepsilon = \left( \frac{5}{6} v_{e, \text{th}} n_e \varepsilon \right)$$

For the heavy species, ions are lost to the wall due to surface reactions and the fact that the electric field is directed toward the wall:

$$\mathbf{n} \cdot \mathbf{j}_k = M_w R_k + M_w c_k Z \mu_k (\mathbf{E} \cdot \mathbf{n}) [Z_k \mu_k (\mathbf{E} \cdot \mathbf{n}) > 0]$$

The walls of the reactor are grounded.

## PLASMA CHEMISTRY

Negative ions are created in certain molecular gaseous discharges (like chlorine, oxygen, hydrogen, fluorocarbons, and so on) and these discharges tend to have complex plasma chemistries with many ions, dissociative products, and excited states.

The plasma chemistry is based on [Ref. 1](#), [Ref. 2](#), and [Ref. 3](#). The electron impact cross sections for SF<sub>6</sub> are from [Ref. 4](#) and [Ref. 6](#), and for F and F<sub>2</sub> are from [Ref. 5](#) and [Ref. 7](#). Argon cross sections are from [Ref. 8](#). All cross-section data were retrieved from [Ref. 9](#). For the rate constants of heavy species reactions were given generic order of magnitude values characteristic of the reaction type. The model includes 28 species: electrons, SF<sub>x</sub><sup>-</sup>, SF<sub>x</sub><sup>+</sup>, SF<sub>x</sub><sup>-</sup>, F<sub>x</sub><sup>-</sup>, F<sub>x</sub><sup>+</sup>, F<sub>x</sub><sup>-</sup>, Ar, Ar<sup>-</sup>, and Ar<sup>+</sup>.

This plasma chemistry is completely prepared in text data files and then imported using the Plasma Chemistry Import feature. Five reaction groups are created as listed in [Table 1](#).

TABLE 1: REACTION GROUP FEATURES INCLUDED IN THIS MODEL.

Reaction group for	Type	No. of reactions included
SF <sub>x</sub>	Electron Impact Reactions	32
F <sub>x</sub>	Electron Impact Reactions	14
Ar	Electron Impact Reactions	5

TABLE 1: REACTION GROUP FEATURES INCLUDED IN THIS MODEL.

Reaction group for	Type	No. of reactions included
Ion-ion	Heavy Species Reactions	56
Neutral-neutral	Heavy Species Reactions	9

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 three species and five electron impact reactions.

SF<sub>6</sub> has a much more rich reaction set that includes vibrational and rotational excitations, excitation of several electronic excited states, electron impact dissociation, dissociative attachment, and many others. Here we have made the simplification that the excited rotational and vibrational states are not treated explicitly but instead lumped into the corresponding neutral species.

In addition to the volume reactions, 11 surface reactions are also implemented. It is by using surface reactions that boundary conditions for heavy species are introduced in the model. If no surface reactions that leads to the lost of a given species at a surface are introduced in the model, that species will not have losses by transport. This can lead to the unbounded growth of a given species and a steady state solution might not be possible.

Atomic recombination (such as  $F \rightarrow 0.5F_2$ ) at a surface is an important aspect of plasma discharges with molecular species since it influences the dissociation degree in the discharge. The sticking coefficient for atomic recombination is a function of the surface type and temperature.

### ELECTRONEGATIVE PLASMAS

Electronegative plasmas are plasmas that contain negative ions. Negative ions are mainly created by electron dissociative attachment (for example,  $e + SF_6 \rightarrow SF_5 + F^-$ ). This reaction tends to be very effective at low electron energies and can reduce the electrons in a discharge to a point that an ion-ion discharge is obtained. The transport and volume creation/destruction mechanisms tend to be more complex than electropositive plasmas in many aspects. Here only a few are mentioned with emphasis on the numerical difficulties that they introduce.

In electronegative discharges negative ions are well confined by the ambipolar electric field and losses by transport are very small. This means that to achieve a steady state volume losses need to be included for negative ions. The mechanisms by which negative ions are lost depend on the gas mixture and pressure and they are: mutual recombination with positive ions, detachment in collisions with excited or neutral atoms or molecules, and electron-impact detachment.

In electronegative discharges it is often possible to identify two spatial regions using the electronegativity (ratio of the negative ion density to the electron density): (i) one in the core of the discharge (the electronegative core) with high electronegativity where the dominant charge species are positive and negative ions; (ii) and the other close to the boundaries (electropositive edges) where the dominant charged species are electron and positive ions. In the transition between these two regions the negative ion density drops abruptly causing a chock-like phenomena. This transition needs to be well resolved spatially. If not, oscillations can be seen in the negative ion density and the model might not converge. Some strategies to deal with this are:

- Increase the negative ion temperature of about 0.3 eV. A higher ion temperature makes the transport numerical easier. The ion temperature is defined in the section **Mobility and Diffusivity Expressions** in the species **Settings**. By default the ion temperature is the gas temperature.
- Enable **Isotropic diffusion for ions** in the **Inconsistent Stabilization** section (the stabilization sections are visible when **Stabilization** is selected in **Show More Options**). This option adds artificial diffusion to all ions and helps smoothing the sharp transition of the negative ion density between the electropositive edge and the electronegative core, and also increase the density of the negative ions in the electropositive edge effectively increasing its losses by transport. This option should be used very carefully since completely wrong results can be obtained if too much diffusion is used (the tuning parameter for ions should not be larger than 0.1). A useful strategy is to start with a large **Tuning parameter for ions** (for example, 0.5) and ramp it down using an **Auxiliary sweep**.

## INFLOW AND OUTFLOW

When solving for plasmas with chemistries that contain more than one element (for example, Ar and SF<sub>6</sub>) with a stationary solver the mass fraction of each element is not conserved if no constraint is used. This problem is similar in nature to the one found when solving for Navier–Stokes equations in steady state without fixing the pressure somewhere. The **Inflow** boundary condition fixes the mass fraction or mole fraction of specified species and it is used in this model as a real inlet for Ar and as a strategy to fix its mass fraction. An important aspect to remember is that no **Inflow** is used for SF<sub>6</sub> since this species is being computed **From mass constraint**. It is assumed that ions are neutralized at the outlet boundary and as such no **Outflow** boundary condition is applied. For the species to which an **Outflow** boundary condition is applied no surface reaction is applied at the same boundary to represent a species that flows out of the system without interacting with a surface.

## *Results and Discussion*

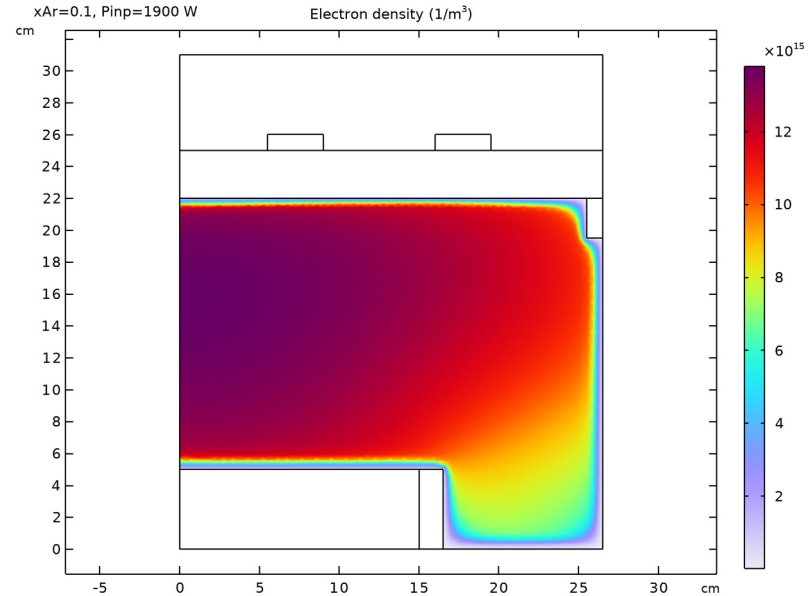
---

Figure 1 to Figure 4 show spatial distributions of the electron density, electron temperature, negative ion density, and power absorbed by the electrons at 1900 W with argon mole fraction of 0.1. The power absorbed by the electrons and electron temperature profiles are typical of an ICP discharge. Most of the power deposition occurs below the coil and is shielded by the high dense plasma, and the electron temperature is relatively flat in the plasma region. The electron and negative ion density have a typical distribution of electronegative plasmas with an electronegative core (where the electrons have a flat profile) and electropositive edges (where the negative ion density drops fast toward the surface). These regions can be better observed in Figure 5 that shows the distribution of the charged species along the axis of symmetry.

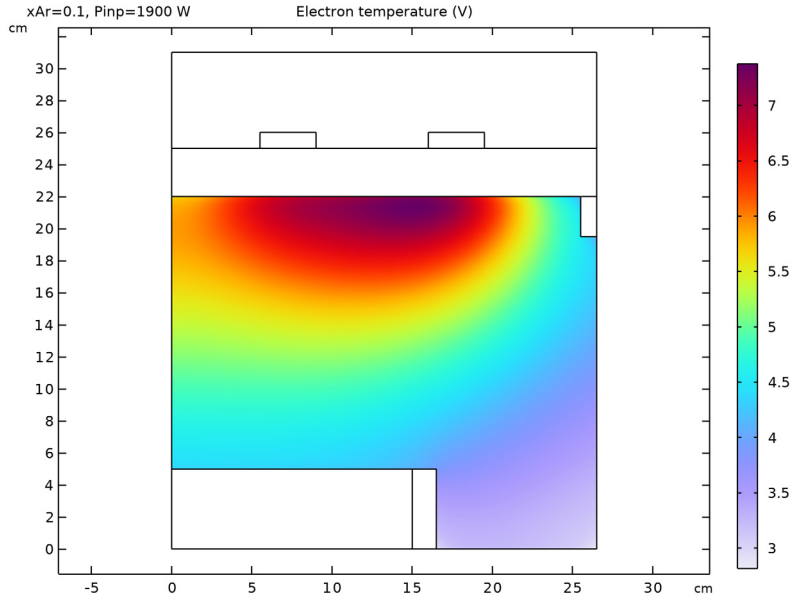
Figure 6 presents the space average number density of charged species as a function of power. For all the power range from 300 W to 1900 W, the negative ions  $\text{SF}_6^-$  and  $\text{F}^-$  are always the dominant species with electrons having 10 to 100 time less density.

Figure 7 presents the space average number density of charged species as a function of the mole fraction of argon. With low argon content the discharge has high electronegativity

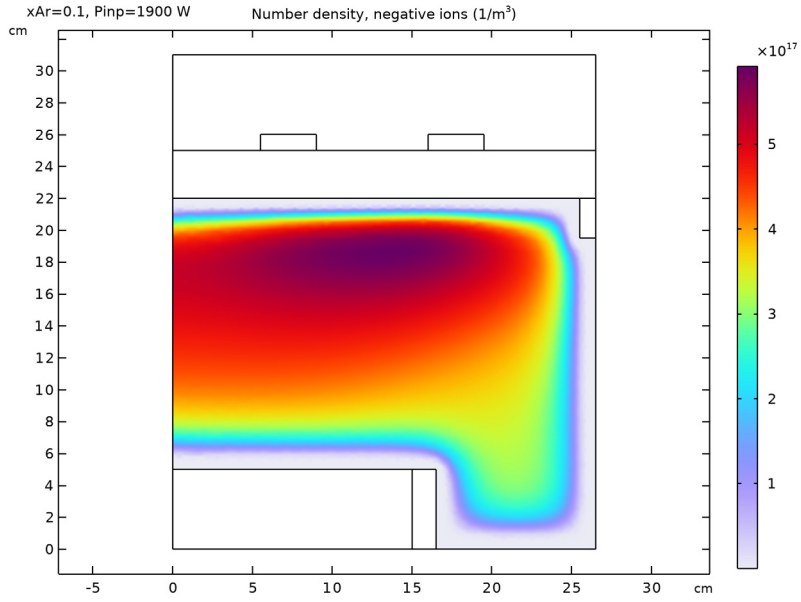
and  $\text{SF}_6^-$  is the dominant negative ion. Increasing the argon content leads to lower electronegativity and a progressive change of the dominant positive ion to be  $\text{Ar}^+$ .



*Figure 1: Electron number density spatial distribution at 1900 W with argon mole fraction of 0.1.*



*Figure 2: Electron temperature spatial distribution at 1900 W with argon mole fraction of 0.1.*



*Figure 3: Negative ions total number density spatial distribution at 1900 W with argon mole fraction of 0.1.*

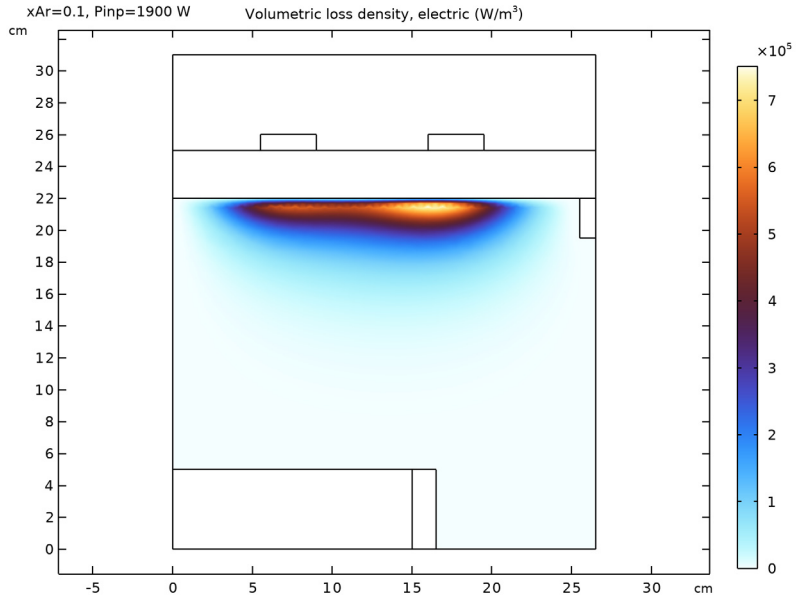


Figure 4: Power absorbed by the electrons at 1900 W with argon mole fraction of 0.1.

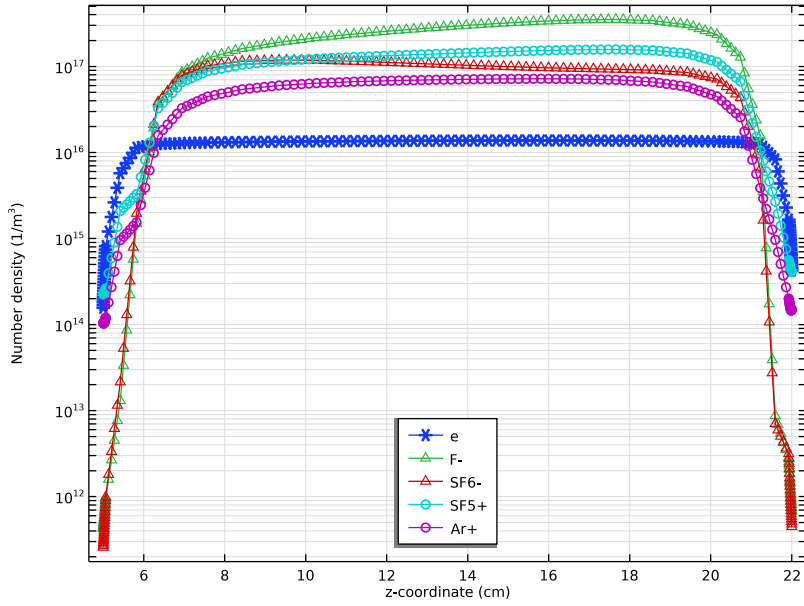


Figure 5: Charged species distribution along the axis-of-symmetry at 1900 W with argon mole fraction of 0.1.

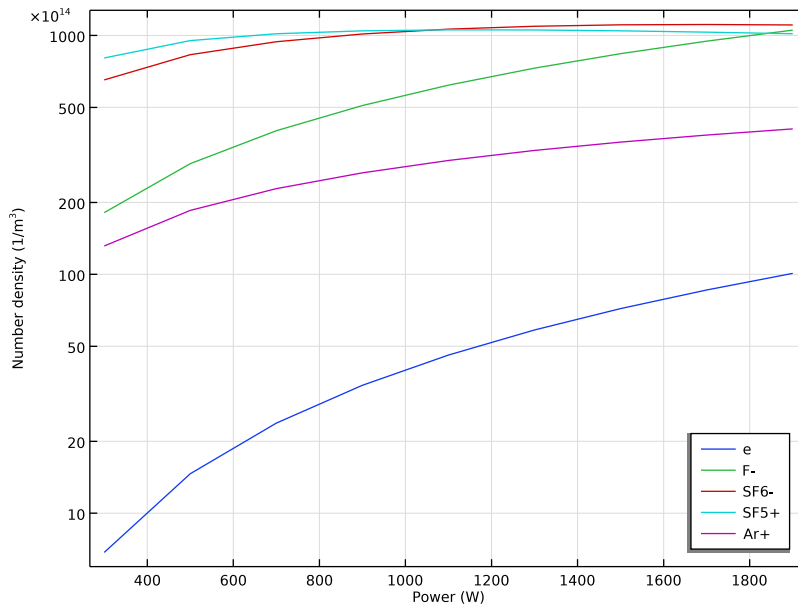


Figure 6: Spatial averaged number density of the charged species as a function of power with argon mole fraction of 0.1.

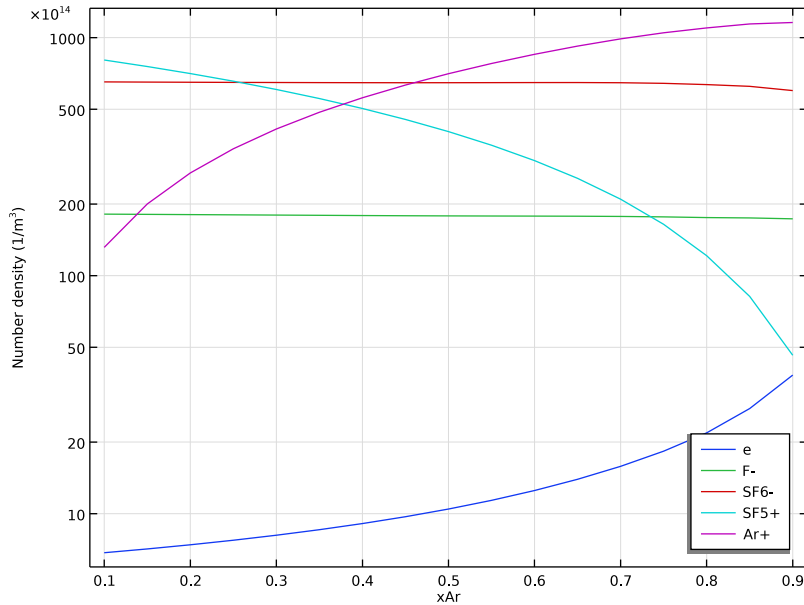


Figure 7: Spatial averaged number density of the charged species as a function of argon mole fraction at 300 W.

## References

1. M. Mao, Y.N. Wang, and A. Bogaerts, “Numerical study of the plasma chemistry in inductively coupled SF6 and SF6/Ar plasmas used for deep silicon etching applications,” *J. Phys. D: Appl. Phys.*, vol. 44, no. 43, p. 435202, 2011.
2. G. Kokkoris and others, “A global model for SF6 plasmas coupling reaction kinetics in the gas phase and on the surface of the reactor walls,” *J. Phys. D: Appl. Phys.*, vol. 42, no. 5, 055209, 2009.
3. L. Lallement, and others, “Global model and diagnostic of a low-pressure SF6/Ar inductively coupled plasma,” *Plasma Sources Sci. Technol.*, vol. 18, no. 2, p. 025001, 2009.
4. L.G. Christophorou and J.K. Olthoff, “Electron interactions with SF6,” *J. Phys. Chem. Ref. Data*, vol. 29, no. 3, pp. 267–330, 2000 ([www.lxcat.net](http://www.lxcat.net), retrieved May 2024).

5. V. Gedeon and others, “B-spline R-matrix-with-pseudostates calculations for electron-impact excitation and ionization of fluorine,” *Phys. Rev. A*, vol. 89, no. 5, p. 052713, 2014.
6. Biagi database, [www.lxcat.net](http://www.lxcat.net), retrieved on May 13, 2024.
7. Morgan database, [www.lxcat.net](http://www.lxcat.net), retrieved on May 17, 2024.
8. Phelps database, [www.lxcat.net](http://www.lxcat.net), retrieved 2017.
9. [www.lxcat.net](http://www.lxcat.net), retrieved 2023.
10. 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, p. 722, 2005.

---

**Application Library path:** Plasma\_Module/Inductively\_Coupled\_Plasmas/  
icp\_sf6\_argon


---

### *Modeling Instructions*




---

From the **File** menu, choose **New**.

#### **NEW**

In the **New** window, click  **Model Wizard**.

#### **MODEL WIZARD**

- 1 In the **Model Wizard** window, click  **2D Axisymmetric**.
- 2 In the **Select Physics** tree, select **Plasma > Nonisothermal Plasma Flow > Inductively Coupled Plasma**.
- 3 Click **Add**.
- 4 Click  **Study**.
- 5 In the **Select Study** tree, select **Preset Studies for Selected Multiphysics > Frequency–Transient**.
- 6 Click  **Done**.

## GLOBAL DEFINITIONS

### Parameters - Geometry

Add some parameters to be used to set up the geometry.

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters I**.
- 2 In the **Settings** window for **Parameters**, type Parameters - Geometry in the **Label** text field.
- 3 Locate the **Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
chamber_w	26.5[cm]	0.265 m	Chamber radius
chamber_h	31[cm]	0.31 m	Chamber height
window_z	22[cm]	0.22 m	Dielectric window z position (lower)
window_h	3[cm]	0.03 m	Dielectric window height
coil_r	5.5[cm]	0.055 m	Coil r position (left)
coil_w	3.5[cm]	0.035 m	Coil width
coil_h	1[cm]	0.01 m	Coil height
coil_dis	10.5[cm]	0.105 m	Coil array displacement
coil_num_r	2	2	Number of coils in r direction
substrate_w	15[cm]	0.15 m	Substrate radius
substrate_h	5[cm]	0.05 m	Substrate height
shell_w	1.5[cm]	0.015 m	Substrate outer shell thickness
inlet_block_w	1[cm]	0.01 m	Inlet block width
inlet_block_h	2.5[cm]	0.025 m	Inlet block height
inlet_h	0.5[cm]	0.005 m	Inlet height
inlet_z	20.3[cm]	0.203 m	Inlet z position (lower)

### Parameters - Physics

- 1 In the **Home** toolbar, click **P<sub>i</sub> Parameters** and choose **Add > Parameters**.  
Add some parameters to be used to set up the physics.
- 2 In the **Settings** window for **Parameters**, type Parameters - Physics in the **Label** text field.

**3** Locate the **Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
Qfeed	100	100	Feed mass flow in SCCM
Temp	315[K]	315 K	Gas temperature
pA	20[mTorr]	2.6664 Pa	Gas pressure
Pinp	300[W]	300 W	Power input from source
xAr	0.9	0.9	Argon mole fraction


## GEOMETRY I

**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**.

### Rectangle 1 (r1)


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

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

**3** In the **Width** text field, type chamber\_w.

**4** In the **Height** text field, type chamber\_h.

### Rectangle 2 (r2)

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


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

**3** In the **Width** text field, type chamber\_w.

**4** In the **Height** text field, type window\_h.

**5** Locate the **Position** section. In the **z** text field, type window\_z.

### Rectangle 3 (r3)

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

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


**3** In the **Width** text field, type coil\_w.

**4** In the **Height** text field, type coil\_h.


**5** Locate the **Position** section. In the **r** text field, type coil\_r.

**6** In the **z** text field, type window\_z+window\_h.


#### *Array 1 (arr1)*

- 1 In the **Geometry** toolbar, click  **Transforms** and choose **Array**.
- 2 Select the object **r3** only.
- 3 In the **Settings** window for **Array**, locate the **Size** section.
- 4 In the **r size** text field, type `coil_num_r`.
- 5 Locate the **Displacement** section. In the **r** text field, type `coil_dis`.


#### *Rectangle 4 (r4)*

- 1 In the **Geometry** toolbar, click  **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type `substrate_w`.
- 4 In the **Height** text field, type `substrate_h`.


#### *Rectangle 5 (r5)*


- 1 In the **Geometry** toolbar, click  **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type `shell_w`.
- 4 In the **Height** text field, type `substrate_h`.
- 5 Locate the **Position** section. In the **r** text field, type `substrate_w`.

#### *Rectangle 6 (r6)*

- 1 In the **Geometry** toolbar, click  **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type `inlet_block_w`.
- 4 In the **Height** text field, type `inlet_block_h`.
- 5 Locate the **Position** section. In the **r** text field, type `chamber_w-inlet_block_w`.
- 6 In the **z** text field, type `window_z-inlet_block_h`.

#### *Line Segment 1 (ls1)*


- 1 In the **Geometry** toolbar, click  **More Primitives** and choose **Line Segment**.
- 2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.
- 3 From the **Specify** list, choose **Coordinates**.
- 4 In the **r** text field, type `chamber_w-inlet_block_w`.
- 5 In the **z** text field, type `inlet_z`.
- 6 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

- 7 In the **r** text field, type `chamber_w-inlet_block_w`.
- 8 In the **z** text field, type `inlet_z+inlet_h`.
- 9 In the **Geometry** toolbar, click  **Build All**.
- 10 In the **Model Builder** window, collapse the **Geometry 1** node.


Define some explicit selections to be used later for boundary conditions.

## DEFINITIONS


### *Walls*

- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type `Walls` in the **Label** text field.
- 3 Select Domain 2 only.
- 4 Locate the **Output Entities** section. From the **Output entities** list, choose **Adjacent boundaries**.


### *Coils*

- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type `Coils` in the **Label** text field.
- 3 Select Domains 5 and 7 only.


### *Coil Boundaries*

- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type `Coil Boundaries` in the **Label** text field.
- 3 Select Domains 5 and 7 only.
- 4 Locate the **Output Entities** section. From the **Output entities** list, choose **Adjacent boundaries**.

### *Inlet*

- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type `Inlet` in the **Label** text field.
- 3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundary 27 only.


### *Outlet*

- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type `Outlet` in the **Label** text field.
- 3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 22 only.

Define an average function to be used later for plotting.

*Average 1 (aveop1)*

- 1 In the **Definitions** toolbar, click  **Nonlocal Couplings** and choose **Average**.
- 2 Select Domain 2 only.
- 3 In the **Model Builder** window, collapse the **Definitions** node.

#### **ADD MATERIAL FROM LIBRARY**


In the **Home** toolbar, click  **Windows** and choose **Add Material from Library**.

#### **ADD MATERIAL**

- 1 Go to the **Add Material** window.
- 2 In the tree, select **Built-in** > **Air**.
- 3 Click the **Add to Component** button in the window toolbar.

#### **MATERIALS**

*Air (mat1)*

- 1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
- 2 Click  **Clear Selection**.
- 3 Select Domain 4 only.

#### **ADD MATERIAL**

- 1 Go to the **Add Material** window.
- 2 In the tree, select **Built-in** > **Glass (quartz)**.
- 3 Click the **Add to Component** button in the window toolbar.

#### **MATERIALS**

*Glass (quartz) (mat2)*

Select Domain 3 only.

#### **ADD MATERIAL**

- 1 Go to the **Add Material** window.
- 2 In the tree, select **Built-in** > **Copper**.
- 3 Click the **Add to Component** button in the window toolbar.

4 In the **Home** toolbar, click  **Add Material** to close the **Add Material** window.

## MATERIALS

*Copper (mat3)*

- 1 Select Domains 5 and 7 only.
- 2 In the **Model Builder** window, collapse the **Component 1 (comp1) > Materials** node.

## THE PLASMA CHEMISTRY IMPORT FEATURE

The next steps have instructions to use the **Plasma Chemistry Import** feature to import a file that automatically creates the SF6/Ar plasma chemistry.





The following is set or created automatically:

- a Species properties
- b Reaction group features for SF6, F2 and Ar
- c Surface reactions


The documentation accompanying the **Plasma Chemistry Import** feature contains more information about the file structure and what can be set automatically.

## PLASMA (PLAS)

*Plasma Chemistry Import 1*

- 1 In the **Physics** toolbar, click  **Global** and choose **Plasma Chemistry Import**.
- 2 In the **Settings** window for **Plasma Chemistry Import**, locate the **Plasma Chemistry Import** section.
- 3 Click  **Browse**.
- 4 Browse to the model's Application Libraries folder and double-click the file SF6\_Ar\_plasma\_chemistry.txt.
- 5 Click  **Import**.
- 6 In the **Model Builder** window, click **Plasma (plas)**.
- 7 In the **Settings** window for **Plasma**, locate the **Domain Selection** section.
- 8 Click  **Clear Selection**.
- 9 Select Domain 2 only.
- 10 Locate the **Transport Settings** section. Select the **Mixture diffusion correction** checkbox.

This model needs stabilization because the density of the negative ions can drop sharply when approaching the reactor edges.

- 11 Click the  **Show More Options** button in the **Model Builder** toolbar.
  - 12 In the **Show More Options** dialog, select **Physics > Stabilization** in the tree.
  - 13 In the tree, select the checkbox for the node **Physics > Stabilization**.
  - 14 Click **OK**.
  - 15 In the **Settings** window for **Plasma**, click to expand the **Inconsistent Stabilization** section.
  - 16 Select the **Isotropic diffusion for ions** checkbox.
  - 17 In the  $\delta_{id,i}$  text field, type 0.1.
- Set properties for some species.

Set SF6 to be the species that the mass fraction is found from mass constraint.

*Species: SF6*

- 1 In the **Model Builder** window, expand the **Component 1 (comp 1) > Plasma (plas) > Group - Species** node, then click **Species: SF6**.
- 2 In the **Settings** window for **Species**, locate the **Species Formula** section.
- 3 Select the **From mass constraint** checkbox.

*Species: Ar*

- 1 In the **Model Builder** window, click **Species: Ar**.
- 2 In the **Settings** window for **Species**, locate the **General Parameters** section.
- 3 In the  $x_0$  text field, type  $xAr$ .

*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** checkbox.

Set the **Additional enthalpy contribution** field for species with internal energy so that the reaction enthalpy is correctly computed.

*Species: F2+*

- 1 In the **Model Builder** window, click **Species: F2+**.
- 2 In the **Settings** window for **Species**, click to expand the **Species Thermodynamic Parameters** section.
- 3 In the  $\Delta h$  text field, type 15.69.

*Species: F+*

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

- 2 In the **Settings** window for **Species**, locate the **Species Thermodynamic Parameters** section.
- 3 In the  $\Delta h$  text field, type 17.687.

*Species: Ars*

- 1 In the **Model Builder** window, click **Species: Ars**.
- 2 In the **Settings** window for **Species**, locate the **Species Thermodynamic Parameters** section.
- 3 In the  $\Delta h$  text field, type 11.5.

*Species: Ar+*

- 1 In the **Model Builder** window, click **Species: Ar+**.
- 2 In the **Settings** window for **Species**, locate the **Species Thermodynamic Parameters** section.
- 3 In the  $\Delta h$  text field, type 15.8.

*Group - Species*

In the **Model Builder** window, collapse the **Component 1 (comp1) > Plasma (plas) > Group - Species** node.

The surface reactions used in the model were created automatically but it is still necessary to specify the boundaries where they take place.

*Surface Reactions - Neutrals*

- 1 In the **Model Builder** window, click **Surface Reactions - Neutrals**.
- 2 In the **Settings** window for **Surface Reaction Group**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Walls**.
- 4 In the list box, select **22**.
- 5 Click **— Remove from Selection**.
- 6 Select Boundaries 3, 4, 6, 17, 21, 25–28, and 30 only.

*Surface Reactions - Ions*

- 1 In the **Model Builder** window, click **Surface Reactions - Ions**.
- 2 In the **Settings** window for **Surface Reaction Group**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Walls**.


*Plasma Model I*

- 1 In the **Model Builder** window, click **Plasma Model I**.
- 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  $1E15 [1/m^3]$ .

*Inflow 1*

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Inflow**.
- 2 In the **Settings** window for **Inflow**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Inlet**.
- 4 Locate the **Inflow** section. Click **+ Add**.
- 5 In the table, enter the following settings:

Species names	Mole fraction (1)
Ar	xAr

- 6 Click **+ Add**.
- 7 In the table, enter the following settings:

Species names	Mole fraction (1)
SF6	1 - xAr

- 8 Click **+ Add**.
- 9 In the table, enter the following settings:

Species names	Mole fraction (1)
SF5	1E - 8

- 10 Click **+ Add**.
- 11 In the table, enter the following settings:

Species names	Mole fraction (1)
SF4	1E - 8

- 12 Click **+ Add**.
- 13 In the table, enter the following settings:

Species names	Mole fraction (1)
SF3	1E - 8

- 14 Click **+ Add**.


15 In the table, enter the following settings:

Species names	Mole fraction (I)
SF2	1E-8

16 Click  Add.

17 In the table, enter the following settings:

Species names	Mole fraction (I)
SF	1E-8

18 Click  Add.

19 In the table, enter the following settings:

Species names	Mole fraction (I)
S	1E-8

20 Click  Add.

21 In the table, enter the following settings:

Species names	Mole fraction (I)
F	1E-8

22 Click  Add.

23 In the table, enter the following settings:

Species names	Mole fraction (I)
F2	1E-8

#### *Outflow 1*

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

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

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


#### *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 list box, select **3**.

- 5 Click  **Remove from Selection**.
- 6 Select Boundaries 4, 6, 17, 21, 22, 25–28, and 30 only.


#### *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**.


#### **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 Click  **Clear Selection**.
- 4 Select Domain 2 only.
- 5 Locate the **Physical Model** section. In the  $p_{\text{ref}}$  text field, type pA.


#### *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 **Inlet**.
- 4 Locate the **Boundary Condition** section. From the list, choose **Mass flow**.
- 5 Locate the **Mass Flow** section. From the **Mass flow type** list, choose **Standard flow rate (SCCM)**.
- 6 In the  $Q_{\text{sccm}}$  text field, type Qfeed.

#### *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**.


#### **HEAT TRANSFER IN FLUIDS (HT)**

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Fluids (ht)**.
- 2 In the **Settings** window for **Heat Transfer in Fluids**, locate the **Domain Selection** section.
- 3 Click  **Clear Selection**.
- 4 Select Domain 2 only.
- 5 Locate the **Physical Model** section. In the  $T_{\text{ref}}$  text field, type Temp.


### *Initial Values I*

- 1 In the **Model Builder** window, under **Component 1 (comp1) > Heat Transfer in Fluids (ht)** click **Initial Values I**.
- 2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.
- 3 In the  $T$  text field, type Temp.


### *Temperature I*

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Temperature**.
- 2 In the **Settings** window for **Temperature**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Walls**.
- 4 Locate the **Temperature** section. In the  $T_0$  text field, type Temp.

## **MAGNETIC FIELDS (MF)**

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Magnetic Fields (mf)**.
- 2 In the **Settings** window for **Magnetic Fields**, locate the **Domain Selection** section.
- 3 Click  **Clear Selection**.
- 4 Select Domains 2–5 and 7 only.

### *Domain Coil I*

- 1 In the **Physics** toolbar, click  **Domains** and choose **Domain Coil**.
- 2 In the **Settings** window for **Domain Coil**, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **Coils**.
- 4 Locate the **Coil** section. Select the **Coil group** checkbox.
- 5 From the **Coil excitation** list, choose **Power**.
- 6 In the  $P_{\text{coil}}$  text field, type Pinp.


## **MESH I**

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh I**.
- 2 In the **Settings** window for **Mesh**, locate the **Sequence Type** section.
- 3 From the list, choose **User-controlled mesh**.


### *Size*

- 1 In the **Model Builder** window, under **Component 1 (comp1) > Mesh I** click **Size**.
- 2 In the **Settings** window for **Size**, locate the **Element Size** section.
- 3 From the **Predefined** list, choose **Finer**.


### Size 1

- 1 In the **Model Builder** window, click **Size 1**.
- 2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
- 3 Click  **Clear Selection**.
- 4 Select Domain 2 only.
- 5 Locate the **Element Size** section. From the **Predefined** list, choose **Coarse**.

### Size 2

- 1 In the **Model Builder** window, click **Size 2**.
- 2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
- 3 From the **Selection** list, choose **Walls**.
- 4 In the list box, select **3**.
- 5 Click  **Remove from Selection**.
- 6 Select Boundaries 4, 6, 17, 21, 22, 25–28, and 30 only.
- 7 Locate the **Element Size** section. From the **Predefined** list, choose **Finer**.

### Mapped 1


- 1 In the **Mesh** toolbar, click  **Mapped**.
- 2 Drag and drop below **Size 2**.
- 3 In the **Settings** window for **Mapped**, locate the **Domain Selection** section.
- 4 From the **Geometric entity level** list, choose **Domain**.
- 5 From the **Selection** list, choose **Coils**.

### Distribution 1


- 1 Right-click **Mapped 1** and choose **Distribution**.
- 2 Select Boundaries 10, 13, 18, and 23 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 4 From the **Distribution type** list, choose **Predefined**.
- 5 In the **Number of elements** text field, type 25.
- 6 In the **Element ratio** text field, type 20.
- 7 Select the **Symmetric distribution** checkbox.

### Distribution 2

- 1 Right-click **Distribution 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Distribution**, locate the **Boundary Selection** section.

- 3 Click  **Clear Selection**.
- 4 Select Boundaries 11, 12, 19, and 20 only.
- 5 Locate the **Distribution** section. In the **Number of elements** text field, type 75.

#### *Edge 1*

- 1 In the **Mesh** toolbar, click  **More Generators** and choose **Edge**.
- 2 Drag and drop below **Mapped 1**.
- 3 In the **Settings** window for **Edge**, locate the **Boundary Selection** section.
- 4 From the **Selection** list, choose **Inlet**.

#### *Distribution 1*

- 1 Right-click **Edge 1** and choose **Distribution**.
- 2 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 3 In the **Number of elements** text field, type 8.

#### *Corner Refinement 1*

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

#### *Free Triangular 1*

- 1 In the **Model Builder** window, click **Free Triangular 1**.
- 2 In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.
- 3 From the **Geometric entity level** list, choose **Remaining**.

#### *Boundary Layers 1*

- 1 In the **Model Builder** window, click **Boundary Layers 1**.
- 2 In the **Settings** window for **Boundary Layers**, click to expand the **Corner Settings** section.
- 3 From the **Handling of sharp corners** list, choose **No special handling**.
- 4 Click to expand the **Transition** section. Clear the **Smooth transition to interior mesh** checkbox.

#### *Boundary Layer Properties 1*

- 1 In the **Model Builder** window, expand the **Boundary Layers 1** node, then click **Boundary Layer Properties 1**.
- 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 **Thickness adjustment factor** text field, type 1.

### *Boundary Layer Properties 2*

- 1 In the **Model Builder** window, right-click **Boundary Layer Properties 2** and choose **Disable**.
- 2 Right-click **Mesh 1** and choose **Build All**.


For the first study we do a decoupled plasma-only solve where the fluid flow and heat transfer are not solved. This is because a fully coupled model with complex electronegative chemistry is quite difficult to converge.

The solution of this study will be used in the next fully coupled study as initial conditions.

### **STUDY 1 - NO FLOW, NO HEAT**

- 1 In the **Model Builder** window, click **Study 1**.
- 2 In the **Settings** window for **Study**, type Study 1 - No Flow, No Heat 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 **Study 1 - No Flow, No Heat** 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 13.56[MHz].
- 4 Locate the **Physics and Variables Selection** section. In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkboxes for **Laminar Flow (spf)** and **Heat Transfer in Fluids (ht)**.
- 5 In the **Solve for** column of the table, under **Component 1 (comp1) > Multiphysics**, clear the checkbox for **Nonisothermal Plasma Flow 1 (nipf1)**.
- 6 In the **Study** toolbar, click  **Compute**.

Add a study to do a fully coupled stationary solve. This study will use the solution of the previous study as initial conditions to save computation time.

The **Initial damping factor** is set to 1 because a solution is used as initial condition.

### **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 - FULLY COUPLED

In the **Settings** window for **Study**, type Study 2 - Fully Coupled in the **Label** text field.


### Step 1: Frequency–Stationary

- 1 In the **Model Builder** window, under **Study 2 - Fully Coupled** click **Step 1: Frequency–Stationary**.
- 2 In the **Settings** window for **Frequency–Stationary**, locate the **Study Settings** section.
- 3 In the **Frequency** text field, type 13.56[MHz].
- 4 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**.
- 5 From the **Method** list, choose **Solution**.
- 6 From the **Study** list, choose **Study 1 - No Flow, No Heat, Frequency–Transient**.
- 7 From the **Time (s)** list, choose **Last**.

### Solution 2 (sol2)

- 1 In the **Study** toolbar, click  **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.
- 3 In the **Model Builder** window, expand the **Study 2 - Fully Coupled > Solver Configurations > Solution 2 (sol2) > Stationary Solver 1** node, then click **Fully Coupled 1**.
- 4 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.
- 5 In the **Initial damping factor** text field, type 1.

### Solver Configurations

- 1 In the **Model Builder** window, collapse the **Study 2 - Fully Coupled > Solver Configurations** node.
- 2 In the **Study** toolbar, click  **Compute**.

## RESULTS

*Electric Potential (plas), Electron Density (plas), Electron Temperature (plas), Magnetic Flux Density (mf), Magnetic Flux Density, Revolved Geometry (mf), Pressure (spf), Temperature (ht), Velocity (spf), Velocity, 3D (spf)*

- 1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Electron Density (plas)**, **Electron Temperature (plas)**, **Electric Potential (plas)**, **Velocity (spf)**, **Pressure (spf)**,

**Velocity, 3D (spf), Temperature (ht), Magnetic Flux Density (mf), and Magnetic Flux Density, Revolved Geometry (mf).**

- 2 Right-click and choose **Group**.



#### *Fully Coupled*

- 1 In the **Settings** window for **Group**, type Fully Coupled in the **Label** text field.
- 2 In the **Model Builder** window, collapse the **Fully Coupled** node.

In the following prepare a parameterization of the argon feed mole fraction  $x_{Ar}$  from 0.9 to 0.1. This study will use the solution of the previous study as initial conditions to save computation time.

The **Initial damping factor** is set to 1 because a solution is used as initial condition and because the next parameter will use the previous solution.


#### **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 3 - XAR SWEEP**

In the **Settings** window for **Study**, type Study 3 -  $x_{Ar}$  Sweep in the **Label** text field.

#### *Step 1: Frequency–Stationary*

- 1 In the **Model Builder** window, under **Study 3 -  $x_{Ar}$  Sweep** click **Step 1: Frequency–Stationary**.
- 2 In the **Settings** window for **Frequency–Stationary**, locate the **Study Settings** section.
- 3 In the **Frequency** text field, type 13.56[MHZ].
- 4 Locate the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.
- 5 From the **Method** list, choose **Solution**.
- 6 From the **Study** list, choose **Study 2 - Fully Coupled, Frequency–Stationary**.
- 7 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** checkbox.
- 8 Click  **Add**.

9 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
xAr (Argon mole fraction)		

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

11 Click  **Range**.

12 In the **Range** dialog, type 0.9 in the **Start** text field.

13 In the **Step** text field, type -0.05.

14 In the **Stop** text field, type 0.1.

15 Click **Add**.

#### *Solution 3 (sol3)*

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

2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node.


3 In the **Model Builder** window, expand the **Study 3 - xAr Sweep > Solver Configurations > Solution 3 (sol3) > Stationary Solver 1** node, then click **Fully Coupled 1**.

4 In the **Settings** window for **Fully Coupled 1**, locate the **Method and Termination** section.

5 In the **Initial damping factor** text field, type 1.

#### *Solver Configurations*

1 In the **Model Builder** window, collapse the **Study 3 - xAr Sweep > Solver Configurations** node.

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

## RESULTS

*Electric Potential (plas) 1, Electron Density (plas) 1, Electron Temperature (plas) 1, Magnetic Flux Density (mf) 1, Magnetic Flux Density, Revolved Geometry (mf) 1, Pressure (spf) 1, Temperature (ht) 1, Velocity (spf) 1, Velocity, 3D (spf) 1*

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Electron Density (plas) 1, Electron Temperature (plas) 1, Electric Potential (plas) 1, Velocity (spf) 1, Pressure (spf) 1, Velocity, 3D (spf) 1, Temperature (ht) 1, Magnetic Flux Density (mf) 1, and Magnetic Flux Density, Revolved Geometry (mf) 1**.

2 Right-click and choose **Group**.

#### *xAr Sweep*



1 In the **Settings** window for **Group**, type xAr Sweep in the **Label** text field.

2 In the **Model Builder** window, collapse the **xAr Sweep** node.

In the following prepare a parameterization of the input power from 300 to 2000 W. This study will use the solution of the previous study as initial conditions to save computation time.

The **Initial damping factor** is set to 1 because a solution is used as initial condition and because the next parameter will use the previous solution.

#### 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 4 - POWER SWEEP

In the **Settings** window for **Study**, type Study 4 - Power Sweep in the **Label** text field.

##### Step 1: Frequency–Stationary

- 1 In the **Model Builder** window, under **Study 4 - Power Sweep** click **Step 1: Frequency–Stationary**.
- 2 In the **Settings** window for **Frequency–Stationary**, locate the **Study Settings** section.
- 3 In the **Frequency** text field, type 13.56[MHZ].
- 4 Locate the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.
- 5 From the **Method** list, choose **Solution**.
- 6 From the **Study** list, choose **Study 3 - xAr Sweep, Frequency–Stationary**.
- 7 From the **Parameter value (xAr)** list, choose **Last**.
- 8 Locate the **Study Extensions** section. Select the **Auxiliary sweep** checkbox.
- 9 Click **+ Add**.
- 10 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Pinp (Power input from source)		W

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

12 Click  **Range**.


13 In the **Range** dialog, type 300 in the **Start** text field.

14 In the **Step** text field, type 200.

15 In the **Stop** text field, type 2000.

16 Click **Add**.

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

18 Click  **Add**.

19 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
xAr (Argon mole fraction)	0.1	

20 From the **Sweep type** list, choose **All combinations**.

*Solution 4 (sol4)*

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

2 In the **Model Builder** window, expand the **Solution 4 (sol4)** node.


3 In the **Model Builder** window, expand the **Study 4 - Power Sweep > Solver Configurations > Solution 4 (sol4) > Stationary Solver 1** node, then click **Fully Coupled 1**.

4 In the **Settings** window for **Fully Coupled**, locate the **Method and Termination** section.

5 In the **Initial damping factor** text field, type 1.

*Solver Configurations*

1 In the **Model Builder** window, collapse the **Study 4 - Power Sweep > Solver Configurations** node.

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

## RESULTS

*Electric Potential (plas) 2, Electron Density (plas) 2, Electron Temperature (plas) 2, Magnetic Flux Density (mf) 2, Magnetic Flux Density, Revolved Geometry (mf) 2, Pressure (spf) 2, Temperature (ht) 2, Velocity (spf) 2, Velocity, 3D (spf) 2*

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Electron Density (plas) 2, Electron Temperature (plas) 2, Electric Potential (plas) 2, Velocity (spf) 2, Pressure (spf) 2, Velocity, 3D (spf) 2, Temperature (ht) 2, Magnetic Flux Density (mf) 2, and Magnetic Flux Density, Revolved Geometry (mf) 2**.

2 Right-click and choose **Group**.

#### *Power Sweep*

In the **Settings** window for **Group**, type Power Sweep in the **Label** text field.

In the following create a surface plot for the total number density of all negative species except electrons.

#### *Negative Ions Density*

1 In the **Model Builder** window, right-click **Electron Density (plas) 2** and choose **Duplicate**.

2 In the **Settings** window for **2D Plot Group**, type Negative Ions Density in the **Label** text field.

#### *Surface 1*

1 In the **Model Builder** window, expand the **Negative Ions Density** node, then click **Surface 1**.

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

3 In the **Expression** text field, type  $p_{1as} \cdot nm$ .

4 In the **Negative Ions Density** toolbar, click  **Plot**.

In the following create a surface plot for the power absorbed by electrons.

#### *Power Absorbed by Electrons*

1 In the **Model Builder** window, right-click **Temperature (ht) 2** and choose **Duplicate**.

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

3 Click to expand the **Selection** section. From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 2 only.

5 Click to collapse the **Selection** section.

#### *Surface 1*

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

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


3 In the **Expression** text field, type  $mf \cdot Qrh$ .

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

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

In the following create a 1D line plot for the charged species number densities along axis-of-symmetry.

#### *Charged Species Along Axis-of-Symmetry*

- 1 In the **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type Charged Species Along Axis-of-Symmetry in the **Label** text field.
- 3 Locate the **Data** section. From the **Dataset** list, choose **Study 4 - Power Sweep/ Solution 4 (sol4)**.
- 4 From the **Parameter selection (Pinp)** list, choose **Last**.
- 5 Click to expand the **Title** section. From the **Title type** list, choose **None**.
- 6 Click to collapse the **Title** section. Locate the **Plot Settings** section.
- 7 Select the **y-axis label** checkbox. In the associated text field, type Number density ( $1/m^{3}$ ).
- 8 Locate the **Axis** section. Select the **y-axis log scale** checkbox.
- 9 Locate the **Legend** section. From the **Position** list, choose **Lower middle**.

#### *Line Graph 1*

- 1 Right-click **Charged Species Along Axis-of-Symmetry** and choose **Line Graph**.
- 2 Select Boundary 3 only.
- 3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
- 4 Select the **Description** checkbox. In the associated text field, type e.
- 5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
- 6 In the **Expression** text field, type z.
- 7 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Asterisk**.
- 8 Click to expand the **Legends** section. Select the **Show legends** checkbox.
- 9 Find the **Include** subsection. Clear the **Solution** checkbox.
- 10 Select the **Description** checkbox.
- 11 Click to collapse the **Legends** section.

#### *Line Graph 2*

- 1 Right-click **Line Graph 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
- 3 In the **Expression** text field, type  $p_{las.n\_wF\_1m}$ .

- 4 In the **Description** text field, type F-.
- 5 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Triangle**.


#### *Line Graph 3*

- 1 Right-click **Line Graph 2** and choose **Duplicate**.
- 2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
- 3 In the **Expression** text field, type `plasn_wSF6_1m`.
- 4 In the **Description** text field, type SF6-.

#### *Line Graph 4*


- 1 Right-click **Line Graph 3** and choose **Duplicate**.
- 2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
- 3 In the **Expression** text field, type `plasn_wSF5_1p`.
- 4 In the **Description** text field, type SF5+.
- 5 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

#### *Line Graph 5*

- 1 Right-click **Line Graph 4** and choose **Duplicate**.
- 2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
- 3 In the **Expression** text field, type `plasn_wAr_1p`.
- 4 In the **Description** text field, type Ar+.
- 5 In the **Charged Species Along Axis-of-Symmetry** toolbar, click  **Plot**.

In the following create a 1D line plot for the spatial averaged number density of the charged species as a function of power.

#### *Space Averaged Charged Species vs. Power*


- 1 In the **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type Space Averaged Charged Species vs. Power in the **Label** text field.
- 3 Locate the **Data** section. From the **Dataset** list, choose **Study 4 - Power Sweep/ Solution 4 (sol4)**.
- 4 Click to expand the **Title** section. From the **Title type** list, choose **None**.
- 5 Click to collapse the **Title** section. Locate the **Plot Settings** section.
- 6 Select the **x-axis label** checkbox. In the associated text field, type Power (W).

- 7 Select the **y-axis label** checkbox. In the associated text field, type Number density ( $1/m^{>3</sup>}$ ).
- 8 Locate the **Axis** section. Select the **y-axis log scale** checkbox.
- 9 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

#### Global 1

- 1 Right-click **Space Averaged Charged Species vs. Power** and choose **Global**.
- 2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.
- 3 In the table, enter the following settings:

Expression	Unit	Description
aveop1(plas.ne)	$1/m^3$	e
aveop1(plas.n_wF_1m)	$1/m^3$	F-
aveop1(plas.n_wSF6_1m)	$1/m^3$	SF6-
aveop1(plas.n_wSF5_1p)	$1/m^3$	SF5+
aveop1(plas.n_wAr_1p)	$1/m^3$	Ar+


- 4 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Pinp**.
- 5 Click to expand the **Legends** section. Find the **Include** subsection. Clear the **Solution** checkbox.
- 6 Click to collapse the **Legends** section. In the **Space Averaged Charged Species vs. Power** toolbar, click  **Plot**.

In the following create a 1D line plot for the spatial averaged number density of the charged species as a function of argon mole fraction.

#### Space Averaged Charged Species vs. Power

In the **Model Builder** window, right-click **Space Averaged Charged Species vs. Power** and choose **Copy**.

#### Space Averaged Charged Species vs. xAr

- 1 In the **Model Builder** window, right-click **xAr Sweep** and choose **Paste ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type Space Averaged Charged Species vs. xAr in the **Label** text field.
- 3 Locate the **Data** section. From the **Dataset** list, choose **Study 3 - xAr Sweep/ Solution 3 (sol3)**.
- 4 Locate the **Plot Settings** section. In the **x-axis label** text field, type xAr.
- 5 In the **Space Averaged Charged Species vs. xAr** toolbar, click  **Plot**.

#### *xAr Sweep*

In the **Model Builder** window, collapse the **Results > xAr Sweep** node.

#### *Power Sweep*

In the **Model Builder** window, collapse the **Results > Power Sweep** node.