Impedance Matching

Introduction

A radio frequency (RF) power supply is used to supply power to a capacitively coupled plasma (CCP). Commercially available RF power supplies are specified to deliver power into a 50 ohm load, and interconnecting coaxial cables and connectors have a 50 ohm characteristic impedance. On the other hand the CCP impedance is typically not equal to 50 ohm and has both resistive and reactive components. For instance in this example the plasma impedance is $Z_p = R_p + jX_p$ at 13.56 MHz and 10 W. A matching network is introduced between the RF power supply and the CCP allowing maximum power transfer and a safe operating region for the power supply. In operation the match components may be fixed or variable and "tuned" either manually or automatically. Modern plasma reactors may be powered by multiple power supplies at different operating frequencies. This note shows how a match can be designed using dimensions of the chamber, properties of the feed gas and the specification of the power supply.

Model Definition

Figure 1 is a schematic of a RF power supply (V_s, R_s) connected to a L–type match network (L_m, C_p) with an additional DC blocking capacitor (C_b). The match network is connected to a plasma load with a parallel stray capacitance (C_s). The plasma voltage and current are V and I.

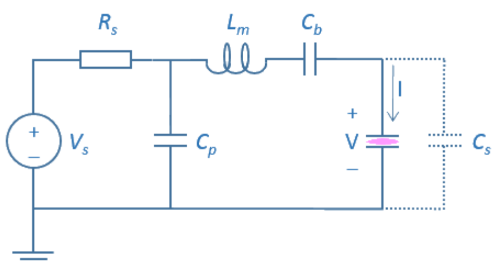

Figure 1: L–match including blocking and stray capacitance.

Figure 2 shows a simplified schematic, in this case the blocking capacitor (C_b) is removed (since the discharge is symmetric and there will be no DC self-bias). The stray capacitance (C_s) is ignored.

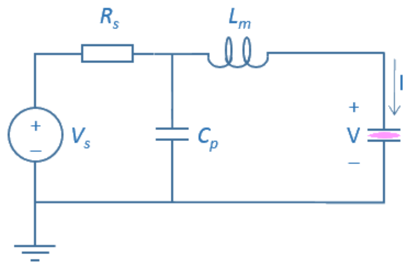

Figure 2: L–match used in this model.

In order to compute the optimum values for the parallel capacitor C_p and the inductor L_m, the plasma impedance is required, Z_p. Using $Z_p = R_p + jX_p$ and the generator source resistance, R_s, the following can be used to compute C_p and L_m. Defining:

$$X_m = \sqrt{R_p R_s - R_p^2} - X_p \tag{1}$$

and

$$B_m = \sqrt{\frac{1}{R_p R_s} - \frac{1}{R_s^2}} \tag{2}$$

then the match inductance is given by:

$$L_m = \frac{X_m}{2\pi f} \tag{3}$$

and the parallel capacitance is given by:

$$C_p = \frac{B_m}{2\pi f}. \tag{4}$$

Results and Discussion

In order to evaluate the results, two meaningful measures are used, the maximum power transfer coefficient and the efficiency. The maximum power transfer is given by:

$$P_{\max} = \frac{V_s^2}{8R_s} \tag{5}$$

where V_s is the generator voltage, and the maximum power transfer coefficient is then given by:

$$\alpha = \frac{P_{\text{plasma}}}{P_{\text{max}}}. \tag{6}$$

The efficiency is then simply:

$$\eta = \frac{P_{\text{plasma}}}{P_{\text{plasma}} + P_s} \tag{7}$$

where P_s is the power lost in the generator.

Figure 3 shows that the maximum of power transfer coefficient is 1 when the plasma power dissipation is 10 W. This is expected because the values of C_p and L_m used to calculated the match components were determined at 10 W plasma power.

Figure 4 and Figure 5 show a plot of maximum power transfer coefficient as a function of frequency and pressure respectively. Matching occurs at 13.56 MHz and 1 torr corresponding to the conditions used to determine C_p and L_m. At low pressure the plasma is more resistive and a significant mismatch occurs. The effect is less pronounced at higher pressures.

Effect of Harmonics

Finally, the plasma impedance was measured at 50 W, a match was designed. Figure 6 shows the power transfer coefficient as a function of power. It is evident that the maximum value is around 45 W, 10% lower than expected.

The match parameters are calculated from the voltage and current amplitudes at the fundamental frequency. However the current waveform has a significant third harmonic contribution (4.5%) which accounts for the difference between the measured and calculated power at maximum power transfer coefficient.

These results show how a matching network for a CCP reactor can be designed using the dimensions of the reactor, the properties of the feed gas and the characteristics of the generator. Estimates of the power transfer coefficient as a function of power frequency and pressure can be made. The same procedure can be used for 2D reactor geometries.

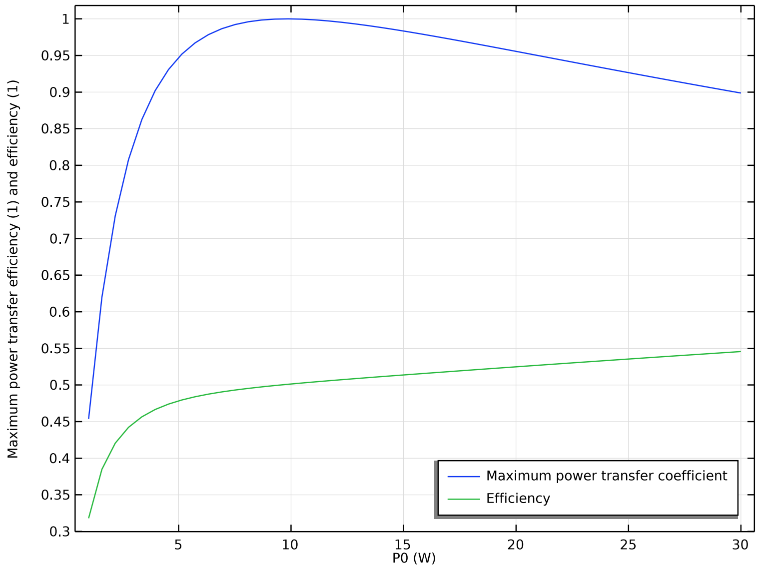

Figure 3: Plot of the power transfer coefficient and efficiency versus power.

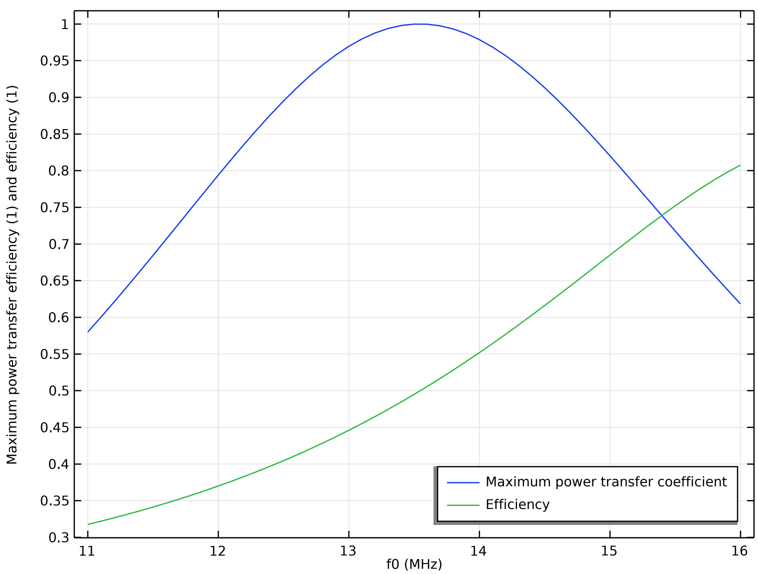

Figure 4: Plot of the power transfer coefficient and efficiency versus frequency.

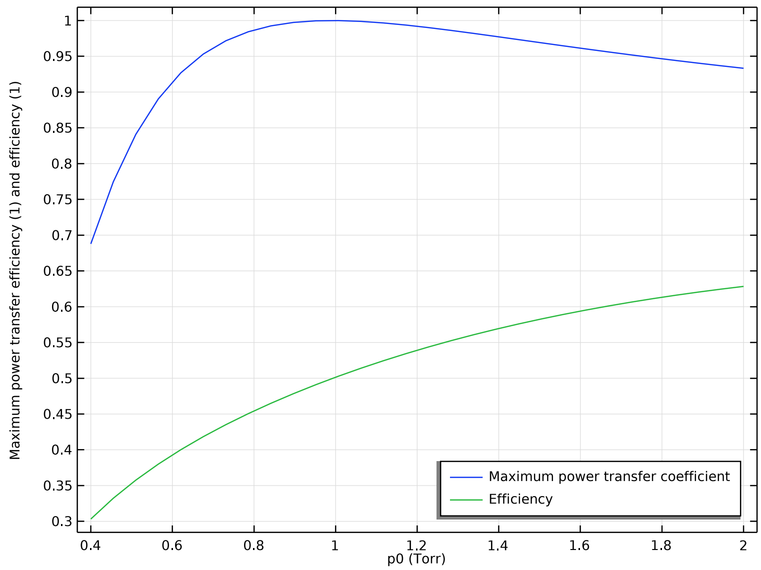

Figure 5: Plot of the power transfer coefficient and efficiency versus pressure.

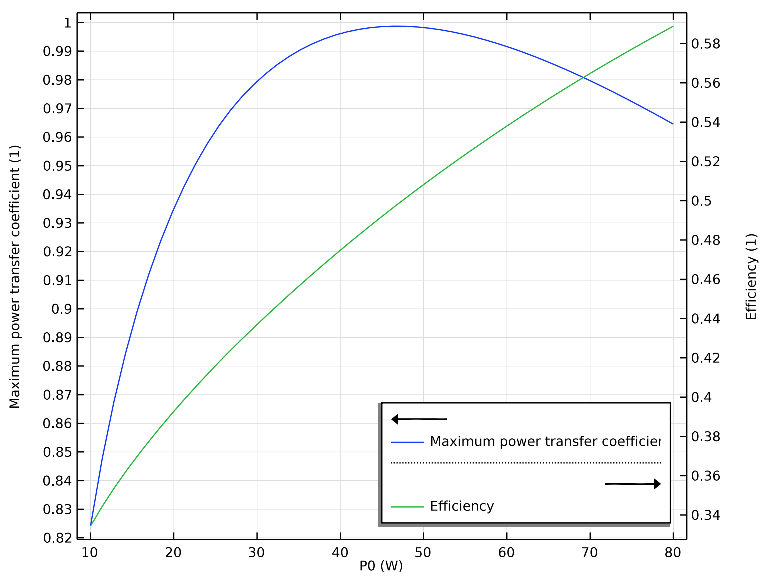

Figure 6: Plot of the power transfer coefficient and efficiency versus power at high power.

Reference

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

Application Library path: `Plasma_Module/Capacitively_Coupled_Plasmas/impedance_matching`

Modeling Instructions

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

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, The model is set up in an identical way to the computing plasma impedance model, except an external circuit is used to drive the discharge.

2 click **1D**.

3 In the **Select Physics** tree, select **Plasma>Plasma, Time Periodic (ptp)**.

4 Click **Add**.

5 Click **Study**.

6 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces> Time Periodic**.

7 Click **Done**.

GEOMETRY 1

Add parameters to compute the match inductance and capacitance.

GLOBAL DEFINITIONS

Parameters 1

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
L	0.025[m]	0.025 m	Discharge gap
de	0.3[m]	0.3 m	Electrode diameter
As	0.25*pi*de^2	0.070686 m^2	Electrode area
P0	10[W]	10 W	Input power
f0	13.56E6[Hz]	1.356E7 Hz	Frequency
p0	1[torr]	133.32 Pa	Pressure
T0	300[K]	300 K	Temperature
Rp	42.696[ohm]	42.696 Ω	Plasma impedance, real part
Xp	-156.62[ohm]	-156.62 Ω	Plasma impedance, imaginary part
Rs	50[ohm]	50 Ω	Generator impedance
fmatch	13.56E6[Hz]	1.356E7 Hz	Frequency at perfect match
Xmd	sqrt(Rp*Rs-Rp^2)-Xp	174.28 Ω	Match help variable
Bm	((1/(Rp*Rs))-(1/Rs^2))^0.5	0.0082721 S	Match help variable
Lm	Xmd/(2*pi*fmatch)	2.0455E-6 H	Inductance
Cp	Bm/(2*pi*fmatch)	9.7091E-11 F	Capacitance

GEOMETRY 1

Interval 1 (i1)

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

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

3 In the table, enter the following settings:

Coordinates (m)
0
L

4 Click **Build All Objects**.

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

PLASMA, TIME PERIODIC (PTP)

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

2 In the **Settings** window for **Plasma, Time Periodic**, locate the **Cross-Section Area** section.

3 In the A text field, type `As`.

4 Locate the **Extra Dimension Settings** section. In the P_{xd} text field, type `1/f0`.

5 In the N text field, type `30`.

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

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 `He_xsecs.txt`.

5 Click **Import**.

Species: He

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

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

3 Select the **From mass constraint** check box.

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

Species: Hes

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

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

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

Species: He+

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

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

3 Select the **Initial value from electroneutrality constraint** check box.

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

5 Locate the **Mobility and Diffusivity Expressions** section. From the **Specification** list, choose **Specify mobility, compute diffusivity**.

6 From the **Ion temperature** list, choose **Use local field approximation**.

7 Locate the **Mobility Specification** section. From the **Specify using** list, choose **Helium ion in helium**.

Plasma Model 1

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

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

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

4 In the p_A text field, type `p0`.

Surface Reaction 1

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

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

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

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

5 Locate the **Reaction Parameters** section. In the γ_f text field, type `0`.

6 Locate the **Secondary Emission Parameters** section. In the γ_i text field, type `0.1`.

7 In the ε_i text field, type `5.8`.

2: He+=>He

1 Right-click **1: He+=>He** and choose **Duplicate**.

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

3 In the **Formula** text field, type `Hes=>He`.

4 Locate the **Reaction Parameters** section. In the γ_f text field, type `1`.

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 **All boundaries**.

Ground 1

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

2 Select Boundary 2 only.

Metal Contact 1

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

2 Select Boundary 1 only.

3 In the **Settings** window for **Metal Contact**, locate the **Terminal** section.

4 From the **Terminal type** list, choose **Circuit**.

5 From the **Circuit type** list, choose **L-network**.

6 From the **Source type** list, choose **Power source**.

7 Locate the **Circuit Settings** section. In the P_s text field, type `P0`.

8 In the R_s text field, type `Rs`.

9 In the C_p text field, type `Cp`.

10 In the L_m text field, type `Lm`.

11 In the f_p text field, type `f0`.

MESH 1

Edge 1

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

Distribution 1

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

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

3 From the **Distribution type** list, choose **Predefined**.

4 In the **Number of elements** text field, type `125`.

5 In the **Element ratio** text field, type `10`.

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

7 Click **Build All**.

Since we're not really interested in the spatial distribution of the plasma variables, disable the default plots.

POWER SWEEP

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

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

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

4 Clear the **Generate convergence plots** check box.

First, sweep over power.

Step 1: Time Periodic

1 In the **Model Builder** window, under **Power Sweep** click **Step 1: Time Periodic**.

2 In the **Settings** window for **Time Periodic**, click to expand the **Study Extensions** section.

3 Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

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

6 Click to select row number 1 in the table.

7 Click **Range**.

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

9 In the **Start** text field, type `1`.

10 In the **Stop** text field, type `30`.

11 In the **Number of values** text field, type `50`.

12 Click **Replace**.

13 In the **Settings** window for **Time Periodic**, locate the **Study Extensions** section.

14 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
P0 (Input power)	`range(1,0.5918367346938775,30)`	W

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

RESULTS

Add a plot to see how the efficiency changes with power.

1D Plot Group 1

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

2 Right-click **Results** and choose **1D Plot Group**.

Global 1

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

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Plasma, Time Periodic>Metal Contact 1>ptp.mct1.alphaP - Maximum power transfer coefficient - 1**.

3 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Plasma, Time Periodic>Metal Contact 1> ptp.mct1.etaP - Efficiency - 1**.

4 In the **1D Plot Group 1** toolbar, click **Plot**.

Power Sweep

1 In the **Model Builder** window, click **1D Plot Group 1**.

2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.

3 Select the **y-axis label** check box. In the associated text field, type `Maximum power transfer efficiency (1) and efficiency (1)`.

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

5 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

6 In the **1D Plot Group 1** toolbar, click **Plot**.

7 In the **Label** text field, type `Power Sweep`.

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 Physics Interfaces>Time Periodic**.

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

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

STUDY 2

Step 1: Time Periodic

Now sweep over frequency.

1 In the **Settings** window for **Time Periodic**, locate the **Study Extensions** section.

2 Select the **Auxiliary sweep** check box.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
f0 (Frequency)		Hz

5 Click to select row number 1 in the table.

6 Click **Range**.

7 In the **Range** dialog box, type `11` in the **Start** text field.

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

9 In the **Stop** text field, type `16`.

10 In the **Number of values** text field, type `51`.

11 Click **Replace**.

12 In the **Settings** window for **Time Periodic**, locate the **Study Extensions** section.

13 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
f0 (Frequency)	`range(11,0.1,16)`	`MHz`

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

15 In the **Settings** window for **Study**, locate the **Study Settings** section.

16 Clear the **Generate default plots** check box.

17 Clear the **Generate convergence plots** check box.

18 In the **Label** text field, type `Frequency Sweep`.

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

RESULTS

Frequency Sweep

1 In the **Model Builder** window, right-click **Power Sweep** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Frequency Sweep` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Frequency Sweep/Solution 2 (sol2)**.

Global 1

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

2 From the **Home** menu, choose **Add Study**.

ADD STUDY

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

2 Find the **Studies** subsection. In the **Select Study** tree, select
Preset Studies for Selected Physics Interfaces>Time Periodic.

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

4 From the **Home** menu, choose **Add Study**.

STUDY 3

Step 1: Time Periodic

Next, sweep over pressure.

1 In the **Settings** window for **Time Periodic**, locate the **Study Extensions** section.

2 Select the **Auxiliary sweep** check box.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
p0 (Pressure)		Pa

5 Click to select row number 1 in the table.

6 Click **Range**.

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

8 In the **Start** text field, type `2`.

9 In the **Stop** text field, type `0.4`.

10 In the **Number of values** text field, type `30`.

11 Click **Replace**.

12 In the **Settings** window for **Time Periodic**, locate the **Study Extensions** section.

13 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
p0 (Pressure)	`range(2,-0.055172413793103454, 0.4)`	`torr`

14 In the **Model Builder** window, expand the **Frequency Sweep** node, then click **Study 3**.

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

16 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

17 Clear the **Generate convergence plots** check box.

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

RESULTS

Pressure Sweep

1 In the **Model Builder** window, right-click **Frequency Sweep** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Pressure Sweep` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Pressure Sweep/Solution 3 (sol3)**.

Global 1

1 In the **Model Builder** window, expand the **Pressure Sweep** node, then click **Global 1**.

2 In the **Settings** window for **Global**, locate the **x-Axis Data** section.

3 From the **Unit** list, choose **Torr**.

4 In the **Pressure Sweep** toolbar, click **Plot**.

Finally, change the impedance so that the match occurs at 50 W, then sweep over power once more.

GLOBAL DEFINITIONS

Parameters 1

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
Rp	29.175[ohm]	29.175 Ω	Plasma impedance, real part
Xp	-126.47[ohm]	-126.47 Ω	Plasma impedance, imaginary part

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 Physics Interfaces>Time Periodic**.

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

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

STUDY 4

Step 1: Time Periodic

1 In the **Settings** window for **Time Periodic**, locate the **Study Extensions** section.

2 Select the **Auxiliary sweep** check box.

3 Click **Add**.

4 In the table, enter the following settings:

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

5 Click to select row number 1 in the table.

6 Click **Range**.

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

8 In the **Start** text field, type `10`.

9 In the **Stop** text field, type `80`.

10 In the **Number of values** text field, type `51`.

11 Click **Replace**.

12 In the **Settings** window for **Time Periodic**, locate the **Study Extensions** section.

13 In the table, enter the following settings:

Parameter name	**Parameter value list**	**Parameter unit**
P0 (Input power)	range(10,1.4,80)	W

14 In the **Model Builder** window, click **Study 4**.

15 In the **Settings** window for **Study**, locate the **Study Settings** section.

16 Clear the **Generate convergence plots** check box.

17 Clear the **Generate default plots** check box.

18 In the **Label** text field, type `High Power Sweep`.

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

RESULTS

1D Plot Group 4

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

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **High Power Sweep/Solution 4 (sol4)**.

Global 1

1 Right-click **1D Plot Group 4** and choose **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Plasma,**

Time Periodic>Metal Contact 1>ptp.mct1.alphaP - Maximum power transfer coefficient - 1.

3 In the **1D Plot Group 4** toolbar, click **Plot**.

Global 2

1 In the **Model Builder** window, right-click **1D Plot Group 4** and choose **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Plasma, Time Periodic>Metal Contact 1>ptp.mct1.etaP - Efficiency - 1**.

High Power Sweep

1 In the **Model Builder** window, under **Results** click **1D Plot Group 4**.

2 In the **Settings** window for **1D Plot Group**, type `High Power Sweep` in the **Label** text field.

3 Locate the **Plot Settings** section. Select the **Two y-axes** check box.

4 In the table, select the **Plot on secondary y-axis** check box for **Global 2**.

5 Locate the **Title** section. From the **Title type** list, choose **None**.

6 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

7 In the **High Power Sweep** toolbar, click **Plot**.

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