Model created in COMSOL Multiphysics 6.4

Mesh Adaptation Study for a Microstrip Patch Antenna

Introduction

In electromagnetic design and simulation, optimizing the mesh configuration is important for achieving accurate results, especially for high-Q devices. This tutorial model showcases mesh adaptation using the Microstrip Patch Antenna model from the RF Module Application Library. It illustrates how to refine the mesh in critical regions to enhance computation accuracy through the Frequency Domain, RF Adaptive Mesh study. The adapted mesh focuses on refining around the edges of the patch radiator, while excluding areas with lower field strength. This approach ensures both computational efficiency and precise characterization of antenna performance.

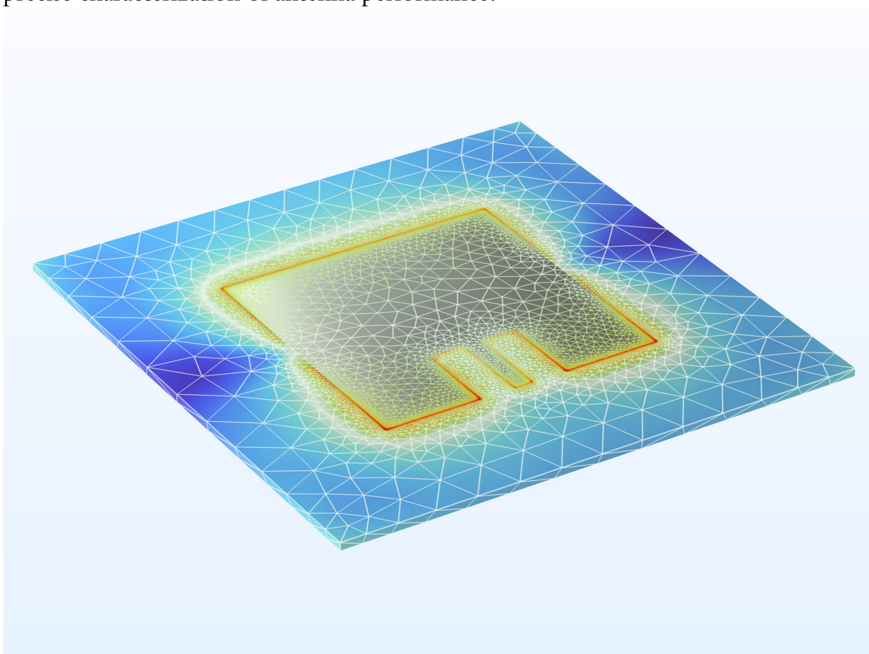


Figure 1: Microstrip patch antenna after a mesh adaptation study.

Model Definition

After loading the Microstrip Patch Antenna model from the Application Library, two additional studies are conducted:

- Frequency Domain, RF Adaptive Mesh for mesh refinement
- Adaptive Frequency Sweep for S-parameter evaluation with fine frequency resolution using the refined mesh

FREQUENCY DOMAIN, RF ADAPTIVE MESH

The Frequency Domain, RF Adaptive Mesh is a dedicated mesh refinement study that dynamically refines the mesh in regions of interest by increasing the resolution around areas with high field variations. Functionally, it is similar to the Adaptation and Error Estimates in a Frequency Domain study, but with more streamlined settings tailored for RF devices. In the Frequency Domain, RF Adaptive Mesh study settings window, the default mesh refinement settings can be reviewed by switching the Mesh refinement option from Physics controlled to User controlled.

Notable settings include the Error estimate, which is set to Functional based on S-parameters. The Adaptation method uses the Rebuild mesh option and employs Goal-oriented termination with a maximum of 20 adaptation steps. The termination expression is the same as the one used in Functional, formulates with S-parameters.

Mesh refinement can be performed at a single frequency or across a bandwidth of the device under test. In this example, refinement is applied at 5 frequency points within the simulation bandwidth.

ADAPTIVE FREQUENCY SWEEP

The Adaptive Frequency Sweep is useful for generating S-parameter plots with very fine frequency resolution. It uses the asymptotic waveform evaluation (AWE) method, which is more efficient than a regular discrete frequency sweep with fine frequency steps especially for single resonance devices or when the result scalar variable changes slowly across the simulation frequency spectrum. To manage the file size, particularly when dealing with many frequency points, consider storing output only from the Lumped port features. This approach significantly reduces file size dramatically preserving the S-parameter analysis results.

When this study is added, the mesh selection is automatically set to the refined mesh obtained from the previous study.

Results and Discussion

The this tutorial model demonstrates how mesh adaptation can be used to increase the result accuracy, by making the mesh finer in regions where it matters most for the results. Figure 2 shows the mesh after running the Frequency Domain, RF Adaptive Mesh study.

Compared to the initial mesh, the adapted mesh is much finer around the patch edges, except where the field strength is low.

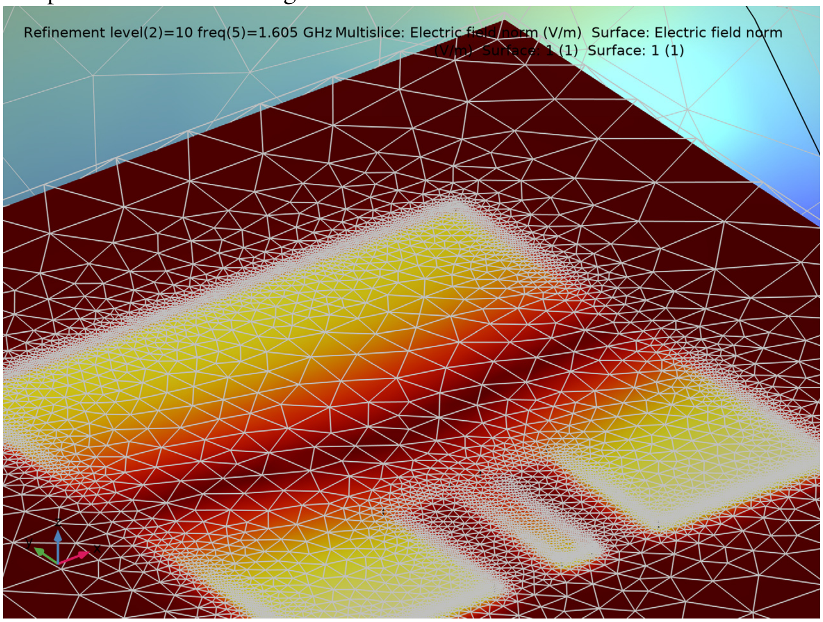


Figure 2: The mesh after running the Frequency Domain, RF Adaptive Mesh study.

[Figure 3](#) shows how the mesh adaptation converges to reach the termination tolerance of 0.02.

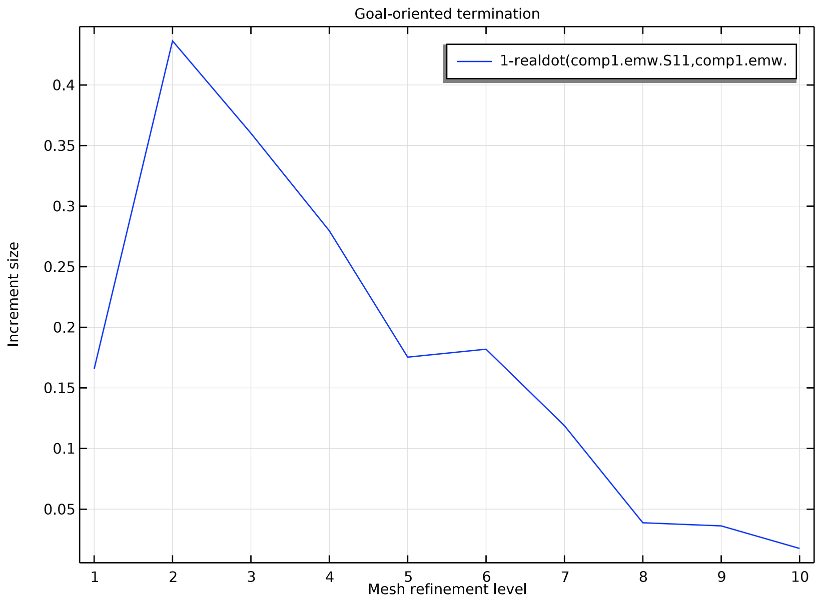


Figure 3: This plot shows the convergence of the mesh adaptation process. The process stops when the tolerance of 0.02 is reached.

Finally, Figure 4 shows that the resonance shifts to higher frequencies when using the finer adapted mesh.

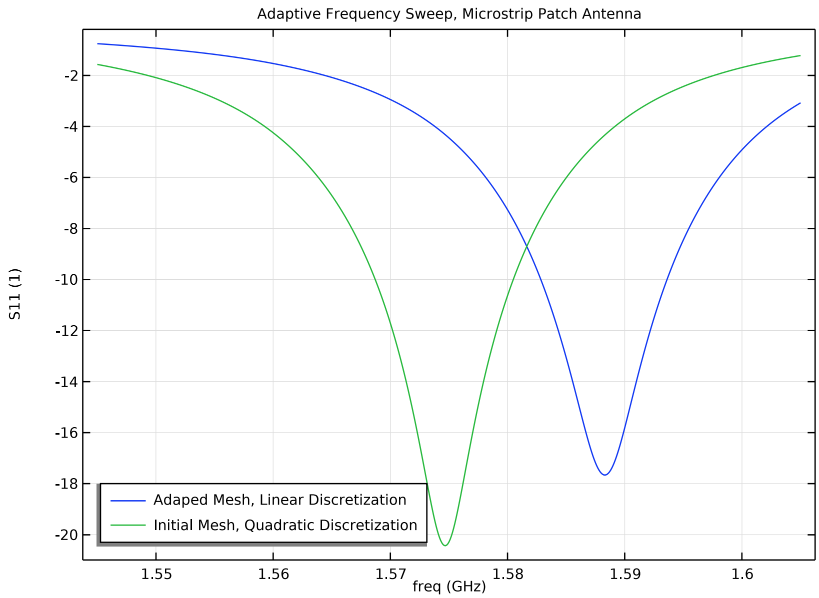


Figure 4: A comparison of the frequency spectra for the initial mesh and the adapted mesh.

Notes About the COMSOL Implementation

This example uses the Adaptive Frequency Sweep study, which employs the asymptotic waveform evaluation (AWE) model order reduction technique to compute the antenna's frequency response with fine frequency resolution. This method is faster than a regular frequency sweep performed in a Frequency Domain study using the same frequency resolution, but it is computationally intensive, and may require more than 6 GB of RAM. The results can be sensitive to the relative tolerance value in the settings window, so using a finer value can improve the accuracy. The Frequency Domain, RF Adaptive Mesh study uses the Direct Solver and may requires more than 7 GB of RAM.

Application Library path: `RF_Module/Antenna_Arrays/microstrip_patch_antenna_mesh_adaptation`

Modeling Instructions

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

Browse to the model's Application Libraries folder and double-click the file `microstrip_patch_antenna_inset.mph`.

STUDY 1, QUADRATIC DISCRETIZATION

1. In the **Model Builder** window, click **Study 1**.
2. In the **Settings** window for **Study**, type `Study 1, Quadratic Discretization` in the **Label** text field.

STUDY 2, QUADRATIC DISCRETIZATION, AWE

1. In the **Model Builder** window, click **Study 2**.
2. In the **Settings** window for **Study**, type `Study 2, Quadratic Discretization, AWE` in the **Label** text field.
3. Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.

ADD STUDY

If the loaded model does not have a solution, run the Study 2. Then, conduct a Frequency Domain, RF Adaptive Mesh study.

1. In the **Study** 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** > **Frequency Domain, RF Adaptive Mesh**.
4. Click the **Add Study** button in the window toolbar.
5. In the **Study** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 3, LINEAR DISCRETIZATION FOR MESH ADAPTATION

In the **Settings** window for **Study**, type `Study 3, Linear Discretization for Mesh Adaptation` in the **Label** text field.

Step 1: Frequency Domain, RF Adaptive Mesh

1. In the **Model Builder** window, under **Study 3, Linear Discretization for Mesh Adaptation** click **Step 1: Frequency Domain, RF Adaptive Mesh**.
2. In the **Settings** window for **Frequency Domain, RF Adaptive Mesh**, locate the **Study Settings** section.

3 Click Range.

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

5 In the **Start** text field, type `freq_min`.

6 In the **Stop** text field, type `freq_max`.

7 In the **Number of values** text field, type `5`.

8 Click **Replace**.

9 In the **Model Builder** window, click **Study 3, Linear Discretization for Mesh Adaptation**.

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

11 Clear the **Generate default plots** checkbox, as the default S-parameter and far-field plots are not of interest for the mesh adaptation study. However, in the following steps a field plot is built, so the mesh adaptation progress can be followed while solving.

Solution 3 (sol3)

In the **Study** toolbar, click **Show Default Solver**, to create the dataset for this study. The field plot that will be created will refer to this dataset.

RESULTS

Electric Field (emw), Mesh Adaptation

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

2 In the **Settings** window for **3D Plot Group**, type `Electric Field (emw), Mesh Adaptation` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 3, Linear Discretization for Mesh Adaptation/Adaptive Mesh Refinement Solutions 1 (sol4)**.

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

Multislice 1

1 In the **Electric Field (emw), Mesh Adaptation** toolbar, click **More Plots** and choose **Multislice**.

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

3 Find the **X-planes** subsection. In the **Planes** text field, type `0`.

4 Find the **Y-planes** subsection. In the **Planes** text field, type `0`.

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **Ctenophora**.

Selection 1

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

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

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

Electric Field (emw), Mesh Adaptation

1 In the **Model Builder** window, under **Results** click **Electric Field (emw), Mesh Adaptation**.

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

3 Clear the **Show legends** checkbox.

Surface 1

1 Right-click **Electric Field (emw), Mesh Adaptation** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 From the **Color table** list, choose **Ctenophora**.

Selection 1

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

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

3 From the **Selection** list, choose **PML, Inside Boundaries**.

4 In the list, choose **10 (hidden)** and **33 (hidden)**.

5 Click **Remove from Selection**, to avoid that the mesh is visualized on the boundaries that otherwise are hidden in the physics view.

6 Select Boundaries 9, 11, 12, 32, 37, and 40 only.

Surface 2

1 In the **Model Builder** window, under **Results** > **Electric Field (emw), Mesh Adaptation** right-click **Surface 1** and choose **Duplicate**, to add the first of two surface plots to visualize the mesh.

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**.

6 Select the **Wireframe** checkbox.

Surface 3

Right-click **Surface 2** and choose **Duplicate**.

Selection 1

1 In the **Model Builder** window, expand the **Surface 3** node, then click **Selection 1**.

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

3 From the **Selection** list, choose **Substrate Boundaries**.

STUDY 3, LINEAR DISCRETIZATION FOR MESH ADAPTATION

Step 1: Frequency Domain, RF Adaptive Mesh

1 In the **Model Builder** window, under **Study 3, Linear Discretization for Mesh Adaptation** click **Step 1: Frequency Domain, RF Adaptive Mesh**.

2 In the **Settings** window for **Frequency Domain, RF Adaptive Mesh**, locate the **Study Settings** section.

3 In the **Damping factor** text field, type `0.05`.

4 In the **Maximum number of adaptations** text field, type `20`.

5 Click to expand the **Results While Solving** section. Select the **Plot** checkbox.

6 In the table, enter the following settings:

Plot group	Plot window
Electric Field (emw), Mesh Adaptation	Graphics

Adjust the view, to clearly see the mesh on both the substrate and the boundary toward the PML.

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

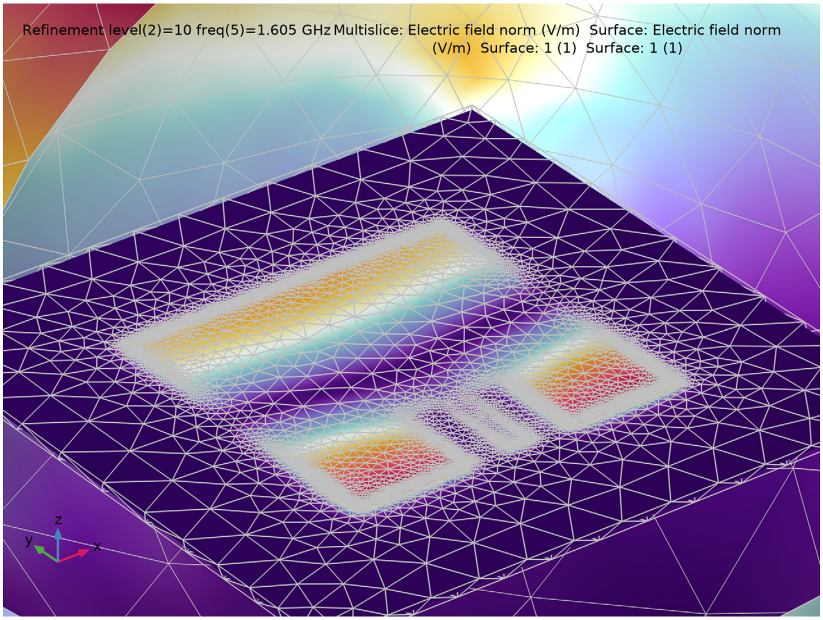
8 Click the **Zoom In** button in the **Graphics** toolbar three times.

9 In the **Study** toolbar, click **Compute**.

In the **Graphics** window, follow how the mesh is adapted while solving.

RESULTS

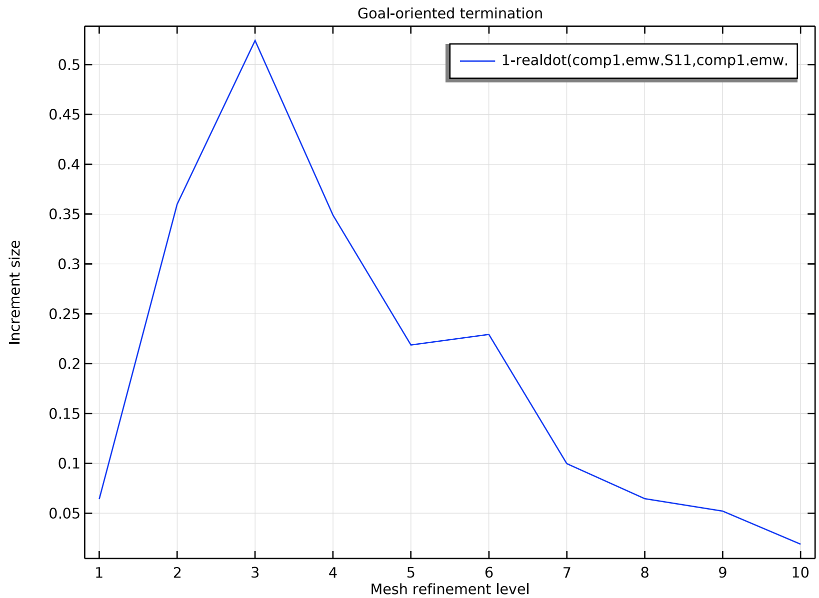
Electric Field (emw), Mesh Adaptation



Notice that the mesh is much denser in the adapted mesh around the patch edges, compared to the original mesh, except where the field strength is low.

Goal-Oriented Termination Expression 3

1 In the **Model Builder** window, click **Goal-Oriented Termination Expression 3**.



This plot shows how the mesh adaptation converges and stops when the increment size is less than the tolerance of 0.02.

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 > Adaptive Frequency Sweep**.

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

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

STUDY 4, LINEAR DISCRETIZATION WITH ADAPTED MESH

1 In the **Settings** window for **Study**, type `Study 4, Linear Discretization with Adapted Mesh` in the **Label** text field.

2 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox, as we again will not be saving the field in this study. Therefore, there is no point in generating the default field plots.

Step 1: Adaptive Frequency Sweep

1 In the **Model Builder** window, under **Study 4, Linear Discretization with Adapted Mesh** click **Step 1: Adaptive Frequency Sweep**.

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

3 In the **Frequencies** text field, type `range(freq_min,100[kHz],freq_max)`.

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 the first row in the table.

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

7 In the **Add** dialog, select **Lumped Port** in the **Selections** list.

8 Click **OK**.

Again, only include the lumped port boundaries in the S-parameter calculation to reduce the size of the model.

9 In the **Study** toolbar, click **Compute**.

RESULTS

S-parameter, Asymptotic Waveform Evaluation on Adapted Mesh

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

2 In the **Settings** window for **1D Plot Group**, type `S-parameter, Asymptotic Waveform Evaluation on Adapted Mesh` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 4, Linear Discretization with Adapted Mesh/Solution 16 (sol16)**.

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

5 In the **Title** text area, type `Adaptive Frequency Sweep, Microstrip Patch Antenna`.

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

Global 1

1 Right-click **S-parameter, Asymptotic Waveform Evaluation on Adapted Mesh** 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, Frequency Domain** > **Ports** > **emw.S11dB - S11 - dB**.

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

4 In the table, enter the following settings:

Legends
`Adapted Mesh, Linear Discretization`

5 Right-click **Global 1** and choose **Copy**.

Global 2

1 In the **Model Builder** window, right-click **S-parameter, Asymptotic Waveform Evaluation on Adapted Mesh** and choose **Paste Global**.

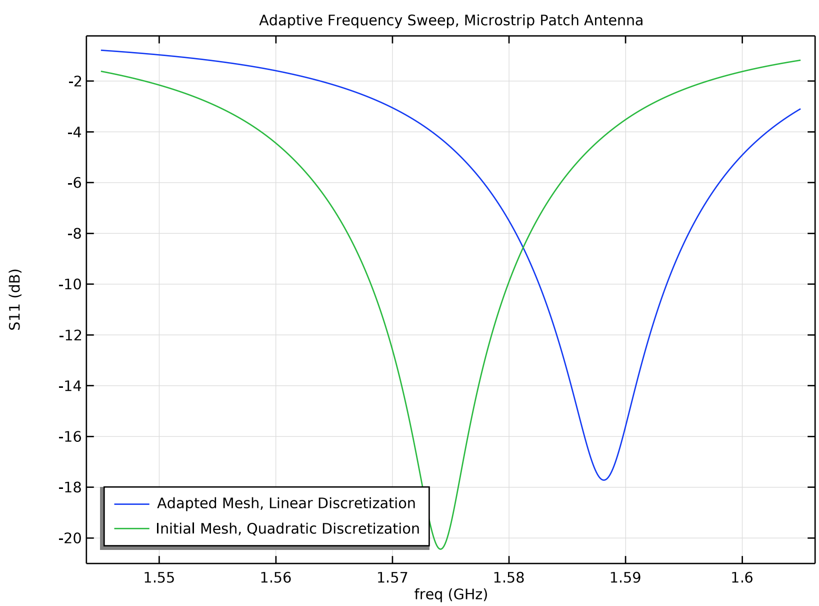
2 In the **Settings** window for **Global**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2, Quadratic Discretization, AWE/Solution 2 (sol2)**.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
`Initial Mesh, Quadratic Discretization`

5 In the **S-parameter, Asymptotic Waveform Evaluation on Adapted Mesh** toolbar, click **Plot**.



For the denser, adapted mesh, the resonance has shifted to a slightly higher frequency.