Created in COMSOL Multiphysics 6.2

Electrostatic Discharge (ESD) Test of a PCB

Introduction

Electronic devices and systems are subject to electrostatic discharge (ESD) and they have to pass ESD tests according to standards. ESD tests have different levels, from component, to PCB and to system. In laboratory, ESD gun is usually used to generate discharge current which is directly or indirectly applied to electronic components. Figure 1 shows an example of an ESD test platform. The aim of the ESD test is to verify if the component or system is able to withstand ESD events. The experimental setup of an ESD test is not simple and it is only possible to test until a component or system is manufactured.

Thanks to numerical simulation, we are now able to identify ESD problems during the design stage. Moreover, numerical simulation is able to provide detailed electromagnetic field distribution which is difficult or even impossible to measure accurately. Such information is needed to understand why an ESD damage happens and how to fix it.

ESD can lead to insulating failure or logic errors to chips. The logic error can be triggered when a pin of the chip at Low state (0) experiences a voltage of amplitude larger than 1.5 V and lasting more than 1 ns, or a voltage of amplitude larger than 0.3 V and lasting more than 5 ns. This example demonstrates how an ESD event causes logic error of a microchip on a PCB board.

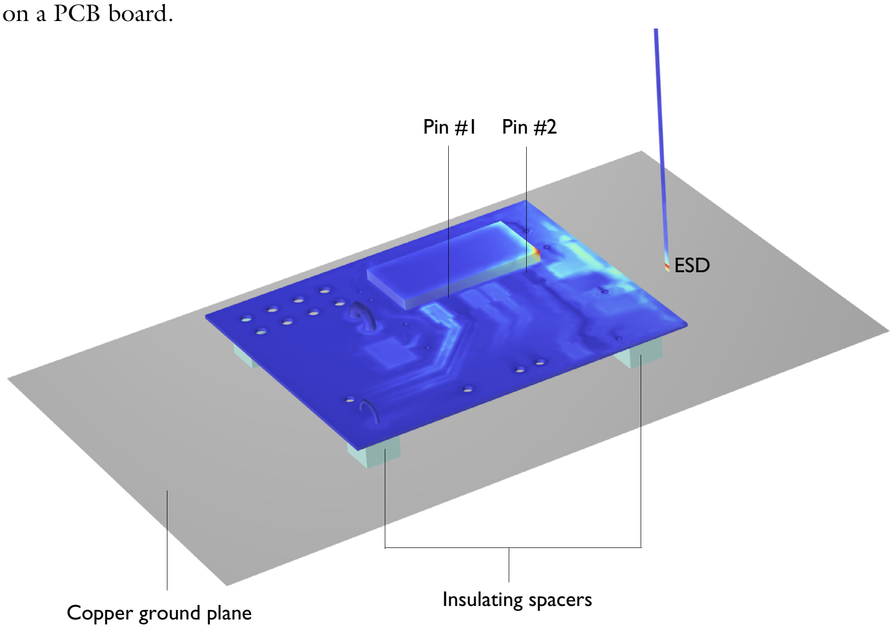


Figure 1: Example of an ESD test platform for a PCB.

Model Definition

ESD currents are generally very nonlinear and require a transient 3D study. The **RF Module** provides the **Electromagnetic Waves, Transient** physics interface dedicated to this purpose. This interface provides four predefined ESD currents according to different standards. These standards are the Human Body Model (HBM), Machine Model (MM), Charged Device Model (CDM), and the Extended HBM (also known as IEC 61000-4-2).

In this example, the ESD current is generated by using the Lumped Port feature with the Extended HBM, as shown in Figure 2. This current is one of the most famous test currents, since it tests against the most severe conditions. The current can easily be visualized by clicking the **Plot Pulse Shape** button in the **Settings** window.

The induced voltage at two pins of a microchip on the PCB are computed by using the **Lumped Element** feature. The exterior boundaries are modeled with the Scattering Boundary Condition.

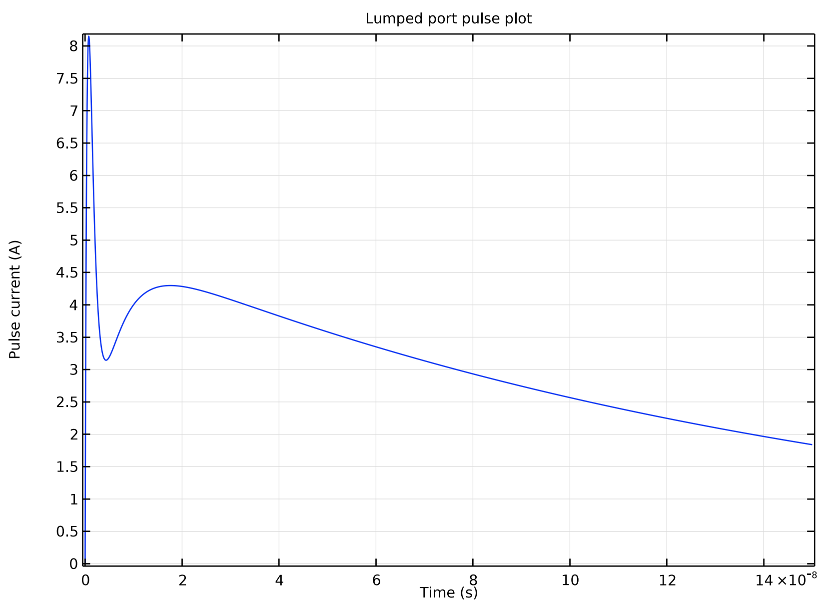


Figure 2: The ESD current based on the Extended HBM.

Results and Discussion

Figure 3 shows the induced surface current density by the ESD current at t = 1.2 ns. Figure 4 shows the computed induced voltage at two pins (#1, #2) of the chip. As can be seen, the induced voltage at pin #2 has an amplitude higher than 1.5 V and is very likely to trigger a logic error.

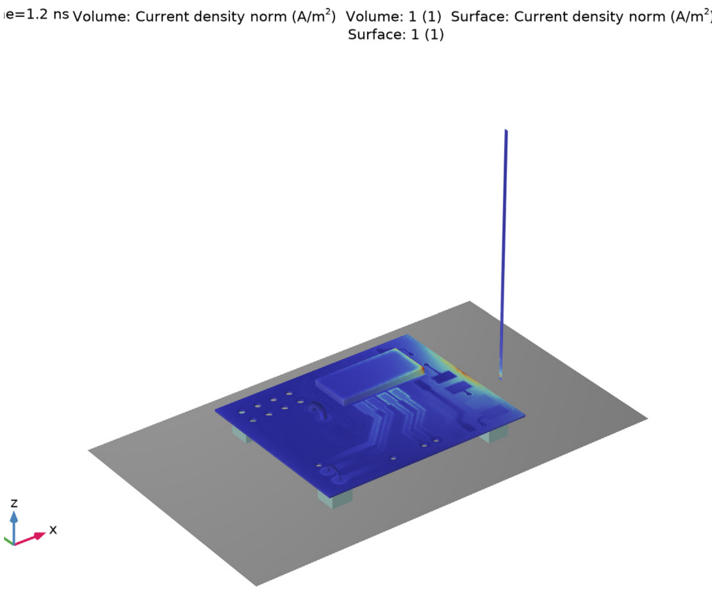


Figure 3: The distribution of induced surface current density of a PCB at t = 1.2 ns due to an ESD event.

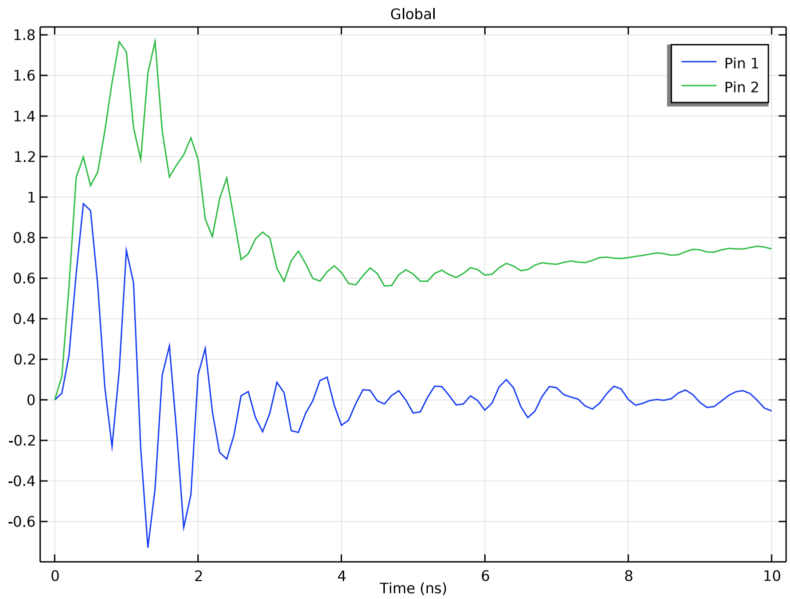


Figure 4: The induced voltage at pin #1 and pin #2 of the microchip.

Notes About the COMSOL Implementation

This example provides only a simplified modeling workflow for the virtual ESD tests. In particular, it does not comply with well-known test standards and protocols. The model setup has to be modified accordingly to perform the standard test.

Reference

1. M. Mardiguian, *Electrostatic Discharge: Understand, Simulate, and Fix ESD Problems*, John Wiley & Sons, 2011.

Application Library path: `RF_Module/ESD_and_Lightning_Surge/esd_test_pcb`

Modeling Instructions

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

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Radio Frequency>Electromagnetic Waves, Transient (temw)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Time Dependent**.

6 Click **Done**.

GEOMETRY 1

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

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

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

Import 1 (imp1)

1 In the **Home** toolbar, click **Import**.

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

3 Click **Browse**.

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

5 Click **Import**.

Block 1 (blk1)

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

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

3 In the **Width** text field, type `21[mm]`.

4 In the **Depth** text field, type `10[mm]`.

5 In the **Height** text field, type `1.5[mm]`.

6 Locate the **Position** section. In the **x** text field, type `-3[mm]`.

7 In the **y** text field, type `5[mm]`.

8 In the **z** text field, type `0.508[mm]`.

9 Click **Build Selected**.

Work Plane 1 (wp1)

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

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

3 In the **z-coordinate** text field, type `-5`.

Work Plane 1 (wp1)>Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 1 (wp1)>Rectangle 1 (r1)

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

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

3 In the **Width** text field, type `100[mm]`.

4 In the **Height** text field, type `60[mm]`.

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

6 Click **Build Selected**.

Work Plane 2 (wp2)

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

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

3 In the **z-coordinate** text field, type `-5`.

Work Plane 2 (wp2)>Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 2 (wp2)>Rectangle 1 (r1)

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

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

3 In the **Width** text field, type `5[mm]`.

4 In the **Height** text field, type `5[mm]`.

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

6 In the **xw** text field, type `-20[mm]`.

7 In the **yw** text field, type `-15[mm]`.

Work Plane 2 (wp2)>Array 1 (arr1)

1 In the **Work Plane** toolbar, click **Transforms** and choose **Array**.

2 Select the object **r1** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **xw size** text field, type `2`.

5 In the **yw size** text field, type `2`.

6 Locate the **Displacement** section. In the **xw** text field, type `40[mm]`.

7 In the **yw** text field, type `30[mm]`.

Extrude 1 (ext1)

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

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (mm)
5[mm]

4 Click **Build Selected**.

Work Plane 3 (wp3)

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

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

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

4 In the **x-coordinate** text field, type `35[mm]`.

Work Plane 3 (wp3)>Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 3 (wp3)>Rectangle 1 (r1)

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

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

3 In the **Width** text field, type `1[mm]`.

4 In the **Height** text field, type `60[mm]`.

5 Locate the **Position** section. In the **yw** text field, type `-5[mm]`.

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

Layer name	Thickness (mm)
`Layer 1`	`1[mm]`

7 Click **Build Selected**.

Work Plane 4 (wp4)

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

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

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

4 In the **y-coordinate** text field, type `4[mm]`.

Work Plane 4 (wp4)>Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 4 (wp4)>Rectangle 1 (r1)

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

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

3 In the **Width** text field, type `0.508[mm]`.

4 In the **Height** text field, type `2[mm]`.

5 Locate the **Position** section. In the **yw** text field, type `2[mm]`.

6 Right-click **Rectangle 1 (r1)** and choose **Duplicate**.

Work Plane 4 (wp4)>Rectangle 2 (r2)

1 In the **Model Builder** window, click **Rectangle 2 (r2)**.

2 In the **Settings** window for **Rectangle**, locate the **Position** section.

3 In the **yw** text field, type `14[mm]`.

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

Block 2 (blk2)

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

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

3 In the **Width** text field, type `150[mm]`.

4 In the **Depth** text field, type `150[mm]`.

5 In the **Height** text field, type `100[mm]`.

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

7 In the **z** text field, type `20[mm]`.

8 Click **Build All Objects**.

DEFINITIONS

In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

Hide for Geometry 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions>View 1** node.

2 Right-click **View 1** and choose **Hide for Geometry**.

3 In the **Settings** window for **Hide for Geometry**, locate the **Selection** section.

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

5 On the object **blk2**, select Boundaries 2–4 only.

Printed Layer

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

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

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `17-19, 24, 26, 35-37, 54-57, 72-75, 86-90, 111, 112, 134, 135, 141, 142, 147-152, 154, 157, 170-172` in the **Selection** text field.

6 Click **OK**.

Vias

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

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

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

Choose all cylindrical boundaries of metalized vias.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `29-32, 38-43, 52, 53, 60-63, 68-71, 78-85, 95-98, 103-110, 113-128, 136-139, 143-146, 155, 156, 158, 159, 174-179, 181-186` in the **Selection** text field.

6 Click **OK**.

Alternatively, with **Group by continuous tangent**, you can select the curved cylindrical surface of a metalized via by selecting one of its boundary elements. The **Group by**

continuous tangent allows you to select adjacent faces or edges that are continuously tangent with the angular tolerance you specified. You may select any part of the cylindrical surfaces of each via, and it will automatically select all surfaces within the angular tolerance range.

Jumpers

1 In the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for **Explicit**, type `Jumpers` in the **Label** text field.
3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4 Click **Paste Selection**.
5 In the **Paste Selection** dialog box, type `44-51, 91-94, 99-102` in the **Selection** text field.
6 Click **OK**.

With **Group by continuous tangent**, you can select the curved surface of the mockup jumpers easily. You may select any surface of each jumper, and it will automatically select all surfaces within the angular tolerance range.

ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.
2 Go to the **Add Material** window.
3 In the tree, select **Built-in>Air**.
4 Click **Add to Component** in the window toolbar.
5 In the tree, select **Built-in>FR4 (Circuit Board)**.
6 Right-click and choose **Add to Component 1 (comp1)**.

MATERIALS

FR4 (Circuit Board) (mat2)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
2 Click **Paste Selection**.
3 In the **Paste Selection** dialog box, type `2` in the **Selection** text field.
4 Click **OK**.

ADD MATERIAL

1 Go to the **Add Material** window.
2 In the tree, select **Built-in>Copper**.

3 Click **Add to Component** in the window toolbar.

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

MATERIALS

Copper (mat3)

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

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

3 From the **Selection** list, choose **Printed Layer**.

Spacer

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

2 In the **Settings** window for **Material**, type `Spacer` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `3, 4, 6, 7` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Material**, locate the **Material Contents** section.

7 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	2.9	1	Basic
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	1	Basic
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	0	S/m	Basic

Chip

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

2 In the **Settings** window for **Material**, type `Chip` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `5` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Material**, locate the **Material Contents** section.

7 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	12	1	Basic
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	1	Basic
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	0	S/m	Basic

ELECTROMAGNETIC WAVES, TRANSIENT (TEMW)

Perfect Electric Conductor 2

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electromagnetic Waves, Transient (temw)** and choose **Perfect Electric Conductor**.

2 In the **Settings** window for **Perfect Electric Conductor**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Printed Layer**.

Perfect Electric Conductor 3

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

2 In the **Settings** window for **Perfect Electric Conductor**, locate the **Boundary Selection** section.

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

Perfect Electric Conductor 4

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

2 In the **Settings** window for **Perfect Electric Conductor**, locate the **Boundary Selection** section.

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

Perfect Electric Conductor 5

1 In the **Physics** toolbar, click **Boundaries** and choose **Perfect Electric Conductor**.
2 In the **Settings** window for **Perfect Electric Conductor**, locate the **Boundary Selection** section.
3 Click **Paste Selection**.
4 In the **Paste Selection** dialog box, type `6-9, 11, 15, 23, 163, 168, 189` in the **Selection** text field.
5 Click **OK**.

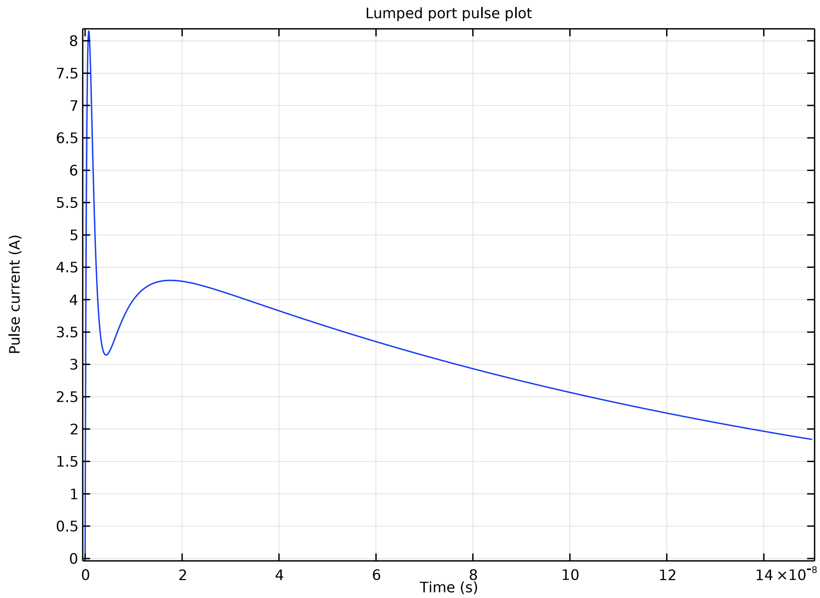
Perfect Electric Conductor 6

1 In the **Physics** toolbar, click **Boundaries** and choose **Perfect Electric Conductor**.
2 In the **Settings** window for **Perfect Electric Conductor**, locate the **Boundary Selection** section.
3 Click **Paste Selection**.
4 In the **Paste Selection** dialog box, type `191` in the **Selection** text field.
5 Click **OK**.

Lumped Port 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Lumped Port**.
2 In the **Settings** window for **Lumped Port**, locate the **Boundary Selection** section.
3 Click **Paste Selection**.
4 In the **Paste Selection** dialog box, type `190` in the **Selection** text field.
5 Click **OK**.
6 In the **Settings** window for **Lumped Port**, locate the **Lumped Port Properties** section.
7 From the **Terminal type** list, choose **Current**.
8 Locate the **Settings** section. From the **Current pulse type** list, choose **Electrostatic discharge**.
9 From the **Electrostatic discharge pulse model** list, choose **Extended human body model**.
10 In the I_{01} text field, type `30[A]`.
11 In the I_{02} text field, type `5[A]`.

12 Click **Plot Pulse Shape** for **Current pulse type**.



Lumped Element 1

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type 140 in the **Selection** text field.

5 Click **OK**.

Lumped Element 2

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type 153 in the **Selection** text field.

5 Click **OK**.

Scattering Boundary Condition 1

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

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

3 Click Paste Selection.

4 In the **Paste Selection** dialog box, type `1-5, 192` in the **Selection** text field.

5 Click **OK**.

MESH 1

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

2 In the **Settings** window for **Mesh**, locate the **Electromagnetic Waves, Transient (temw)** section.

3 From the **Maximum mesh element size control parameter** list, choose **Frequency**.

4 In the **Maximum frequency** text field, type `2[GHz]`.

5 Click **Build All**.

STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 From the **Time unit** list, choose **ns**.

4 In the **Output times** text field, type `range(0,0.1,10)`.

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

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

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

Solution 1 (sol1)

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

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

4 From the **Steps taken by solver** list, choose **Manual**.

5 In the **Time step** text field, type `0.1[ns]`.

6 Click **Compute**.

RESULTS

Current Density

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

3 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Volume 1

1 Right-click **Current Density** and choose **Volume**.

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

3 In the **Expression** text field, type `temw.normJ`.

Selection 1

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `2,5` in the **Selection** text field.

5 Click **OK**.

Volume 2

1 In the **Model Builder** window, right-click **Current Density** and choose **Volume**.

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

3 In the **Expression** text field, type `1`.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

5 From the **Color** list, choose **Custom**.

6 On Windows, click the colored bar underneath, or — if you are running the cross-platform desktop — the **Color** button.

7 Click **Define custom colors**.

8 Set the RGB values to 166, 214, and 208, respectively.

9 Click **Add to custom colors**.

10 Click **Show color palette only** or **OK** on the cross-platform desktop.

Selection 1

1 Right-click **Volume 2** and choose **Selection**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `3, 4, 6, 7` in the **Selection** text field.

5 Click **OK**.

Surface 1

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

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

3 In the **Expression** text field, type `temw.normJ`.

Selection 1

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `190, 191` in the **Selection** text field.

5 Click **OK**.

Surface 2

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

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

3 In the **Expression** text field, type `1`.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

5 From the **Color** list, choose **Gray**.

Selection 1

1 Right-click **Surface 2** and choose **Selection**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `6` in the **Selection** text field.

5 Click **OK**.

Current Density

1 In the **Model Builder** window, under **Results** click **Current Density**.

2 In the **Current Density** toolbar, click **Plot**.

3 Click the **Show Legends** button in the **Graphics** toolbar.

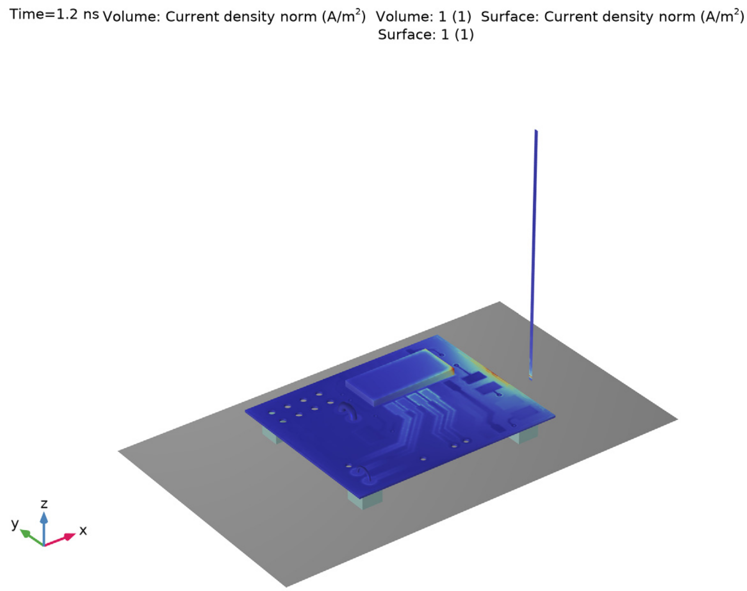
4 Click the **Show Grid** button in the **Graphics** toolbar.

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

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

7 From the **Time (ns)** list, choose **1.2**.

8 In the **Current Density** toolbar, click **Plot**.



Induced Voltage

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

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

Global 1

1 Right-click **Induced Voltage** and choose **Global**.

2 In the **Settings** window for **Global**, click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)> Electromagnetic Waves, Transient>Lumped elements>Voltage>temw.Velement_1 - Lumped element 1 voltage - V**.

3 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Electromagnetic Waves, Transient> Lumped elements>Voltage>temw.Velement_2 - Lumped element 2 voltage - V**.

4 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

5 In the table, enter the following settings:

Legends
Pin 1
Pin 2

6 In the **Induced Voltage** toolbar, click **Plot**.