



Microwave Filter on PCB with Stress (IPC-2581, ECAD Import)

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. The entire layout including the dielectric layer is imported from an IPC-2581 file using the ECAD Import feature.

Model Definition

The model uses the Electromagnetic Waves interface that solves the vector Helmholtz wave equation. The PCB layout of the Chebyshev filter is imported using the ECAD Import feature, which creates the entire 3D geometry from the layout and stackup information of an IPC-2581 file.

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 ECAD import has an option that causes the import engine to ignore the thickness of metal layers and insert them as faces on the dielectric layer. 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](#).

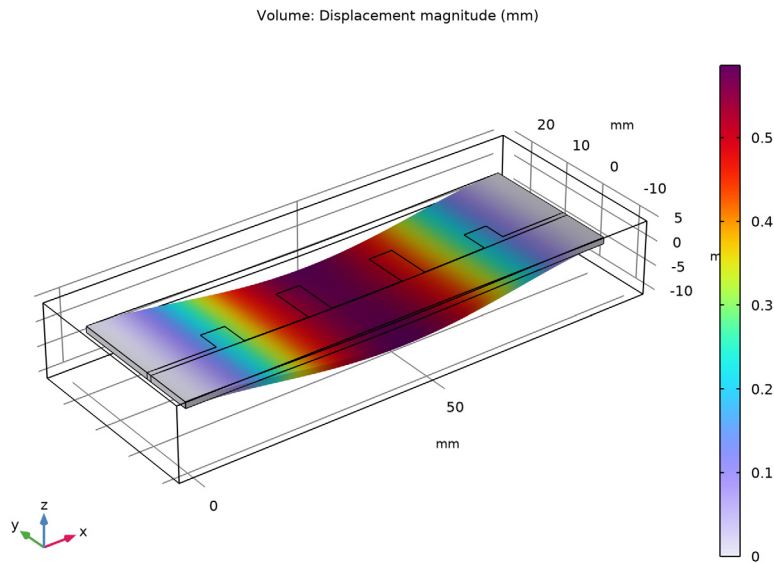


Figure 1: The graph shows the total displacement of the PCB, due to the load.

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.

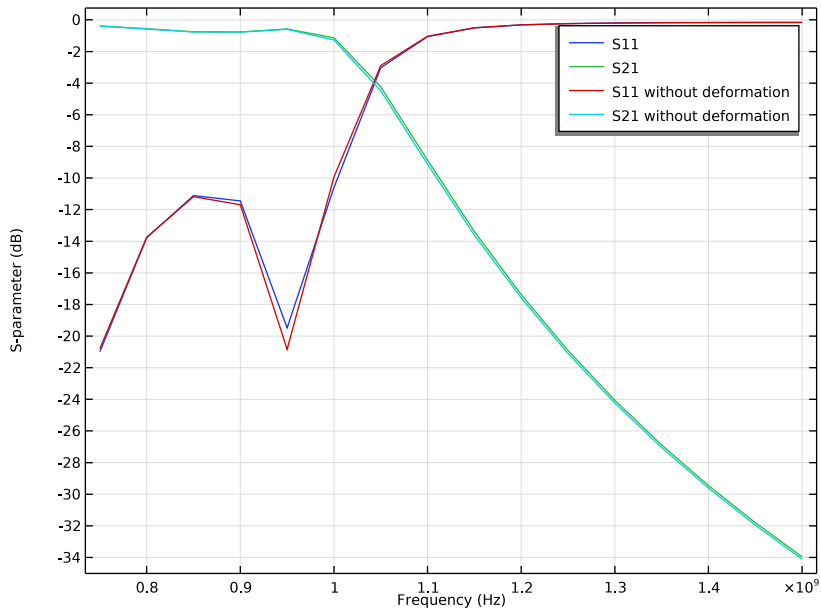


Figure 2: A comparison between the S_{11} and S_{21} parameters before and after a force of 40 N has been applied on the PCB. The red and turquoise lines correspond to the S-parameter curves for the filter with an applied force.

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_ecad




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, Frequency Domain (emw)**.
- 3 Click **Add**.
- 4 Click  **Study**.
- 5 In the **Select Study** tree, select **General Studies>Frequency Domain**.
- 6 Click  **Done**.

GLOBAL DEFINITIONS

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


GEOMETRY 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**.

The major part of the geometry is imported using the ECAD Import tool, which automatically reads a PCB layout and extrudes the layers to a 3D geometry.


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 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.

Import 1 (imp1)


- 1 In the **Geometry** toolbar, click  **Import**.
 - 2 In the **Settings** window for **Import**, locate the **Import** section.
 - 3 From the **Source** list, choose **ECAD file**.
 - 4 Click  **Browse**.
 - 5 Browse to the model's Application Libraries folder and double-click the file `pcb_microwave_filter_with_stress_ecad.xml`.

It is located in the same folder as specified in the Application Library Path on page 3.
This path originates from the folder **applications** under the COMSOL Multiphysics installation folder.
 - 6 From the **Type of import** list, choose **Metal shell**.

This will import all metal layers as faces, which drastically reduce the problem size in this model without compromising the accuracy.
 - 7 Click  **Import**.

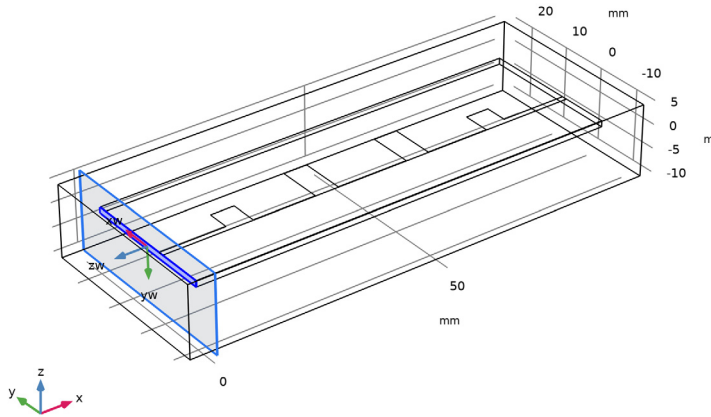
The PCB is now placed within the drawn block. To see the PCB, select wireframe rendering.
 - 8 Click the  **Wireframe Rendering** button in the **Graphics** toolbar.
- The next step is to add boundary faces for the input and output ports.

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 From the **Plane type** list, choose **Face parallel**.

- 4 On the object **impl.SIGNAL_LAYER_DIEL**, select Boundary 1 only.


It might be easier to select the correct boundary by using the **Selection List** window. To open this window, in the **Home** toolbar click **Windows** and choose **Selection List**. (If you are running the cross-platform desktop, you find **Windows** in the main menu.)



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**.

The size and position of the rectangle, as defined below, will perfectly fit it to the microstrip line.

- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type 1.125.
- 4 In the **Height** text field, type 1.27.
- 5 Locate the **Position** section. In the **xw** text field, type -4.77.
- 6 In the **yw** text field, type -0.635.


Work Plane 1 (wp1)

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Work Plane 1 (wp1)**.

- 2 In the **Settings** window for **Work Plane**, click  **Build Selected**.

This action embeds the 2D rectangle in the 3D geometry.


Work Plane 2 (wp2)

- 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 type** list, choose **Face parallel**.
- 4 On the object **imp1.SIGNAL_LAYER_DIEL**, select Boundary 6 only.


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 1.125.
- 4 In the **Height** text field, type 1.27.
- 5 Locate the **Position** section. In the **xw** text field, type 3.645.
- 6 In the **yw** text field, type -0.635.

Work Plane 2 (wp2)

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Work Plane 2 (wp2)**.
- 2 In the **Settings** window for **Work Plane**, click  **Build Selected**.

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 Click **Add to Component** in the window toolbar.
- 7 In the **Home** toolbar, click  **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	epsilon _{nr_iso} ; epsilon _{nr_ii} = epsilon _{nr_iso} , epsilon _{nr_ij} = 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  **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

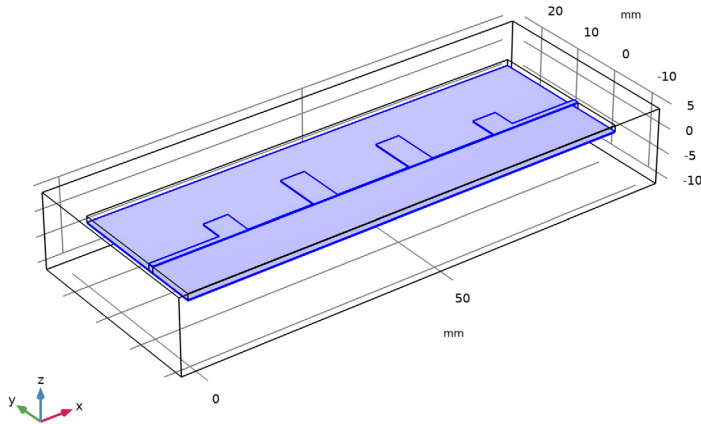
- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Lumped Port**.
- 2 Select Boundary 16 only.

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 (COMPI)

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:

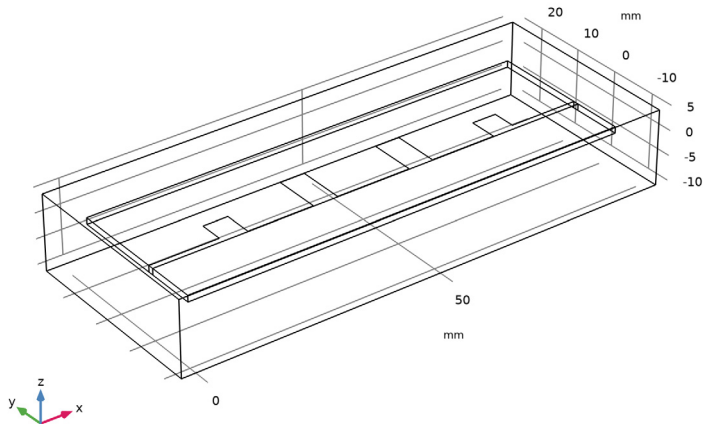
Name	Expression	Value	Description
fload	40[N]	40 N	Load on PCB


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

GEOMETRY I

```
Import I (impl I)
```

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Import 1 (imp1)**.
- 2 In the **Graphics** window toolbar, click  next to  **Select Boundaries**, then choose **Select Domains**.



- 3 On the object **impl.SIGNAL_LAYER_DIEL**, select Domain 1 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

Parameters 1

(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 (COMPI)

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)**.


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 Some Physics Interfaces>Stationary**.
- 4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Electromagnetic Waves, Frequency Domain (emw)**.
- 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 f_{start} in the **Start** text field.
- 6 In the **Step** text field, type f_{step} .
- 7 In the **Stop** text field, type f_{stop} .
- 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](#).

Stress (solid)


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

Volume 1

- 1 In the **Model Builder** window, expand the **Results>Stress (solid)>Volume 1** 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 In the **Stress (solid)** toolbar, click  **Plot**.

You should now see the plot in [Figure 1](#).