Multipactor

Introduction

The multipactor effect is a phenomenon in radio frequency (RF) amplifier vacuum tubes and waveguides, where, under certain conditions, secondary electron emission in resonance with an alternating electric field leads to exponential electron multiplication, possibly damaging and even destroying the RF device (Ref. 1). This effect can be modeled in COMSOL for a device with metallic electrodes using a particle based approach.

Note: This model requires the Particle Tracing Module.

Model Definition

The model is solved in two stages. First, the electric potential is computed inside the multipactor using a frequency domain study. This solution is then stored, then used to compute the electric field for a second time dependent study which computes the particle trajectories. The multipaction effect is implemented using the secondary emission boundary condition for the particles.

The multipaction effect is strongest when the applied voltage, V_0, the gap thickness, d, and the RF frequency, f, are in a specific ratio:

$$f = \frac{1}{2d\sqrt{\pi}}\sqrt{\frac{qV_0}{m}} \tag{1}$$

When setting up the model, the gap thickness is set to 1 cm, the applied voltage is 100 V and the applied frequency is set using Equation 1.

MODEL EQUATIONS

The electric field is computed using an electric currents formulation in the frequency domain:

$$-\nabla \cdot j\omega\varepsilon_0\nabla V = 0 \tag{2}$$

where V is the electric potential, ω is the angular frequency, j is the square root of -1 and ε_0 is the vacuum permittivity.

For the particle trajectories, a time dependent study is employed and the equations of motion are solved assuming no damping of electron motion due to collisions with the background gas:

$$\frac{d}{dt}(m\mathbf{v}) = q\mathbf{E}\,. \tag{3}$$

The electric field must be real and be sinusoidal in time and is given by the expression

$$\mathbf{E} = \mathrm{real}(-\nabla V \exp(j\omega t))\,. \tag{4}$$

When a primary particle strikes the wall, it is assumed to be absorbed into the wall and thus disappears from the modeling domain. However, each primary particle ejects 3 secondary particles, each with 1/3 of the energy of the primary particle. The secondary particles are released with a hemispherical velocity distribution whose north pole points away from the surface normal. Mathematically, the velocity of the secondary particles is given by

$$\begin{aligned} v_x &= c\cos\theta\sin\varphi \\ v_y &= c\sin\theta\sin\varphi \\ v_z &= c\cos\varphi \end{aligned} \tag{5}$$

where θ goes from 0 to 2π and φ goes from 0 to $\pi/2$. The variable c is related to the energy of the primary particle:

$$c = \sqrt{\frac{2E}{m}} \tag{6}$$

where E is the energy of the primary particle.

Results and Discussion

Initially only 7 particles are released with zero initial velocity. Figure 1 plots the particle locations at subsequent RF cycles. After 1 RF cycle the number of particles has increased to around 50. After 2 cycles this has increased to over 200. By the 4th RF cycle the number has increased to 1600. Clearly the multipactor is well into the exponential growth phase by the 4th RF cycle.

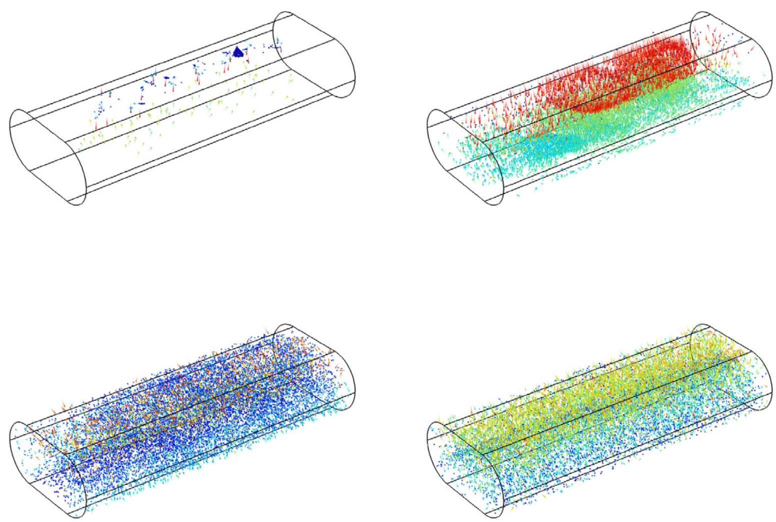

Figure 1: The locations of the electrons after 1 RF cycle (top left), 2 RF cycles (top right), 3 RF cycles (bottom left), and 4 RF cycles (bottom right).

Figure 2 shows the number of particles present inside the multipactor versus time. The number increases in discrete steps corresponding to the moment of impact of the primary particles on the emitting surfaces.

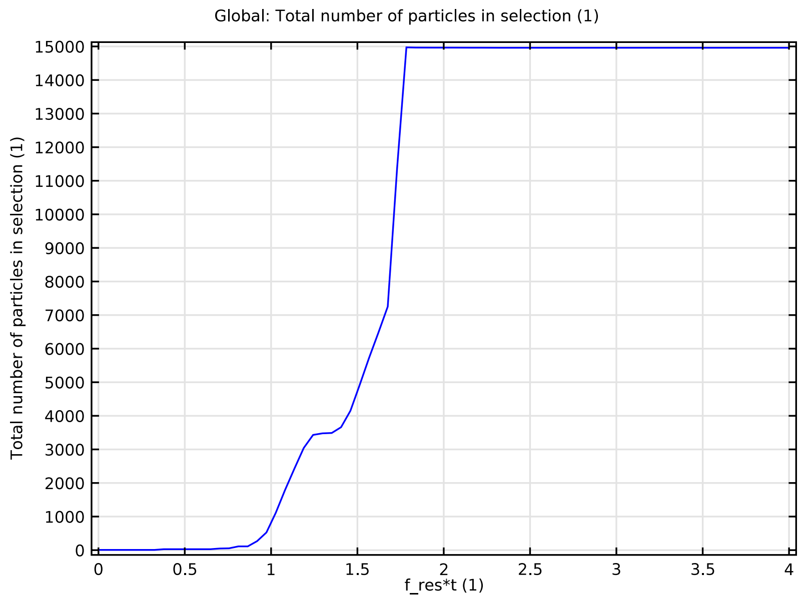

Figure 2: The number of electrons in the multipactor versus RF period.

Reference

1. http://en.wikipedia.org/wiki/Multipactor_effect

Model Library path: `ACDC_Module/Particle_Tracing/multipactor`

Modeling Instructions

Start by solving for the electric currents in the multipactor.

MODEL WIZARD

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

2 Click **Next**.

3 In the **Add physics** tree, select **AC/DC>Electric Currents (ec)**.

4 Click **Add Selected**.

5 Click **Next**.

6 Find the **Studies** subsection. In the tree, select **Preset Studies>Frequency Domain**.

7 Click **Finish**.

GLOBAL DEFINITIONS

Add some parameters to compute the resonant frequency which should be used in the model.

1 In the **Model Builder** window, expand the **Global Definitions** node.

Parameters

1 Right-click **Global Definitions** and choose **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Description
`V0`	`100[V]`	`Applied voltage`
`d`	`1[cm]`	`Gap thickness`
`f_res`	`1/(d*2*sqrt(pi))*sqrt(e_const*V0/me_const)`	`Resonant frequency`

GEOMETRY 1

Work Plane 1

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

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

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

Rectangle 1

1 In the **Model Builder** window, under **Model 1>Geometry 1>Work Plane 1** right-click **Plane Geometry** and choose **Rectangle**.

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

3 In the **Width** edit field, type `d`.

4 In the **Height** edit field, type `d`.

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

Circle 1

1 Right-click **Plane Geometry** and choose **Circle**.

2 In the **Circle** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type `d/2`.

4 Locate the **Position** section. In the **xw** edit field, type `-d/2`.

Circle 2

1 Right-click **Plane Geometry** and choose **Circle**.

2 In the **Circle** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type `d/2`.

4 Locate the **Position** section. In the **xw** edit field, type `d/2`.

5 Click the **Build All** button.

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

Union 1

1 Right-click **Plane Geometry** and choose **Boolean Operations>Union**.

2 Click in the **Graphics** window, press Ctrl+A to highlight all objects, and then right-click to confirm the selection.

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

4 Clear the **Keep interior boundaries** check box.

5 Click the **Build Selected** button.

Extrude 1

1 In the **Model Builder** window, under **Model 1>Geometry 1** right-click **Work Plane 1** and choose **Extrude**.

2 In the **Extrude** settings window, locate the **Distances from Plane** section.

3 In the table, enter the following settings:

Distances (m)
`5*d`

4 Click the **Build All** button.

5 Click the **Go to Default 3D View** button on the Graphics toolbar.

MATERIALS

Material 1

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

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

3 In the table, enter the following settings:

Property	Name	Value
Electrical conductivity	sigma	0
Relative permittivity	epsilonr	1

ELECTRIC CURRENTS

Ground 1

1 In the **Model Builder** window, under **Model 1** right-click **Electric Currents** and choose **Ground**.

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

3 Select Boundary 4 only.

Electric Potential 1

1 In the **Model Builder** window, right-click **Electric Currents** and choose **Electric Potential**.

2 Select Boundary 5 only.

3 In the **Electric Potential** settings window, locate the **Electric Potential** section.

4 In the V_0 edit field, type `V0`.

MESH 1

Free Triangular 1

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

2 Select Boundary 1 only.

Size 1

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

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

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

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

Size

1 In the **Model Builder** window, under **Model 1>Mesh 1** click **Size**.
2 In the **Size** settings window, locate the **Element Size** section.
3 From the **Predefined** list, choose **Extra fine**.
4 Click the **Build All** button.

STUDY 1

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 1** click **Step 1: Frequency Domain**.
2 In the **Frequency Domain** settings window, locate the **Study Settings** section.
3 In the **Frequencies** edit field, type `f_res`.
4 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

MODEL 1

Now add the interface to compute the electron trajectories.

1 In the **Model Builder** window, expand the **Electric Potential (ec)** node.
2 Right-click **Model 1** and choose **Add Physics**.

MODEL WIZARD

1 Go to the **Model Wizard** window.
2 In the **Add physics** tree, select **AC/DC>Charged Particle Tracing (cpt)**.
3 Click **Add Selected**.
4 Click **Next**.
5 Find the **Studies** subsection. In the tree, select **Preset Studies for Selected Physics>Time Dependent**.
6 Click **Finish**.

CHARGED PARTICLE TRACING

1 In the **Model Builder** window, under **Model 1** click **Charged Particle Tracing**.
2 In the **Charged Particle Tracing** settings window, locate the **Advanced Settings** section.
3 In the **Maximum number of secondary particles** edit field, type `15000`.

Release from Grid 1

Release 7 seed particles initially. The initial velocity is 0.

1 Right-click **Model 1>Charged Particle Tracing** and choose **Release from Grid**.
2 In the **Release from Grid** settings window, locate the **Initial Coordinates** section.
3 In the $q_{x,0}$ edit field, type `10e-3 15e-3 20e-3 25e-3 30e-3 35e-3 40e-3`.

Electric Force 1

1 In the **Model Builder** window, right-click **Charged Particle Tracing** and choose **Electric Force**.
2 Select Domain 1 only.
3 In the **Electric Force** settings window, locate the **Electric Force** section.
4 From the **Specify force using** list, choose **Electric potential**.
5 From the V list, choose **Electric potential (ec)**.
6 Locate the **Advanced Settings** section. Select the **Multiply force by phase angle** check box.

Wall 2

1 Right-click **Charged Particle Tracing** and choose **Wall**.
2 Select Boundaries 4 and 5 only.
3 In the **Wall** settings window, locate the **Wall Condition** section.
4 From the **Wall condition** list, choose **Bounce**.
5 Locate the **Secondary Particles** section. Select the **Include secondary emission** check box.
6 In the N_s edit field, type `3*(cpt.Ep>7[eV])`.

Wall 1

1 In the **Model Builder** window, under **Model 1>Charged Particle Tracing** click **Wall 1**.
2 In the **Wall** settings window, locate the **Wall Condition** section.
3 From the **Wall condition** list, choose **Bounce**.

STUDY 2

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 2** click **Step 1: Time Dependent**.
2 In the **Time Dependent** settings window, locate the **Physics and Variables Selection** section.

3 In the table, enter the following settings:

Physics	Solve for
Electric Currents (ec)	×

4 Click to expand the **Values of Dependent Variables** section. Select the **Values of variables not solved for** check box.

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

6 From the **Study** list, choose **Study 1, Frequency Domain**.

7 Locate the **Study Settings** section. Click the **Range** button.

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

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

10 In the **Stop** edit field, type `4/f_res`.

11 In the **Number of values** edit field, type `75`.

12 Click the **Replace** button.

13 In the **Model Builder** window, right-click **Study 2** and choose **Compute**.

RESULTS

Particle Trajectories (cpt)

Reproduce the electron trajectory plots in Figure 1 as follows.

1 In the **Model Builder** window, expand the **Particle Trajectories (cpt)** node.

2 In the **Model Builder** window, under **Results>Particle Trajectories (cpt)** click **Particle Trajectories 1**.

3 In the **Particle Trajectories** settings window, locate the **Coloring and Style** section.

4 Find the **Point style** subsection. From the **Type** list, choose **Comet tail**.

5 In the **Model Builder** window, click **Particle Trajectories (cpt)**.

6 In the **3D Plot Group** settings window, locate the **Data** section.

7 From the **Time** list, choose **8.681127e-9**.

8 Click the **Plot** button.

9 From the **Time** list, choose **1.690535e-8**.

10 Click the **Plot** button.

11 From the **Time** list, choose **2.558648e-8**.

12 Click the **Plot** button.

13 From the **Time** list, choose **3.38107e-8**.

14 Click the **Plot** button.

Data Sets

Finally, reproduce the plot in Figure 2.

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

2 Right-click **Particle 1** and choose **Add Selection**.

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

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

5 Select Domain 1 only.

1D Plot Group 3

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

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

3 From the **Data set** list, choose **Particle 1**.

4 Right-click **Results>1D Plot Group 3** and choose **Global**.

5 In the **Global** settings window, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Charged Particle Tracing>Particle statistics>Total number of particles in selection (cpt.Nsel)**.

6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

7 In the **Expression** edit field, type `f_res*t`.

8 Click to expand the **Legends** section. Clear the **Show legends** check box.

9 Click the **Plot** button.