Microwave Filter on PCB with Stress

This example analyzes the transfer function of a low-pass filter on a printed circuit board.

Introduction

Microstrip filters can be fabricated directly on a printed circuit board (PCB) with a microstrip line going from the input to the output. Along the microstrip line there are a number of stubs of certain lengths and widths. The design of the filter involves choosing the impedance of the microstrip line, the impedance of the stub microstrips, and the stub lengths. This particular filter is based on a textbook example from Ref. 2. It is also used as example in Ref. 1, which contains results from other simulation tools and methods and is freely available online. The filter has a seven-pole low-pass Chebyshev response with a cutoff frequency of 1 GHz.

Model Definition

The model uses the Electromagnetic Waves interface that solves the vector Helmholtz wave equation. The cutoff frequency of the filter is 1 GHz by design, and the dielectric layer of the PCB has a relative permittivity of 10.8. The metal layers are modeled as perfect electric conductors with zero thickness, thereby avoiding a dense meshing of thin conductive layers. The width of the microstrip line is 0.1 mm and the width of the stubs is 5 mm.

The characteristics of the filter are sensitive to the placement and length of the stubs; therefore this example also analyzes the change in filter characteristics as a function of mechanical deformation. This is done by adding Solid Mechanics and Moving Mesh interfaces. The Moving Mesh interface is necessary to enable the Electromagnetic Waves interface to account for the deformation of the PCB. The deformation comes from a uniform load across the board with fixed input and output faces.

Because the filter cutoff should be close to 1 GHz, the frequency is swept from 750 MHz up to 1.5 GHz. The first solution step performs this sweep for the Electromagnetic Waves interface without any mechanical deformation. Then a uniform load of 40 N is applied to the PCB, generating a large deformation of the board. The Solid Mechanics interface calculates the deformation, and the Moving Mesh interface applies this deformation to the coordinate system that the Electromagnetic Waves interface uses. After this step, the frequency sweep is performed again for the Electromagnetic Waves interface using the parametric solver.

This example accounts for the structural deformation in the sense that it solves for the electromagnetic fields on the deformed geometry, as if the PCB was manufactured in the deformed shape — free of stress.

Results and Discussion

The purpose of this simulation is to analyze how the S-parameter curve changes when a force of 40 N is applied on the circuit board. This force bends the PCB significantly, as you can see in Figure 1.

Volume: Displacement magnitude (mm)
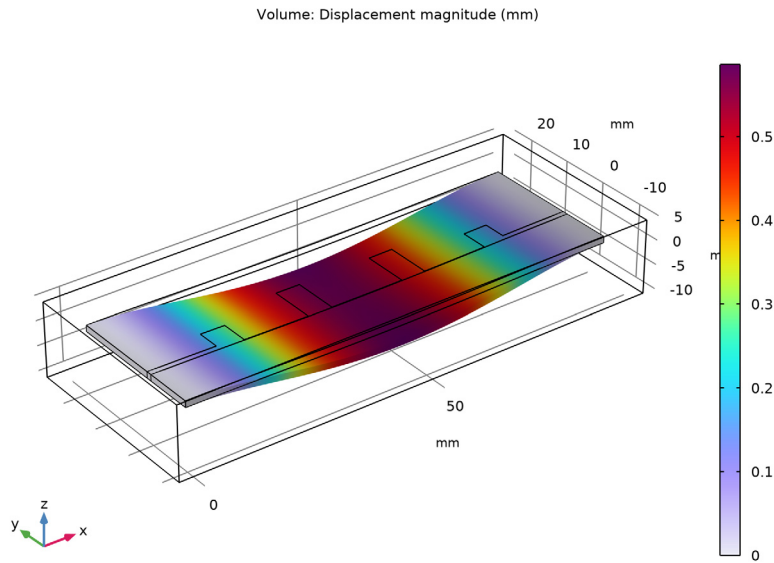
Figure 1: The graph shows the total displacement of the PCB, due to the load.

Although the PCB deformation is fairly large the S-parameter curve does not change that much. The cutoff frequency is shifted less than 10 MHz when the force is applied.

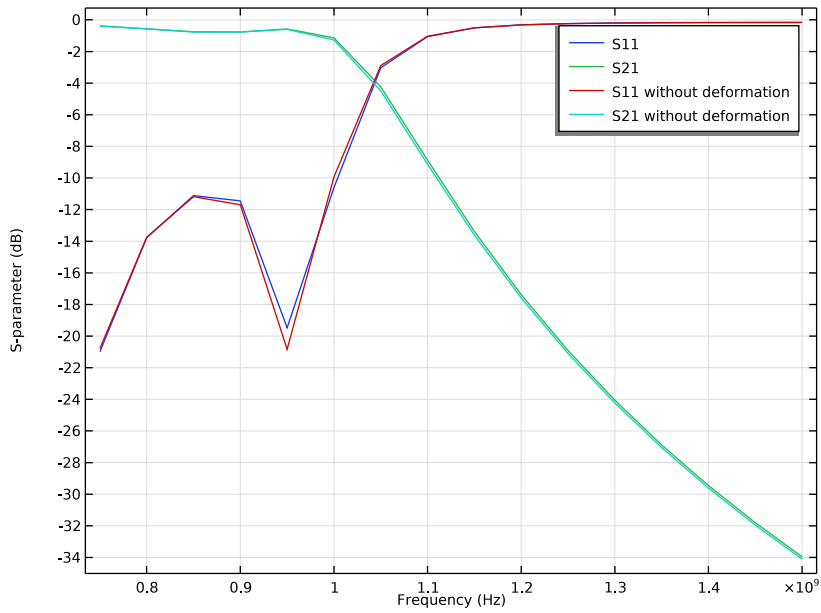Figure 2 displays the difference between the S-parameter curves with and without an applied force.

Figure 2: A comparison between the S$_{11}$ and S$_{21}$ parameters before and after a force of 40 N has been applied on the PCB. The red and turquoise lines correspond to the S-parameter curves for the filter with an applied force.

References

1. D.V. Tosic and M. Potrebic, "Software Tools for Research and Education," *Microwave Review*, vol. 12, no. 2, pp. 45–54, 2006.

2. J.-S.G. Hong and M.J. Lancaster, *Microstrip Filters for RF/Microwave Applications*, John Wiley & Sons, 2001.

Application Library path: RF_Module/Filters/
pcb_microwave_filter_with_stress

Modeling Instructions

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

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

M O D E L   W I Z A R D

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

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

3 Click **Add**.

4 Click → **Study**.

5 In the **Select Study** tree, select **General Studies>Frequency Domain**.

6 Click ☑ **Done**.

G L O B A L   D E F I N I T I O N S

The following steps define the parameters for the frequency sweep.

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
fstart	750[MHz]	7.5E8 Hz	Start frequency
fstop	1.5[GHz]	1.5E9 Hz	Stop frequency
fstep	50[MHz]	5E7 Hz	Frequency step

G E O M E T R Y   1

Set mm as the default unit for length.

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**.

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 89.49.

4 In the **Depth** text field, type 29.54.

5 In the **Height** text field, type 1.27.

6 Locate the **Position** section. In the **y** text field, type -10.

7 Click **Build Selected**.

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

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 1.27.

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 13.88.

4 In the **Height** text field, type 1.125.

Work Plane 1 (wp1)>Rectangle 2 (r2)

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.

4 In the **Height** text field, type 5.86.

5 Locate the **Position** section. In the **xw** text field, type 13.88.

Work Plane 1 (wp1)>Rectangle 3 (r3)

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 13.32.

4 In the **Height** text field, type 0.1.

5 Locate the **Position** section. In the **xw** text field, type 18.88.

Work Plane 1 (wp1)>Rectangle 4 (r4)

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.

4 In the **Height** text field, type 9.54.

5 Locate the **Position** section. In the **xw** text field, type 32.2.

Work Plane 1 (wp1)>Rectangle 5 (r5)

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 15.09.

4 In the **Height** text field, type 0.1.

5 Locate the **Position** section. In the **xw** text field, type 37.2.

Work Plane 1 (wp1)>Mirror 1 (mir1)

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

2 Select the objects **r1**, **r2**, **r3**, and **r4** only.

3 In the **Settings** window for **Mirror**, locate the **Input** section.

4 Select the **Keep input objects** check box.

5 Locate the **Point on Line of Reflection** section. In the **xw** text field, type 44.745.

Work Plane 1 (wp1)>Union 1 (uni1)

1 In the **Work Plane** toolbar, click ▦ **Booleans and Partitions** and choose **Union**.

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

3 In the **Settings** window for **Union**, locate the **Union** section.

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

5 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 1.27.

Work Plane 2 (wp2)>Plane Geometry

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

Work Plane 2 (wp2)>Line Segment 1 (ls1)

1 In the **Work Plane** 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 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

5 In the **yw** text field, type 1.125.

Work Plane 2 (wp2)>Mirror 1 (mir1)

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

2 Select the object **ls1** only.

3 In the **Settings** window for **Mirror**, locate the **Input** section.

4 Select the **Keep input objects** check box.

5 Locate the **Point on Line of Reflection** section. In the **xw** text field, type 44.745.

6 Click ▦ **Build Selected**.

The next step is to add boundary faces for the input and output ports by extruding the lines.

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)
-1.27

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 100.

4 In the **Depth** text field, type 40.

5 In the **Height** text field, type 15.

6 Locate the **Position** section. In the **x** text field, type -5.

7 In the **y** text field, type -15.

8 In the **z** text field, type -10.

9 In the **Geometry** toolbar, click ▦ **Build All**.

ADD MATERIAL

1 In the **Home** toolbar, click ![icon] **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 Click **Add to Component** in the window toolbar.

7 In the **Home** toolbar, click ![icon] **Add Material** to close the **Add Material** window.

MATERIALS

FR4 (Circuit Board) (mat2)

1 Select Domain 2 only.

The relative permittivity is modified to agree with the value used in Ref. 1. The FR4 material is selected to provide parameters for the solid mechanics simulation.

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

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	10.8	1	Basic

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Scattering Boundary Condition 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Scattering Boundary Condition**.

2 Select Boundaries 1–5 and 18 only.

Lumped Port 1

1 In the **Physics** toolbar, click ![icon] **Boundaries** and choose **Lumped Port**.

2 Select Boundary 10 only.

For the first port, wave excitation is **on** by default. This port excites the microstrip line.

Lumped Port 2

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

2 Select Boundary 16 only.

Perfect Electric Conductor 2

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

2 Select Boundaries 8 and 11 only.

These boundaries represent the microstrip line and the ground plane on the PCB.

MESH 1

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

DEFINITIONS

Probe plotting is a convenient technique to plot while solving, which is very useful for parameter sweeps. It is possible to discover problems before the solution step has finished, and then stop the sweep to save time. It is also useful in situations when the solver does more steps than it stores in the output. The probe plot will contain all steps that the solver takes.

Global Variable Probe 1 (var1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Global Variable Probe**.

2 In the **Settings** window for **Global Variable Probe**, type S11 in the **Variable name** text field.

3 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain>Ports> S-parameter, dB>emw.S11dB - S11**.

Global Variable Probe 2 (var2)

1 In the **Model Builder** window, right-click **Definitions** and choose **Global Variable Probe**.

2 In the **Settings** window for **Global Variable Probe**, type S21 in the **Variable name** text field.

3 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain>Ports> S-parameter, dB>emw.S21dB - S21**.

STUDY 1

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 1** click **Step 1: Frequency Domain**.

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

3 Locate the **Study Settings** section. Click ⌁ **Range**.

4 In the **Range** dialog box, type fstart in the **Start** text field.

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

6 In the **Stop** text field, type fstop.

7 Click **Replace**.

8 In the **Home** toolbar, click = **Compute**.

RESULTS

The **Electric Field** plot group under the **Results** node, shows the norm of the electric field. You can change the frequency by selecting another value from the Parameter value (freq) list box.

The Probe 1D Plot Group 2 displays the S_{11}- and S_{21}-parameters for the frequency sweep.

COMPONENT 1 (COMP1)

The following instructions adds physics from the Solid Mechanics and the Moving Mesh interfaces for the simulations of the deformed PCB.

ADD PHYSICS

1 In the **Home** toolbar, click ⚛ **Add Physics** to open the **Add Physics** window.

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

3 In the tree, select **Structural Mechanics>Solid Mechanics (solid)**.

4 Click **Add to Component 1** in the window toolbar.

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

SOLID MECHANICS (SOLID)

Select Domain 2 only.

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
fload	40[N]	40 N	Load on PCB

The following steps describe how to measure the volume of the PCB and then copy and paste the value in a parameter definition.

GEOMETRY 1

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

2 In the **Graphics** window toolbar, click ▼ next to 🔲 **Select Objects**, then choose **Select Domains**.

3 On the object **fin**, select Domain 2 only.

4 In the **Geometry** toolbar, click ⬜ **Measure**.

5 Click the **Measure** button from the toolbar. The volume of the PCB domain is displayed in the Messages window.

Copy the volume of the PCB from the Messages table.

GLOBAL DEFINITIONS

(by pasting in the previously copied volume)

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

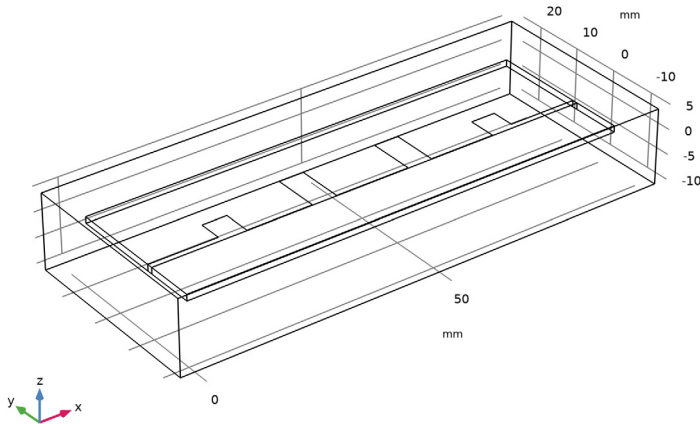**2** In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
V	3357.0[mm^3]	3.357E-6 m³	Volume of PCB

SOLID MECHANICS (SOLID)

In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

Body Load 1

1 In the **Physics** toolbar, click ▬ **Domains** and choose **Body Load**.

2 Select Domain 2 only.

3 In the **Settings** window for **Body Load**, locate the **Force** section.

4 Specify the $\mathbf{F}_V$ vector as

0	x
0	y
-fload/V	z

Fixed Constraint 1

1 In the **Physics** toolbar, click ▬ **Boundaries** and choose **Fixed Constraint**.

2 Select Boundaries 6, 10, 12, and 15–17 only.

COMPONENT 1 (COMP1)

Deforming Domain 1

1 In the **Physics** toolbar, click ▦ **Moving Mesh** and choose **Free Deformation**.

2 Select Domain 1 only.

STUDY 1

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 1** click **Step 1: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check boxes for **Solid Mechanics (solid)** and **Moving mesh (Component 1)**.

1 In the **Home** toolbar, click ⬡ **Add Study** to open the **Add Study** window.

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

3 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Electromagnetic Waves, Frequency Domain (emw)**.

4 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

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

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

STUDY 2

Step 2: Frequency Domain

1 In the **Study** toolbar, click ⬡ **Study Steps** and choose **Frequency Domain> Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check boxes for **Solid Mechanics (solid)** and **Moving mesh (Component 1)**.

4 Locate the **Study Settings** section. Click ⬡ **Range**.

5 In the **Range** dialog box, type `fstart` in the **Start** text field.

6 In the **Step** text field, type `fstep`.

7 In the **Stop** text field, type `fstop`.

8 Click **Replace**.

9 In the **Settings** window for **Frequency Domain**, locate the **Results While Solving** section.

10 Select the **Plot** check box.

11 From the **Plot group** list, choose **Electric Field (emw)**.

12 In the **Study** toolbar, click ⬡ **Compute**.

RESULTS

Electric Field (emw) 1
The default plot shows a Multislice plot of the norm of the electric field for the last frequency in the sweep. The plot can be updated for any of the frequencies used, by selecting another frequency from the Parameter value (freq) list box.

S-parameter (emw) 1

To compare the S-parameters for the initial and the stressed PCB, add the S-parameter from the first different solutions.

Global 1

1 In the **Model Builder** window, expand the **S-parameter (emw) 1** node.

2 Right-click **Global 1** and choose **Duplicate**.

Global 2

1 In the **Model Builder** window, click **Global 2**.

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

3 From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
emw.S11dB	1	S11 without deformation
emw.S21dB	1	S21 without deformation

5 In the **S-parameter (emw) 1** toolbar, click ▦ **Plot**.

You should now see the plot in Figure 2.

Volume 1

1 In the **Model Builder** window, expand the **Results>Stress (solid)** node, then click **Volume 1**.

2 In the **Settings** window for **Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics> Displacement>solid.disp - Displacement magnitude - m**.

Stress (solid)

1 In the **Model Builder** window, click **Stress (solid)**.

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

3 From the **Dataset** list, choose **Study 2/Solution Store 1 (sol3)**.

4 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (x, y, z)**.

5 In the **Stress (solid)** toolbar, click ▦ **Plot**.

You should now see the plot in Figure 1.

Analyze the same filter model with a much finer frequency resolution using **Adaptive Frequency Sweep** based on asymptotic waveform evaluation (AWE). When a device presents a slowly varying frequency response, the AWE method provides a faster solution time

when running the simulation on many frequency points. The following example with the Adaptive Frequency Sweep can be computed about ten times faster than regular Frequency Domain sweeps with a same finer frequency resolution.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Lumped Port 1

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

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

3 Click **Create Selection**.

4 In the **Create Selection** dialog box, type Lumped port 1 in the **Selection name** text field.

5 Click **OK**.

Lumped Port 2

1 In the **Model Builder** window, click **Lumped Port 2**.

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

3 Click **Create Selection**.

4 In the **Create Selection** dialog box, type Lumped port 2 in the **Selection name** text field.

5 Click **OK**.

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 **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Solid Mechanics (solid)**.

4 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Adaptive Frequency Sweep**.

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

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

STUDY 3

Step 1: Adaptive Frequency Sweep

1 In the **Settings** window for **Adaptive Frequency Sweep**, locate the **Study Settings** section.

2 In the **Frequencies** text field, type `range(fstart,fstep/20,fstop)`.

Use a 10 times finer frequency resolution.

A slowly varying scalar value curve works well for AWE expressions. When **AWE expression type** is set to **Physics controlled** in the **Adaptive Frequency Sweep** study settings, `abs(comp1.emw.S21)` is used automatically for two-port devices.

3 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check boxes for **Solid Mechanics (solid)** and **Moving mesh (Component 1)**.

Because such a fine frequency step generates a memory-intensive solution, the model file size will increase tremendously when it is saved. When only the frequency response of port related variables are of interest, it is not necessary to store all of the field solutions. By selecting the **Store in Output** check box in the **Values of Dependent Variables** section, we can control the part of the model on which the computed solution is saved. We only add the selection containing these boundaries where the port variables are calculated. The lumped port size is typically very small compared to the entire modeling domain, and the saved file size with the fine frequency step is more or less that of the regular discrete frequency sweep model when only the solutions on the port boundaries are stored.

4 Click to expand the **Store in Output** section. In the table, enter the following settings:

Interface	Output
Electromagnetic Waves, Frequency Domain (emw)	Selection

5 Click to select row number 1 in the table.

6 Under **Selections**, click ╋ **Add**.

7 In the **Add** dialog box, in the **Selections** list, choose **Lumped port 1** and **Lumped port 2**.

8 Click **OK**.

It is necessary to include the lumped port boundaries to calculate S-parameters. By choosing only the lumped port boundaries for **Store in Output** settings, it is possible to reduce the size of a model file a lot.

9 In the **Settings** window for **Adaptive Frequency Sweep**, locate the **Store in Output** section.

10 In the table, enter the following settings:

Interface	Output
Solid Mechanics (solid)	Selection

11 Click to select row number 2 in the table.

12 Under **Selections**, click ╋ **Add**.

13 In the **Add** dialog box, in the **Selections** list, choose **Lumped port 1** and **Lumped port 2**.

14 Click **OK**.

15 In the **Settings** window for **Adaptive Frequency Sweep**, locate the **Store in Output** section.

16 In the table, enter the following settings:

Interface	Output
Moving mesh (Component 1)	Selection

17 Click to select row number 3 in the table.

18 Under **Selections**, click ✛ **Add**.

19 In the **Add** dialog box, in the **Selections** list, choose **Lumped port 1** and **Lumped port 2**.

20 Click **OK**.

21 In the **Home** toolbar, click ▬ **Compute**.

RESULTS

Multislice

1 In the **Model Builder** window, expand the **Electric Field (emw) 2** node.

2 Right-click **Multislice** and choose **Delete**.

Surface 1

In the **Model Builder** window, right-click **Electric Field (emw) 2** and choose **Surface**.

Selection 1

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

2 Select Boundaries 10 and 16 only.

3 In the **Electric Field (emw) 2** toolbar, click 🔲 **Plot**.

Global 1

1 In the **Model Builder** window, expand the **S-parameter (emw) 2** node, then click **Global 1**.

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

3 In the table, enter the following settings:

| Expression	Unit	Description
emw.S11dB	1	S11 Adaptive Frequency Sweep
emw.S21dB	1	S21 Adaptive Frequency Sweep

4 Right-click **Global 1** and choose **Duplicate**.

Global 2

1 In the **Model Builder** window, click **Global 2**.

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

3 From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
emw.S11dB	1	S11 Regular Sweep
emw.S21dB	1	S21 Regular Sweep

5 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Cycle**.

6 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

7 In the **S-parameter (emw) 2** toolbar, click ![plot icon] **Plot**.
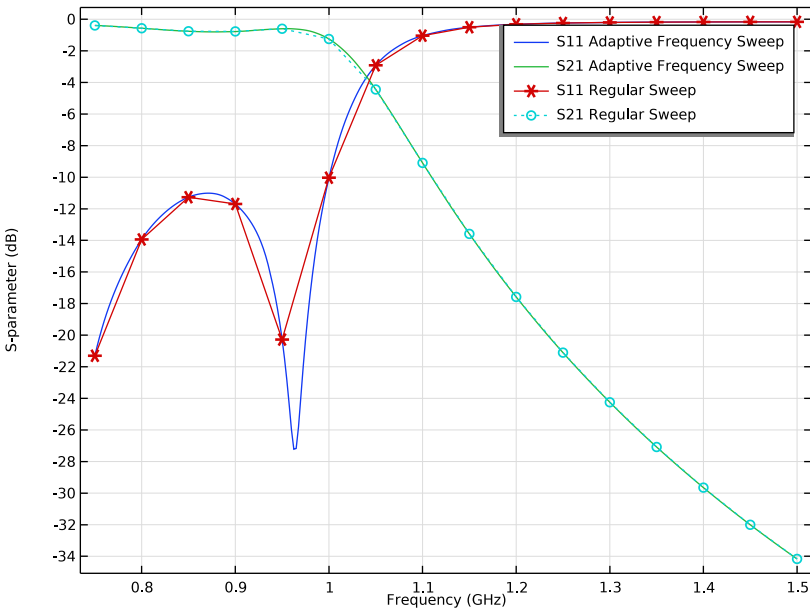
Compare the results from **Adaptive Frequency Sweep** to those from the regular discrete sweep.