



Full-wave, 3D, Electromagnetic  
Analysis Software  
**USER'S GUIDE**





# XFdtd® User's Guide

Release 7.9.2



Remcom Inc.  
315 S. Allen St., Suite 416  
State College, PA 16801

+1.888.7.REMCOM  
+1.814.861.1299  
+1.814.861.1308 fax  
[www.remcom.com](http://www.remcom.com)

XFdtd is versioned with four numbers as *major.minor.feature.bugfix*. A change in any number except the *bugfix* number indicates that new features have been added to the software.

This document has been updated for, prepared and delivered with XFdtd 7.9.2.2, June 2021.

# Contents

<b>List of Figures</b>	<b>vii</b>
<b>1 Creating Simulations with XFDTD: An Overview</b>	<b>1</b>
1.1 Constructing the Geometry . . . . .	1
1.2 Defining the Grid and Creating the Mesh . . . . .	2
1.3 Defining Run Parameters . . . . .	3
1.4 Requesting Results . . . . .	3
1.5 Running a Simulation . . . . .	3
1.6 Viewing Output . . . . .	4
1.7 Other Tools . . . . .	4
<b>2 Quick Example: A Bow Tie Antenna</b>	<b>7</b>
2.1 Getting Started . . . . .	8
2.2 Creating the Bow Tie Antenna Geometry . . . . .	8
2.3 Setting the Material . . . . .	10
2.4 Create the Feed Circuitry . . . . .	12
2.5 Far Zone Sensor . . . . .	14
2.6 Run the Simulation . . . . .	14
2.7 Viewing Results . . . . .	15
<b>3 Example: A Microstrip Patch Antenna</b>	<b>21</b>
3.1 Getting Started . . . . .	22
3.2 Creating the Patch Antenna Geometry . . . . .	22
3.2.1 Modeling the Substrate . . . . .	22
3.2.2 Modeling the Microstrip Patch . . . . .	24
3.3 Creating Materials . . . . .	27
3.4 Assigning Materials . . . . .	29
3.5 Defining the Outer Boundary . . . . .	30
3.6 Defining the Grid . . . . .	31
3.7 Adding a Feed . . . . .	32
3.8 Requesting Output Data . . . . .	33
3.9 Running a Simulation . . . . .	35
3.10 Viewing the Results . . . . .	36
3.11 Adding a Parameter Sweep . . . . .	39
3.11.1 Parameterizing the Geometry . . . . .	40
3.11.2 Setting the Mesh Priority . . . . .	42

3.11.3 Parameterizing the Feed . . . . .	43
3.11.4 Adding Automatic Fixed Points to the Microstrip Object . . . . .	44
3.12 Running a Simulation with Parameter Sweep . . . . .	46
3.13 Viewing Results of the Parameter Sweep . . . . .	47
<b>4 Example: A Low Pass Filter</b>	<b>49</b>
4.1 Getting Started . . . . .	50
4.2 Creating Materials . . . . .	50
4.3 Creating the Low Pass Filter Geometry . . . . .	53
4.3.1 Modeling the Substrate . . . . .	53
4.3.2 Modeling the Strip Line . . . . .	56
4.4 Defining the Outer Boundary . . . . .	59
4.5 Defining the Grid . . . . .	60
4.6 Adding a Feed . . . . .	62
4.7 Adding a Load . . . . .	64
4.8 Requesting Output Data . . . . .	65
4.9 Running the Calculation . . . . .	67
4.10 Viewing the Results . . . . .	68
<b>5 Example: A Rectangular Waveguide</b>	<b>71</b>
5.1 Getting Started . . . . .	72
5.2 Creating the Waveguide Geometry . . . . .	72
5.2.1 Modeling the Waveguide . . . . .	72
5.3 Creating Materials . . . . .	76
5.4 Defining the Outer Boundary . . . . .	81
5.5 Defining the Grid . . . . .	81
5.6 Adding a Waveform . . . . .	85
5.7 Add a Waveguide Interface and Evaluate the Modes . . . . .	86
5.8 Requesting Results . . . . .	91
5.9 Running the Simulation . . . . .	92
5.10 Viewing the Results . . . . .	93
<b>6 Example: A Monopole Antenna on a Conducting Box</b>	<b>99</b>
6.1 Getting Started . . . . .	100
6.2 Parameterizing the Project . . . . .	100
6.3 Creating the Monopole Antenna Geometry . . . . .	101
6.3.1 Modeling the Box . . . . .	101
6.3.2 Modeling the Monopole . . . . .	103
6.4 Creating Materials . . . . .	105
6.5 Assigning Materials . . . . .	106
6.6 Defining the Outer Boundary . . . . .	107
6.7 Defining the Grid . . . . .	108
6.8 Adding a Feed . . . . .	109
6.9 Viewing the Waveform . . . . .	111
6.10 Requesting Output Data . . . . .	111
6.11 Running the Calculation . . . . .	113
6.12 Viewing the Results . . . . .	115
<b>7 Example: A Pyramidal Horn</b>	<b>121</b>

---

7.1	Getting Started . . . . .	122
7.2	Creating the Pyramidal Horn Geometry . . . . .	122
7.2.1	Modeling the Horn . . . . .	122
7.2.2	Modeling the Excitation Wire . . . . .	124
7.3	Creating Materials . . . . .	125
7.4	Assigning Materials . . . . .	127
7.5	Defining the Outer Boundary . . . . .	128
7.6	Defining the Grid . . . . .	129
7.7	Configuring the Mesh . . . . .	129
7.8	Adding a Feed . . . . .	130
7.9	Running a Simulation . . . . .	131
7.10	Viewing the Results . . . . .	132
<b>8</b>	<b>Example: A Simple SAR Calculation</b>	<b>137</b>
8.1	Getting Started . . . . .	138
8.2	Creating the SAR Geometry . . . . .	138
8.2.1	Modeling the Tissue . . . . .	138
8.2.2	Modeling the Dipole . . . . .	141
8.3	Creating Materials . . . . .	141
8.4	Assigning Materials . . . . .	144
8.5	Defining the Outer Boundary . . . . .	145
8.6	Defining the Grid . . . . .	146
8.7	Adding a Feed to the Dipole Wire . . . . .	147
8.8	Requesting Output Data . . . . .	148
8.9	Running the Calculation . . . . .	150
8.10	Viewing the Results . . . . .	151
<b>9</b>	<b>Example: A Validation of SAR Calculations</b>	<b>157</b>
9.1	Getting Started . . . . .	158
9.2	Creating the Geometry . . . . .	158
9.2.1	Modeling the Flat Phantom . . . . .	158
9.2.2	Modeling the Phantom Shell . . . . .	160
9.2.3	Modeling the Dipole . . . . .	161
9.3	Creating Materials . . . . .	164
9.4	Assigning Materials . . . . .	168
9.5	Defining the Outer Boundary . . . . .	168
9.6	Defining the Grid . . . . .	169
9.7	Adding a Feed . . . . .	170
9.8	Requesting Output Data . . . . .	171
9.9	Running the Calculation . . . . .	174
9.10	Viewing the Results . . . . .	175
<b>10</b>	<b>Appendix: Application Preferences</b>	<b>187</b>
<b>Bibliography</b>		<b>189</b>



# List of Figures

1.1 XF Project Workflow . . . . .	2
2.1 Bow tie antenna . . . . .	7
2.2 Bow tie vertices . . . . .	8
2.3 Sheet body creation with completed bow tie . . . . .	10
2.4 Create a New Material of Type PEC . . . . .	11
2.5 The profile of the bow tie antenna geometry . . . . .	12
2.6 Feed Attached to the Bow Tie . . . . .	13
2.7 Circuit Component Definition for Bow Tie Feed . . . . .	14
2.8 Simulation Dialog . . . . .	15
2.9 Results Browser . . . . .	16
2.10 Available Results with “S Parameters” Selected . . . . .	16
2.11 S Parameters for the Bow Tie with a $300\Omega$ Feed . . . . .	17
2.12 Gain of the Bow Tie Antenna . . . . .	18
2.13 Available Results with “Impedance” Selected . . . . .	18
2.14 Creating a Line Graph of Real and Imaginary Frequency-Domain Impedance Data . . . . .	19
2.15 Complex Impedance vs Frequency for the Bowtie . . . . .	19
3.1 The microstrip patch antenna . . . . .	21
3.2 Selecting the geometry tool to perform an extrusion . . . . .	23
3.3 Selecting the rectangle tool . . . . .	23
3.4 Specifying the coordinates of the rectangle corner . . . . .	23
3.5 Specifying the coordinates of the rectangle corner . . . . .	24
3.6 Specifying the extrude distance . . . . .	24
3.7 Selecting the geometry tool to create a sheet body . . . . .	25
3.8 Picking the Origin . . . . .	25
3.9 Selecting the Trim Edges tool . . . . .	26
3.10 Manually setting the meshing order of the Microstrip . . . . .	27
3.11 Creating a new material . . . . .	27
3.12 Editing the color of the PEC material . . . . .	28
3.13 Editing the properties of the Duroid material . . . . .	29
3.14 Assigning materials to the patch antenna geometry . . . . .	30
3.15 The profile of the patch antenna geometry . . . . .	30
3.16 Defining the outer boundary for the patch antenna . . . . .	31
3.17 Defining the grid extents on the Grid Editor . . . . .	32
3.18 Adding a feed to the project . . . . .	32

3.19 Adding a feed with the Picker tool . . . . .	33
3.20 Adding a planar sensor to the project . . . . .	34
3.21 Adding the sensor definition . . . . .	35
3.22 Adding a new simulation to the patch antenna project . . . . .	36
3.23 Viewing S-Parameters in the Results window . . . . .	37
3.24 Viewing S-Parameters v. Frequency plot . . . . .	38
3.25 Viewing E-Field in the Geometry window . . . . .	39
3.26 Defining a global parameter in the parameters window . . . . .	40
3.27 Navigating to the Sheet Body object . . . . .	40
3.28 Deleting an edge of the stub . . . . .	41
3.29 Parameterizing the stub position . . . . .	42
3.30 Manually setting the meshing order of the Stub . . . . .	43
3.31 Parameterizing the position of the feed . . . . .	43
3.32 Patch Antenna Mesh with and without Automatic Fixed Points . . . . .	44
3.33 Selecting Gridding Properties from the context menu . . . . .	45
3.34 Adding automatic fixed points to the Microstrip . . . . .	46
3.35 Adding the parameter to the simulation . . . . .	47
3.36 The results of the parameter sweep . . . . .	48
 4.1 The low pass filter . . . . .	49
4.2 Editing the color of the PEC material . . . . .	51
4.3 Defining the properties of the Substrate material . . . . .	52
4.4 PEC and Substrate materials in the Project Tree . . . . .	52
4.5 The substrate script in the Scripting workspace window . . . . .	55
4.6 Applying Substrate material through scripting interface . . . . .	56
4.7 The Substrate block in the Project Tree . . . . .	56
4.8 Strip Line geometry created from script . . . . .	57
4.9 Low pass filter geometry . . . . .	58
4.10 Manually setting the meshing order of the Strip Line . . . . .	59
4.11 Defining the outer boundary for the low pass filter . . . . .	60
4.12 Defining cell sizes in the Grid Editor . . . . .	60
4.13 Defining free space padding in the Grid Editor . . . . .	61
4.14 Selecting Gridding Properties from the context menu . . . . .	61
4.15 Adding automatic fixed points to the StripLine . . . . .	62
4.16 Adding a feed to the project . . . . .	63
4.17 Adding a feed with the Picker tool . . . . .	64
4.18 Zooming in on the location of the Load . . . . .	65
4.19 Centering the planar sensor on the Strip Line . . . . .	65
4.20 Adding the sensor definition . . . . .	66
4.21 Adding a new simulation to the low pass filter project . . . . .	67
4.22 Viewing results plot to ensure convergence at the Load . . . . .	68
4.23 Viewing results plot to ensure convergence at the Feed . . . . .	69
4.24 Viewing E-field results for the surface sensor near the end of the time sequence . . . . .	70
 5.1 Teflon Filled WR-42 Rectangular Waveguide . . . . .	71
5.2 Creating Cuboid . . . . .	73
5.3 Selecting Open Faces for Shell Feature . . . . .	74
5.4 Specifying the Shell Thickness . . . . .	75

---

5.5	Creating a Cuboid for the Dielectric Insert . . . . .	76
5.6	Libraries Workspace Window . . . . .	77
5.7	Pulling in Material Definition from Materials Library . . . . .	78
5.8	Copper (Pure)[ND] from the Materials Library . . . . .	79
5.9	Material Properties of Teflon . . . . .	80
5.10	Parts List with Associated Materials . . . . .	80
5.11	Defining the Outer Boundary for the Waveguide Simulation . . . . .	81
5.12	Defining the grid extents on the Grid Editor . . . . .	82
5.13	Mesh Viewing Controls . . . . .	83
5.14	Gridding Properties Menu Item . . . . .	84
5.15	Gridding Properties for WR-42 . . . . .	85
5.16	Automatic Waveform . . . . .	86
5.17	Waveguide Interfaces Project Tree Location . . . . .	86
5.18	Waveguide Interface Properties Tab . . . . .	87
5.19	Waveguide Interface Geometry Tab . . . . .	87
5.20	Waveguide Port Specification Tab with Mode Results . . . . .	88
5.21	Calculated TE <sub>20</sub> Mode . . . . .	89
5.22	Waveguide Receiver Interface Geometry Tab . . . . .	90
5.23	Solid Sensor Definition . . . . .	91
5.24	Setting up S-Parameters on the New Simulation Window . . . . .	92
5.25	Disabling far zone data collection on the New Simulation Window . . . . .	93
5.26	Field Control Panel . . . . .	94
5.27	X Bounds for Solid Sensor Results . . . . .	94
5.28	Change the Viewpoint to the View from the Left . . . . .	95
5.29	Solid Sensor Results in the YZ Plane . . . . .	95
5.30	Solid Sensor Results in the XY Plane . . . . .	96
5.31	Magnetic Fields in the ZX Plane . . . . .	97
6.1	The Monopole Antenna on a Conducting Box . . . . .	99
6.2	Defining parameterized values . . . . .	101
6.3	Selecting the geometry tool to perform an extrusion . . . . .	101
6.4	Selecting the Distance constraint tool . . . . .	102
6.5	Adding the length constraint to the rectangle . . . . .	102
6.6	The Box with completed side constraints . . . . .	102
6.7	Setting the extrude options for the Box . . . . .	103
6.8	Setting the extrude options for the Monopole . . . . .	103
6.9	Placing the origin at the center of the Monopole . . . . .	104
6.10	Adjusting the orientation plane to sketch the Monopole . . . . .	104
6.11	Selecting the correct orientation . . . . .	104
6.12	Constraining the length of the Monopole wire . . . . .	105
6.13	Editing the color of the PEC material . . . . .	106
6.14	Assigning materials to the Monopole and Box parts . . . . .	107
6.15	The profile of the Monopole Box geometry . . . . .	107
6.16	Defining the outer boundary for the monopole box project . . . . .	108
6.17	Defining cell size on the Grid Editor . . . . .	108
6.18	Adding a feed to the project . . . . .	109
6.19	Using the Pick tool to place the first endpoint of the Feed . . . . .	110
6.20	Defining the location the feed . . . . .	110

---

6.21 Viewing the sinusoidal waveform . . . . .	111
6.22 Adding the sensor definition . . . . .	112
6.23 Setting up the simulation for the monopole box project . . . . .	114
6.24 Viewing power and efficiency results . . . . .	115
6.25 Viewing the conduction current plot for the monopole project . . . . .	116
6.26 Unloading surface current results . . . . .	117
6.27 Setting up far zone post-processing . . . . .	118
6.28 Far zone gain data for the monopole project . . . . .	119
7.1 The pyramidal horn . . . . .	121
7.2 CAD import options for the horn . . . . .	123
7.3 The imported horn geometry . . . . .	124
7.4 Specifying the orientation of the Excitation Wire inside the Horn . . . . .	125
7.5 Editing the color of the PEC material . . . . .	126
7.6 Editing the properties of the Copper material . . . . .	127
7.7 The profile of the pyramidal horn geometry . . . . .	128
7.8 Defining the outer boundary for the pyramidal horn . . . . .	128
7.9 Disabling ProGrid boundary refinement in the Grid Editor . . . . .	129
7.10 The Meshing Properties editor shown with the “Use XACT Mesh” box selected for the Horn object . . . . .	130
7.11 Adding a feed to the project . . . . .	130
7.12 Adding a new simulation to the pyramidal horn project . . . . .	132
7.13 Checking the convergence of the voltage waveform . . . . .	133
7.14 Setting up far zone post-processing for the horn . . . . .	134
7.15 Far zone gain data for the horn antenna project . . . . .	135
8.1 A Simple SAR example . . . . .	137
8.2 Selecting the geometry tool to perform an extrusion . . . . .	139
8.3 Selecting the circle tool . . . . .	139
8.4 Sketching the circular cross-section of the cylinder . . . . .	140
8.5 Sketching the circular cross-section of the cylinder . . . . .	140
8.6 The finished SAR geometry . . . . .	141
8.7 Editing the color of the PEC material . . . . .	142
8.8 Editing the Fat, Yellow Marrow material . . . . .	143
8.9 Assigning Physical Parameters to Material . . . . .	144
8.10 Finished geometry with applied material definitions . . . . .	145
8.11 Defining the outer boundary for the SAR calculation . . . . .	146
8.12 Defining cell size on the Grid Editor . . . . .	146
8.13 Adding a feed to the project . . . . .	147
8.14 Adding the SAR sensor definition . . . . .	148
8.15 Adding the sensor definition . . . . .	149
8.16 Setting up the simulation for the SAR project . . . . .	151
8.17 Viewing the E-field at the center of the Tissue . . . . .	152
8.18 Viewing SAR results at the Feed . . . . .	153
8.19 Setting the view of the SAR sensor data . . . . .	154
8.20 SAR sensor data . . . . .	155
9.1 The Flat Phantom . . . . .	157

---

9.2 Selecting the geometry tool to perform an extrusion . . . . .	159
9.3 Selecting the rectangle tool . . . . .	159
9.4 Specifying the extrude distance . . . . .	160
9.5 Specifying the phantom shell size . . . . .	161
9.6 Specifying the cylinder sketching plane orientation . . . . .	162
9.7 A larger view of the cross-section in relation to the phantom geometry . . . . .	162
9.8 The finished Phantom geometry . . . . .	164
9.9 Editing the color of the PEC material . . . . .	165
9.10 Editing the electric properties of the Phantom Liquid material . . . . .	166
9.11 Editing the physical parameters Phantom Liquid material . . . . .	167
9.12 Defining the outer boundary for the SAR Phantom calculation . . . . .	168
9.13 Defining cell size and min feature size on the Grid Editor . . . . .	169
9.14 Adding a feed to the project . . . . .	170
9.15 Defining Feed Properties on the Circuit Component Editor . . . . .	171
9.16 Adding the SAR Averaging sensor definition . . . . .	172
9.17 Adding the sensor definition . . . . .	173
9.18 Setting up the simulation for the SAR Phantom project . . . . .	175
9.19 Viewing the E-field at the center of the Flat Phantom . . . . .	176
9.20 The power table from the SAR Phantom simulation, including input and dissipated powers	177
9.21 The power table after the input power has been adjusted to 1 W . . . . .	178
9.22 The steady-state output data for the feed . . . . .	179
9.23 Setting the view of the SAR sensor data . . . . .	181
9.24 The SAR statistics for the simulation showing the peak values and locations . . . . .	182
9.25 The display of the feed statistics after scaling the feed point current to 200 mA . . . . .	183
9.26 Creating a line plot of the SAR above the feed point . . . . .	184
9.27 A line plot of the SAR as a function of distance above the feed point . . . . .	185
10.1 Setting the project display units . . . . .	188



# Chapter 1

## Creating Simulations with XFDTD: An Overview

*In this chapter, you will learn about...*

- the major steps you will take to create a simulation in XFDTD
- extra tools you can use to make your XFDTD simulation easier to set up

This User's Guide provides several how-to examples to familiarize the new users to the XFDTD interface. Each example will take the user through a step-by-step process to set up and run a full XFDTD simulation.

One of the most powerful aspects of XFDTD 7 is the flexibility it offers users to customize and organize their projects. Features such as scripting and parameterization make it possible to quickly and efficiently create or modify projects without carrying out tedious steps in the Graphical User Interface (GUI). As a result, there are a variety of ways to create an accurate simulation. This guide provides some basic instructions for how to set up and run simulations in XFDTD. Each of the following sections contain a brief description of the main steps to create a simulation, as summarized in the XFDTD project workflow of Figure 1.1.

- ▶ For a more thorough explanation of the features available in XFDTD, please see the XFDTD Reference Manual included with the software package.

### 1.1 Constructing the Geometry

XFDTD uses solid, dimension-based modeling to create geometries. It employs a concept called **Feature Based Modeling**, in which geometric objects are created as a set of repeatable actions so that operations can be undone and redone quickly. All modeling begins with a simple 2-D cross-section that is manipulated as needed to create the intended object. For projects that require common geometries, it is possible to create  TEMPLATES out of geometric objects or export them to  LIBRARIES to make it easy to import them into new projects. CAD files from third-party solid modeling packages may also be imported.

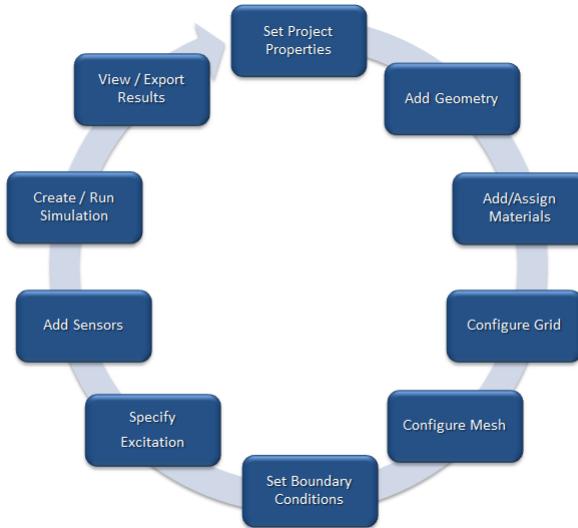


Figure 1.1: XF Project Workflow

After building or importing the geometrical objects, you can assign materials to them by creating MATERIAL definition objects and applying them via drag-and-drop.

Discrete CIRCUIT COMPONENTS may be added to the geometry as well. In previous versions, circuit components were defined in terms of their placement in the mesh, but this method has been revised so that their location (as well as all other physical objects) is defined in terms of their global position in the simulation space. This eliminates the chance that a circuit component's location is altered during meshing, when cells tend to shift.

## 1.2 Defining the Grid and Creating the Mesh

Once the geometry has been created and materials have been applied, the grid should be initialized within the GRID TOOLS interface. It is good to keep several considerations in mind when choosing an appropriate cell size for the grid:

1. **Wavelength:** The primary constraint on cell size is wavelength. The FDTD cell can be no larger than 1/10 of the smallest wavelength used to excite the model. Hence the maximum cell size may be determined from:

$$L_{max} = \frac{c}{10 * f}$$

Where

- $L_{max}$  is the maximum cell dimension
- $c$  is the speed of light,  $3 \times 10^8$  m/s in free space
- $f$  is the frequency of excitation (Hz)

If materials other than good conductors are included in the calculation, the velocity of light will be reduced in those materials and the FDTD cell size must be reduced accordingly.

2. **Geometry features:** The FDTD cell can be no larger than the smallest feature of your geometry. For example, if the distance between two wires in the geometry is smaller than the maximum cell size, a smaller cell size is needed.
3. **Accuracy:** Smaller cell sizes result in greater accuracy in the simulation.

After initializing the grid, you can create a MESH for the project, and then run calculations.

## 1.3 Defining Run Parameters

In order to run a calculation, it is important to set up the necessary parameters. XFDTD 7 allows you to specify the following run parameters:

- CIRCUIT COMPONENTS
- WAVEGUIDES
- EXTERNAL EXCITATIONS
- WAVEFORMS
- OUTER BOUNDARIES

A CIRCUIT COMPONENT DEFINITION is automatically added to the project as soon as a new CIRCUIT COMPONENT is added. The CIRCUIT COMPONENT DEFINITION EDITOR is used to make modifications to this definition.

EXTERNAL EXCITATIONS are added using the PLANE WAVE EDITOR and GAUSSIAN BEAM EDITOR. The source type, whether it be a discrete source or an external excitation, is set in the SIMULATIONS workspace window prior to running the calculation.

WAVEFORMS are created or edited within the WAVEFORM EDITOR. If a discrete circuit component was already added to the project, a default waveform is automatically added to the project.

OUTER BOUNDARIES are defined within the OUTER BOUNDARY EDITOR. Defining the characteristics of the outer boundary enables the calculation engine to provide accurate results.

## 1.4 Requesting Results

Results are collected and stored with objects called SENSORS. There are different types of sensors available depending on what type of data is to be collected.

## 1.5 Running a Simulation

Simulations are conveniently created, defined, and stored in the SIMULATIONS workspace window. Any number of simulations may be queued at one time in this window. They will run one at a time until all

simulations are finished calculating. This workspace window is superior to past releases since run parameters are manipulated within one common place so that multiple simulations can be queued without revisiting many different parts of the GUI. Specifications such as SOURCE TYPE, PARAMETER SWEEPS, S-PARAMETER CALCULATIONS, FREQUENCIES OF INTEREST, TOTAL/SCATTERED FIELD INTERFACES, and TERMINATION CRITERIA are defined during this step.

## 1.6 Viewing Output

After running the calculation, view the results from the RESULTS workspace window. Some results will be in the form of numerical values. Other results will be displayed in the form of plots. There are several types of plots available to view results based on whether they are time-dependent, frequency-dependent, or angle-dependent. Finally, some results will be available to review as colored field displays.

## 1.7 Other Tools

There are several optional tools available in XFDTD that are used throughout the simulation creation process.

## Scripting

SCRIPTING allows users to customize XFDTD to perform virtually any task. Scripts are typically used to automate repetitive or tedious tasks (that could otherwise be done through the GUI) with speed and precision.

- ▶ The low pass filter example in Chapter 4 is a good introduction to scripting. It uses 2 simple scripts to build the geometry of the project.

## Parameterization

PARAMETERS are global variables that are defined and stored in one common workspace window and can be referenced anywhere in the XFDTD interface. Furthermore, they can be used to perform a PARAMETER SWEEP, which is a new feature that increments a specific parameter and perform a calculation at every iteration.

- ▶ The Monopole Antenna Example in Chapter 6 is a good introduction to parameters. It creates a geometry that is defined entirely in terms of parameters so that its dimensions can be modified very quickly in one workspace window.
- ▶ The Patch Antenna Example in Chapter 3 is a good introduction to parameter sweeps. It derives the optimal location of the attached feed by running a calculation over several different positions.

## Libraries

 LIBRARIES are essentially databases of project definitions that are saved so that they can be used over and over again in subsequent projects. They are especially useful for users that create many similar projects.



## Chapter 2

### Quick Example: A Bow Tie Antenna

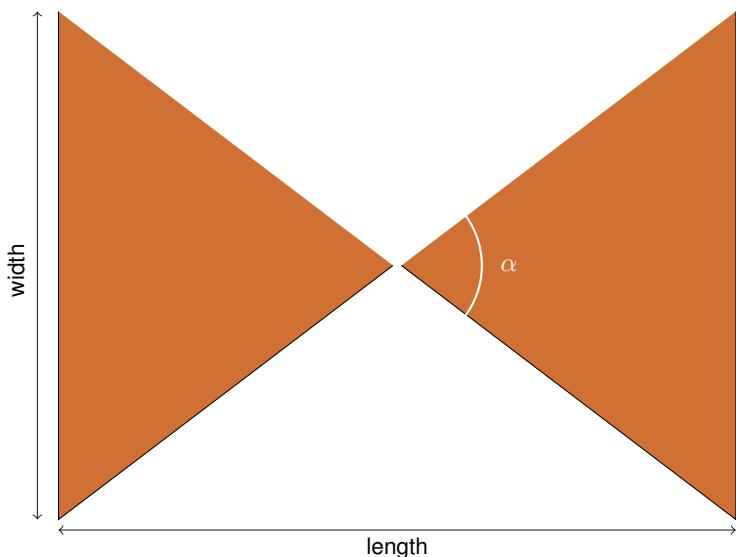


Figure 2.1: Bow tie antenna

Time to create: 15:00 (approx.)

In this chapter, you will learn how to...

- create the shape and set the material properties of the bow tie antenna
- add a feed to the antenna and simulate its effects
- view the plotted results of your simulation

The bow tie antenna is a common type of broad band antenna ([1], pp. 506 – 508). It can be thought of as a 2-dimensional variation of the conical dipole. Its lower frequency is a function of its length and flare angle,  $\alpha$ , and is usually a little lower than a thin wire dipole of the same length. The input impedance is a function of frequency and  $\alpha$ , and the real part is typically between  $70\Omega$  and  $500\Omega$ .

## 2.1 Getting Started

→ Start XFdtd and for Frequency Range of Interest enter Minimum=2 GHz and Maximum=25 GHz.

## 2.2 Creating the Bow Tie Antenna Geometry

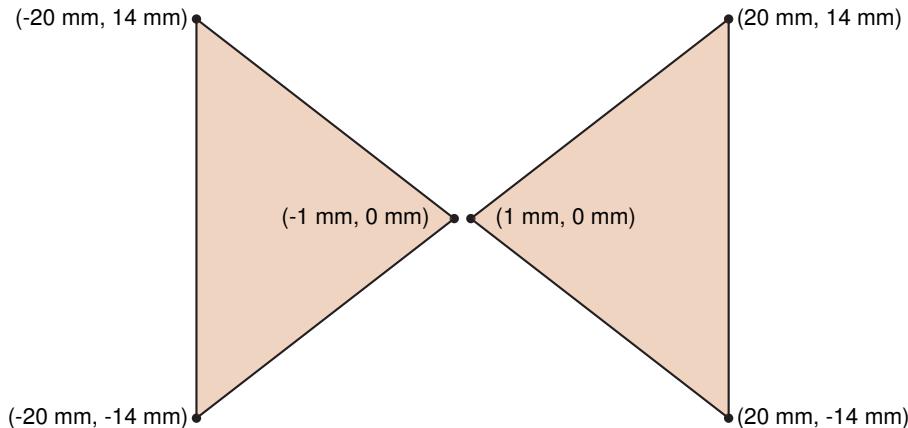
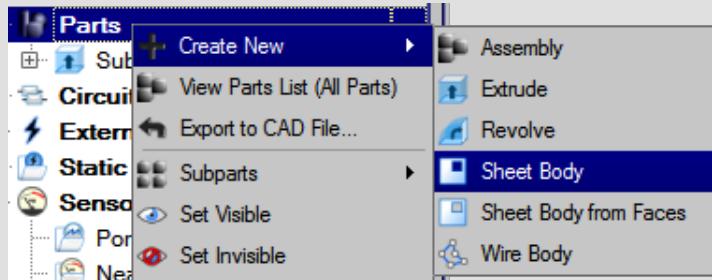


Figure 2.2: Bow tie vertices

We will use a flare angle  $\alpha \approx 70$  degrees, which should give an average input impedance of about  $300\Omega$ , and an overall length of about 40 mm which sets a lower frequency of about 3 GHz. This gives width = 28 mm though the exact values are not critical. The  $(x, y)$  vertices of the two triangles which make up the bow tie are shown in figure 2.2 (all lie in the  $z=0$  plane).

For this example, we will use the GEOMETRY TOOLS interface to create the bow tie using a SHEET BODY.

- Right-click on the **PARTS** branch of the **PROJECT TREE**. Choose **CREATE NEW > SHEET BODY** from the context menu.



- By default, the orientation of the drawing plane will be in the  $xy$  plane at  $z=0$ , which is fine for this case.
- Click on the **EDIT PROFILE** tab.
- Name the part by typing "Bow Tie" in the **NAME** box in the upper-right corner of the window.
- Choose the **POLYGON** tool from the **SHAPES** toolbar as shown in Figure 2.3.
- Zoom out a little in the drawing area so that all the vertices will fit on the screen, as shown in Figure 2.3. The mouse wheel is a convenient way to do this, or click on the view control tool at the top of the right-side pushbuttons and use control-left-drag to zoom in and out.
- In the drawing area click on the three vertices of the left-hand triangle from figure 2.2, then press **Enter** to close the triangle. This should draw the left-hand triangle.
- Repeat for the right-hand triangle.
- Press **DONE** to finish the **Bow Tie** geometry.

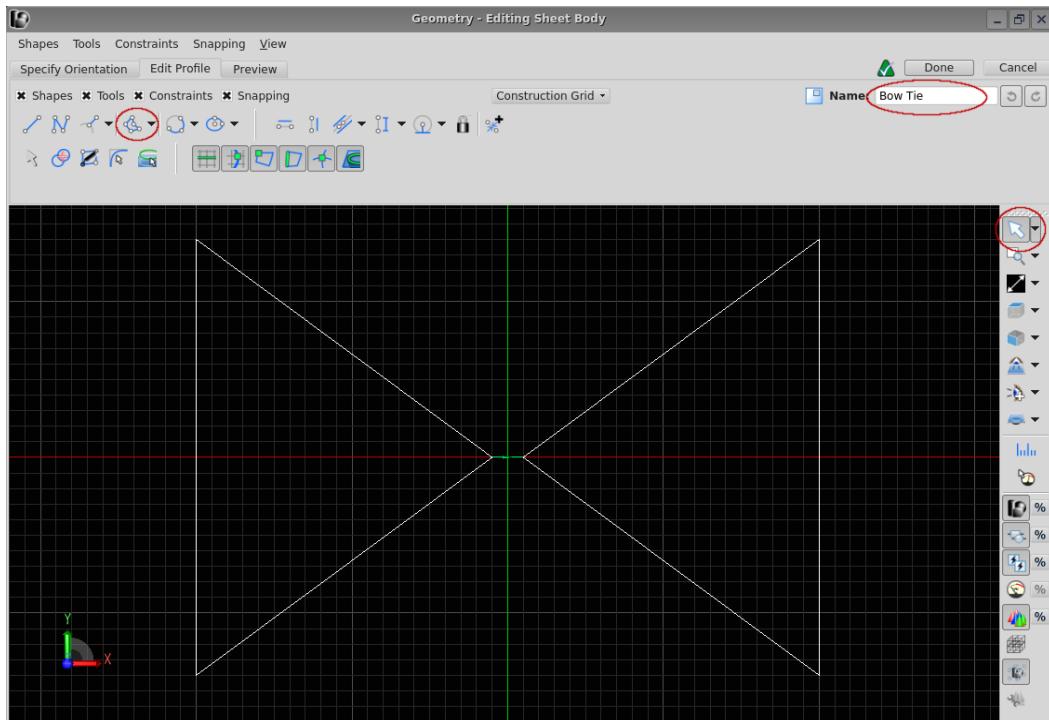


Figure 2.3: Sheet body creation with completed bow tie

## 2.3 Setting the Material

The created geometry needs to have a material associated with it which defines the electromagnetic properties of the antenna material. A good approximation for this case is to assume the bow tie is made of a perfect electric conductor (PEC) material, as the input impedance and radiation pattern will be essentially the same as it would be if a very good conductor such as copper were used.

### Define a PEC material

- Create a perfect electric conductor material. Right-click on the DEFINITIONS: MATERIALS branch of the PROJECT TREE. Choose NEW MATERIAL DEFINITION from the context menu.
- Double-click the new material to edit its properties. Set the perfect electric conductor material properties as shown in Figure 2.4:
  - NAME: PEC
  - ELECTRIC: Perfect Conductor
  - MAGNETIC: Freespace
- If desired, navigate to the APPEARANCE tab to set the PEC material's display color.

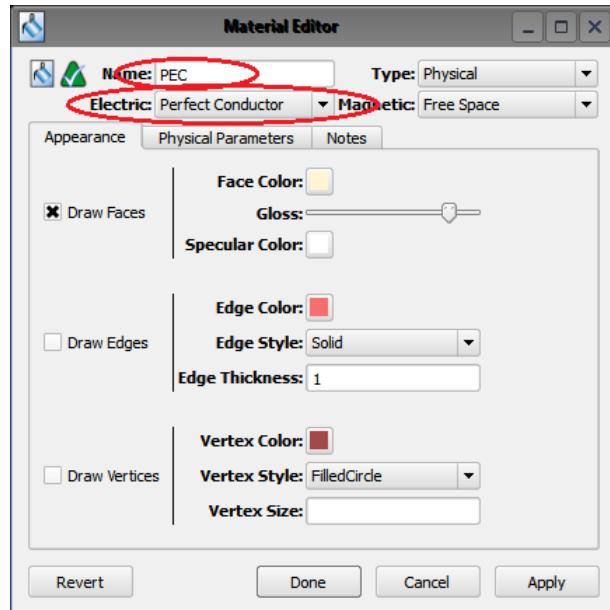


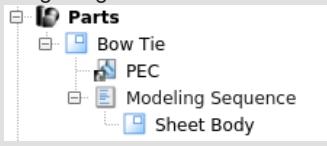
Figure 2.4: Create a New Material of Type PEC

→ Click Done to finish the PEC material.

### Assign the PEC Material to the Bowtie

→ Click-and-drag the PEC material object located in the PROJECT TREE and drop it on top of the BOW TIE object in the PARTS branch of the tree. The color of the bow tie in the geometry should change to match the color defined for the PEC material, indicating that material is associated with the bow tie.

The following image shows the PROJECT TREE after the material has been assigned to the bow tie.



The completed bow tie should look something like Figure 2.5.

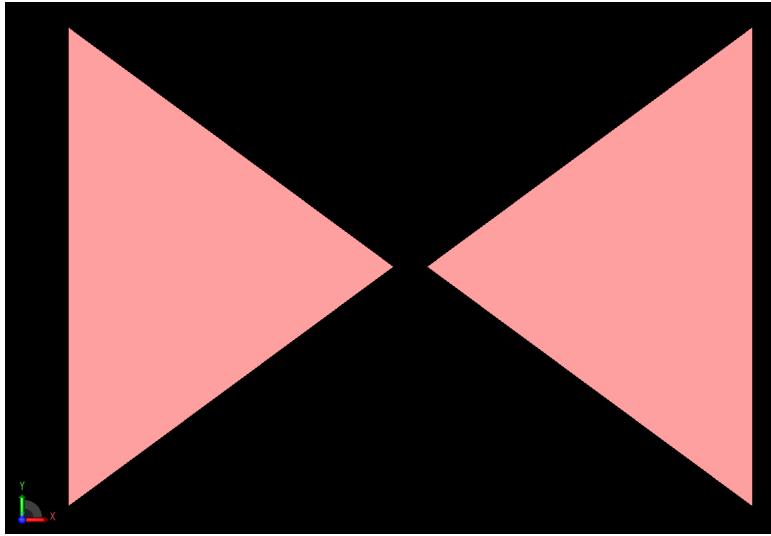


Figure 2.5: The profile of the bow tie antenna geometry

## 2.4 Create the Feed Circuitry

The feed circuitry is added between the two halves of the bow tie. In this case, this is just a voltage source in series with a source resistor, and represents the point where a feed line would be connected to the antenna. This represents a situation where a generator is connected directly to the antenna, and is similar to a generator attached to a perfectly matched feed line, which is connected to the antenna. Unless interaction with the feed line itself is important to the problem this is an efficient and common way to simulate an antenna.

### Create the Feed

- Right-click on the CIRCUIT COMPONENTS branch of the PROJECT TREE and select “New Circuit Component with: New Feed Definition”. Attach the component to the near corners of the two halves of the bowtie, as shown in figure 2.6. If the coordinates of figure 2.2 were used for the antenna, then the  $x$ ,  $y$ , and  $z$  values will be as shown in figure 2.6. The values may simply be typed into their text boxes, or the arrow keys can be used to enable mouse selection of each point.

### Change Feed Resistance to $300\Omega$

By default, the “New Feed Definition” added above will have a  $50\Omega$  resistor in series with an ideal voltage source. Since the bow tie is expected to have an average input impedance closer to  $300\Omega$ , we will change the source resistor to more closely match the input impedance.

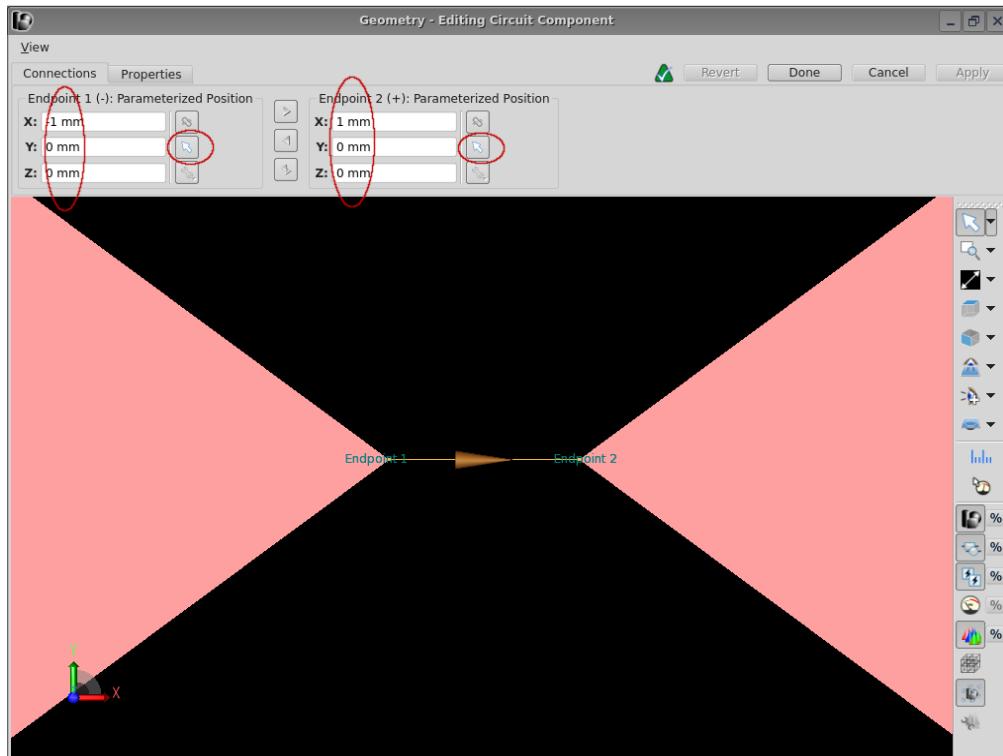


Figure 2.6: Feed Attached to the Bow Tie

→ In the DEFINITIONS branch of the PROJECT TREE open “Circuit Component Definitions” and double-click on the “50 ohm Voltage Source” that should have been added when the feed was created above. This should open a dialog to edit the definition of that component, as shown in Figure 2.7. Change the Resistance to  $300\Omega$ . It is also a good idea, but not required, to change the Name, which was automatically set to reflect the default values, to indicate  $300\Omega$ , as shown.

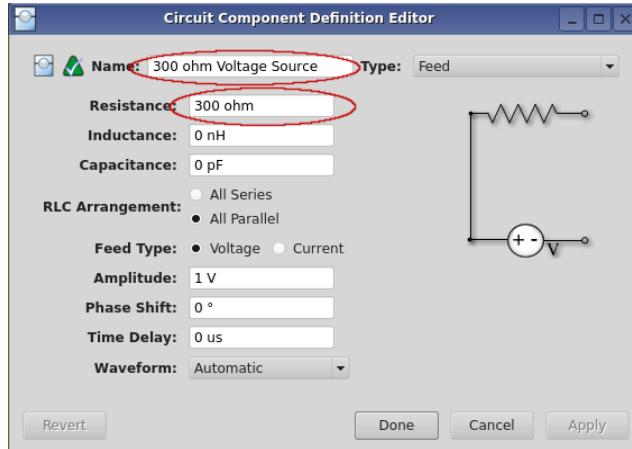


Figure 2.7: Circuit Component Definition for Bow Tie Feed

## 2.5 Far Zone Sensor

Next, a far-zone sensor is added, which just tells the project to save the far-zone radiation pattern at certain frequencies when the simulation is performed. The desired frequencies will be input later, at the time the simulation is launched.

- To add the sensor, right-click on Far Zone Sensors in the Sensors branch of the PROJECT TREE, and select “New Far Zone Sensor”. Accept the default values for the sensor, which will save far-zone fields every 5 degrees in  $\theta$  and  $\phi$ . In particular, make sure the “Collect Broadband Data” check box in the “Properties” tab remains off (unchecked).

This completes the creation of the project.

## 2.6 Run the Simulation

- If you have not already saved your project, do so by selecting FILE > SAVE PROJECT. Once the project is saved, a new simulation can be created to send to the calculation engine.
- To start a simulation, click the SIMULATIONS button on the upper-right side of XFdtd, then click the FDTD in the upper-left of the Simulations dialog to set up a new simulation as shown in Figure 2.8.
- In the Simulation Dialog click on Frequencies of Interest to add frequencies at which the far-zone pattern will be saved. Four frequencies have been added in the figure, but any number may be added. They should all be within the frequency range of interest entered at the start of this project.
- After all frequencies of interest are added, select CREATE AND QUEUE SIMULATION to close the dialog and run the simulation. In the SIMULATIONS dialog, click on the Output tab to see the progress of the simulation.

The simulation should take a few seconds to a couple of minutes to complete, depending on computer hardware.

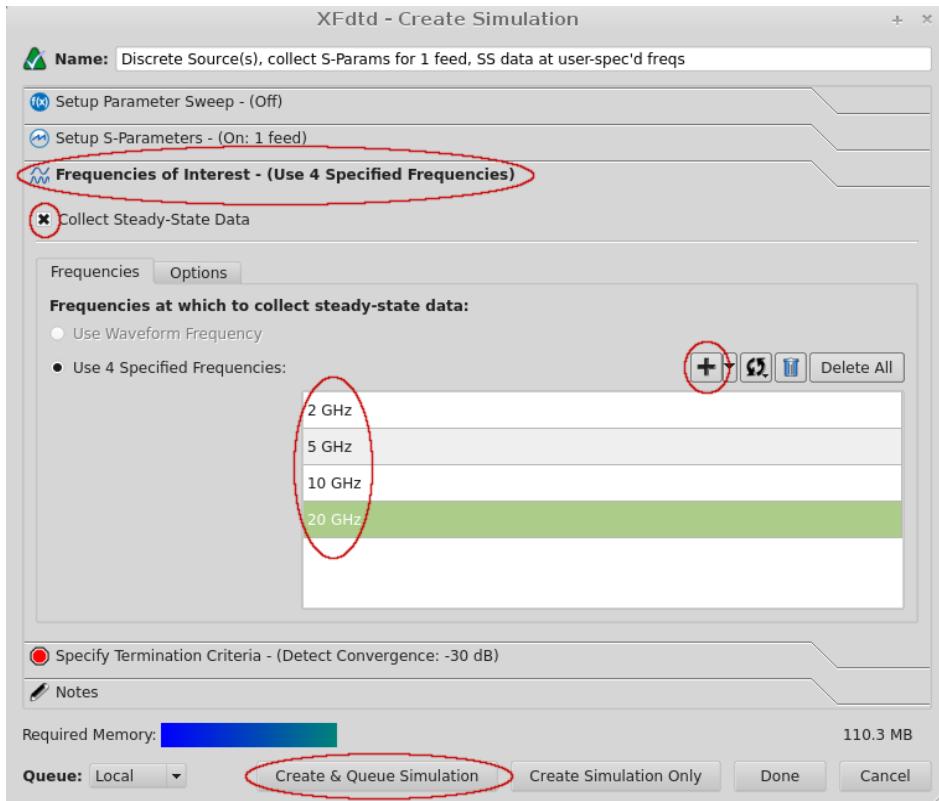


Figure 2.8: Simulation Dialog

## 2.7 Viewing Results

Once the simulation has completed, results may be viewed by using the RESULTS browser, as shown in Figure 2.9. The appearance of this dialog may vary depending on how the configurable columns along the top are set up. The lower half of the dialog contains a list of all the results that meet the criterion of the selections in the four columns of the upper half.

→ Click on the RESULTS button on the right-hand side of XFdtd to bring up the Results Browser.

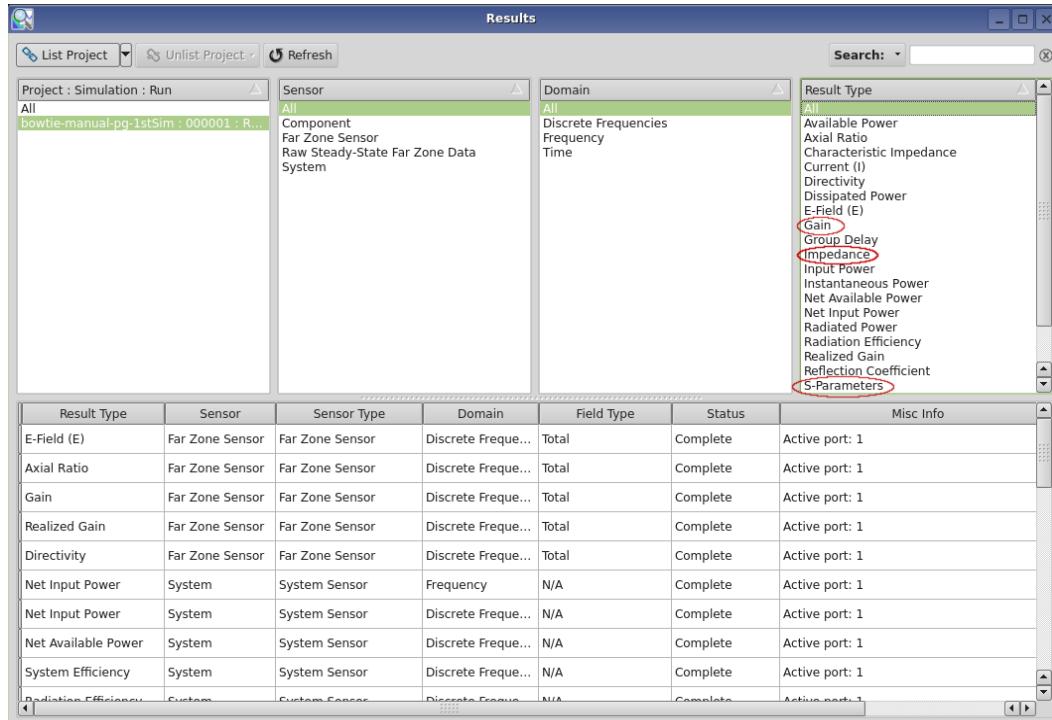


Figure 2.9: Results Browser

## S Parameters

Result Type	Sensor	Sensor Type	Domain	Field Type	Status	Misc Info
S-Parameters - S[1,1]	Component	Circuit Component	Frequency	N/A	Complete	S[1,1]
S-Parameters - S[1,1]	Component	Circuit Component	Discrete Frequency	N/A	Complete	S[1,1]

Figure 2.10: Available Results with "S Parameters" Selected

- Click on “S-Parameters” in the right-hand column to narrow down the possible results to just S Parameter results, which should cause the lower half to look something like that in Figure 2.10.
- Double-clicking on a line containing a set of results will cause XFdtd to perform what it considers to be the action most likely desired by the user for the given result type. Additional choices are usually available by right-clicking on the result instead. In this case the default action is what we want, so double-click on the line containing Frequency-domain data S Parameter, as indicated in the figure. This should bring up a plot of S Parameters over the frequency range of interest, as shown in Figure 2.11.

The return loss is below about -6 dB from about 3.5 GHz through 25 GHz, indicating a pretty good broadband match.

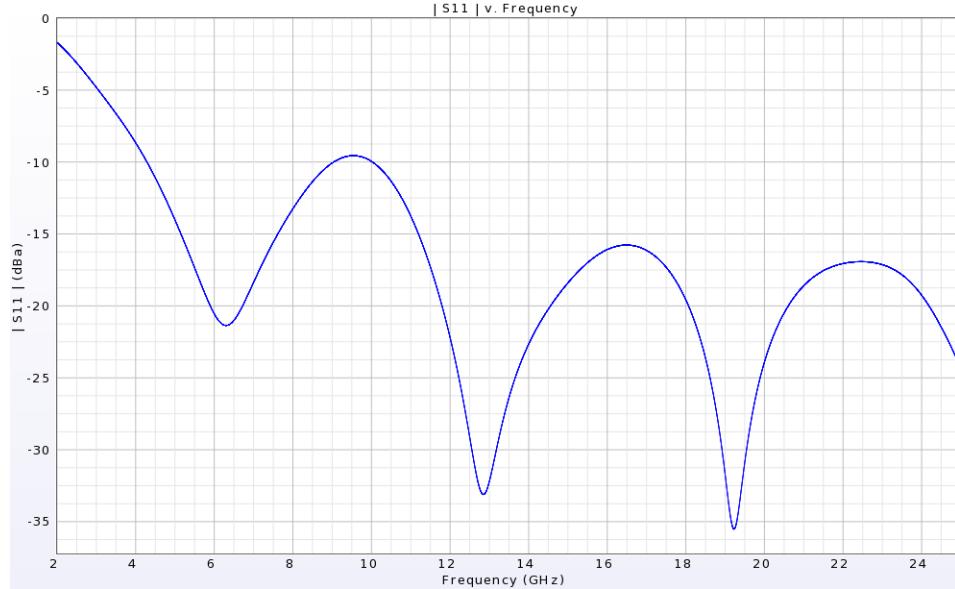


Figure 2.11: S Parameters for the Bow Tie with a  $300\Omega$  Feed

## Far Zone Gain

Click on “Gain” in the right-hand column of the RESULTS browser to narrow the choices to gain in the lower half of the dialog. Double-click on this Gain line to load the 3D gain into the geometry view, as shown in Figure 2.12. This causes the gain pattern to be plotted in 3D in the Geometry View. The opacity of the gain pattern has been turned down via the “%” control next to the Field Output Visibility button highlighted on the right side in the figure. A gain pattern for each frequency entered when running the simulation should be available from the “Frequency” drop-box in the lower left. Additional data is available in the “Statistics” tab. Note that the “XPD”, “PDF”, and “Diversity” tabs apply to antenna diversity measurements, which do not apply here as they are intended for projects with multiple antennas at overlapping frequency ranges.

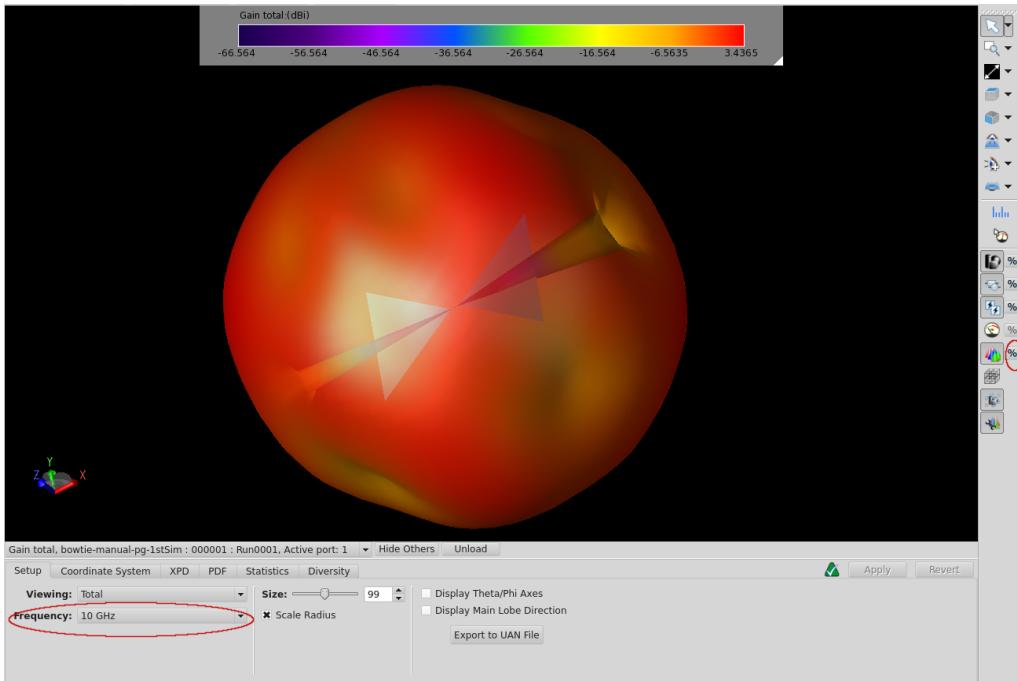


Figure 2.12: Gain of the Bow Tie Antenna

## Input Impedance

Click on “Impedance” in the right-hand column of the RESULTS browser to display only the input impedance data in the lower half of the dialog, as shown in Figure 2.13. We want to plot real and

Result Type	Sensor	Sensor Type	Domain	Field Type	Status	Misc Info
Impedance	Component	Circuit Component	Frequency	N/A	Complete	Active port: 1
Impedance	Component	Circuit Component	Discrete Freque...	N/A	Complete	Active port: 1

Figure 2.13: Available Results with “Impedance” Selected

imaginary input impedance vs frequency for the bow tie antenna. The frequency-domain data line is the one needed, but in this case, double-clicking on that line will plot the magnitude of the impedance vs frequency, which is not what we want. To see more options right-click on the frequency-domain impedance line and choose “Create Line Graph...”. This will bring up a dialog similar to that in Figure 2.14a. Change the Complex Part to “Real”, as shown and select View to view the line graph of the real part of the impedance.

Now add a plot of the imaginary part of the impedance by right-clicking on the frequency-domain impedance data line and selecting “Create Line Graph...” to bring up the Create Line Graph dialog again. This time choose the Imaginary part and for Target Graph, select the graph created when plotting the real part, which will probably be named as shown in Figure 2.14b if that was the first impedance plot created. This should produce a graph of real and imaginary impedance vs frequency as shown in Figure 2.15

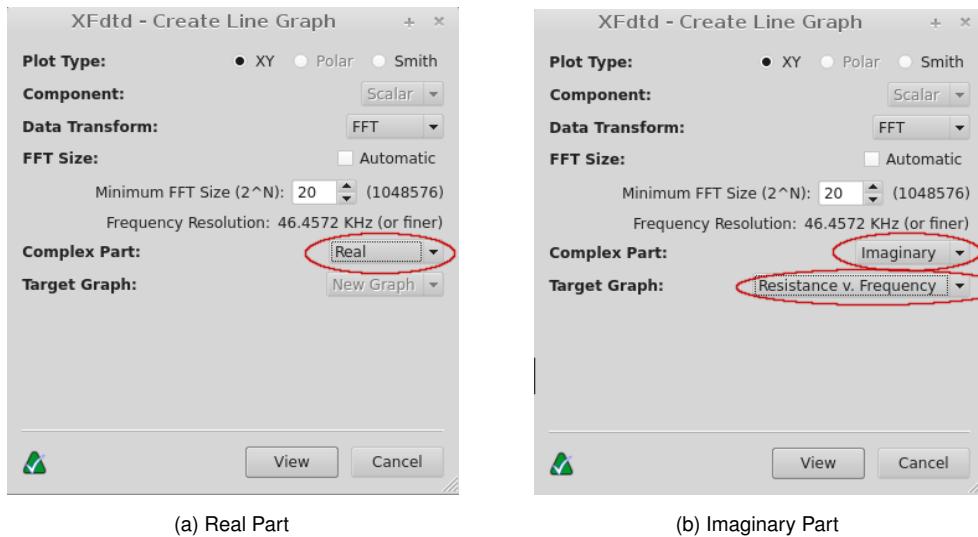


Figure 2.14: Creating a Line Graph of Real and Imaginary Frequency-Domain Impedance Data

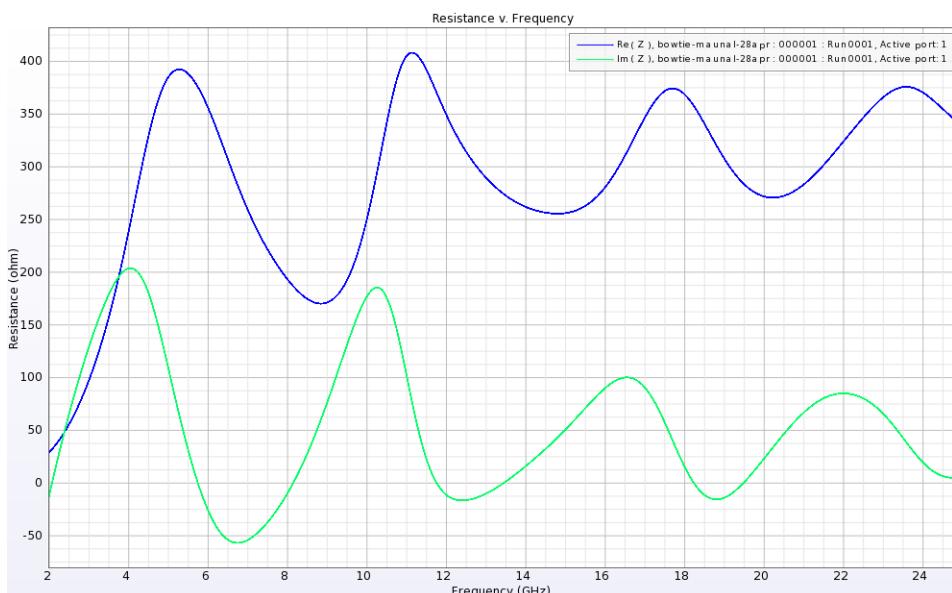


Figure 2.15: Complex Impedance vs Frequency for the Bowtie



## Chapter 3

### Example: A Microstrip Patch Antenna

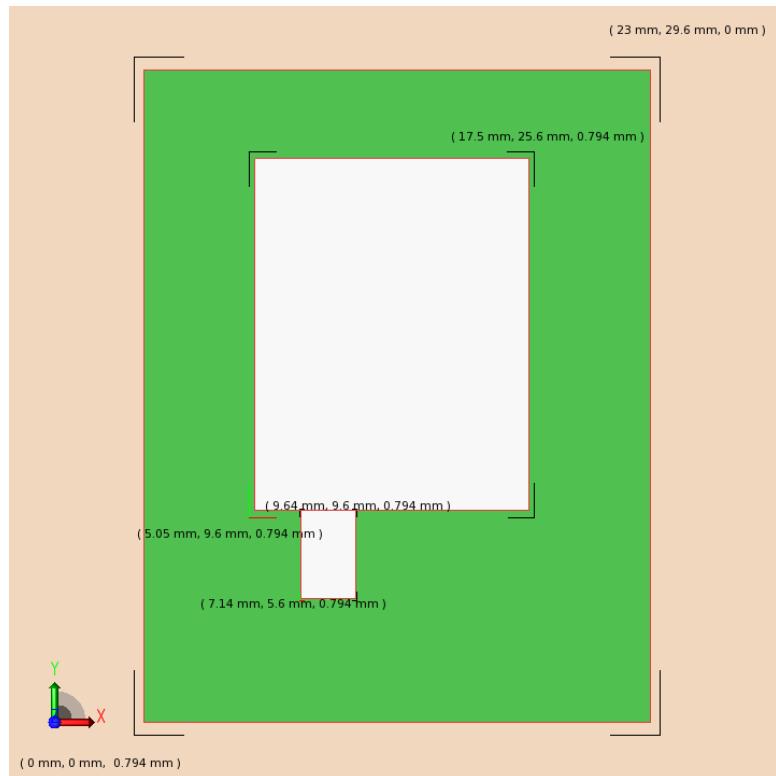


Figure 3.1: The microstrip patch antenna

Time to create: 50:00 (approx.)

*In this chapter, you will learn how to...*

- set project preferences and display units to initialize any XFdtd project
- create the shape and set the material properties of your patch antenna
- add a feed to the antenna and simulate its effects
- view the plotted results of your simulation
- run an additional simulation with a parameter sweep

This microstrip patch antenna example is based on a paper [2] by Sheen et al. The patch antenna from Figure 3 of the paper will be constructed and the S-parameters compared with the measured and computed return loss of Figure 5 of the paper. The antenna geometry is built as shown in Figure 3.1. The substrate thickness is 0.794 mm with a relative permittivity  $\epsilon_r = 2.2$ .

## 3.1 Getting Started

First, a few Project Properties are set up for the Patch Antenna project.

- The PROJECT PROPERTIES EDITOR opens automatically whenever XFdtd is started or when a new project is created. If the editor is not currently visible, double-click on PROJECT at the top of the PROJECT TREE to open the editor.
- On the FREQUENCY RANGE OF INTEREST tab of the PROJECT PROPERTIES EDITOR, specify a frequency range of 0 - 20 GHz by setting MINIMUM to "0 GHz" and MAXIMUM to "20 GHz".
- Navigate to the DISPLAY UNITS tab, where a few relevant units should be set:
  - Set FREQUENCY to "gigahertz (GHz)"
  - Set LENGTH to "millimeters (mm)"
- Press DONE on the PROJECT PROPERTIES EDITOR.

## 3.2 Creating the Patch Antenna Geometry

Now we will create the patch antenna geometry out of two simple components: a rectangular substrate and a microstrip patch. For this example, we will use the GEOMETRY TOOLS interface to create a rectangular EXTRUSION and a SHEET BODY comprised of two rectangles.

### 3.2.1 Modeling the Substrate

First, we will create the rectangular substrate named SUBSTRATE. This object will stretch from (0, 0, 0) to (23, 29.6, 0) and have a .794 mm extrusion in the +Z direction.

- Right-click on the PARTS branch of the PROJECT TREE. Choose CREATE NEW > EXTRUDE from the context menu.

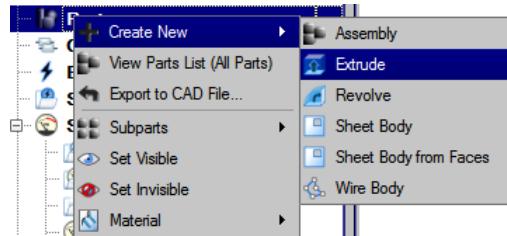


Figure 3.2: Selecting the geometry tool to perform an extrusion

- Name the part by typing “Substrate” in the NAME box in the upper-right corner of the window.
- Choose the RECTANGLE tool from the SHAPES toolbar.

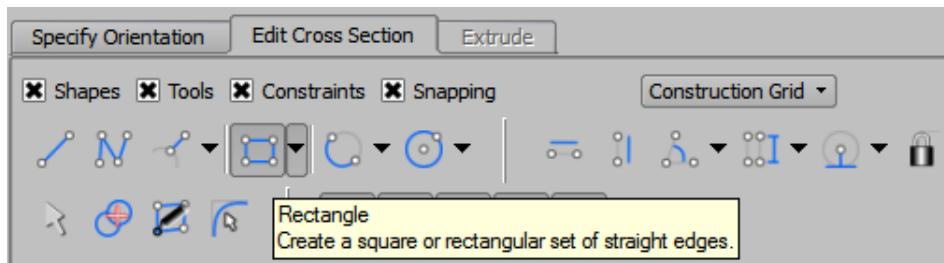


Figure 3.3: Selecting the rectangle tool

- The creation dialog allows exact entry of coordinates. Press **TAB** to display the creation dialog for the first point. Enter (0 mm, 0 mm) and press OK.
- !** If the creation dialog does not appear, right-click in the geometry space to “activate” the window.

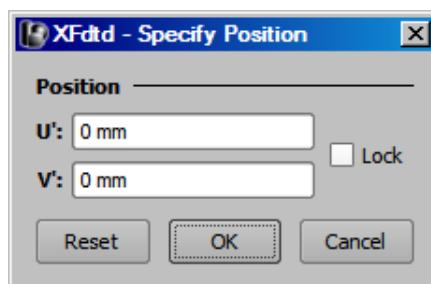


Figure 3.4: Specifying the coordinates of the rectangle corner

- Press **TAB** to display the creation dialog for the second point. Enter (23 mm, 29.6 mm) and press OK to complete the rectangle.

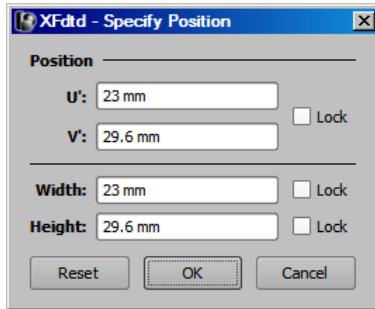


Figure 3.5: Specifying the coordinates of the rectangle corner

- Navigate to the EXTRUDE tab to extrude the rectangular region. Enter a distance of 0.794 mm.

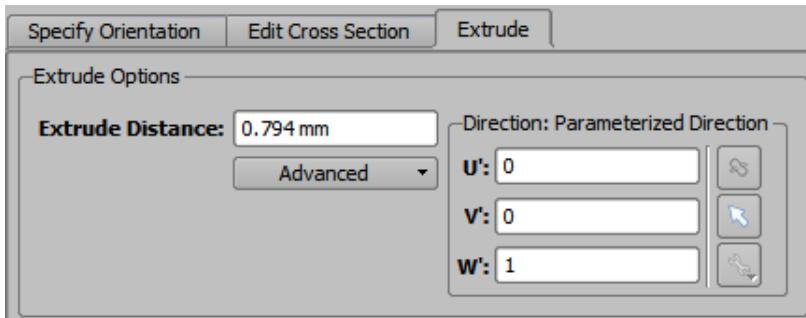


Figure 3.6: Specifying the extrude distance

- Press DONE to finish the SUBSTRATE geometry.

### 3.2.2 Modeling the Microstrip Patch

The microstrip patch will be created with a SHEET BODY object that rests on top of the SUBSTRATE. This shape will be comprised of two rectangles. The patch will stretch from (5.05, 9.6, 0.794) to (17.5, 25.6, 0.794). The stub will stretch from (7.2, 5.6, 0.794) to (9.6, 9.6, 0.794).

- Right-click on the PARTS branch of the PROJECT TREE. Choose +CREATE NEW > SHEET BODY from the context menu.

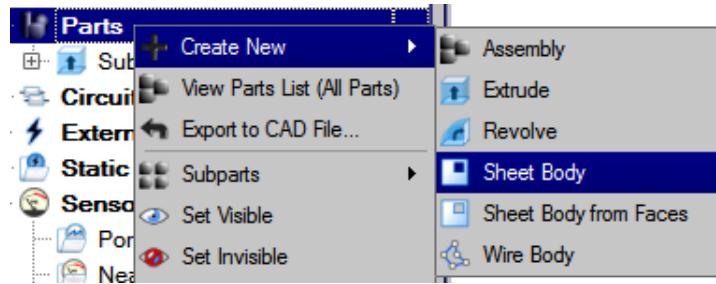


Figure 3.7: Selecting the geometry tool to create a sheet body

- Navigate to the SPECIFY ORIENTATION tab. Click the PICKER TOOL.
- Select ORIGIN from the drop-down list.
- Mouse over the SUBSTRATE and click on the Lower X, Lower Y corner (Figure 3.8) to set the orientation for the correct placement of the SHEET BODY. If placed correctly, the data in the ORIGIN section of the SPECIFY ORIENTATION tab should be as follows: X: 0 mm, Y: 0mm, Z: 0.794 mm.

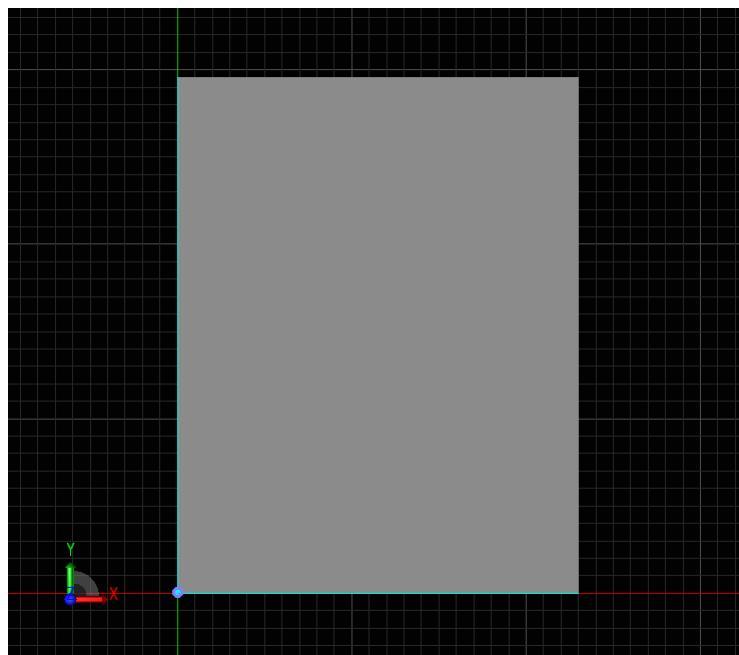


Figure 3.8: Picking the Origin

- Navigate to the EDIT PROFILE tab. Type “Microstrip” into the NAME box.

We will draw the microstrip and its stub individually and then combine them into a single polygon.

- Select the RECTANGLE tool. Use the creation dialog to enter the corners of the microstrip rectangle:

- Endpoint 1: (5.05 mm, 9.6 mm)
  - Endpoint 2: (17.5 mm, 25.6 mm)
  - Now use the creation dialog to enter the corners of the stub rectangle:
    - Endpoint 1: (7.2 mm, 5.6 mm)
    - Endpoint 2: (9.6 mm, 9.6 mm)
- Select the TRIM EDGES tool.

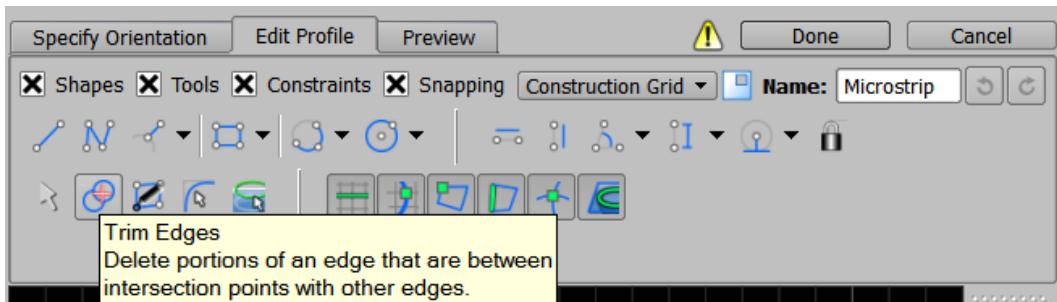


Figure 3.9: Selecting the Trim Edges tool

- Remove the line segments between the microstrip and the stub by clicking twice on the overlapping edges.
- Click DONE to finish the MICROSTRIP geometry.

### Meshing Priority

Ensure that the meshing priority of the MICROSTRIP is greater than the SUBSTRATE for an accurate calculation.

- Right-click on the Microstrip in the PROJECT TREE. Under GRIDDING / MESHING > MESHING ORDER, select MOVE TO TOP if it is an available option.

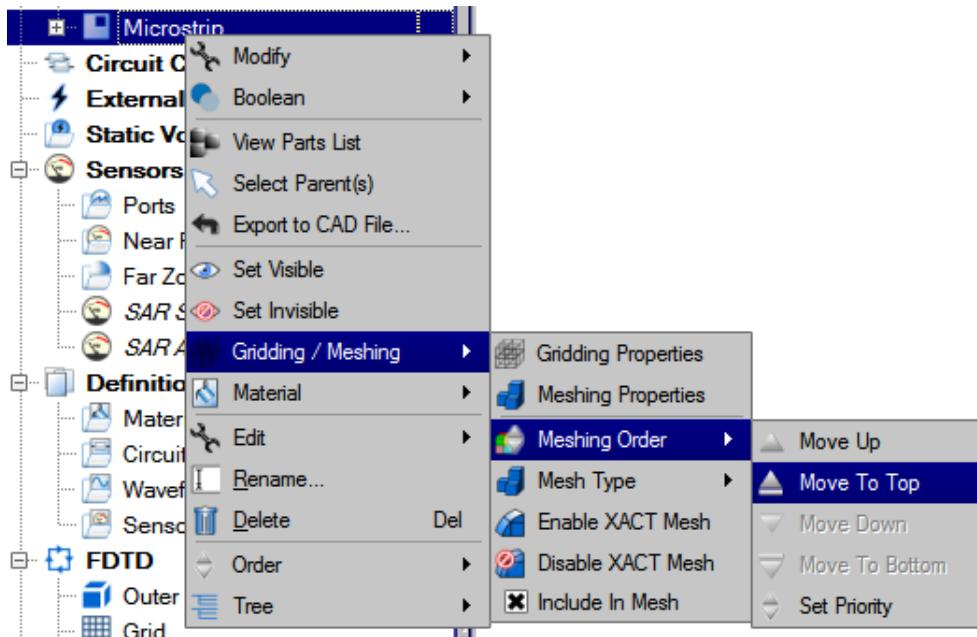


Figure 3.10: Manually setting the meshing order of the Microstrip

### 3.3 Creating Materials

#### Define material, PEC

- First, create a perfect electric conductor material. Right-click on the DEFINITIONS: MATERIALS branch of the PROJECT TREE. Choose NEW MATERIAL DEFINITION from the context menu.

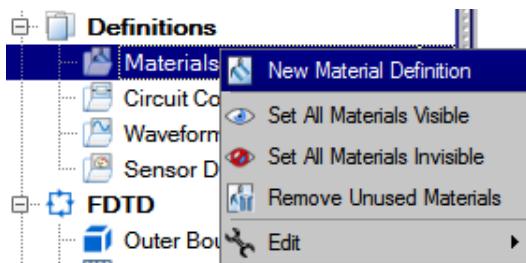


Figure 3.11: Creating a new material

- Double-click the new material to edit its properties. Set the perfect electric conductor material properties as follows:

- NAME: PEC
- ELECTRIC: Perfect Conductor
- MAGNETIC: Freespace

- If desired, navigate to the APPEARANCE tab to set the PEC material's display color.

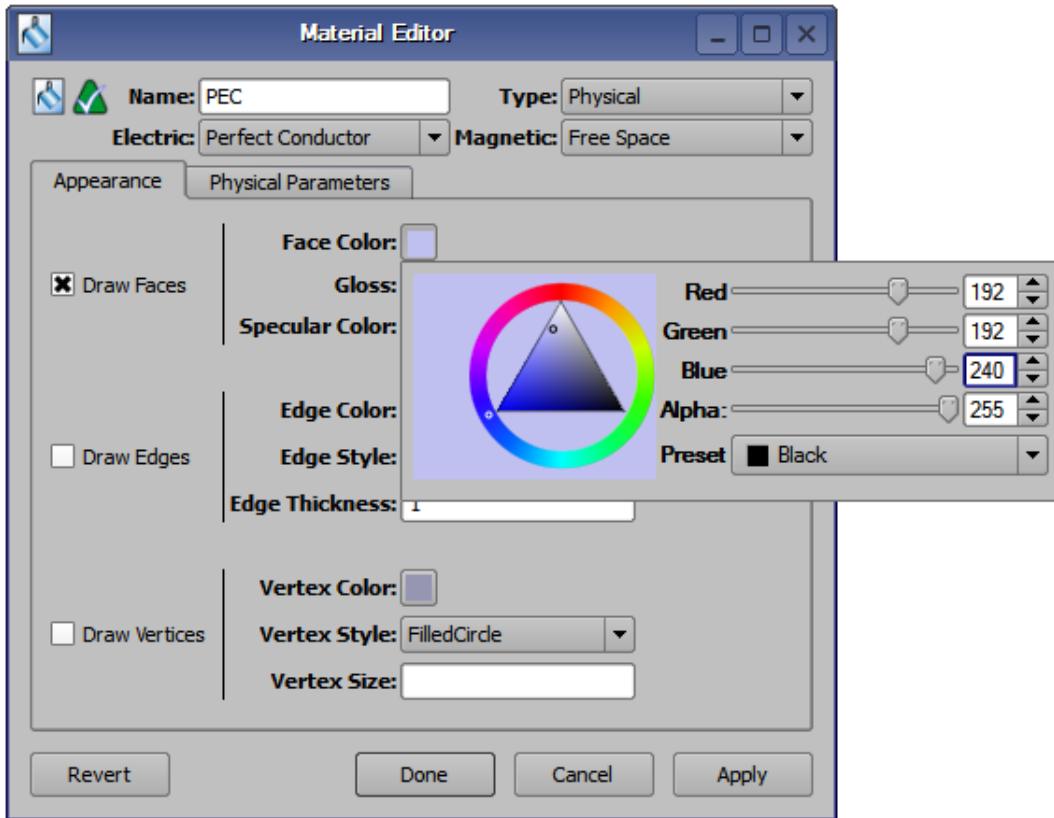


Figure 3.12: Editing the color of the PEC material

## Define material, DUROID

- Right-click on the DEFINITIONS: MATERIALS branch of the PROJECT TREE. Choose NEW MATERIAL DEFINITION from the context menu.
- Double-click the new material to edit its properties. Set the duroid material properties as follows:
- NAME: Duroid
  - ELECTRIC: Isotropic
  - MAGNETIC: Freespace
- Under the ELECTRIC tab:
- TYPE: Nondispersive
  - ENTRY METHOD: Normal
  - CONDUCTIVITY: 0 S/m

- RELATIVE PERMITTIVITY: 2.2

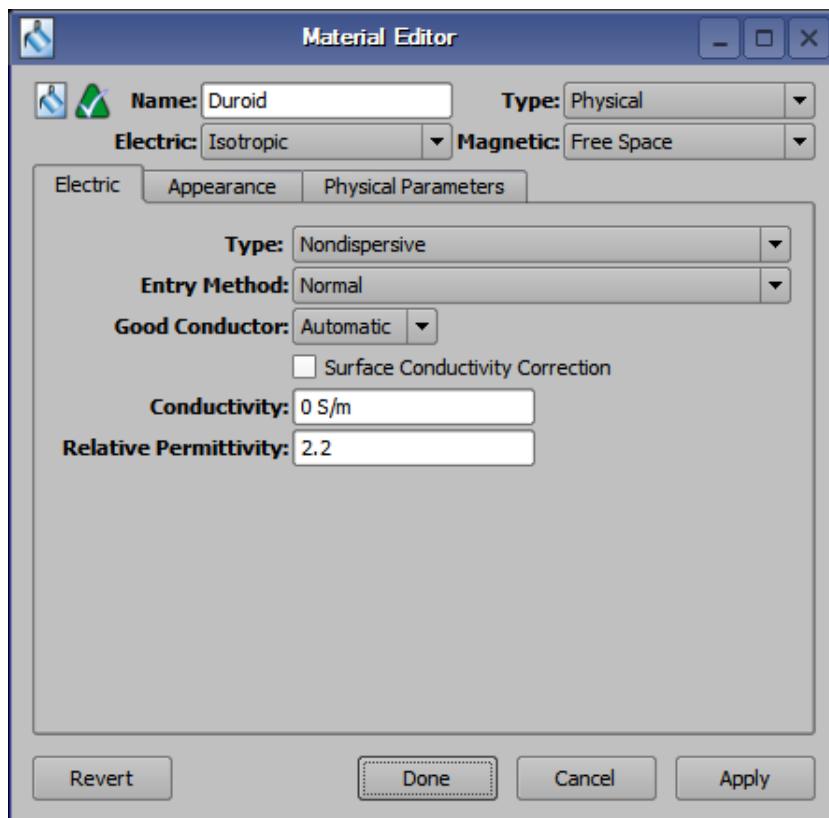


Figure 3.13: Editing the properties of the Duroid material

→ If desired, navigate to the APPEARANCE tab to set the DUROID material's display color.

### 3.4 Assigning Materials

- Click-and-drag the PEC material object located in the PROJECT TREE and drop it on top of the MICROSTRIP objects in the PARTS branch of the tree.
- Assign the DUROID material to the SUBSTRATE object using the same procedure.

The following image shows the PROJECT TREE after material objects have been dropped on their respective parts.

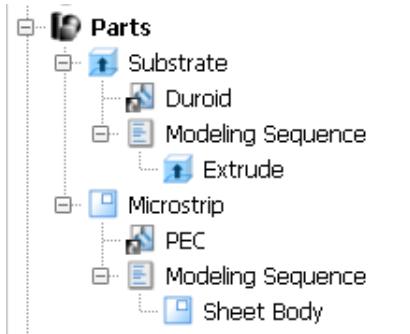


Figure 3.14: Assigning materials to the patch antenna geometry

This image shows the microstrip patch antenna geometry with materials applied and colors set for each.

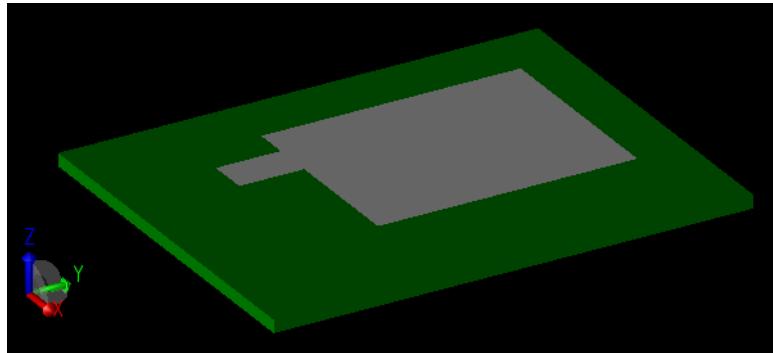


Figure 3.15: The profile of the patch antenna geometry

### 3.5 Defining the Outer Boundary

- Double-click on the **FDTD:OUTER BOUNDARY** branch of the **PROJECT TREE** to open the **OUTER BOUNDARY EDITOR**.
- Set the outer boundary properties as follows:
  - **BOUNDARY**: “Absorbing” for all boundaries except **LOWER BOUNDARY Z**, which should be “PEC”
  - **ABSORPTION TYPE**: PML
  - **LAYERS**: 7

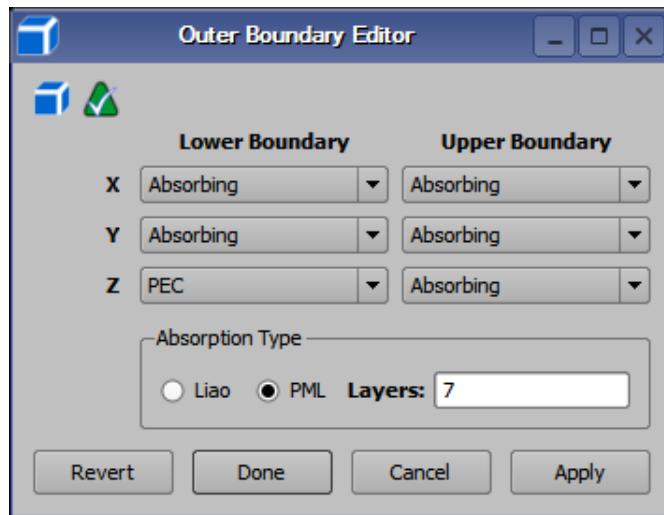


Figure 3.16: Defining the outer boundary for the patch antenna

- Click DONE to apply the outer boundary settings.

## 3.6 Defining the Grid

Now we will define characteristics of the calculation grid.

- Double-click on the FDTD:GRID branch of the PROJECT TREE to open the GRID EDITOR.
- The default settings on the CELL SIZE tab are sufficient to obtain an accurate result.
- Navigate to the EXTENTS tab. We will apply free space padding to all sides of the simulation space except the Lower Z boundary (where a PEC boundary condition was previously assigned).
  - Free space padding on the five absorbing boundaries is present with the default settings (both SPECIFY PADDING and PROGRID PADDING ON ABSORBING BOUNDARIES enabled).
  - Remove all padding from the lower Z boundary by setting LOWER Z FREE SPACE PADDING (BASE CELLS) to "0" (Figure 3.17). This will cause the PEC boundary condition to lie exactly on the bottom surface of the Substrate as a ground plane.
- Click DONE to apply the grid settings.

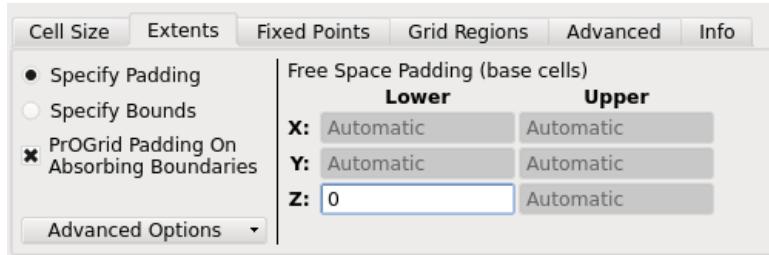


Figure 3.17: Defining the grid extents on the Grid Editor

### 3.7 Adding a Feed

We will now add a FEED to the patch antenna geometry. It will consist of a voltage source and series  $50\Omega$  resistor connected between the base of the stub portion of the MICROSTRIP and the ground plane.

- Right-click on the CIRCUIT COMPONENTS branch in the PROJECT TREE. Choose NEW CIRCUIT COMPONENT WITH > NEW FEED DEFINITION from the context menu.

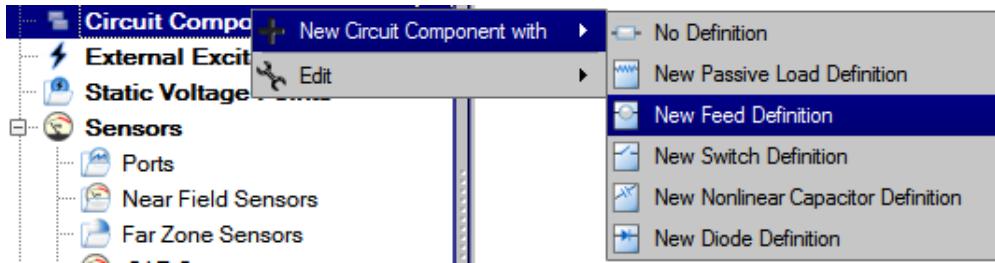


Figure 3.18: Adding a feed to the project

- Define the endpoints of the feed.

1. Click the PICKER TOOL for ENDPOINT 1. The cursor will change to an arrow with a thumbtack to indicate that the endpoint will be attached to the selected geometry (see Note on being "attached" below).
  2. Click on the center of the bottom edge of the stub (See Figure 3.34). The X, Y, Z fields for Endpoint 1 populate.
  3. Click the COPY ENDPOINT 1 TO ENDPOINT 2 button. The X, Y, Z fields for Endpoint 2 populate.
  4. Open the ADVANCED OPTIONS for ENDPOINT 2.
  5. Enter -0.794 mm in the Z field.
  6. Press the ENTER key. The Z field in the ENDPOINT 2 section of the CONNECTIONS tab displays 0 mm.
- Attached endpoints detach when their locations are changed manually. Therefore, typing data in the Z field is not recommended because it will detach ENDPOINT 2 from the part. If the

orientation of the part changes the detached feed will not move with it. To avoid this issue, it is best to use the ADVANCED OPTIONS to offset the position.

- Navigate to the PROPERTIES tab, and name the component “Feed”.
- Click DONE to add the FEED.

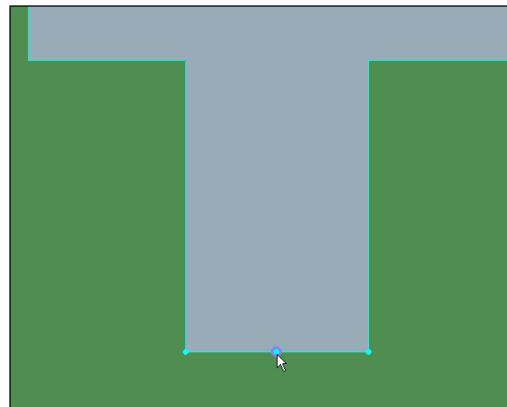


Figure 3.19: Adding a feed with the Picker tool

## 3.8 Requesting Output Data

We will add a PLANAR SENSOR at the surface of the MICROSTRIP plate to retrieve electric field sampling data.

- Right-click on the SENSORS: NEAR FIELD SENSORS branch of the PROJECT TREE. Select NEW PLANAR SENSOR from the context menu.
  1. Click the PICK SIMPLE PLANE button.
  2. Mouse over the top of the substrate in the left corner closest to the Stub (Figure 3.20). The directional arrow appears.
  3. Click the Space bar until the arrow is pointing straight up
  4. Click on the top of the substrate in the left corner closest to the Stub to place the PLANAR SENSOR (Figure 3.20).

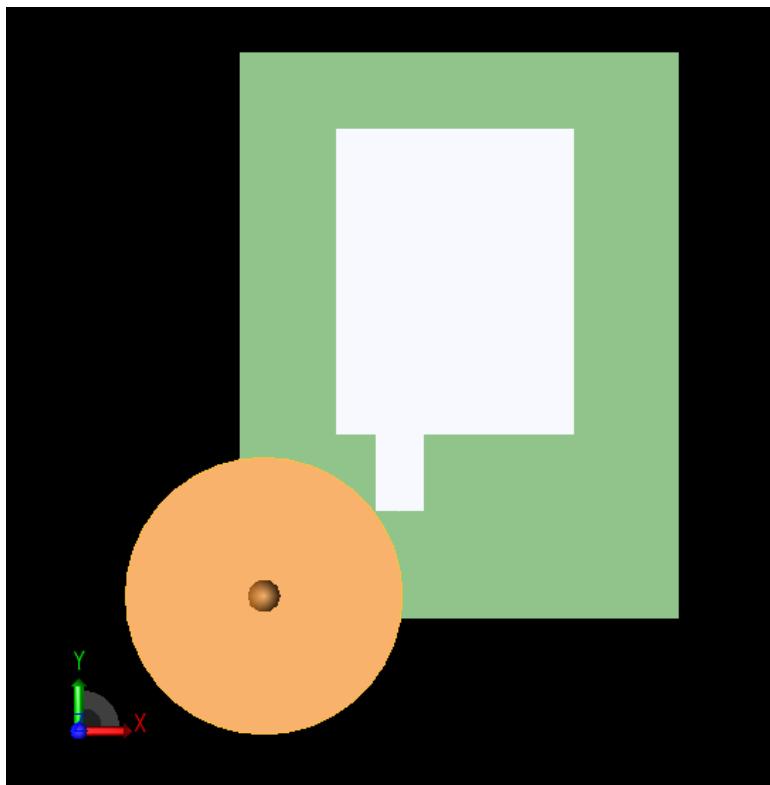


Figure 3.20: Adding a planar sensor to the project

→ Press DONE to add the planar sensor.

This sensor requires a data definition.

→ Open the DEFINITIONS: SENSOR DATA DEFINITIONS branch of the PROJECT TREE. Double-click on SURFACE SENSOR DEFINITION to edit its properties.

→ Set the properties of the surface sensor definition as follows:

- NAME: Field Sampling
- FIELD VS. TIME: E, H, B, and J
- START TIME: 20 \* timestep
- END TIME: 1000 \* timestep
- SAMPLING INTERVAL: 10 \* timestep

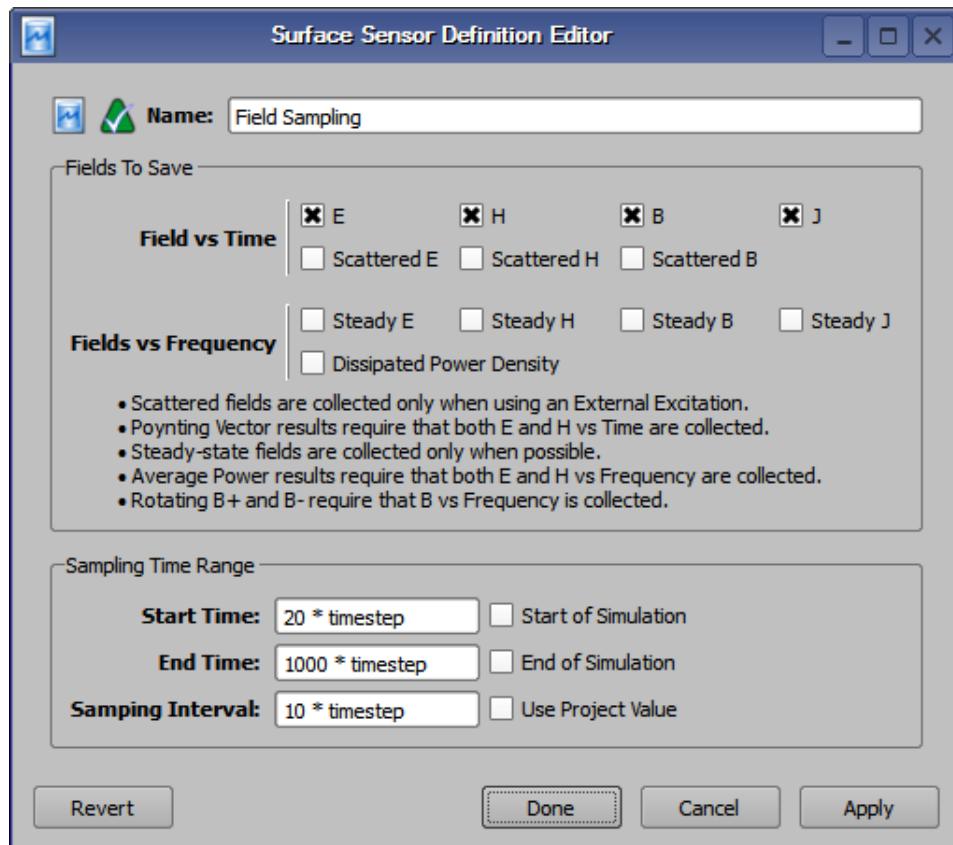


Figure 3.21: Adding the sensor definition

→ Press DONE to finish editing the surface sensor definition.

## 3.9 Running a Simulation

If you have not already saved your project, do so by selecting FILE > SAVE PROJECT. Once the project is saved, a new simulation can be created to send to the calculation engine.

→ Open the SIMULATIONS workspace window. Click the FDTD button in the upper-left to set up a new simulation.

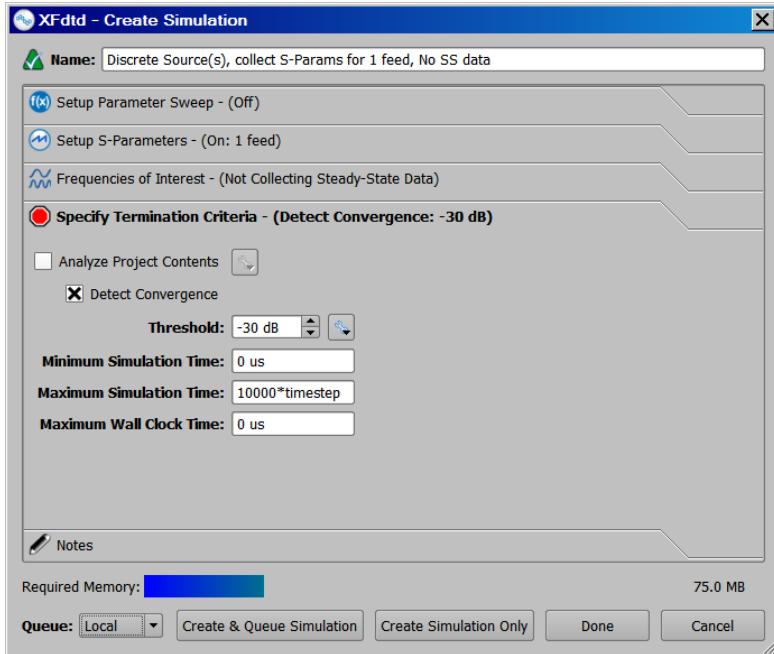


Figure 3.22: Adding a new simulation to the patch antenna project

- Under **FREQUENCIES OF INTEREST**, **COLLECT STEADY-STATE DATA** should be unchecked.
- Under **SPECIFY TERMINATION CRITERIA**, ensure that the fields are defined as follows:
  - ANALYZE PROJECT CONTENTS is unchecked
  - DETECT CONVERGENCE is checked
  - THRESHOLD: -30 dB
  - MINIMUM SIMULATION TIME: 0 s
  - MAXIMUM SIMULATION TIME: 10000 \* timestep
- Select **CREATE AND QUEUE SIMULATION** to close the dialog and run the new simulation.

## 3.10 Viewing the Results

The **OUTPUT** tab of the **SIMULATIONS** workspace window displays the progress of the simulation. Once the **STATUS** column shows that the simulation has completed, we can view its results in the **RESULTS** workspace window.

## S-Parameter Results from the Port Sensor

First, we will view the S-parameter results retrieved with the port sensor placed at the location of the  FEED.

- In the top pane of the  RESULTS window, right-click on each column heading to select the correct category. Then highlight the following options within the columns:
  - SENSOR: Feed
  - SENSOR TYPE: Circuit Component
  - RESULT TYPE: S-Parameters

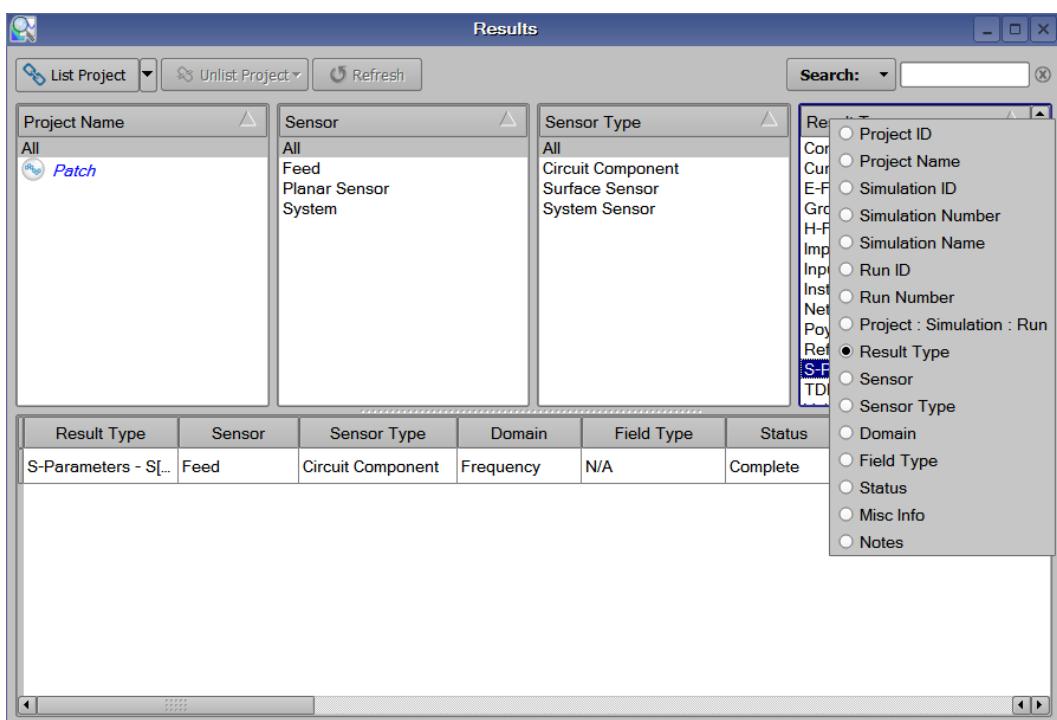


Figure 3.23: Viewing S-Parameters in the Results window

- Double-click on the result with a DOMAIN value of “Frequency” to view transient S-parameter results. The following plot will appear:

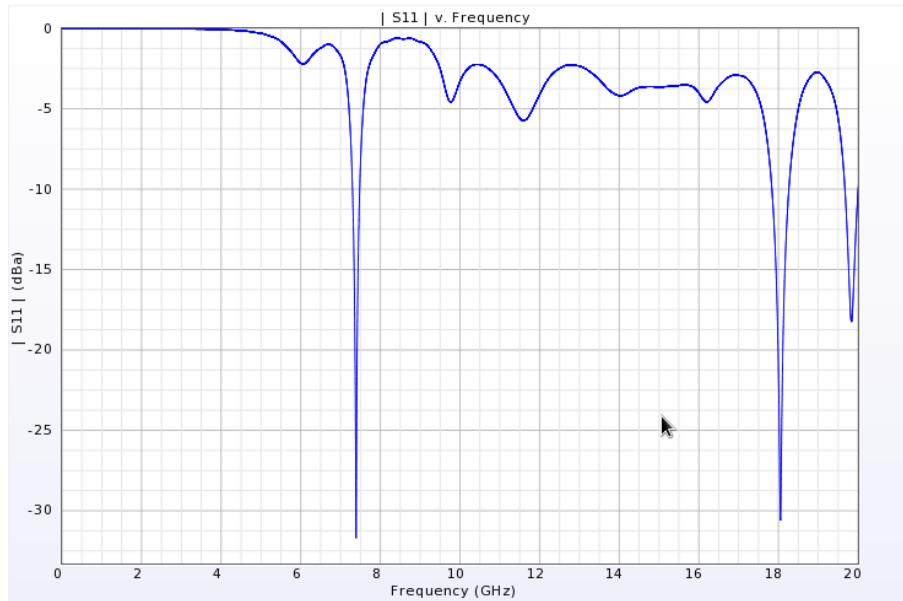


Figure 3.24: Viewing S-Parameters v. Frequency plot

## E-Field Results from the Planar Sensor

Now we can view the results retrieved from the PLANAR SENSOR.

→ To filter the E-field results, select the following options:

- SENSOR: Planar Sensor
- SENSOR TYPE: Surface Sensor
- RESULT TYPE: E-Field (E)

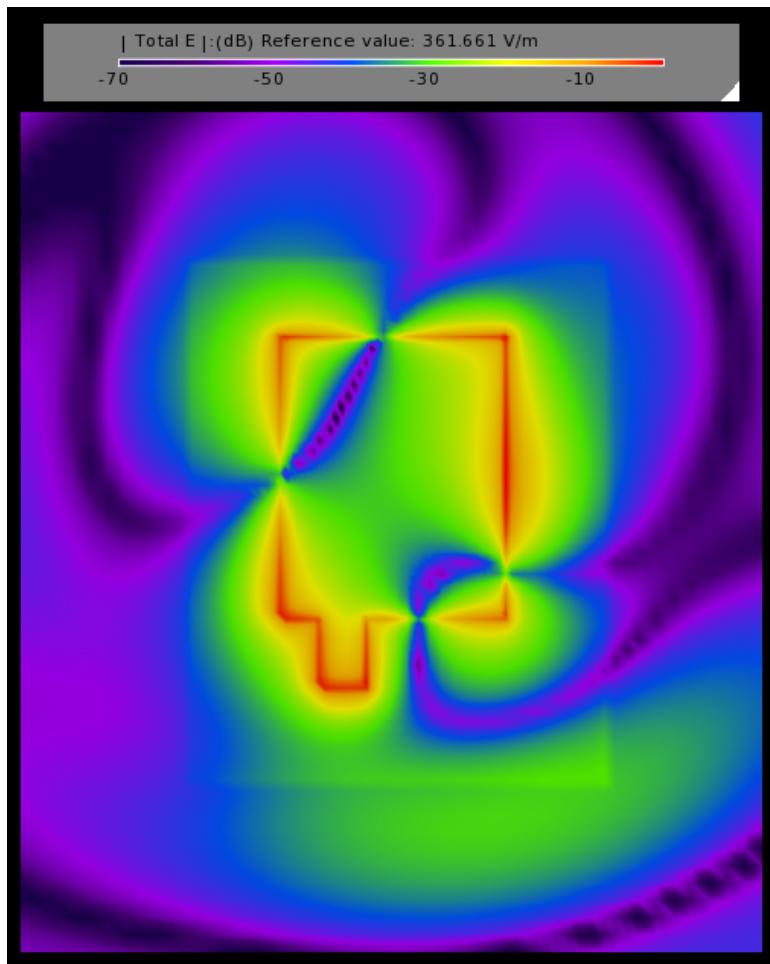


Figure 3.25: Viewing E-Field in the Geometry window

- Double-click on the E-FIELD (E) result in the filtered list. The  GEOMETRY workspace window will appear to view the electric field time sequence.
- Navigate to the  SEQUENCE tab to view the results. You can play back the results as an animation or step through them with the SHOWING control. If you wish, change the MINIMUM and MAXIMUM settings to only display a certain range of the sequence.
- Press the UNLOAD button when you are finished viewing the E-field results.

### 3.11 Adding a Parameter Sweep

XF allows you to parameterize your project by defining variables within the  PARAMETERS workspace window so that you can reference them in any editor or dialog window. Additionally, it incorporates the

ability to perform a “Parameter Sweep” so that a calculation will increment the value of a variable in order to perform a calculation at every iteration.

For this patch antenna example, we will define a parameter called **x** that will control the position of the **FEED** and antenna **STUB**. Later, we will set up a parameter sweep so that the calculation engine will retrieve values for several incremented locations of the feed.

- Open the **PARAMETERS** workspace window. Press the **+** button to add a new parameter.

- NAME: **x**
- FORMULA: **7.14**
- DESCRIPTION: Position of Patch

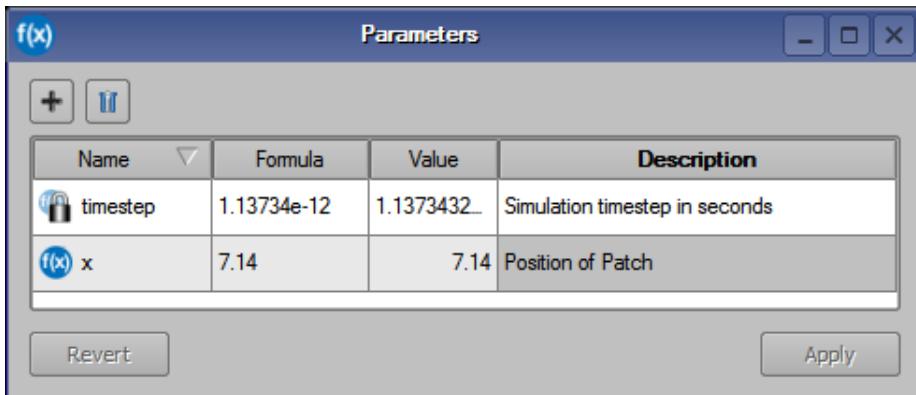


Figure 3.26: Defining a global parameter in the parameters window

- Press **APPLY** to add the parameter to the project.

### 3.11.1 Parameterizing the Geometry

For brevity, we originally created the antenna **MICROSTRIP** in one piece. To parameterize the **STUB** location, we will redraw it as two separate sheet bodies.

- In the **PROJECT TREE**, navigate to the **MICROSTRIP: MODELING SEQUENCE** branch and double-click on the **SHEET BODY** object.

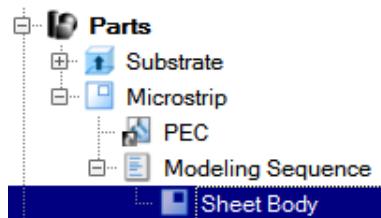


Figure 3.27: Navigating to the Sheet Body object

- Select the SELECT/MANIPULATE tool at the top left of the GEOMETRY workspace window. Right-click on an edge of the stub extension, and select DELETE EDGE.

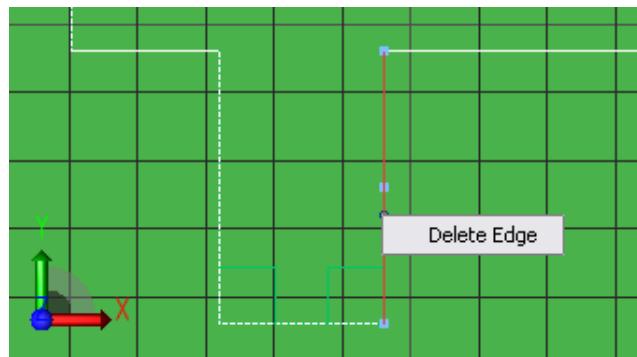


Figure 3.28: Deleting an edge of the stub

- Repeat this process to remove the other two stub edges.
- Left-click on the left endpoint in the bottom edge of the sketch. Drag it to connect with the neighboring endpoint and close the gap.
- Press DONE to apply your changes.

Now, we will add a STUB sheet body with a parameterized location.

- Right-click on the PARTS branch in the PROJECT TREE. Choose +CREATE NEW > SHEET BODY from the context menu.
- Navigate to the SPECIFY ORIENTATION tab. Set the origin to (0, 0, 0.794 mm) to place the SHEET BODY on top of the SUBSTRATE.
- Navigate to the EDIT PROFILE tab. Type “Stub” into the NAME box.
- Select the RECTANGLE tool. Use the creation dialog to enter the corners of the stub rectangle:
  - Endpoint 1: (7.2 mm, 5.6 mm)
  - Endpoint 2: (9.6 mm, 9.6 mm)

Because of the sketcher’s automatic constraint behavior, we must parameterize all four corners of the stub so it can move whenever the value of **x** changes.

- To change the position of a vertex, choose the SELECT/MANIPULATE tool again and right-click on the vertex. Choose EDIT POSITION from the menu.

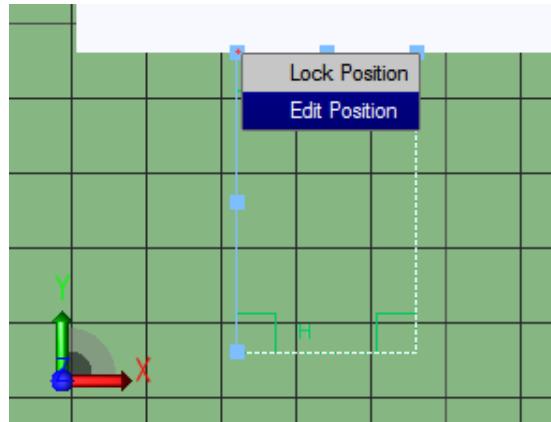


Figure 3.29: Parameterizing the stub position

- Set the stub rectangle corner positions as follows (be sure to add “mm” where necessary):
  - Upper left: (x mm, 9.6 mm)
  - Upper right: (x mm + 2.5 mm, 9.6 mm)
  - Lower left: (x mm, 5.6 mm)
  - Lower right: (x mm + 2.5 mm, 5.6 mm)
- Press DONE to finish the STUB geometry.
- Assign the PEC material to the STUB by drag-and-dropping it onto the object.

### 3.11.2 Setting the Mesh Priority

Now we will set the meshing priority of the STUB so it is greater than the SUBSTRATE. This ensures an accurate calculation.

- Right-click on the Stub in the PROJECT TREE. Under GRIDDING / MESHING > MESHING ORDER, select MOVE TO TOP.

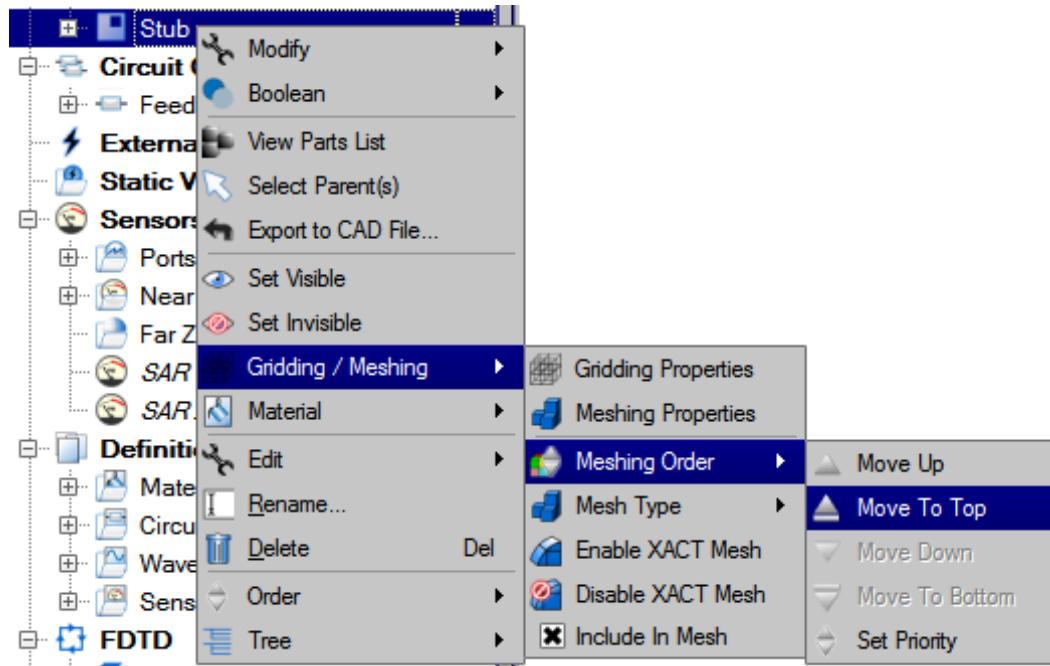


Figure 3.30: Manually setting the meshing order of the Stub

### 3.11.3 Parameterizing the Feed

- Locate the CIRCUIT COMPONENTS branch of the PROJECT TREE, and double-click on the FEED object to edit its position.
- Set the endpoints of the feed as follows:
  - ENDPOINT 1: X: “x mm + 1.2 mm”, Y: “5.6 mm”, Z: “0.794 mm”
  - ENDPOINT 2: X: “x mm + 1.2 mm”, Y: “5.6 mm”, Z: “0 mm”

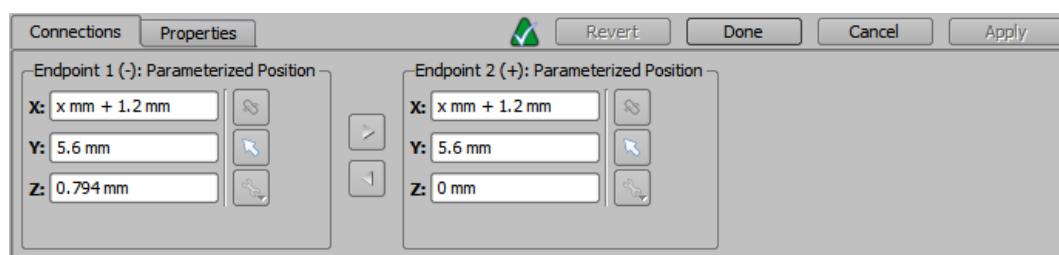


Figure 3.31: Parameterizing the position of the feed

- Click DONE to finish editing the FEED.

### 3.11.4 Adding Automatic Fixed Points to the Microstrip Object

A fixed point indicates a location of geometric significance to the grid generation process and enables the mesh to accurately represent the underlying geometry. These points can be placed automatically or manually at any location in space. With Automatic Fixed Points, XFDTD places fixed points at locations where a part has one of several key geometric features. In this Patch Antenna project, for example, the width of the Stub will not be captured exactly unless automatic fixed points are used. This is illustrated at the default original location of the Stub (with Parameter “x” set to “7.14”) in Figure 3.34 below. Here, the Meshed representation of the Stub is wider than its CAD representation, which may lead to an inaccurate result. Adding fixed points will allow us to accurately capture the Stub width. Furthermore, when we move the Stub (by sweeping the Parameter “x”), the Automatic Fixed Points will move along with the changes to the geometry, and the Mesh remains accurate.

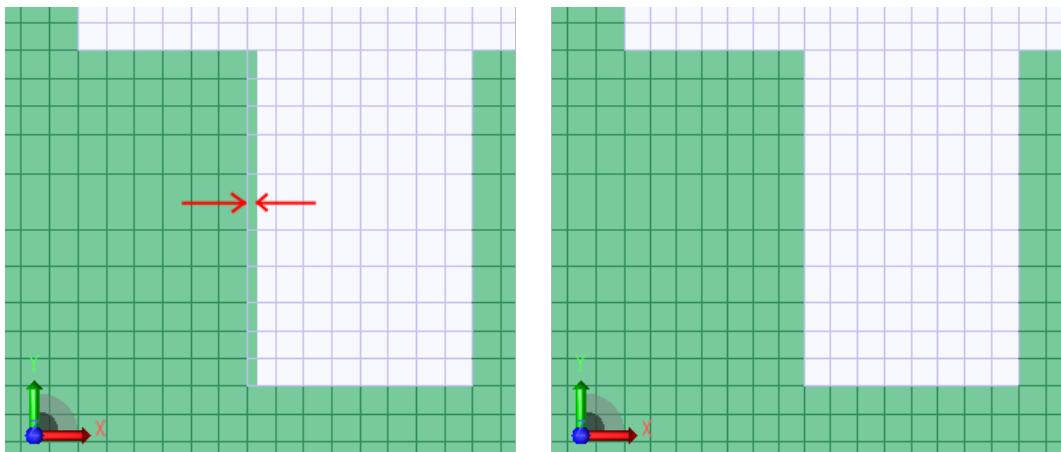


Figure 3.32: Patch Antenna Mesh with and without Automatic Fixed Points. Without Automatic Fixed Points (left), the Mesh does not accurately capture the width of the Stub, because there is a gap between the lower X edge of the Stub and the nearest grid line. Adding Automatic Fixed Points to the Stub (right) adjusts cell sizes near the Stub so that grid lines coincide with the Stub’s corners.

To add automatic fixed points to the microstrip and the stub, follow this procedure:

- Right-click on the MICROSTRIP in the PROJECT TREE. Select GRIDDING / MESHING > GRIDDING PROPERTIES.

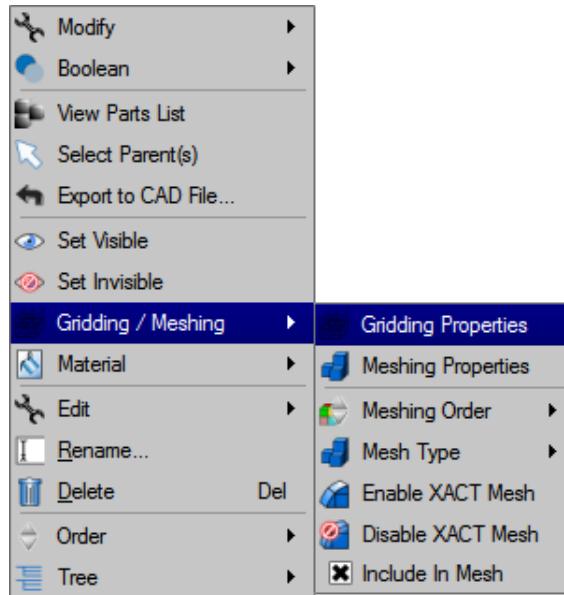


Figure 3.33: Selecting Gridding Properties from the context menu

- Click the USE AUTOMATIC FIXED POINTS checkbox as shown in Figure 3.34.
- In the AUTOMATIC DISCOVERY OPTIONS region, click the EDGE CORNERS checkbox.
- Deselect all other items in the AUTOMATIC DISCOVERY OPTIONS region.
- Click the Done button. XFDTD extracts fixed points from the object and uses them to define the grid.
- Repeat the above procedure to add Automatic Fixed Points to the STUB.

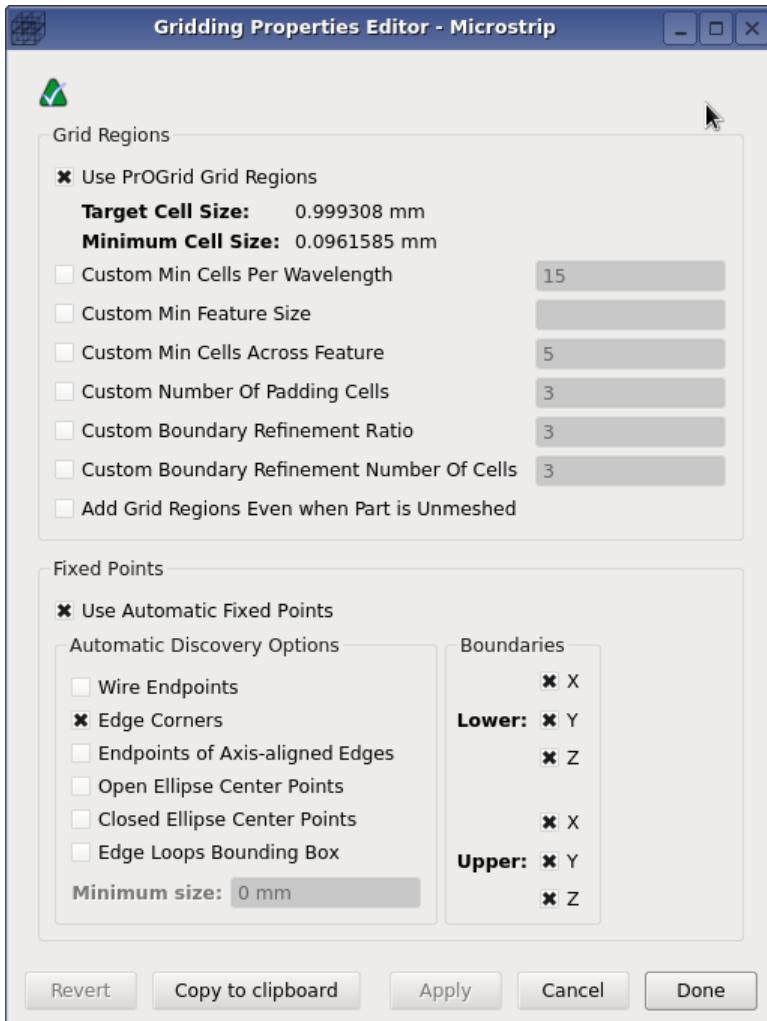


Figure 3.34: Adding automatic fixed points to the Microstrip

### 3.12 Running a Simulation with Parameter Sweep

If you have not already saved your project, do so by selecting FILE > SAVE PROJECT.

- Open the SIMULATIONS workspace window. Click the FDTD button in the upper-left to set up a new simulation.

Most of the default settings are sufficient. For this simulation, we will define a parameter sweep so that the calculation engine will collect 5 sets of results, each based on an incremented value of our global parameter, **x**.

- Navigate to the SETUP PARAMETER SWEEP tab. Check the PERFORM PARAMETER SWEEP box. Click the ADD A NEW PARAMETER SEQUENCE button. Set up the sweep as follows:

- PARAMETER TO SWEEP: x
- SWEEP TYPE: Start, Incr, Count
- COUNT: 5
- STARTING VALUE: 6.9
- INCREMENT: 0.1

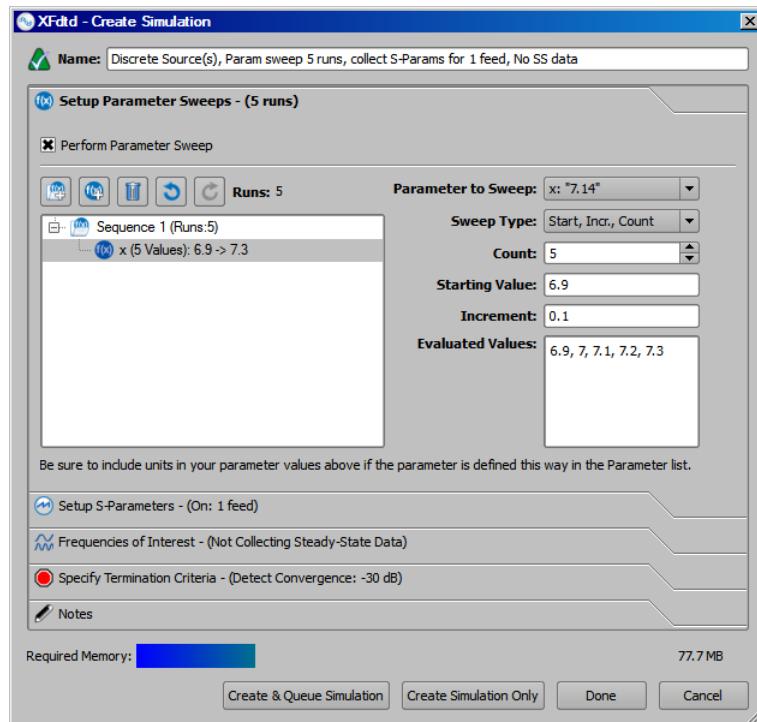


Figure 3.35: Adding the parameter to the simulation

As shown above, these settings will produce a range of values from 6.9 to 7.3.

→ Select CREATE AND QUEUE SIMULATION to run this simulation.

### 3.13 Viewing Results of the Parameter Sweep

As before, once the STATUS column in the **SIMULATIONS** workspace window shows that the simulation has completed, we can view its results in the **RESULTS** window. Under the PROJECT:SIMULATION:RUN column, notice that within the simulation, a new run is created for each parameter value.

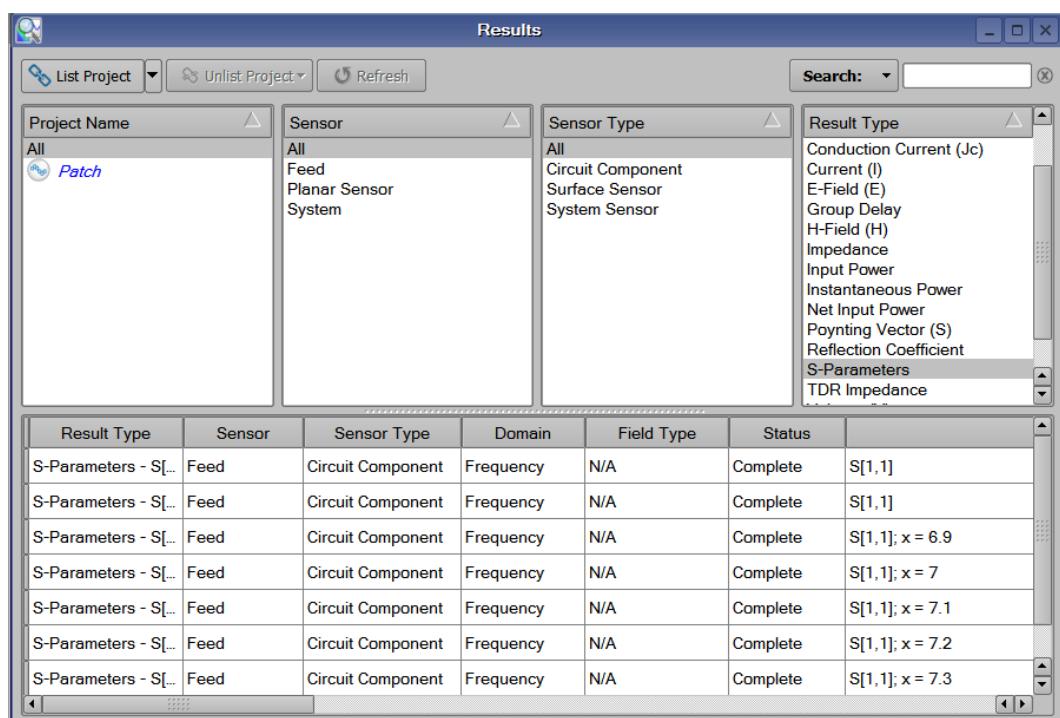


Figure 3.36: The results of the parameter sweep

## Chapter 4

### Example: A Low Pass Filter

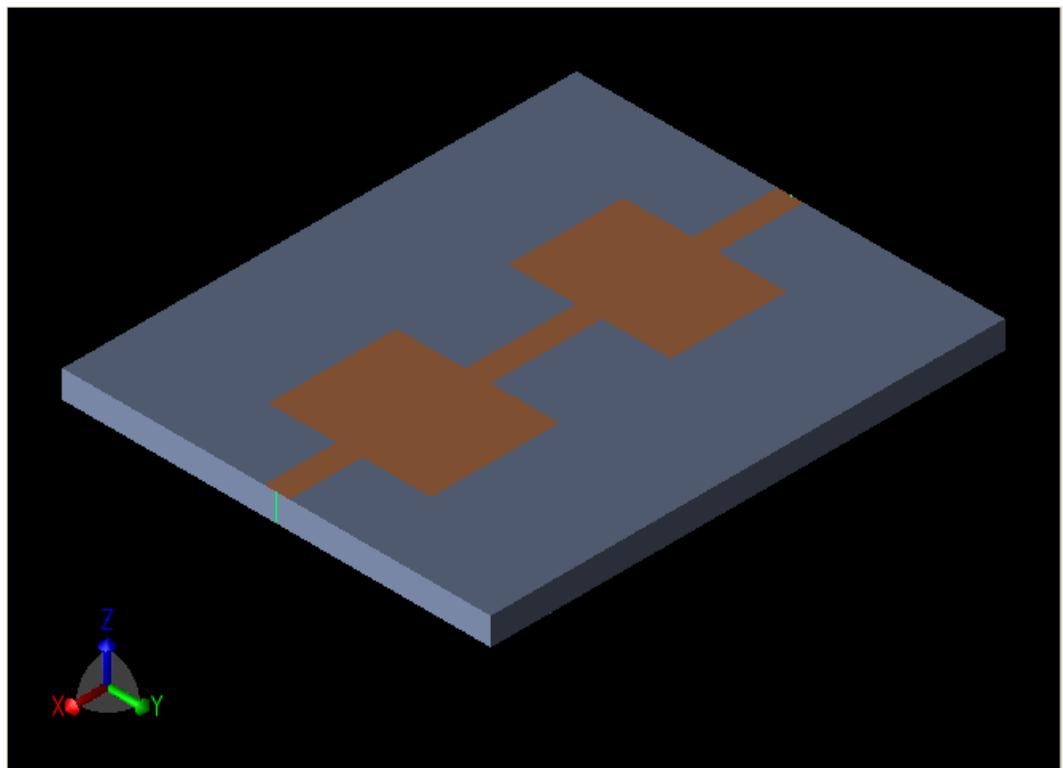


Figure 4.1: The low pass filter

Time to create: 35:00 (approx.)

In this chapter, you will learn how to...

- set the material properties of your low pass filter and create the geometry through scripts
- define the properties of the low pass filter environment
- add a feed and a load to the filter and simulate their effects
- retrieve port sensor and planar surface sensor data after running the calculation

## 4.1 Getting Started

First, a few Project Properties are set up for the low pass filter project.

- The PROJECT PROPERTIES EDITOR opens automatically whenever XFDTD is started or when a new project is created. If the editor is not currently visible, double-click on PROJECT at the top of the PROJECT TREE to open the editor.
- On the FREQUENCY RANGE OF INTEREST tab of the PROJECT PROPERTIES EDITOR, specify a frequency range of 0 - 13 GHz by setting MINIMUM to "0 GHz" and MAXIMUM to "13 GHz".
- Navigate to the DISPLAY UNITS tab, where a few relevant units should be set:
  - Set FREQUENCY to "gigahertz (GHz)"
  - Set LENGTH to "millimeters (mm)"
- Press DONE on the PROJECT PROPERTIES EDITOR.

## 4.2 Creating Materials

For this example, we will create material definitions before creating the geometry. We do this so that the script we execute to build the SUBSTRATE block can access the material definitions. The low pass filter will consist of a PERFECT ELECTRIC CONDUCTOR and a SUBSTRATE.

### Define material, PEC

- Right-click on the DEFINITIONS: MATERIALS branch of the PROJECT TREE and select NEW MATERIAL DEFINITION.
- Double-click the new material to edit its properties. Set the perfect electric conductor material properties as follows:
  - NAME: PEC
  - ELECTRIC: Perfect Conductor
  - MAGNETIC: Freespace
- If desired, navigate to the APPEARANCE tab to set the PEC material's display color.

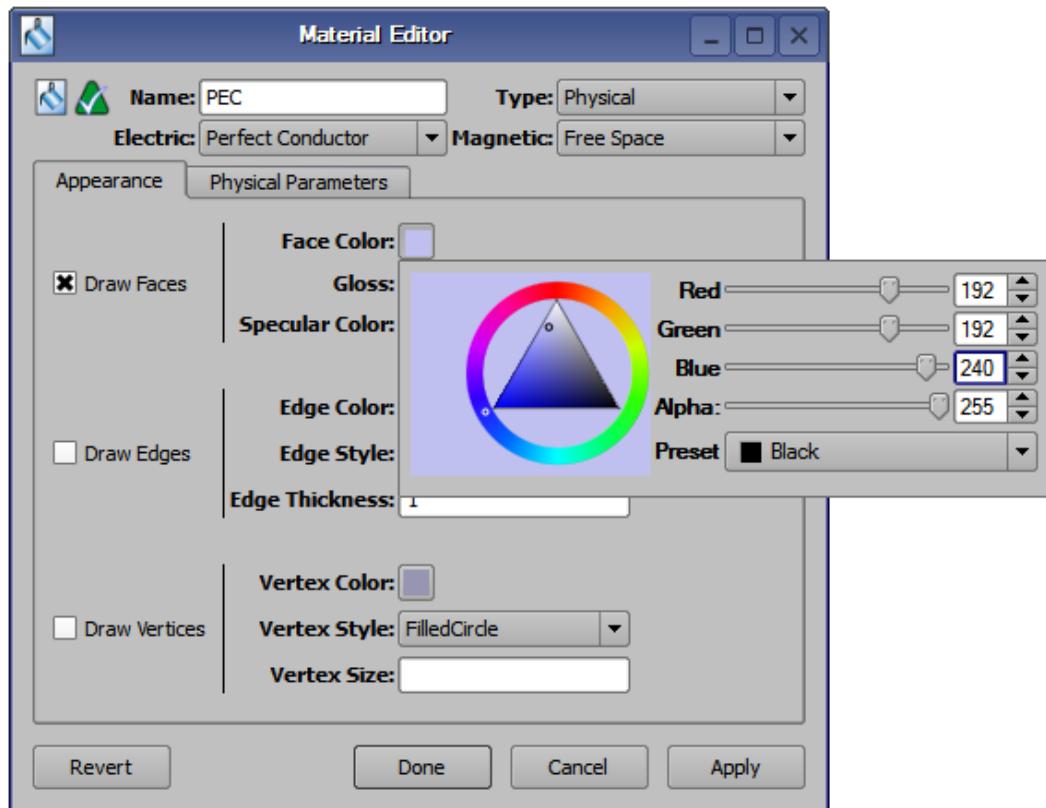


Figure 4.2: Editing the color of the PEC material

### Define material, SUBSTRATE

- Right-click on the DEFINITIONS: MATERIALS branch of the PROJECT TREE and select NEW MATERIAL DEFINITION.
- Double-click the new material to edit its properties. Set the substrate material properties as follows:
  - NAME: Substrate
  - ELECTRIC: Isotropic
  - MAGNETIC: Freespace
 Under the ELECTRIC tab:
  - TYPE: Nondispersive
  - ENTRY METHOD: Normal
  - CONDUCTIVITY: 0 S/m
  - RELATIVE PERMITTIVITY: 3

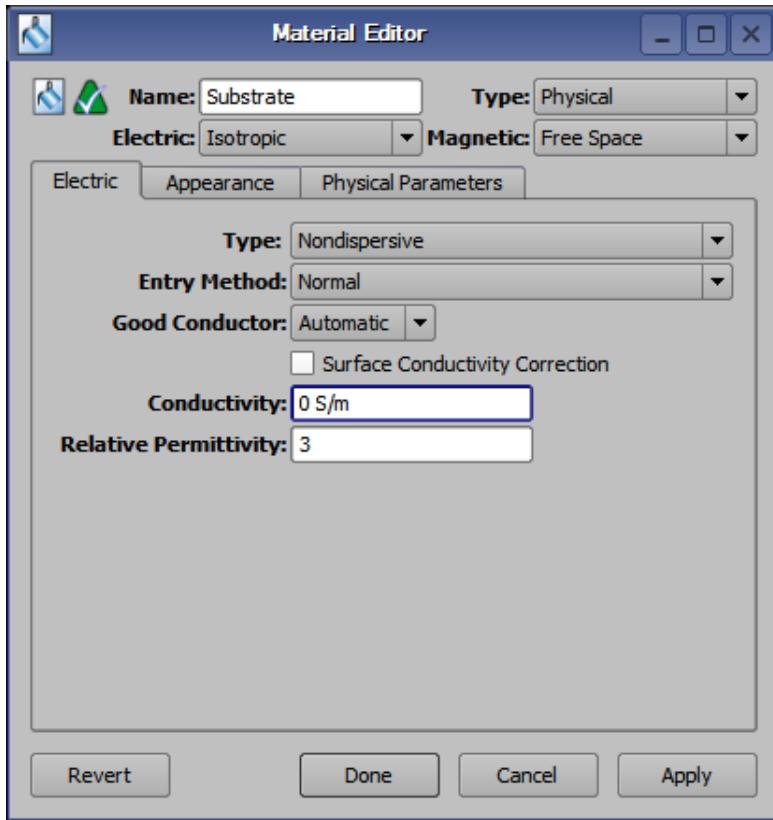


Figure 4.3: Defining the properties of the Substrate material

- In the APPEARANCE tab, assign a new color to this material to distinguish it from the first material, PEC.
- Click DONE to add the new material, SUBSTRATE.



Figure 4.4: PEC and Substrate materials in the Project Tree

## 4.3 Creating the Low Pass Filter Geometry

### 4.3.1 Modeling the Substrate

The  SUBSTRATE block geometry for the low pass filter is a simple rectangular block. For this example, we will use a script to prompt an interface where we can create a rectangular block with an applied material.

- Right-click on the  SCRIPTS branch of the  PROJECT TREE and select  NEW MACRO SCRIPT.
- This automatically adds a  NEW MACRO SCRIPT object to the branch. Right-click on the object, select RENAME, and type "Rectangular Block".
- Copy-and-paste the following script into the  SCRIPTING workspace window.

```
var proj = App.getActiveProject();

var d = new SimpleDialog;
d.windowTitle = "RectangularBlock";

var nameEdit = new LabeledLineEdit();
nameEdit.label = "Name";
nameEdit.text = "Substrate";
d.add( nameEdit );

var positionLabel = new Label();
positionLabel.text = "Corner Point";
d.add( positionLabel );

var positionGroupBox = new SimpleGroupBox();

var xPos = new LabeledLineEdit();
xPos.label = "X";
xPos.text = "0 mm";
positionGroupBox.add( xPos );

var yPos = new LabeledLineEdit();
yPos.label = "Y";
yPos.text = "0 mm";
positionGroupBox.add( yPos );

var zPos = new LabeledLineEdit();
zPos.label = "Z";
zPos.text = "-0.64 mm";
positionGroupBox.add( zPos );
d.add( positionGroupBox );

var offsetLabel = new Label();
offsetLabel.text = "Change in";
d.add( offsetLabel );

var offsetGroupBox = new SimpleGroupBox();

var xOff = new LabeledLineEdit();
xOff.label = "X";
xOff.text = "12 mm";
offsetGroupBox.add( xOff );
```

```

var yOff = new LabeledLineEdit();
yOff.label = "Y";
yOff.text = "10 mm";
offsetGroupBox.add( yOff );

var zOff = new LabeledLineEdit();
zOff.label = "Z";
zOff.text = "0.64 mm";
offsetGroupBox.add( zOff );

d.add( offsetGroupBox );

var materialsCombo = new LabeledComboBox;
materialsCombo.label = "Material: ";
var ml = proj.getMaterialList();
var materialNames = ml.getAllMaterialNames();
materialNames.push( "--no material--" );
materialsCombo.setItemList( materialNames );
d.add( materialsCombo );

if( d.exec() )
{
    var name = nameEdit.text;
    var s = new Sketch();
    var origin = new Cartesian3D( xPos.text, yPos.text, zPos.text );
    var offset = new Cartesian3D( xOff.text, yOff.text, zOff.text );

    var p1 = new Cartesian3D( origin.x,
                            origin.y,
                            origin.z );
    var p2 = new Cartesian3D( origin.x,
                            MathUtils.addFormula( origin.y, offset.y ),
                            origin.z );
    var p3 = new Cartesian3D( MathUtils.addFormula( origin.x, offset.x ),
                            MathUtils.addFormula( origin.y, offset.y ),
                            origin.z );
    var p4 = new Cartesian3D( MathUtils.addFormula( origin.x, offset.x ),
                            origin.y,
                            origin.z );

    var l = new Line( p1, p2 );
    s.addEdge( new Line( p1, p2 ) );
    s.addEdge( new Line( p2, p3 ) );
    s.addEdge( new Line( p3, p4 ) );
    s.addEdge( new Line( p4, p1 ) );

    var e = new Extrude( s, offset.z );
    var r = new Recipe();

    r.append( e );

    var m = new Model();
    m.setRecipe( r );
    m.name = name;
    var objectInProject = App.getActiveProject()
        .getGeometryAssembly()
        .append( m );

    var mname = materialsCombo.currentText;
    if( mname != "--no material--" )

```

```

    {
        var material = proj.getMaterialList().getMaterial( mname );
        App.getActiveProject().setMaterial( objectInProject, material );
    }
}

```

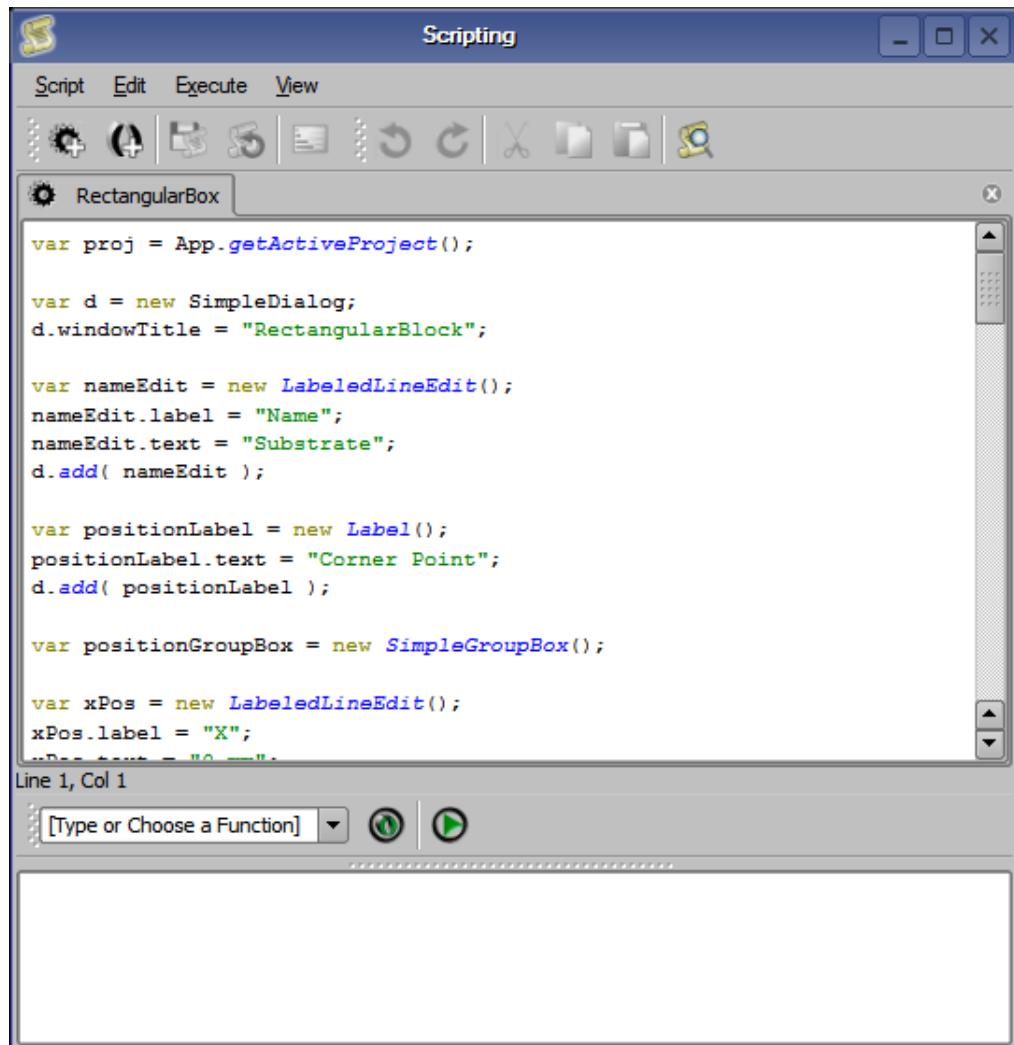


Figure 4.5: The substrate script in the Scripting workspace window

- Press the COMMIT icon to commit your new macro script to the project.
- Press the EXECUTE button to run the script. This will prompt you with a dialog to create a rectangular block. The correct values have already been defined in the form.
- In the MATERIAL drop-down list of the window, select SUBSTRATE to apply the SUBSTRATE material to the rectangular box.

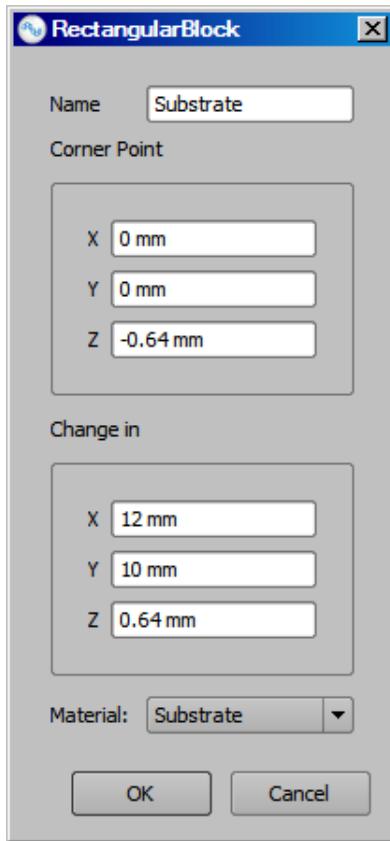


Figure 4.6: Applying Substrate material through scripting interface

→ Press OK to close the window and add the SUBSTRATE block to the project.

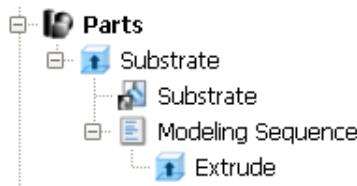


Figure 4.7: The Substrate block in the Project Tree

### 4.3.2 Modeling the Strip Line

The STRIP LINE will be modeled as a polygon SHEET BODY. To simplify this operation, we will use another script to add the Strip Line to the project.

→ Right-click on the SCRIPTS branch of the PROJECT TREE and select NEW MACRO SCRIPT.

- Right-click on the NEW MACRO SCRIPT object, select RENAME, and type “Strip Line”.
- Copy-and-paste the following script into the SCRIPTING workspace window.

```

var MicroStripWire = new Sketch();

var v = [];
v[0] = new Cartesian3D( 0, "5.3 mm", 0 );
v[1] = new Cartesian3D( "2 mm", "5.3 mm", 0 );
v[2] = new Cartesian3D( "2 mm", "6.9 mm", 0 );
v[3] = new Cartesian3D( "4.7 mm", "6.9 mm", 0 );
v[4] = new Cartesian3D( "4.7 mm", "5.3 mm", 0 );
v[5] = new Cartesian3D( "7.3 mm", "5.3 mm", 0 );
v[6] = new Cartesian3D( "7.3 mm", "6.9 mm", 0 );
v[7] = new Cartesian3D( "10.3 mm", "6.9 mm", 0 );
v[8] = new Cartesian3D( "10.3 mm", "5.3 mm", 0 );
v[9] = new Cartesian3D( "12 mm", "5.3 mm", 0 );
v[10] = new Cartesian3D( "12 mm", "4.7 mm", 0 );
v[11] = new Cartesian3D( "10.3 mm", "4.7 mm", 0 );
v[12] = new Cartesian3D( "10.3 mm", "3.1 mm", 0 );
v[13] = new Cartesian3D( "7.3 mm", "3.1 mm", 0 );
v[14] = new Cartesian3D( "7.3 mm", "4.7 mm", 0 );
v[15] = new Cartesian3D( "4.7 mm", "4.7 mm", 0 );
v[16] = new Cartesian3D( "4.7 mm", "3.1 mm", 0 );
v[17] = new Cartesian3D( "2 mm", "3.1 mm", 0 );
v[18] = new Cartesian3D( "2 mm", "4.7 mm", 0 );
v[19] = new Cartesian3D( 0, "4.7 mm", 0 );

MicroStripWire.addPolygon( v );

var cover = new Cover( MicroStripWire );
var MicroStripRecipe = new Recipe();
MicroStripRecipe.append( cover );

var MicroStripModel = new Model();
MicroStripModel.setRecipe( MicroStripRecipe );
MicroStripModel.name = "StripLine"

var MicroStripInProject = App.getActiveProject()
    .getGeometryAssembly()
    .append( MicroStripModel );

```

- Press the COMMIT icon to commit your new macro script to the project.
- Press the EXECUTE button to run the script and create the STRIP LINE object.

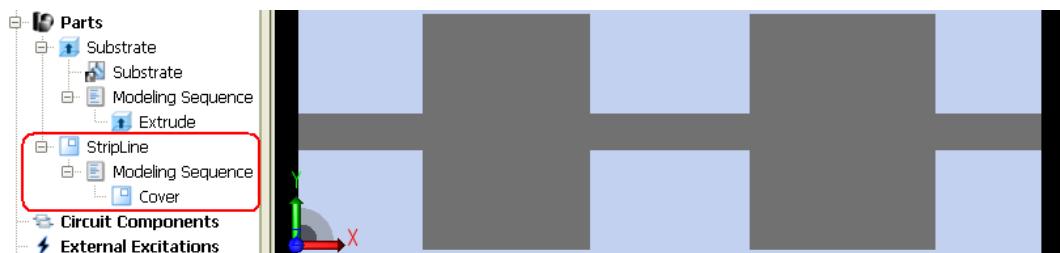


Figure 4.8: Strip Line geometry created from script

- Drag-and-drop the material object PEC onto the object STRIP LINE to assign a material to this

object.

The completed low pass filter geometry will appear in the GEOMETRY workspace window.

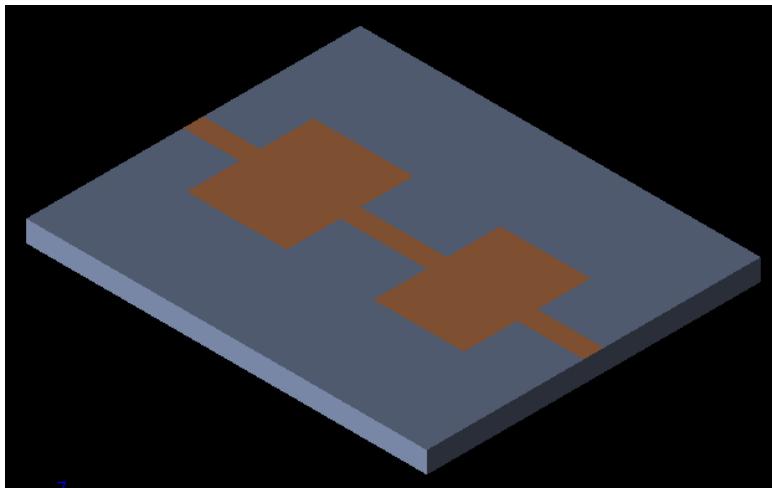


Figure 4.9: Low pass filter geometry

### Meshing Priority

Ensure that the meshing priority of the STRIP LINE is greater than the SUBSTRATE for an accurate calculation.

- Right-click on the Strip Line in the PROJECT TREE. Under MESHING ORDER, select MOVE TO TOP if it is an available option.

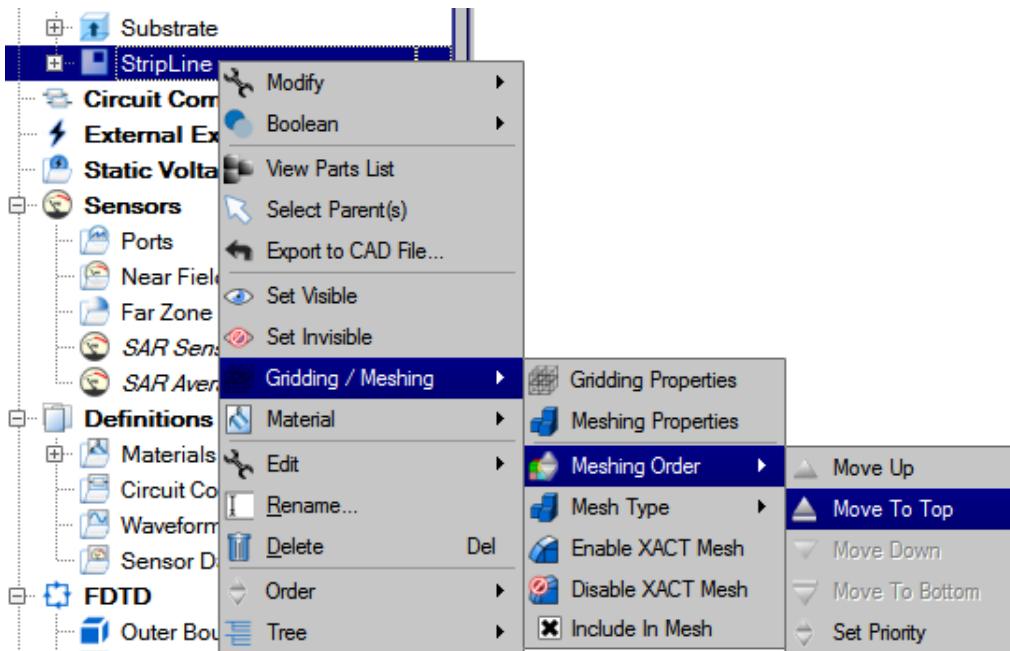


Figure 4.10: Manually setting the meshing order of the Strip Line

## 4.4 Defining the Outer Boundary

- Double-click on the **FDTD: OUTER BOUNDARY** branch of the **PROJECT TREE** to open the **OUTER BOUNDARY EDITOR**.
- Set the outer boundary properties as follows:
  - **BOUNDARY:** All boundaries “Absorbing” except LOWER BOUNDARY  $Z$ , which is “PEC”
  - **ABSORPTION TYPE:** PML
  - **LAYERS:** 7

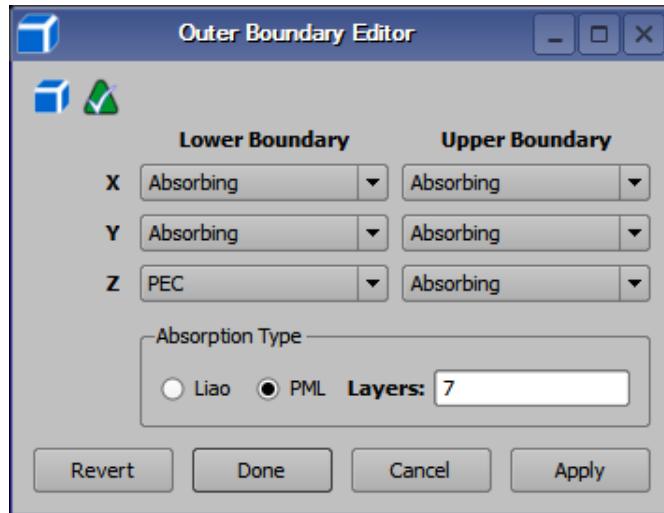


Figure 4.11: Defining the outer boundary for the low pass filter

→ Click DONE to apply the outer boundary settings.

## 4.5 Defining the Grid

Now we will define characteristics of the calculation grid.

- Double-click on the FDTD:GRID branch of the PROJECT TREE to open the GRID EDITOR.
- On the CELL SIZE tab (Figure 4.12), set MIN CELLS PER WAVELENGTH to "60". This will increase the grid resolution (improving accuracy) without significantly increasing the amount of memory or runtime required for the simulation.

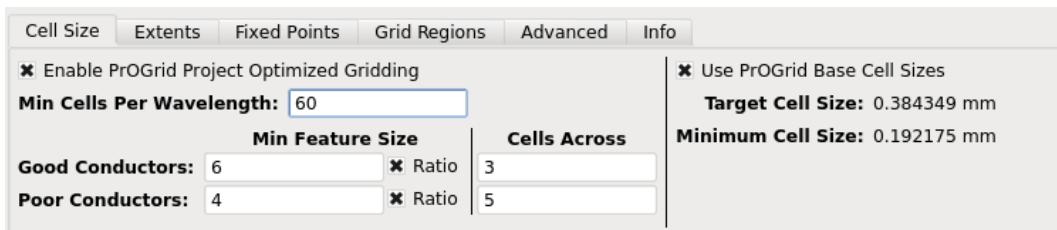


Figure 4.12: Defining cell sizes in the Grid Editor

- Navigate to the EXTENTS tab (Figure 4.13). We will apply free space padding to all sides of the simulation space except the Lower Z boundary (where a PEC boundary condition was previously assigned).
  - Free space padding on the five absorbing boundaries is present with the default settings (both SPECIFY PADDING and PROGRID PADDING ON ABSORBING BOUNDARIES enabled).

- Remove all padding from the lower Z boundary by setting LOWER Z FREE SPACE PADDING (BASE CELLS) to “0” (Figure 4.13). This will cause the PEC boundary condition to lie exactly on the bottom surface of the Substrate as a ground plane.

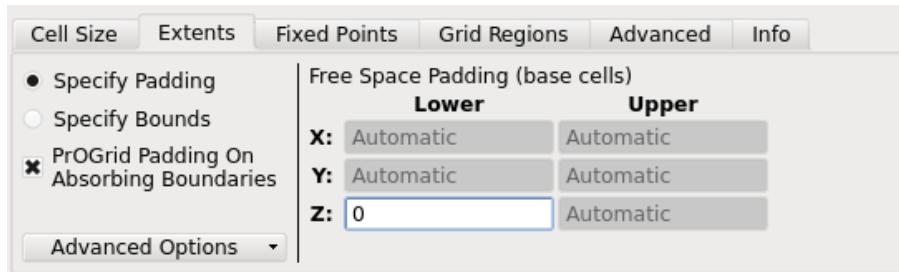


Figure 4.13: Defining free space padding in the Grid Editor

→ Click DONE to apply the grid settings.

At this point, some of the key geometrical features do not fall exactly on grid lines. To properly capture the impedance of each section of the strip line, we must resolve the strip widths and the distance between the strip line and the ground plane accurately. Adding Automatic Fixed Points to the “StripLine” part will ensure that this is the case.

→ Right-click on the STRIPLINE in the PROJECT TREE. Select GRIDDING / MESHING > GRIDDING PROPERTIES.

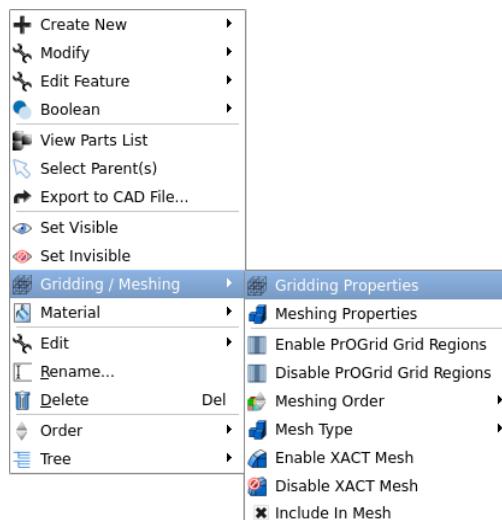


Figure 4.14: Selecting Gridding Properties from the context menu

- Click the USE AUTOMATIC FIXED POINTS checkbox as shown in Figure 3.34. The default AUTOMATIC DISCOVERY OPTIONS are sufficient for this example.
- Click the **Done** button. XFdtd extracts fixed points from the object and uses them to define the grid.

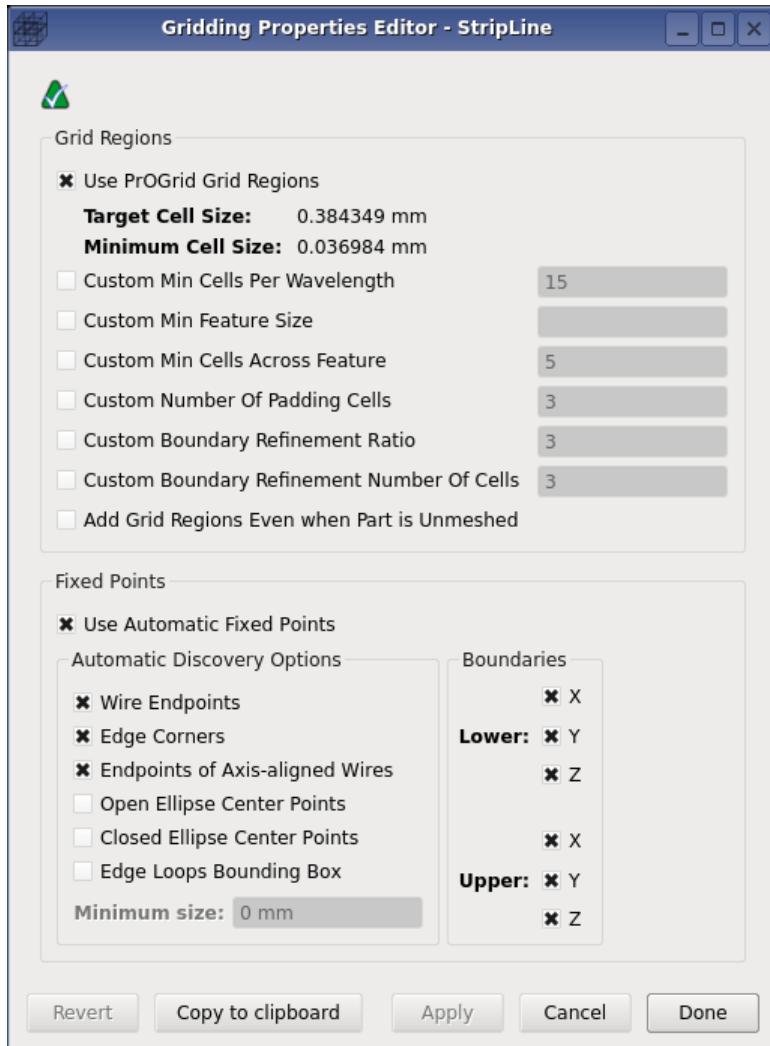


Figure 4.15: Adding automatic fixed points to the StripLine

## 4.6 Adding a Feed

- Right-click on the CIRCUIT COMPONENTS branch of the PROJECT TREE. Choose NEW CIRCUIT COMPONENT WITH > NEW FEED DEFINITION from the context menu.

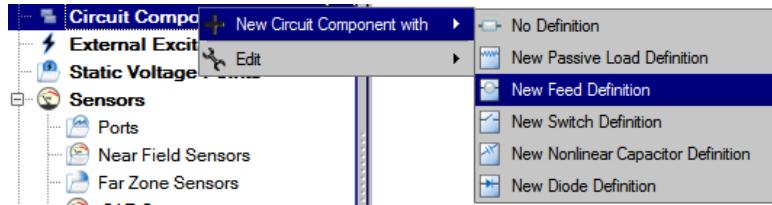


Figure 4.16: Adding a feed to the project

→ Define the endpoints of the feed.

1. Click the PICKER TOOL for Endpoint 1.
  2. Click on the center of the Strip Line at the Lower X end of the part (See Figure 4.17). The X, Y, Z fields for Endpoint 1 populate.
  3. Click the COPY ENDPOINT 1 TO ENDPOINT 2 button. The X, Y, Z fields for Endpoint 2 populate.
  4. Open the ADVANCED OPTIONS for ENDPOINT 2.
  5. Enter -0.64 mm in the Z field.
  6. Press the ENTER key. The Z field in the ENDPOINT 2 section of the CONNECTIONS tab displays -0.64 mm.
- Endpoints detach when their locations are changed manually. Therefore, typing data in the Z field is not recommended because it will detach ENDPOINT 2 from the part. If the orientation of the part changes the detached feed will not move with it. To avoid this issue, it is best to use the ADVANCED OPTIONS window to modify data.

→ Under the PROPERTIES tab, type “Feed” in the NAME box.

→ Click DONE to add the FEED.

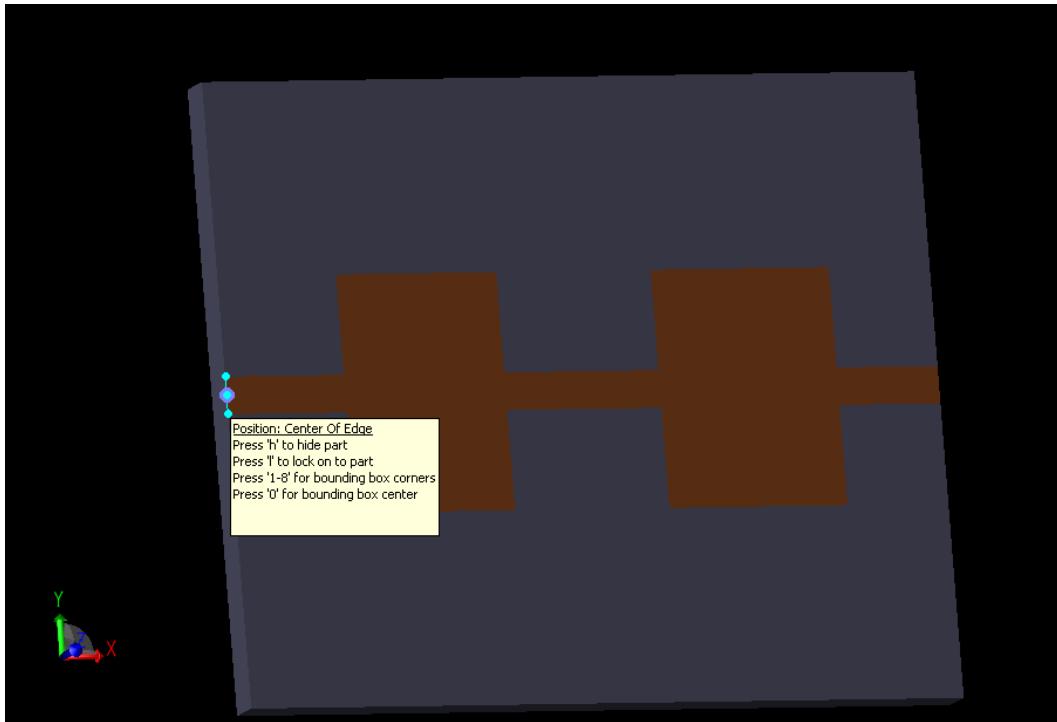


Figure 4.17: Adding a feed with the Picker tool

## 4.7 Adding a Load

The LOAD will be placed at the opposite end of the low pass filter geometry as the FEED.

- Right-click on the CIRCUIT COMPONENTS branch of the PROJECT TREE, and select NEW CIRCUIT COMPONENT WITH > NEW PASSIVE LOAD DEFINITION.
- Define the endpoints of the passive load.
  1. Click the PICKER TOOL for Endpoint 1.
  2. Click on the center of the Strip Line at the Upper X end of the part. The X, Y, Z fields for Endpoint 1 populate.
  3. Click the COPY ENDPOINT 1 TO ENDPOINT 2 button. The X, Y, Z fields for Endpoint 2 populate.
  4. Open the ADVANCED OPTIONS for ENDPOINT 2.
  5. Enter -0.64 mm in the Z field.
  6. Press the ENTER key. The Z field in the ENDPOINT 2 section of the CONNECTIONS tab displays -0.64 mm.

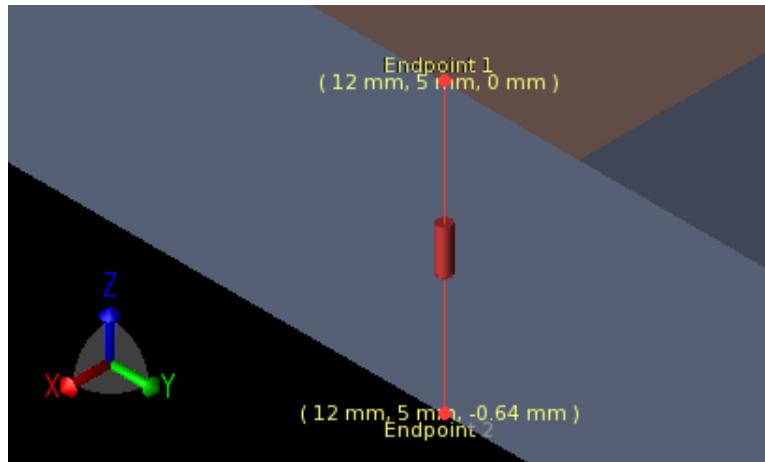


Figure 4.18: Zooming in on the location of the Load

- Under the PROPERTIES tab, type “Load” in the NAME box.
- Click DONE to add the LOAD to the project.

## 4.8 Requesting Output Data

Recall that the project already contains two port sensors that will request results ( FEED and LOAD). We also wish to collect field samplings at discrete intervals of time throughout the calculation. To retrieve this data, add a PLANAR SENSOR at the surface of the STRIP LINE.

- Right-click on the SENSORS: NEAR FIELD SENSORS branch of the PROJECT TREE. Select NEW PLANAR SENSOR from the context menu.
- In the Specify Orientation tab, select the PICK SIMPLE PLANE button. Mouse over the STRIP LINE and press C to center the normal on the face. Click on this location to set the PLANAR SENSOR in place.

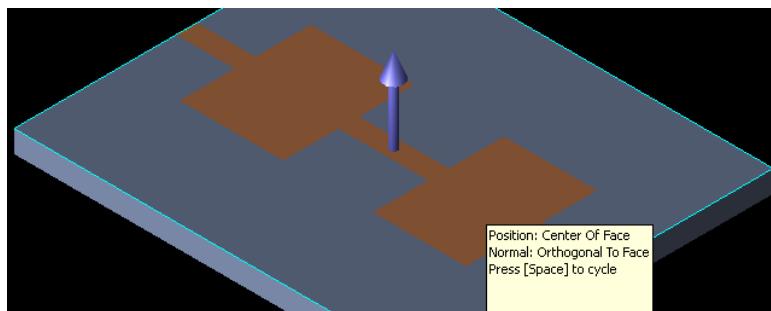


Figure 4.19: Centering the planar sensor on the Strip Line

- Click DONE to add the PLANAR SENSOR to the project.

This sensor requires a data definition.

- Open on the DEFINITIONS: SENSOR DATA DEFINITIONS branch of the PROJECT TREE. Double-click on the NEW SURFACE SENSOR DEFINITION item in the tree.
- Set the properties of the surface sensor definition as follows:
  - NAME: Field Sampling
  - FIELD VS. TIME: E, H, B, and J
  - START TIME: 50 \* timestep
  - END TIME: 2000 \* timestep
  - SAMPLING INTERVAL: 50 \* timestep

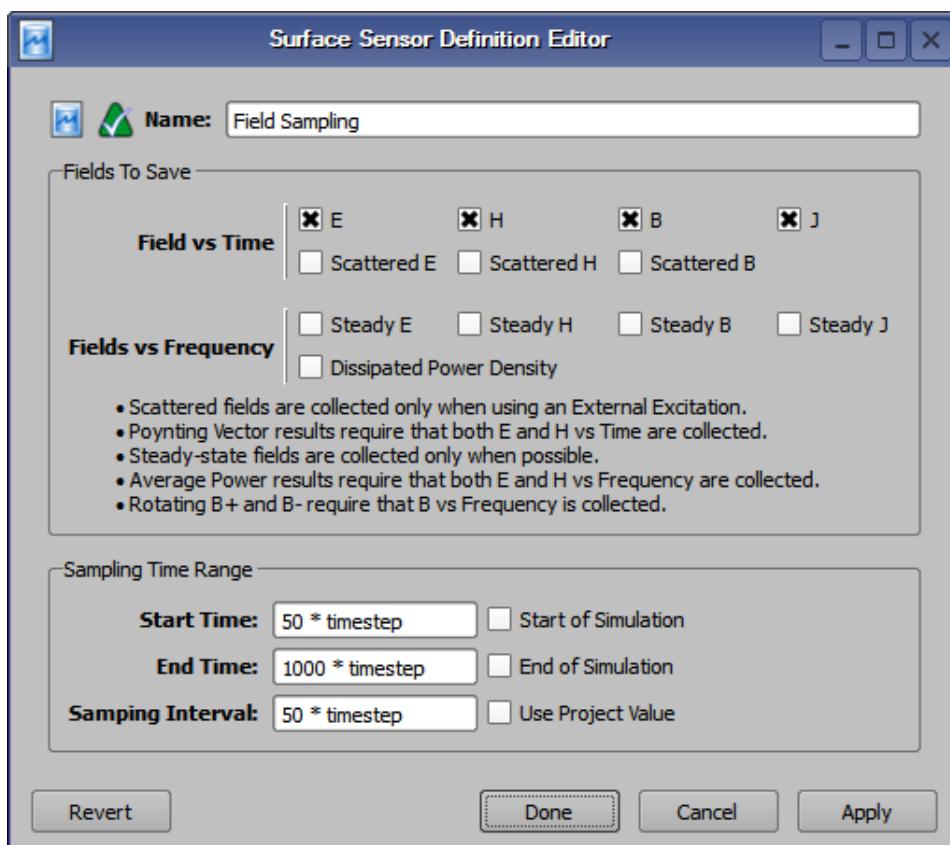


Figure 4.20: Adding the sensor definition

- Press DONE to finish editing the surface sensor definition.

Now, assign the new definition to the surface sensor.

- Click-and-drag the FIELD SAMPLING definition located in the PROJECT TREE and drop it on top of the SURFACE SENSOR in the SENSORS: NEAR FIELD SENSORS branch.

## 4.9 Running the Calculation

If you have not already saved your project, do so by selecting FILE > SAVE PROJECT. Once the project is saved, a new simulation can be created to send to the calculation engine.

- Open the SIMULATIONS workspace window. Click the CREATE SIMULATION button in the upper-left to set up a new simulation.
- Most of the default settings are sufficient. Navigate to the SPECIFY TERMINATION CRITERIA tab (Figure 4.21). Set up the termination criteria as follows:
  - ANALYZE PROJECT CONTENTS: Unchecked
  - DETECT CONVERGENCE: Checked
  - THRESHOLD: -40 dB
  - MAXIMUM SIMULATION TIME: 10000 \* timestep

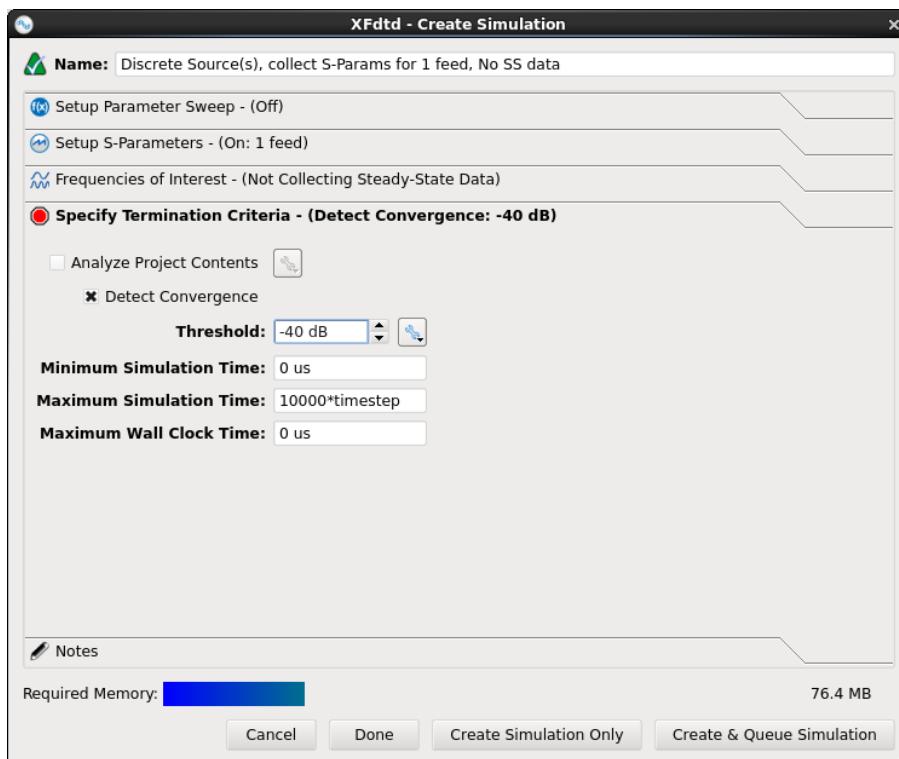


Figure 4.21: Adding a new simulation to the low pass filter project

- Click the CREATE AND QUEUE SIMULATION button to close the dialog and run the new simulation.

## 4.10 Viewing the Results

First, we will view the results retrieved with the port sensor placed at the location of the  FEED.

### Ensuring convergence has been reached

Although automatic convergence has been set, it is good practice to view the waveforms in the model to ensure that the energy has completely dissipated, providing complete convergence.

- To filter the list accordingly, select the following options in the columns in the top pane of the  RESULTS window. (You may need to change your column headings first.)
  - SENSOR TYPE: Circuit Component
  - DOMAIN: Time
  - RESULT TYPE: Voltage (V)

This will filter all time-dependent voltage data collected by the  FEED circuit component.

- Double-click on the  LOAD result to view a 2-D plot to ensure convergence has been met.

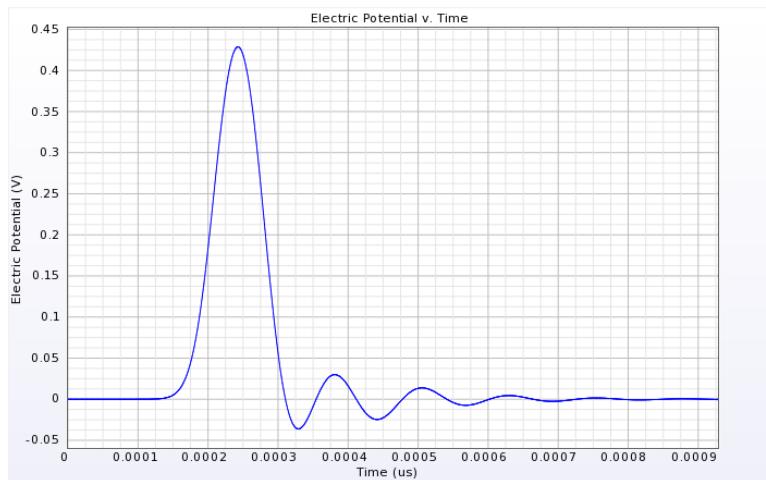


Figure 4.22: Viewing results plot to ensure convergence at the Load

- Repeat to view the results at the  FEED.

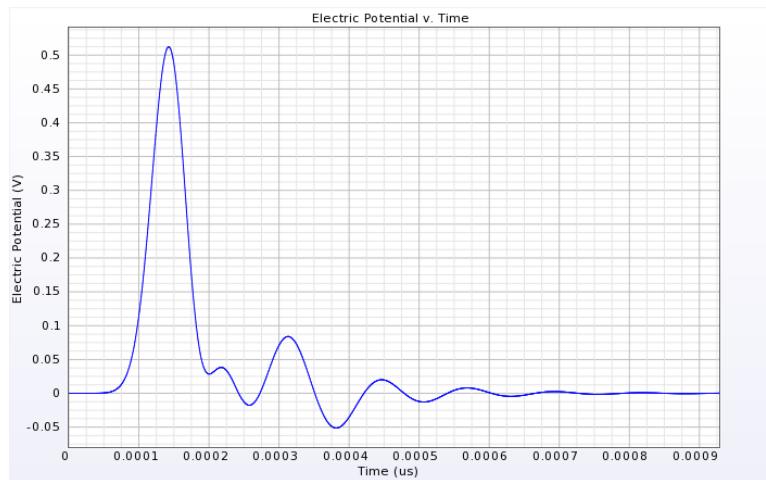


Figure 4.23: Viewing results plot to ensure convergence at the Feed

## E-Field Results from the Surface Sensor

Now we will view the field sequences collected by the surface sensor that was placed at the surface of the STRIP LINE.

- To filter the E-field results, select:
  - SENSOR TYPE: Surface Sensor
  - RESULT TYPE: E-Field (E)
- Double-click on the result to open the interface and view the 3-D field sequence.
- Navigate to the SEQUENCE tab to view the results. You can play back the results as an animation or step through them with the SHOWING control. If you wish, change the MINIMUM and MAXIMUM settings to only display a certain range of the sequence.

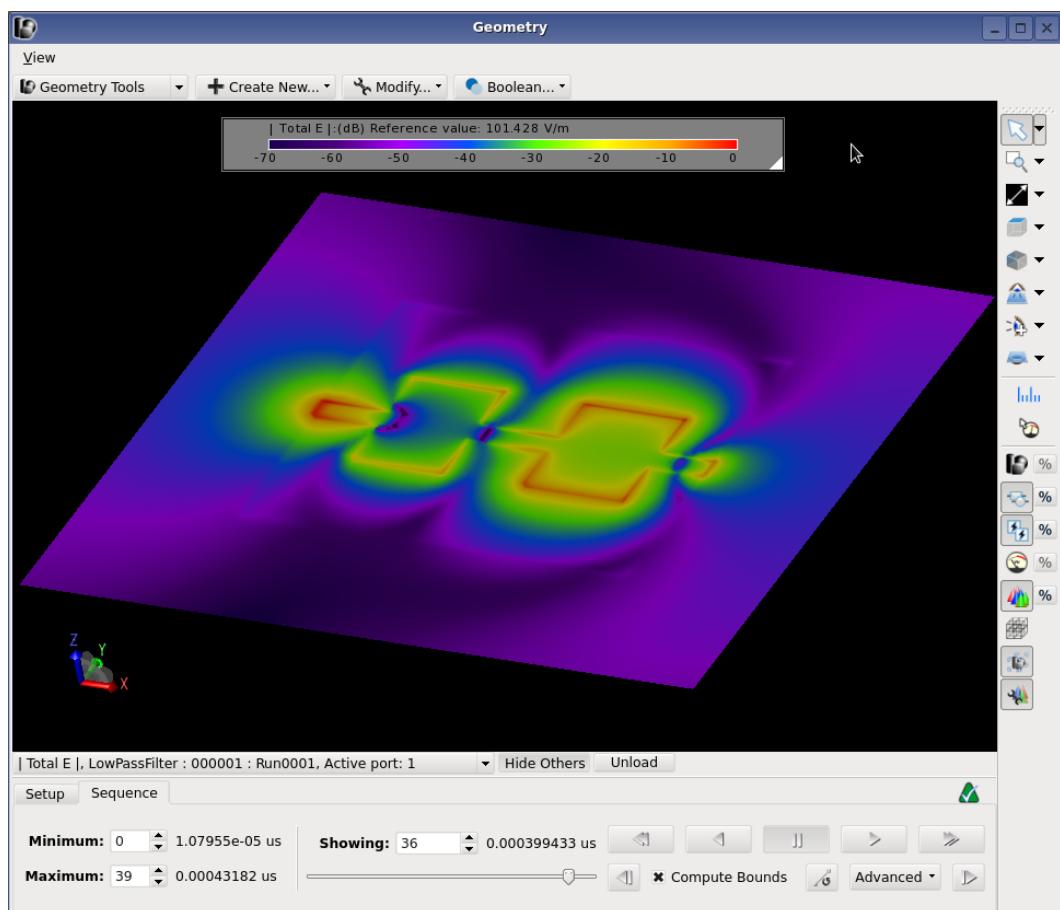


Figure 4.24: Viewing E-field results for the surface sensor near the end of the time sequence

## Chapter 5

### Example: A Rectangular Waveguide

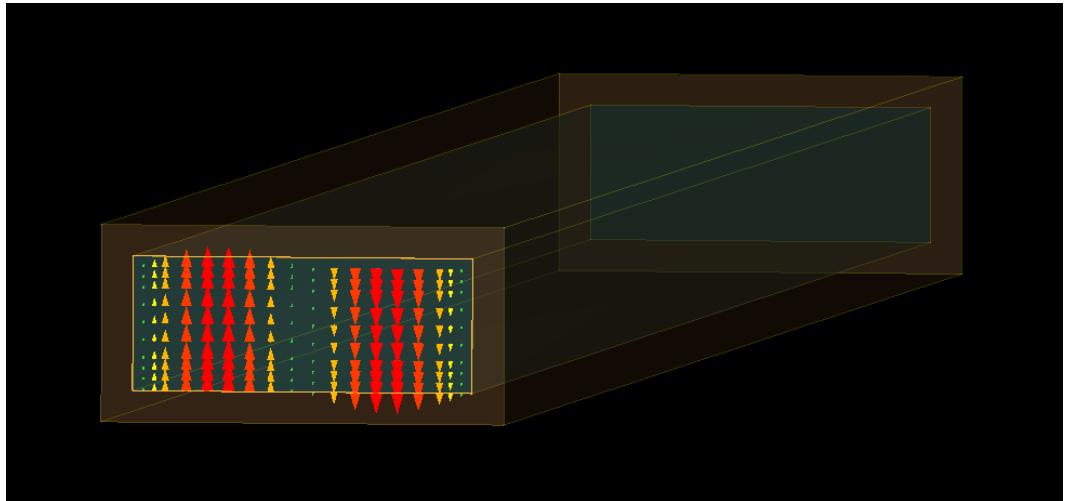


Figure 5.1: Teflon Filled WR-42 Rectangular Waveguide

Time to create: **25:00** (approx.)

In this chapter, you will learn how to...

- Build a simple rectangular dielectric filled waveguide using the XFDTD solid modeling techniques
- Use the XFDTD material library
- Create a waveguide interface to transmit and receive energy
- Solve for and display the two lowest order TE modes of the waveguide
- Use and display field results from a solid part sensor

This example is taken from Microwave Engineering 4th edition by David M. Pozar Chapter 3 Example 3.1 on page 116. We will create a Teflon filled, copper K-band waveguide operates at 19.44 GHz.

## 5.1 Getting Started

First, a few Project Properties are set up for the waveguide project.

- The PROJECT PROPERTIES EDITOR opens automatically whenever XFDTD is started or when a new project is created. If the editor is not currently visible, double-click on PROJECT at the top of the PROJECT TREE to open the editor.
- Our K-band waveguide corresponds to the dimensions of WR-42, which has an operating frequency of 18 GHz to 26.5 GHz. However, the field distributions in the reference book correspond to single frequency operation, 19.44 GHz. So we will set our frequency range of interest to this specific frequency. On the FREQUENCY RANGE OF INTEREST tab of the PROJECT PROPERTIES EDITOR, specify a single frequency of interest by setting both the MINIMUM and MAXIMUM controls to “19.44 GHz”.
- Navigate to the DISPLAY UNITS tab, where a few relevant units should be set:
  - Set ANGLE to “degrees ( $^{\circ}$ , deg)”
  - Set FREQUENCY to “gigahertz (GHz)”
  - Set LENGTH to “centimeters (cm)”
- Press DONE on the PROJECT PROPERTIES EDITOR.

## 5.2 Creating the Waveguide Geometry

The waveguide conductor will be created using a Cuboid and then shelled to make it hollow with open ends.

### 5.2.1 Modeling the Waveguide

First we will create the copper tube geometry.

- At the top of the Geometry window click on **CREATE NEW > CUBOID**.
- Type “WR-42” into the NAME entry.
- Type “1.07 cm” for the WIDTH, “0.43 cm” for the DEPTH and “10 cm” for the HEIGHT.

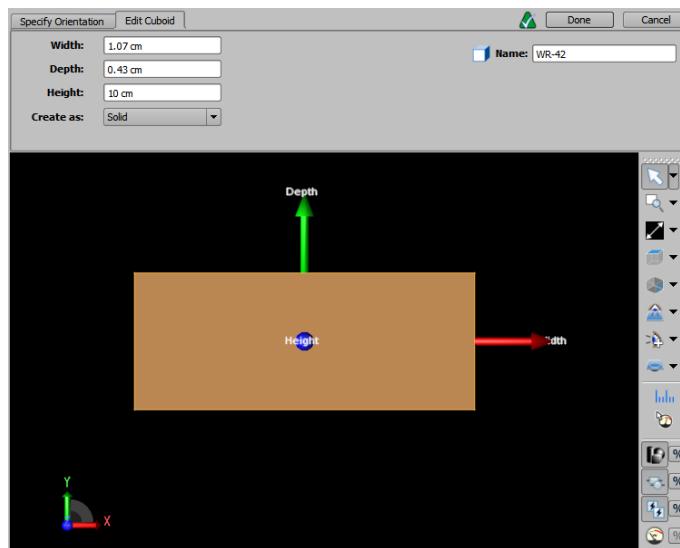


Figure 5.2: Creating Cuboid

- Click the DONE button.

In order to change the solid box we created into a hollow shape we will use the **GEOLOGY** window click **MODIFY > SHELL**.

- Select the “WR-42” cuboid that was previously created.
- Navigate to the **SELECT OPEN FACES** after “WR-42” has been selected.
- Select the end face pointing in the +Z direction using the mouse.
- Rotate your view so you can see an unobstructed view of the -Z face.
- Hold the **CTRL** key and select the -Z face, both faces should now be selected.

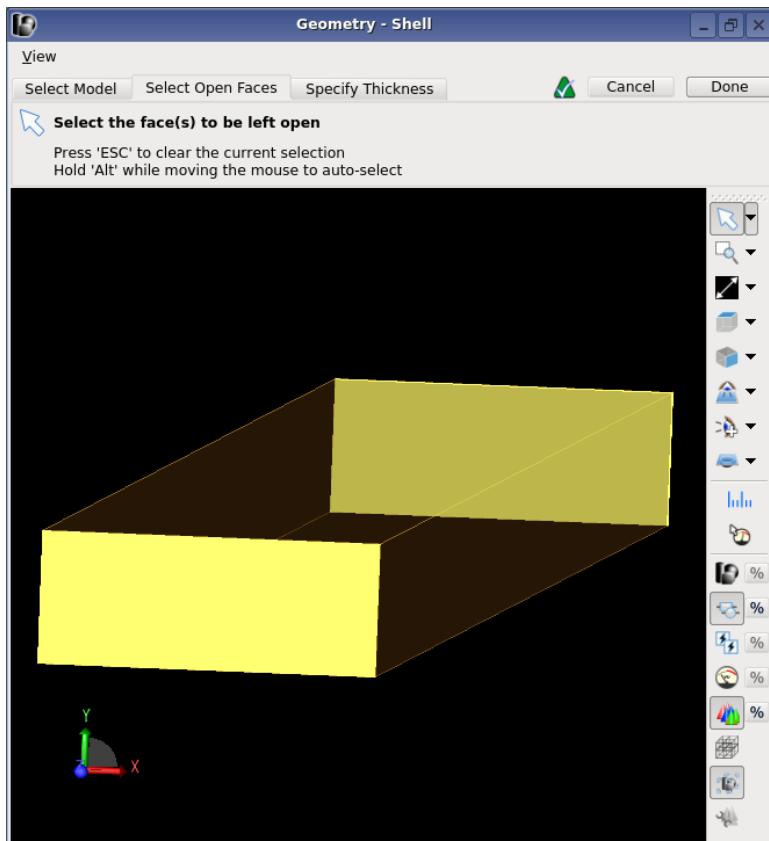


Figure 5.3: Selecting Open Faces for Shell Feature

- Navigate to SPECIFY THICKNESS step
- Specify a SHELL THICKNESS of “0.1 cm” and press DONE

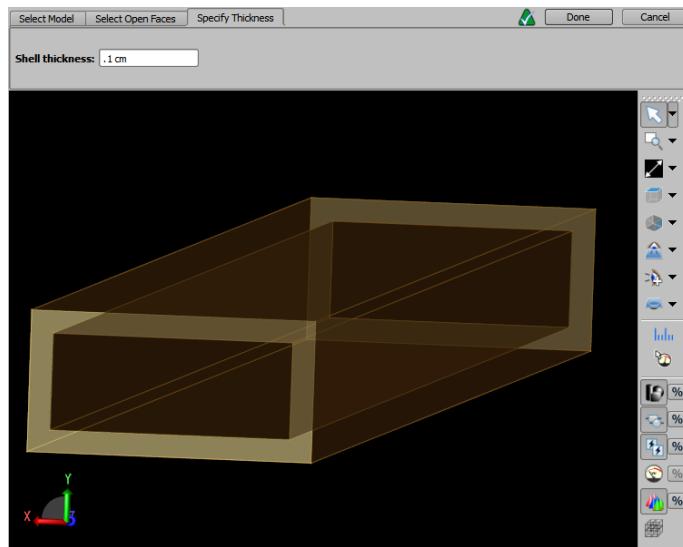


Figure 5.4: Specifying the Shell Thickness

Now we will create the part representing the dielectric fill also using the CUBOID primitive.

- At the top of the Geometry window click on CREATE NEW > CUBOID.
- Type “Dielectric Insert” into the NAME entry.
- Type “1.07 cm” for the WIDTH, “0.43 cm” for the DEPTH and “10 cm” for the HEIGHT.

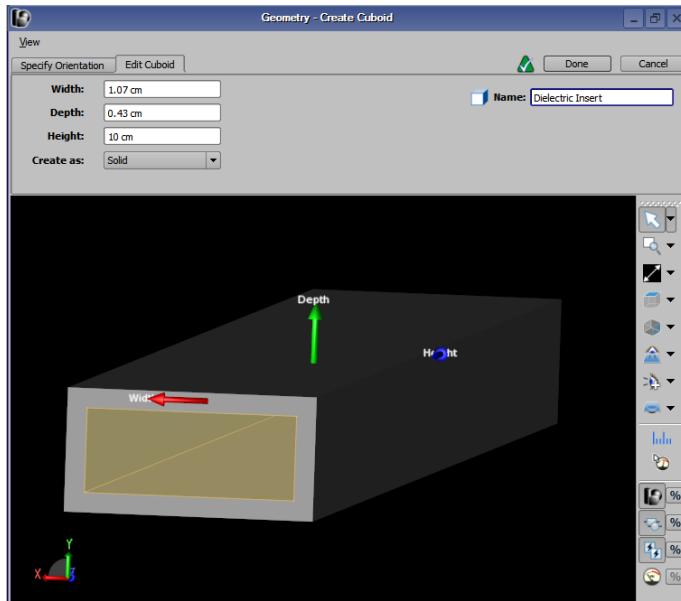


Figure 5.5: Creating a Cuboid for the Dielectric Insert

- Click the DONE button.

### 5.3 Creating Materials

For this project we will use two materials. We will pull in Copper from the materials library, and we will create a new material Teflon which we will define from the relative permittivity and loss tangent provided in the reference.

#### Defining COPPER

- In the WORKSPACE WINDOWS TOOLBAR located on the right hand side of the XF UI choose the LIBRARIES window.
- The default view of the LIBRARIES window is shown in Figure 5.6. XFdtd has several built in materials LIBRARIES which appear in the upper left portion of the LIBRARIES window. The upper right portion of the LIBRARIES window contains filters which restrict the results displayed in the lower portion of the window to only display those types which match the filters. In addition to the built in libraries you can create your own library which you can populate with any object or definition type that appears in the filter pane. This includes CAD parts, circuit components, materials etc.
- Select the “Materials- Pure Metals” library in the LIBRARIES pane.
- Make sure that “Materials” is highlighted in the FILTERS pane.

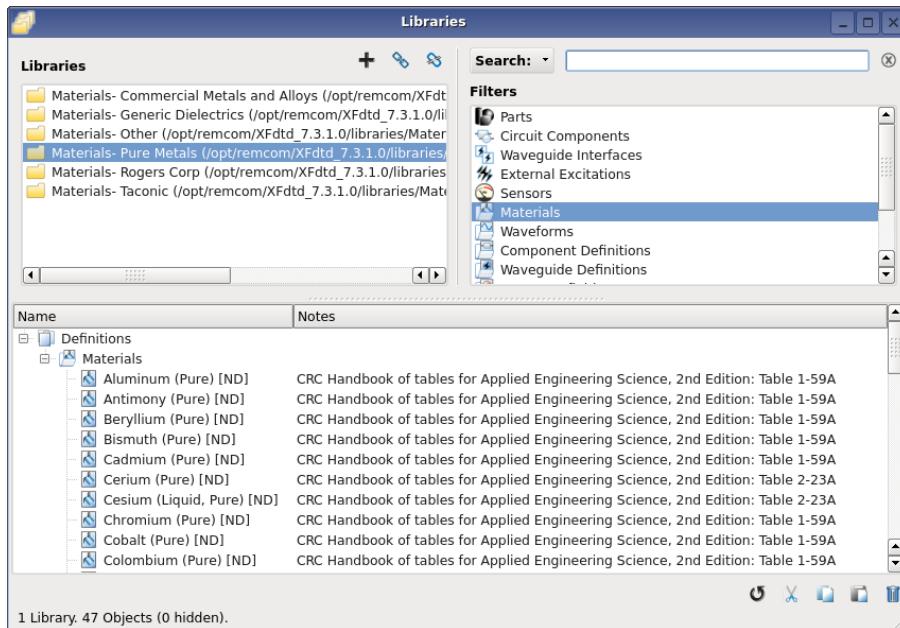


Figure 5.6: Libraries Workspace Window

- All materials from the “Materials- Pure Metals” are now visible in the lower pane. Type “copper” into the SEARCH editor to further limit the number of displayed materials.
- Find “Copper (Pure) [ND]” in the lower pane and drag it into the PROJECT TREE on the DEFINITIONS: MATERIALS.

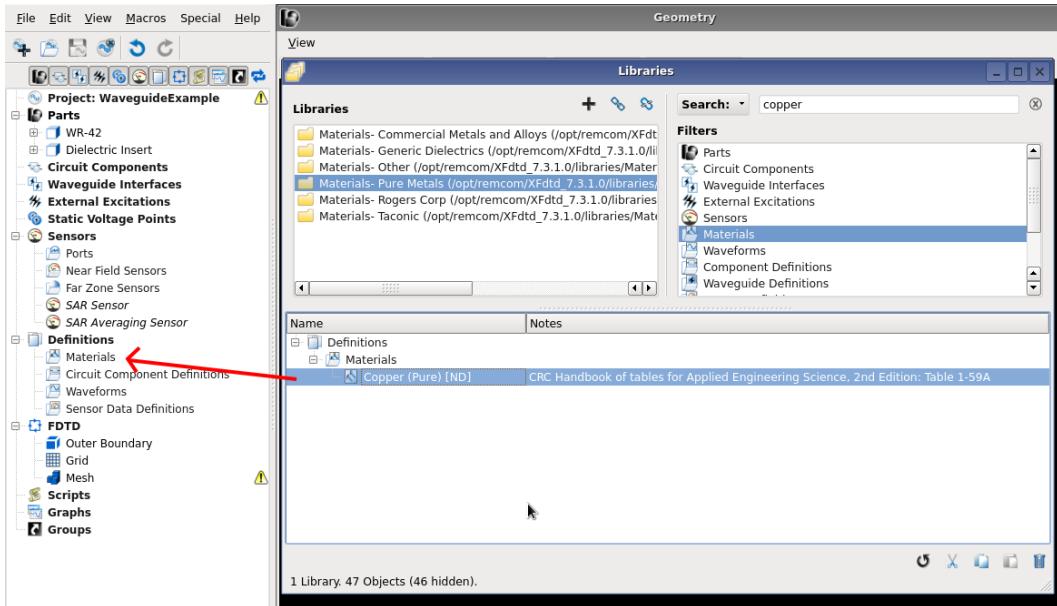


Figure 5.7: Pulling in Material Definition from Materials Library

You will notice there is an error symbol next to the material that was pulled into the project. If you place your mouse over the error a tool tip will say “Copper (Pure) [ND]: Evaluation frequency for surface conductivity correction is invalid”. This copper material has frequency dependent properties necessitating the definition of an evaluation frequency. If you double click on COPPER (PURE) [ND] in the PROJECT TREE the MATERIALS EDITOR Window will appear as in Figure 5.8. There is a red box that contains the parameter “materialFrequency” which is not defined in the PARAMETERS window. Replace this parameter with a value of “19.44 GHz”. After the EVALUATION FREQUENCY is defined the material will become valid as shown by the green check in the upper left.

→ Then finish editing by clicking DONE.

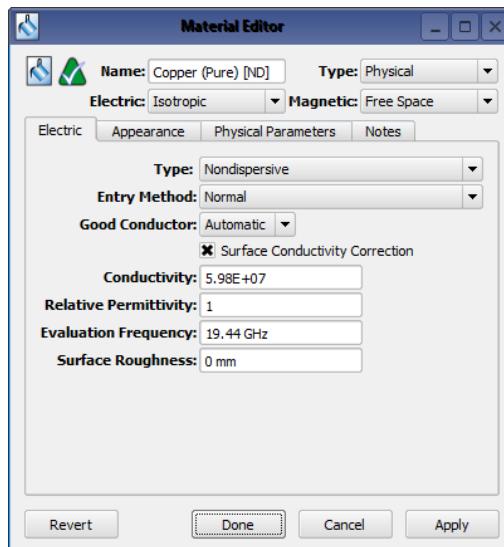


Figure 5.8: Copper (Pure)[ND] from the Materials Library

### Defining TEFLO

Since the relative permittivity of “2.08” and the loss tangent of “0.0004” were defined in the reference example we will define this material with those properties directly. The reference defines these values at “10 GHz”, but we will evaluate them at “19.44 GHz”. We can do this because the loss tangent should be relatively stable over frequency. Therefore, using the loss tangent to calculate the conductivity at “19.44 GHz” based on the data at “10 GHz” should provide an accurate result.

- In the PROJECT TREE under DEFINITIONS, right click on MATERIALS and choose NEW MATERIAL DEFINITION. By default XFdtd will place an entry under MATERIALS and immediately have the user edit the material name.
- If the MATERIAL EDITOR for the newly created material is not already open, open it by double-clicking on the node created in the MATERIALS in the PROJECT TREE
- Change the NAME entry to “Teflon”
- From the pull-down menu next to ENTRY METHOD choose the “Loss Tangent” item. This entry method will have fields for RELATIVE PERMITTIVITY, LOSS TANGENT, and EVALUATION FREQUENCY.
- Fill in the values for RELATIVE PERMITTIVITY of “2.08”, LOSS TANGENT of “0.0004” and EVALUATION FREQUENCY of “19.44 GHz”. This should match Figure 5.9.
- If desired, navigate to the APPEARANCE Tab and change the FACE COLOR to Dark Cyan.

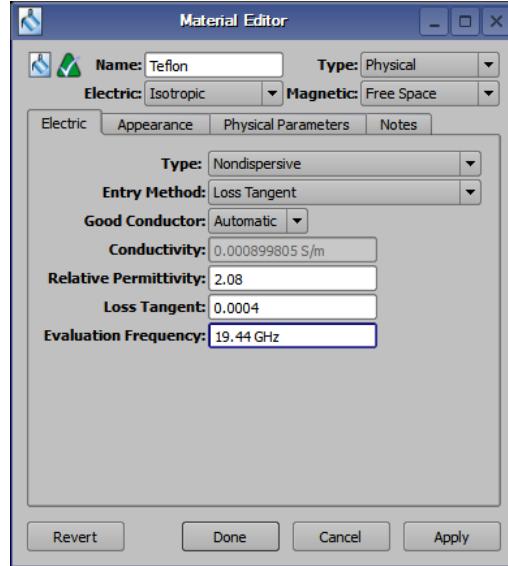


Figure 5.9: Material Properties of Teflon

- Press DONE to finish editing the Teflon material.

## Assigning Materials

The next task is to assign these materials to our waveguide geometry. Do this by:

- Left click and hold on “Copper (Pure) [ND]” under MATERIALS in the PROJECT TREE. Drag and drop the material onto the “WR-42” model in the PARTS branch.
- Similarly, left click and hold on “Teflon” under MATERIALS in the PROJECT TREE. Drag and drop the material onto the “Dielectric Insert” model in the PARTS branch.
- Expand the parts under the PARTS by pressing the (+) symbol. The material associated with each part is displayed as shown in Figure 5.10

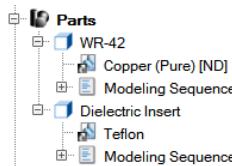


Figure 5.10: Parts List with Associated Materials

## 5.4 Defining the Outer Boundary

The entire simulation space is contained within the waveguide and energy should not propagate to the outer boundary. For this reason we can change the boundaries to PEC as it is more computationally efficient than the absorbing boundaries.

- Double-click on the FDTD: OUTER BOUNDARY branch of the PROJECT TREE to open the OUTER BOUNDARY EDITOR.
- Set the outer boundary properties as follows:
  - BOUNDARY: “PEC” for all boundaries

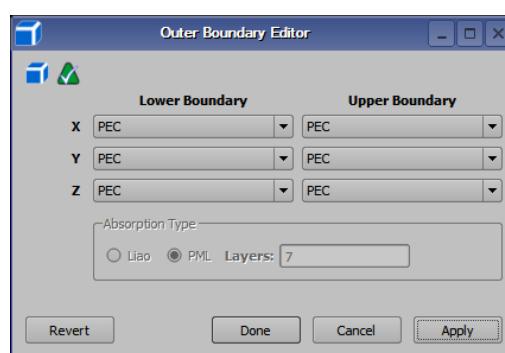


Figure 5.11: Defining the Outer Boundary for the Waveguide Simulation

- Click DONE to apply the outer boundary settings.

## 5.5 Defining the Grid

Now we will define characteristics of the calculation grid.

- Double-click on the FDTD: GRID branch of the PROJECT TREE to open the GRID EDITOR.
- The default settings on the CELL SIZE tab are sufficient to obtain an accurate result.
- Navigate to the EXTENTS tab. We will remove free space padding from all sides of the simulation space, so that the PEC boundary conditions are applied exactly on the outer boundaries of the geometry. Set all six FREE SPACE PADDING (BASE CELLS) values to “0” as shown in Figure 5.12.
- Click DONE to apply the grid settings.

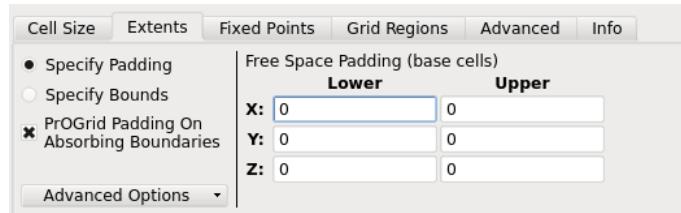


Figure 5.12: Defining the grid extents on the Grid Editor

At this point, the base cell sizes and extents are set for the project, but we need to check the mesh to make sure the geometry is properly represented. In particular, the cross-sectional dimensions of the waveguide must be resolved exactly in order to accurately predict the propagation characteristics of each waveguide mode.

- In the VIEW TOOLS toolbar, select the VIEW FROM -Z (BOTTOM) orientation.
- On the right hand side of the GEOMETRY window press the TOGGLE MESH VIEWING CONTROLS button.
- The mesh viewing controls will appear at the bottom of the screen as shown in Figure 5.13.
- Make sure MESH CUTPLANES is selected in the lower left corner.
- Check the XY PLANE in the upper left corner.
- Move the slider, or use the arrows, or type into the box to move the view to slice “0” to display an XY mesh slice giving a cross-section of the waveguide at the lower Z end. As shown in Figure 5.13, the mesh edges line up exactly with the boundary between the Copper and Teflon materials. However, there is no guarantee that this will remain true if any changes are made to the geometry or grid settings. For example, changing the MIN CELLS PER WAVELENGTH setting to “18” on the GRID EDITOR will cause the meshed representation of the Copper shell to be slightly larger than modelled, reducing the inner waveguide dimensions. To prevent this from happening, we can apply Fixed Points to the WR-42’s Copper shell.

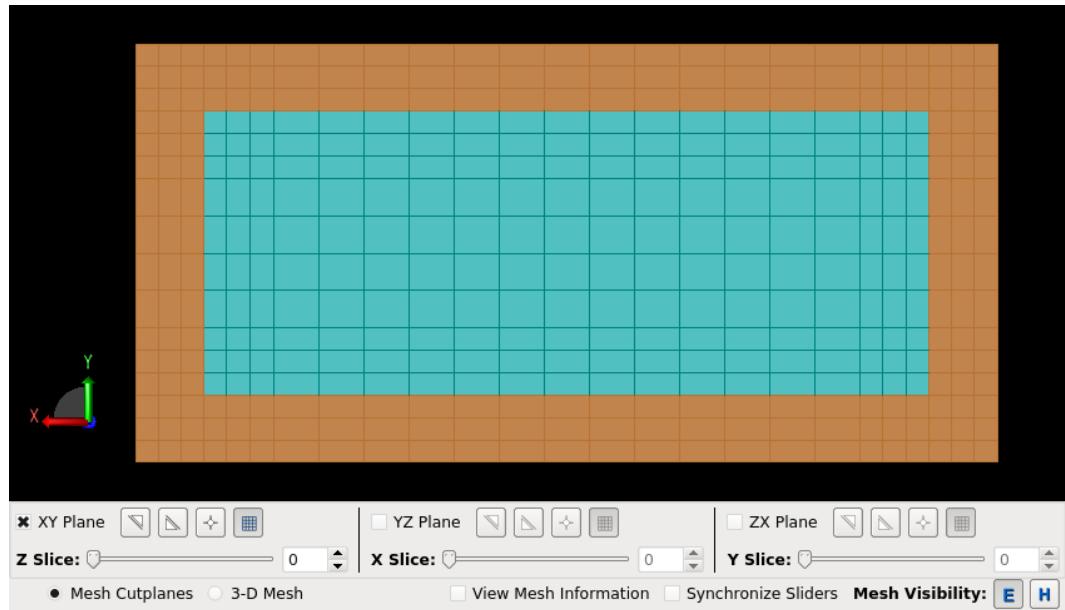


Figure 5.13: Mesh Viewing Controls

- In the PROJECT TREE under the PARTS branch right click on the “WR-42”, navigate to GRIDDING/MESHING and choose GRIDDING PROPERTIES as shown in Figure 5.14.

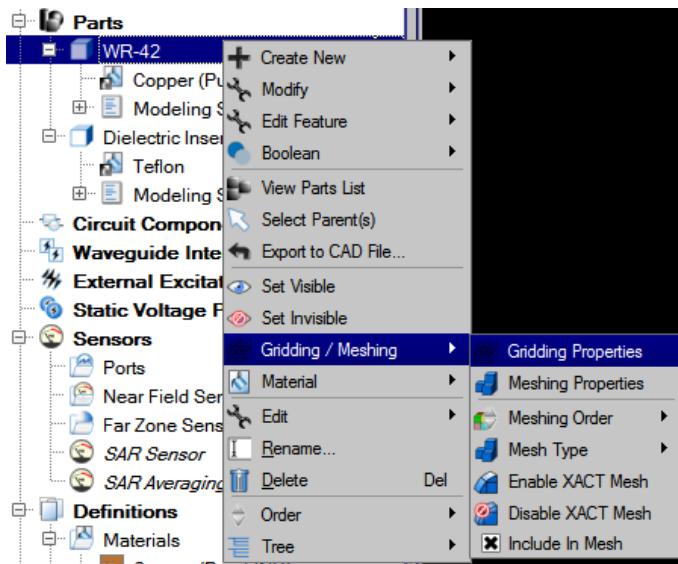


Figure 5.14: Gridding Properties Menu Item

- Check the box next to USE AUTOMATIC FIXED POINTS.
- One item to note is that at the top of Figure 5.15 there is a check box for USE AUTOMATIC GRID REGIONS. This option allows you to apply a smaller grid to this part than the base cell size. This can help speed up calculations and reduce the overall memory usage in cases where a smaller mesh size is needed only on a particular part in order to resolve gaps, traces, etc.
- Press DONE to apply the changes to the gridding properties.
- Press the TOGGLE MESH VIEWING CONTROLS button to hide the mesh.

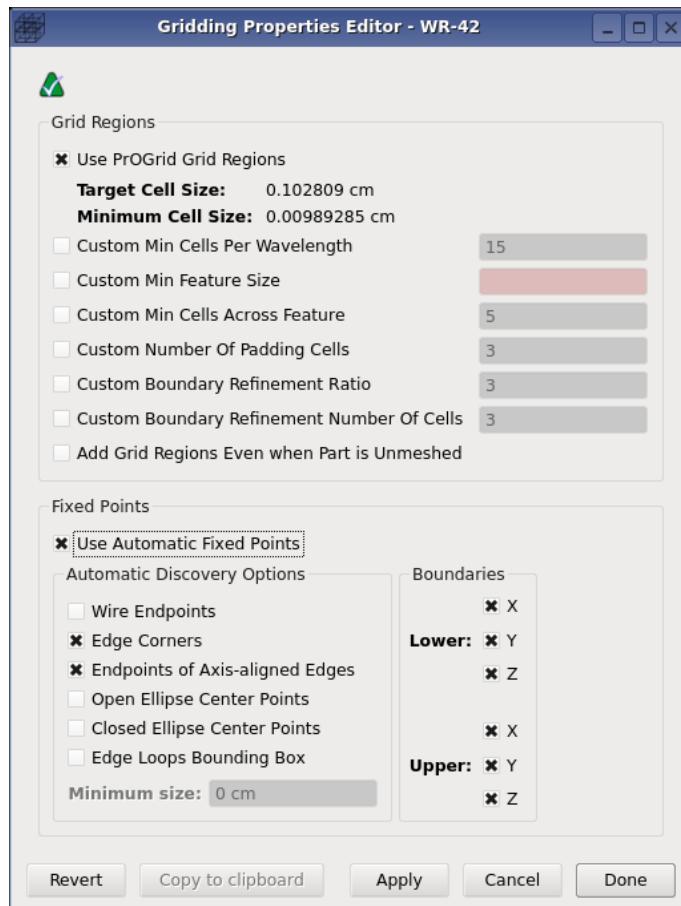


Figure 5.15: Gridding Properties for WR-42

## 5.6 Adding a Waveform

The next step is to define a waveform that will be used to excite the waveguide. Do this by:

- In the PROJECT TREE right click on WAVEFORMS and choose NEW WAVEFORM DEFINITION
- Open the WAVEFORM by double clicking on the node in the PROJECT TREE. It should look like 5.16 and was setup automatically based on the project's frequency range of interest.

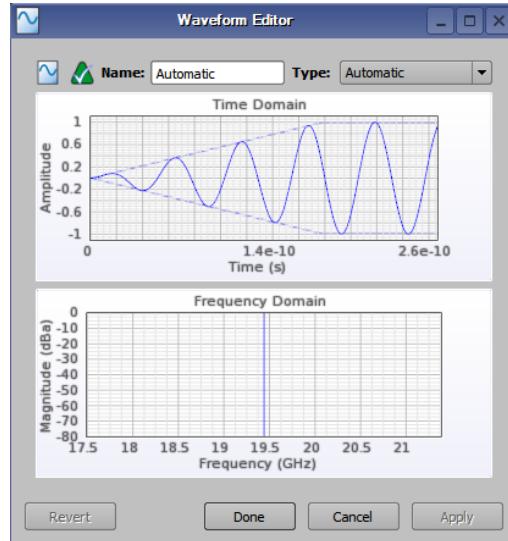


Figure 5.16: Automatic Waveform

## 5.7 Add a Waveguide Interface and Evaluate the Modes

This section will discuss adding a waveguide interface and evaluating the modes. The grid must be set up prior to this step, as in Section 5.5 so that XFdtd can properly evaluate the modes.

We will add two waveguide interfaces, one to excite the waveguide at one end, and the other at the opposite end to receive energy.

### Add Excitation Waveguide Interface

- Locate the WAVEGUIDE INTERFACES node in the PROJECT TREE. Right click on WAVEGUIDE INTERFACES and choose NEW MODAL WAVEGUIDE INTERFACE as shown in Figure 5.17. A waveguide interface editor will be shown in the GEOMETRY WINDOW.

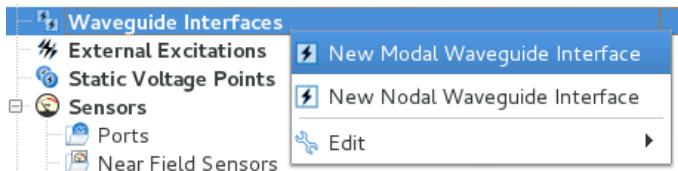


Figure 5.17: Waveguide Interfaces Project Tree Location

- Type “WR-42 Excitation” in the NAME box.
- In the WAVEFORM pull down menu select the “Automatic” Waveform.

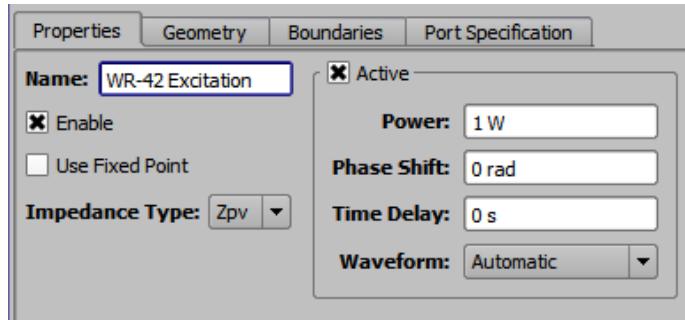


Figure 5.18: Waveguide Interface Properties Tab

- Click on the GEOMETRY tab. We will leave the center of the waveguide interface at (0,0,0) which is at the end of the waveguide. The values in the EXTENSIONS group determine the size of the waveguide interface. Set the values in the EXTENSIONS box as shown in Figure 5.19.

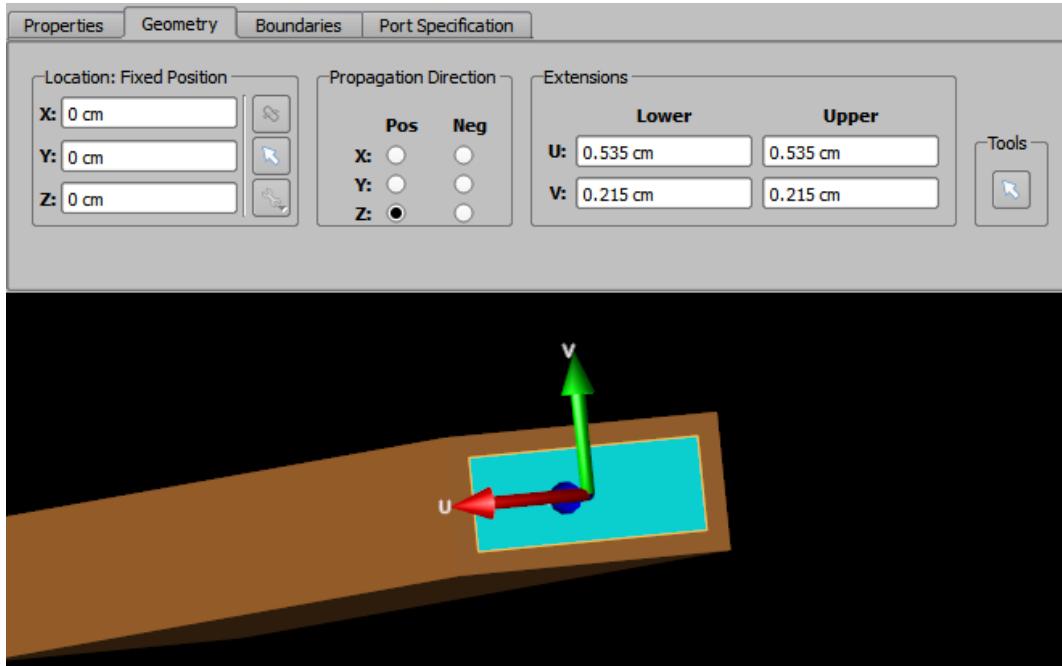


Figure 5.19: Waveguide Interface Geometry Tab

We will not change any settings in the BOUNDARIES tab. All boundaries will be left as PEC.

- Click on the PORT SPECIFICATION tab. Click the button twice on the upper left of the screen to add two ports to the waveguide interface. Each new port added will be of successively increasing order. In this example we will add the first two modes corresponding to the two lowest order modes in the WR-42 waveguide. Notice that these ports have no data associated with them because we have not yet computed the associated modes.

Select the first port WAVEGUIDE PORT. Left click or press **F2** to rename this port to “TE10” because the port will use a mode that corresponds to the TE<sub>10</sub> mode of a rectangular waveguide. Do the same for the second port but name this mode “TE20”.

- We will now compute the modes associated with each defined port. Change the EVALUATION FREQUENCY in the upper right corner to “19.44 GHz”. This is the frequency that the eigensolver will use to compute the requested modes. 19.44 GHz corresponds to our frequency of interest. As a general rule it is best to us an evaluation frequency that is near your frequencies of interest because the accuracy of results decrease as one moves away from the evaluation frequency in non-TEM modes.
- In the **PORT INFO** tab there is a MODE entry. The inputs for this entry are integer indices starting at zero corresponding to the modes position when ordered from the lowest to highest in order of cutoff frequency. We are currently interested in modes 0 and 1.
- The reference source contains data for the TE<sub>10</sub>, TE<sub>20</sub>, TE<sub>01</sub>, TE<sub>11</sub>, TM<sub>11</sub>, TE<sub>21</sub> and TM<sub>21</sub> but for this example we'll focus on the TE<sub>10</sub> and TE<sub>20</sub> modes.
- Now click COMPUTE MODES to calculate the fields and information for each of the defined modes that exist at or below the evaluation frequency. If the interface has not yet been added to the project answer YES to add it before computing the mode information.

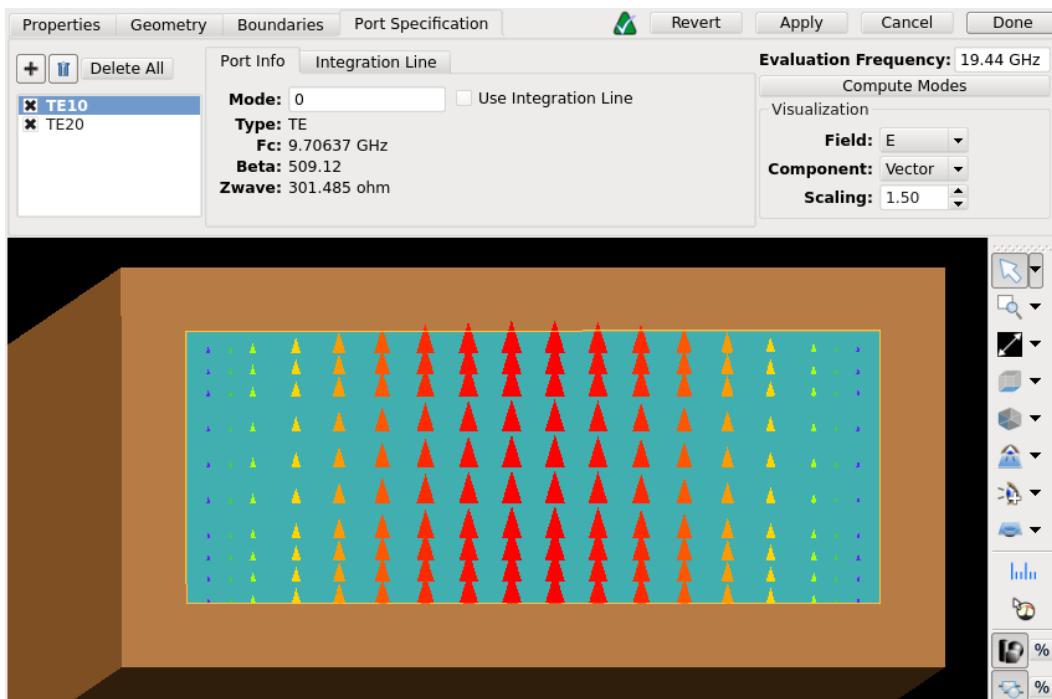


Figure 5.20: Waveguide Port Specification Tab with Mode Results

- The fields for the selected modes will now be shown in the GEOMETRY window and should be as shown in Figure 5.20. The statistics displayed in the **PORT INFO** tab are TYPE OF MODE, which is TE in this case, the Cutoff Frequency (Fc), the propagation constant (BETA) and the Wave

Impedance ( $Z_{WAVE}$ ) value. This is the pre-simulation data computed by the eigensolver.

- Figures 5.20 and 5.21 show the field distributions from the eigensolver on the selected waveguide interface.
- To compare the field distributions from XFdtd to the predicted results, the interested reader should refer to figure 3.9 in the reference.

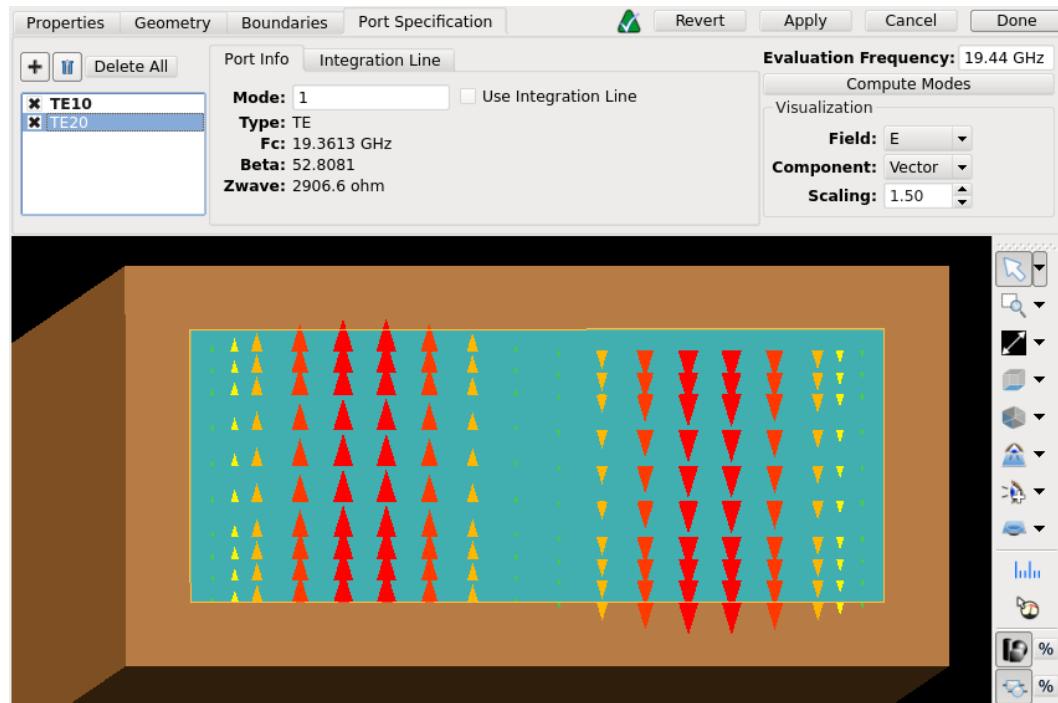


Figure 5.21: Calculated TE<sub>20</sub> Mode

- Press DONE to finish editing the waveguide interface.

## Adding a Waveguide Interface to Receive Energy

- Right click on WAVEGUIDE INTERFACES in the PROJECT TREE and choose NEW MODAL WAVEGUIDE INTERFACE
- Enter “WR-42 Receiver” in the NAME box
- Next to the NAME box uncheck the box labeled ACTIVE.
- Navigate to the GEOMETRY tab.
- Orient the view of the GEOMETRY so that the +Z side of the waveguide structure is visible.
- In the LOCATION group next to the box that indicates the Y position there is a button that has this arrow . Hovering over the arrow brings up a tool tip that states “Specify this position by selecting a reference point on existing geometry”. Click this button.

- Hover over the Dielectric Insert geometry in the GEOMETRY window. Hovering over this object will show a tool tip with additional picking options. Press C to center the position on the center of the waveguide insert face, then left-click on the Dielectric Insert. The LOCATION should now be updated with the coordinates of the center of the face.
- In the PROPAGATION DIRECTION group, choose the negative Z direction. We need to point the waveguide port into the waveguide. If we point the waveguide port out of the waveguide, any fields that strike the back side of the waveguide port will be reflected off as if they struck a PEC object.
- Update the EXTENSIONS as shown in Figure 5.22.

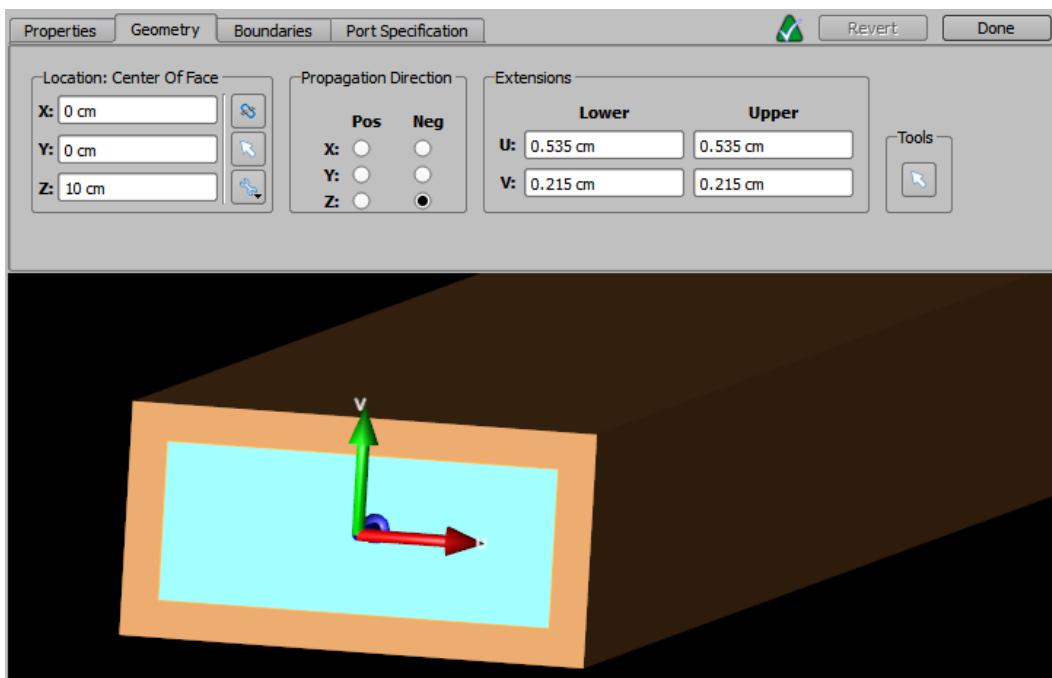


Figure 5.22: Waveguide Receiver Interface Geometry Tab

- Navigate to the PORT SPECIFICATION tab. Refer to the previous section for details on how to add waveguide ports.
- Click the + button to add a waveguide port.
- Change the MODE entry in the PORT INFO to “1” and rename it to “TE20”.
- Change the EVALUATION FREQUENCY to “19.44 GHz” corresponding to the excitation frequency.
- Press COMPUTE MODES. You will be asked to add this waveguide to the project if you have not already done so. Select YES. Once the mode computation has complete select the port to ensure the desired field distribution and port information.
- Click DONE

## 5.8 Requesting Results

To compare our results to the field distributions in the text book example we will need 3 different field cuts. Unfortunately, not all of the field slices from the different modes are taken from the same plane. Instead of creating 3 different planar sensors, we will create a solid part sensor so all possible field data is captured.

### Adding a Solid Sensor Definition

- First create the solid sensor definition. Right-click on the DEFINITIONS: SENSOR DATA DEFINITIONS branch of the PROJECT TREE. Choose the NEW SOLID SENSOR DEFINITION from the context menu.
- Set the properties of the solid sensor definition as follows:
  - NAME: Field Sampling
  - FIELD VS. FREQUENCY: Steady E and Steady H

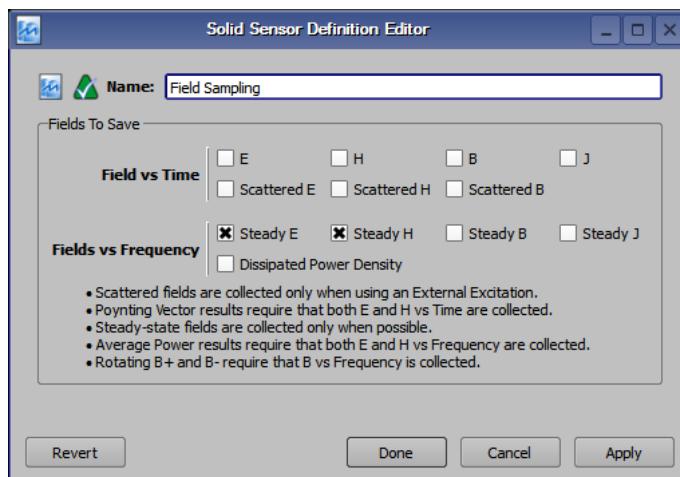


Figure 5.23: Solid Sensor Definition

- Press DONE to finish editing the Solid Sensor Definition.

### Adding a Solid Part Sensor

- Right click on the SENSORS: NEAR FIELD SENSORS branch of the PROJECT TREE. Select NEW SOLID PART SENSOR.
- The SELECT MODEL tab of the SOLID PART SENSOR editor allows you to select the model to attach the sensor to. In the PROJECT TREE under PARTS select Dielectric Insert.
- Navigate to the PROPERTIES tab and name the sensor "Fields in Waveguide".
- The SENSOR DEFINITION should be "Field Sampling".

- Press DONE to add the sensor to your project.

## 5.9 Running the Simulation

If you have not already saved your project, do so by selecting FILE > SAVE PROJECT. Once the project is saved, a new simulation can be created to send to the XFsolver.

- Open the SIMULATIONS WORKSPACE WINDOW.
- If you have not already done so, click on the QUEUE button on the top. This menu allows you to choose XStream® acceleration if you have a CUDA enabled GPU, or use multiple processors.
- Click the FDTD button in the upper left to set up a new simulation.
- In the SETUP S-PARAMETERS tab check the COMPUTE S-PARAMETERS box. Uncheck the box next to WR-42 EXCITATION:TE10 [PORT 1], and check the box next to WR-42 EXCITATION:TE20 [PORT 2], as shown in Figure 5.24. We do this to excite the TE<sub>20</sub> mode.

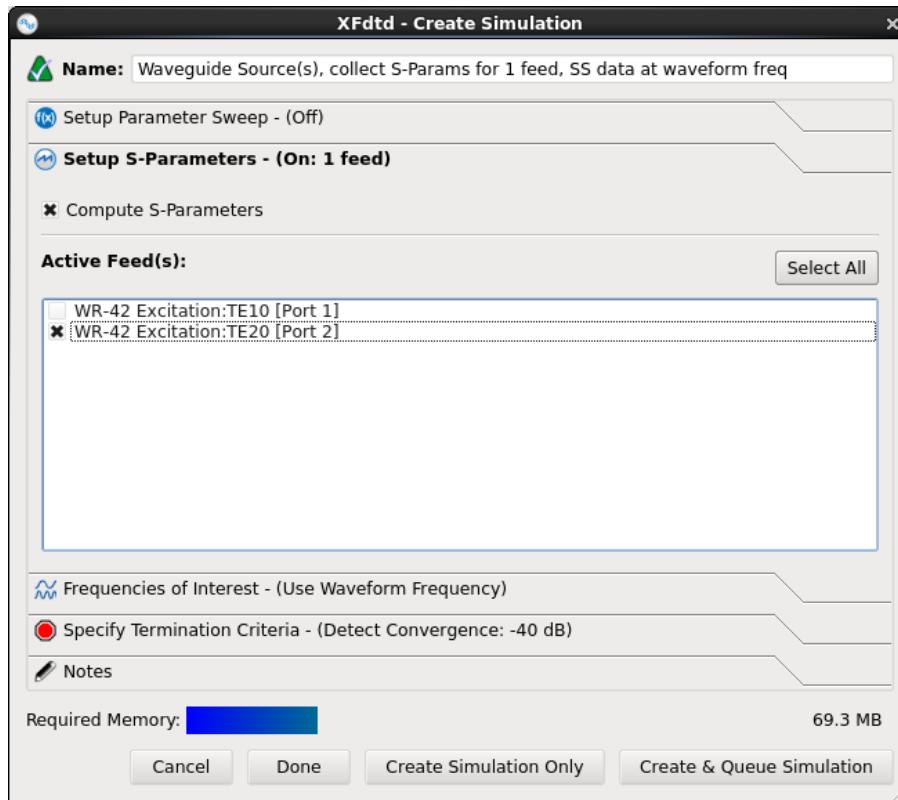


Figure 5.24: Setting up S-Parameters on the New Simulation Window

- Navigate to the FREQUENCIES OF INTEREST tab. This should already be set up to collect steady state data at the waveform frequency of 19.44 GHz. On the OPTIONS sub-tab, uncheck the SAVE DATA FOR POST-SIMULATION FAR ZONE STEADY-STATE PROCESSING, as shown in 5.25 No

steady-state far zone data is generated for simulations with six PEC boundary conditions, as is the case here.

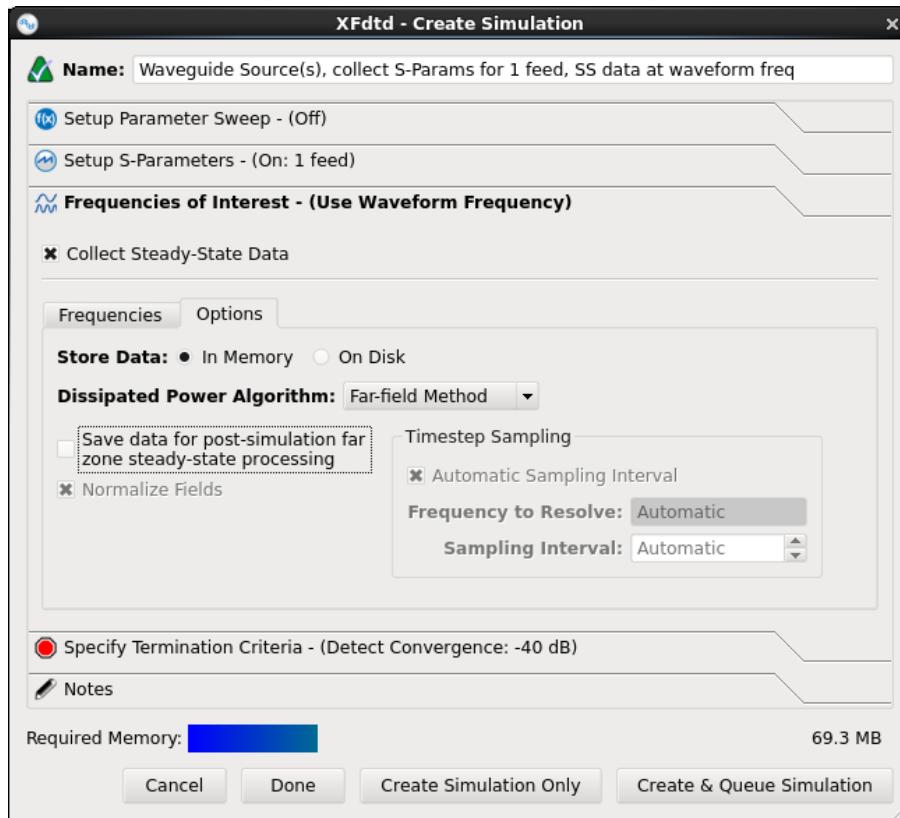


Figure 5.25: Disabling far zone data collection on the New Simulation Window

- Press CREATE & QUEUE SIMULATION Button. While the simulation is running click on the OUTPUT tab to view the convergence criteria.

## 5.10 Viewing the Results

All of the results are accessed from the RESULTS workspace window. Keep in mind that each heading in this window can be customized by right clicking and choosing a new option from the pull down menu.

When the simulation has completed, the steady-state results we requested will be ready to be viewed.

### Viewing Field Distributions

- Open the RESULTS workspace window.
- In the second column labeled SENSOR choose our FIELDS IN WAVEGUIDE sensor.

- In the DOMAIN column choose DISCRETE FREQUENCIES.
- In the RESULT TYPE column choose ALL.
- Double click on E-FIELD (E) in the bottom pane. This will bring up the FIELD VIEWER in the GEOMETRY window.
- The way this result works is that you save all the fields in the 3D geometry. Then you choose the X, Y, and Z ranges to display a 2-D cut through this volume. In the middle column on the toolbar in Figure 5.26 you can see yellow bars indicating the ranges of the X, Y, and Z values. By default we are showing all values of Y and Z, and a single value in X indicating that we are showing the YZ plane.
- Click on the  button next to the X Range.

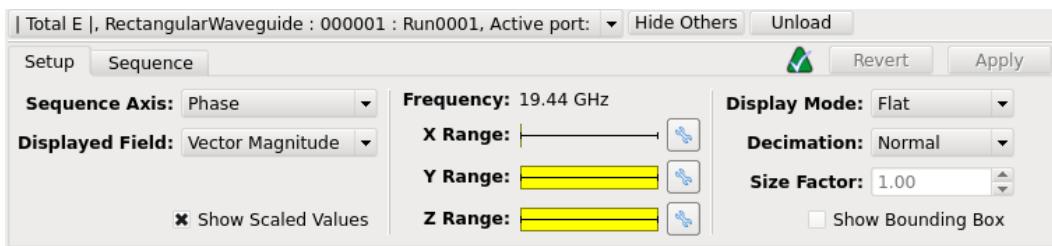


Figure 5.26: Field Control Panel

- Slide the min and max values to slice 6 as shown in Figure 5.27. The fields displayed in the reference example for the TE<sub>20</sub> mode are located about one fourth of the way into the waveguide.

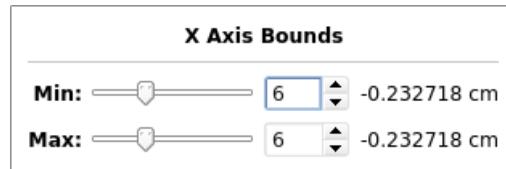


Figure 5.27: X Bounds for Solid Sensor Results

- For the DISPLAY MODE choose “Vector Field” to display vector arrows.
- For DECIMATION choose “Finest”.
- Press APPLY to update the display with the changes.
- Press the  TOGGLE PARTS VISIBILITY button to hide the solid geometry.
- Reorient the view to  VIEW FROM -X (LEFT)

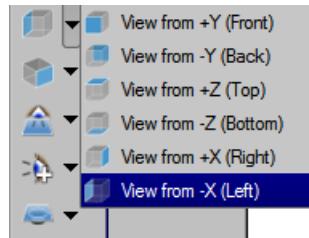


Figure 5.28: Change the Viewpoint to the View from the Left

- On the SEQUENCE tab, set the SHOWING editor to “54” to view E-field data at a  $270^\circ$  phase offset into the sinusoidal excitation.
- Comparing our result to figure 3.9 in the reference book, TE<sub>20</sub> mode, slice 2 we see the same distribution of field lines (Figure 5.29).

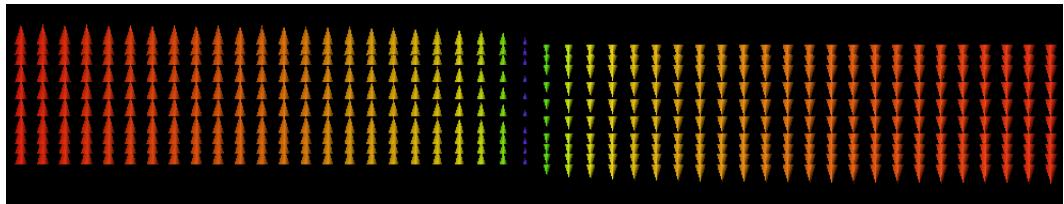


Figure 5.29: Solid Sensor Results in the YZ Plane (detail near the lower Z end of the waveguide, rotated  $90^\circ$ )

Now we will look at the fields in the XY plane of our waveguide.

- Select the button next to the Z-RANGE. Set both the MAX and MIN values to “72” to show a slice about halfway along the waveguide length.
- Select the button next to the X-RANGE. Set the MIN to “0” and the MAX to “19”, to show the full range of X.
- Press APPLY to update the view.
- Rotate the view to see the XY plane. You can also select VIEW FROM +Z (TOP) from the VIEW CONTROLS.
- Figure 5.30 shows the electric field lines in the XY plane of our waveguide. They match figure 3.9 in the reference.

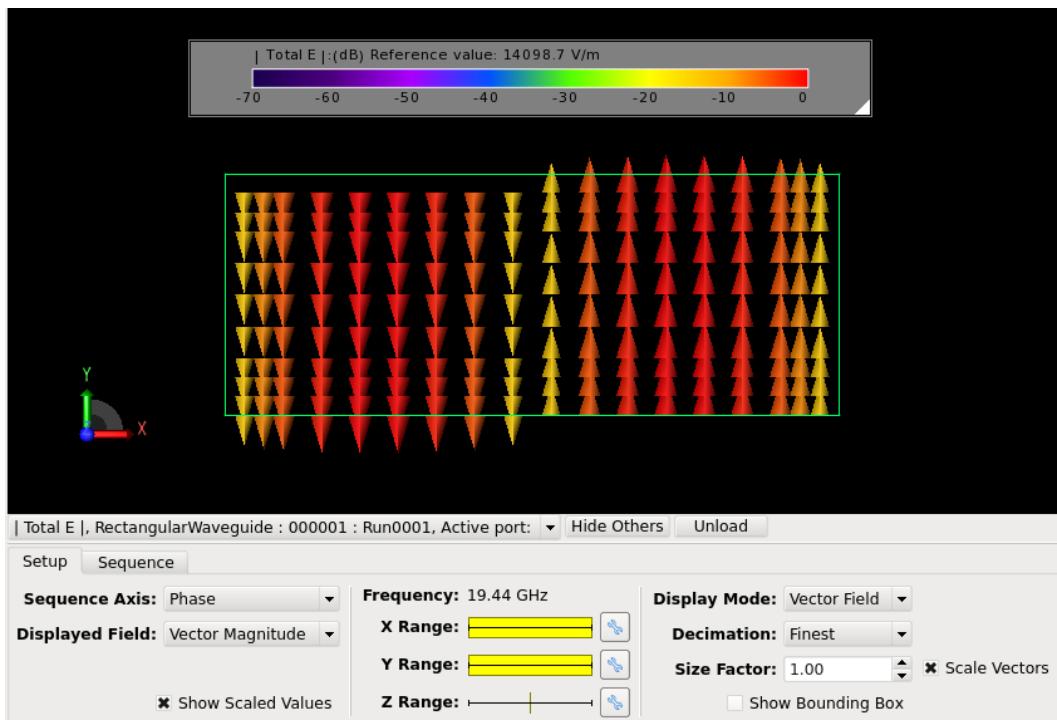


Figure 5.30: Solid Sensor Results in the XY Plane

- The last set of field lines we are comparing to are in the XZ plane. These are the magnetic field lines.
- Click UNLOAD to unload the results from the field control panel. Alternatively we could load the magnetic field lines at the same time and use the HIDE OTHERS button to only display one set of fields at a time. We will unload them to simplify this process and reduce confusion.
- In the bottom of the RESULTS window double click on H-FIELD (H). This will re-open the field control panel showing the new result.
- Click on the button next to Y-RANGE and enter “5” for both the MAX and MIN values.
- Click on the button next to X-RANGE and move the max slider to the end of the range which is “19”.
- Change DISPLAY MODE to “Vector Field”.
- Change DECIMATION to “Finest”.
- Press APPLY to update the display.
- Rotate the view to look at the XZ plane. Alternatively, you can also choose VIEW FROM +Y (FRONT) from the View Tools
- Right click on the scalebar at the top of the screen and choose PROPERTIES. On the left hand side in the LIMITS group uncheck AUTOMATIC RANGE and set the MINIMUM to “-40 dB”, then click the

DONE button. This will exaggerate the difference in the vector colors to more easily view what is happening in the figure.

- On the  SEQUENCE tab, set the SHOWING editor to “54” to view H-field data at a  $270^\circ$  phase offset into the sinusoidal excitation (at the same point in time where E-fields were viewed above).
- Comparing figure 3.9 in the reference to the zoomed in view of the fields shown in Figure 5.31 we see that in both cases, H-field vectors point inward toward the central nodal point from the top and bottom (+X and -X directions), and outward from the central nodal point from the left and right (-Z and +Z) directions.

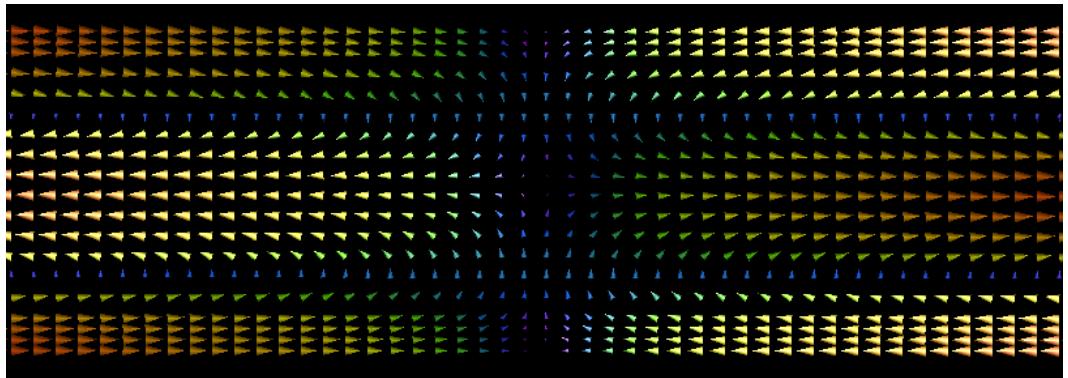


Figure 5.31: Magnetic Fields in the ZX Plane (detail near the center of the waveguide, rotated  $90^\circ$ )

More information on using waveguide interfaces can be found in the XFdtd Reference Manual Section 7.3.



## Chapter 6

### Example: A Monopole Antenna on a Conducting Box

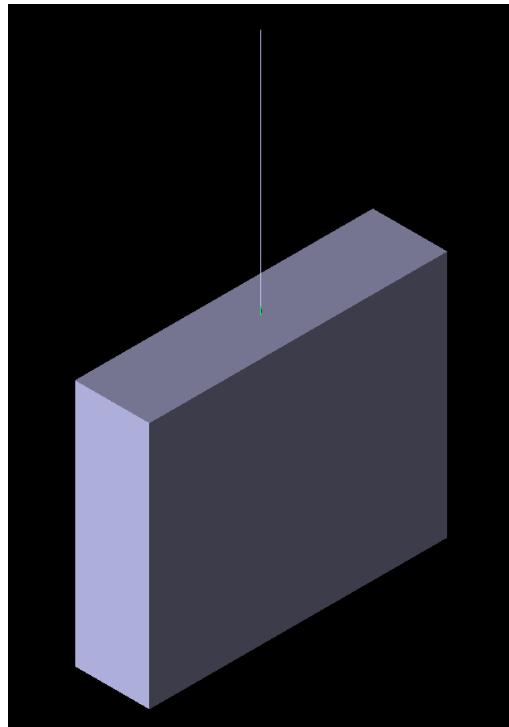


Figure 6.1: The Monopole Antenna on a Conducting Box

Time to create: 20:00 (approx.)

In this chapter, you will learn how to...

- build a monopole antenna using solid modeling techniques
- define the properties of the antenna environment
- add a feed to the antenna and simulate its effects
- add a surface sensor to the box and view the calculated surface current
- retrieve far zone results after running the calculation

In this example, a wire monopole is connected to a conducting box and fed at the junction. The radiation pattern will be calculated at a frequency of 1.47 GHz.

## 6.1 Getting Started

First, a few Project Properties are set up for the Monopole Antenna project.

- The PROJECT PROPERTIES EDITOR opens automatically whenever XFdtd is started or when a new project is created. If the editor is not currently visible, double-click on PROJECT at the top of the PROJECT TREE to open the editor.
- On the FREQUENCY RANGE OF INTEREST tab of the PROJECT PROPERTIES EDITOR, specify a single frequency of interest by setting both the MINIMUM and MAXIMUM controls to "1.47 GHz".
- Navigate to the DISPLAY UNITS tab, where a few relevant units should be set:
  - Set FREQUENCY to "gigahertz (GHz)"
  - Set LENGTH to "millimeters (mm)"
- Press DONE on the PROJECT PROPERTIES EDITOR.

## 6.2 Parameterizing the Project

For this example, we will parameterize the dimensions of the monopole antenna geometry so that any value can be easily changed in the PARAMETERS browser window.

- Open the PARAMETERS workspace window. Press the + icon to add a new parameter.
  - NAME: width
  - FORMULA: 60 mm
  - DESCRIPTION: width (x-direction) of Box
- Add parameters named "length", "height", and "wireLength" in the same manner, according to Figure 6.2.
- Press APPLY to add the parameters to the project.

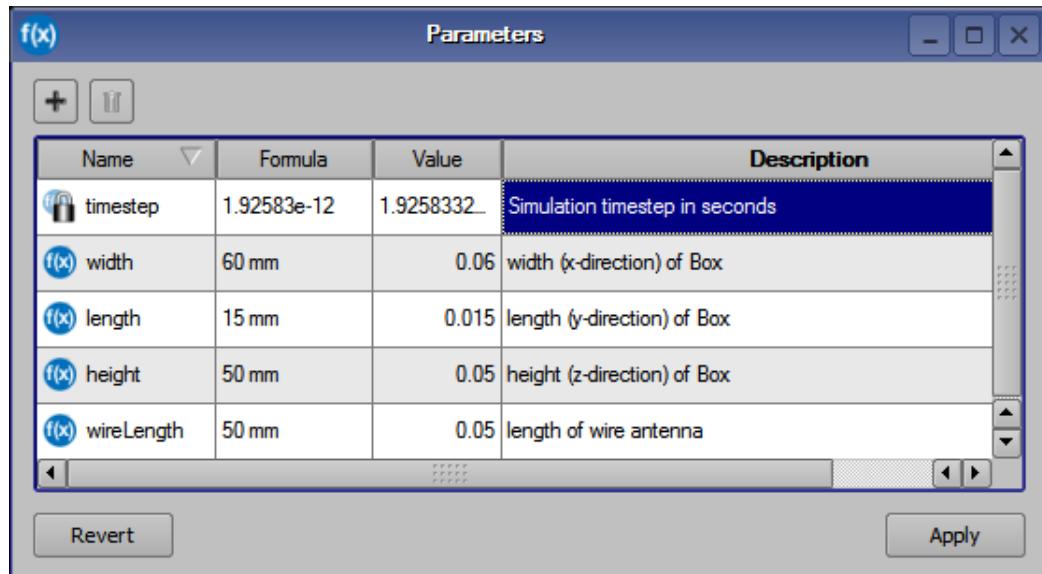


Figure 6.2: Defining parameterized values

## 6.3 Creating the Monopole Antenna Geometry

The Monopole Antenna geometry is created with a simple Box and a MONOPOLE antenna. The dimensions of the Box as well as the antenna's length will be defined with parameters.

### 6.3.1 Modeling the Box

First, we will create the rectangular substrate named Box. This object will use the parameters “length”, “width” and “height” for its dimensions with an extrusion in the  $+Z$  direction.

- Right-click on the PARTS branch of the PROJECT TREE. Choose CREATE NEW > EXTRUDE from the context menu.

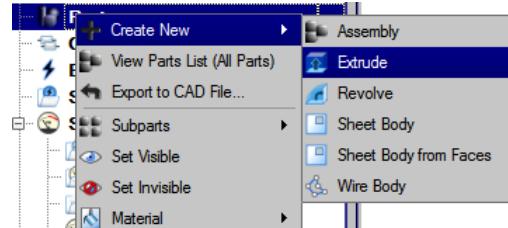


Figure 6.3: Selecting the geometry tool to perform an extrusion

- Type “Box” into the NAME box.
- Choose the RECTANGLE tool from the SHAPES toolbar, and draw a rectangle in the sketching plane. (Dimensions are unimportant.)

- Select the  DISTANCE constraint tool from the CONSTRAINTS toolbar.

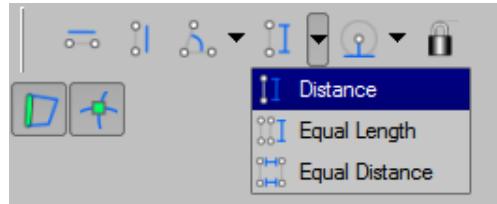


Figure 6.4: Selecting the Distance constraint tool

- Click on the left vertical side of the rectangle (it will turn blue), then move the mouse slightly to its left and click again.
- Type “length” in the dialog box, to set its value equal to the LENGTH parameter.
- Press **ENTER** to add the constraint.



Figure 6.5: Adding the length constraint to the rectangle

- Add a constraint to the bottom horizontal side of the rectangle, defining its value as “width”. The finished 2-D cross-section is seen in Figure 6.6.

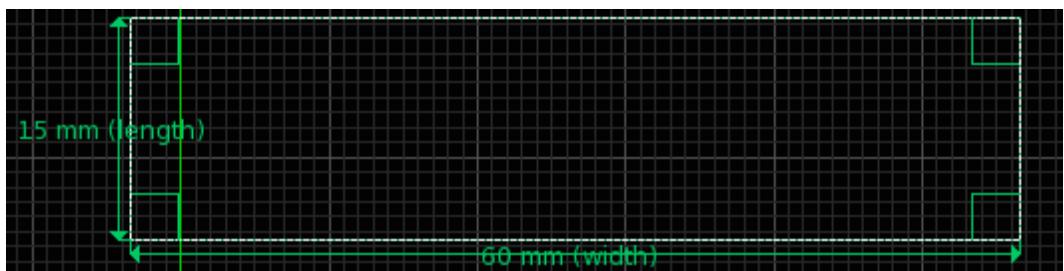


Figure 6.6: The Box with completed side constraints

- Navigate to the  EXTRUDE tab to extrude the rectangular region. Enter “height” as the distance, in the +Z direction.

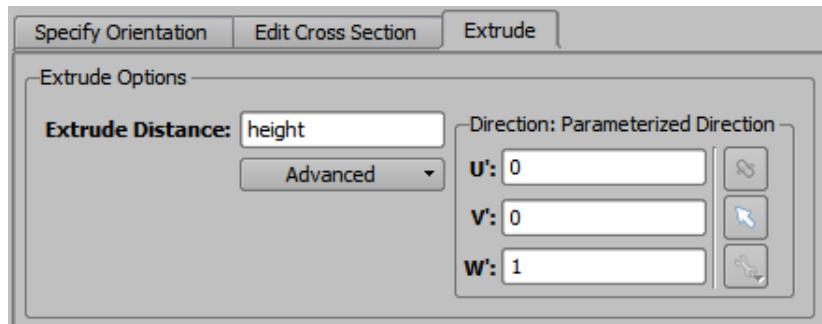


Figure 6.7: Setting the extrude options for the Box

→ Press DONE to complete the Box geometry.

Now the dimensions of the Box are completely parameterized and adjustable from the PARAMETERS workspace window.

### 6.3.2 Modeling the Monopole

The Monopole will be created with a WIRE BODY object that is locked to the top center of the Box. Its length will be defined by the parameter “wireLength”.

- Right-click on the PARTS branch of the PROJECT TREE. Choose CREATE NEW > WIRE BODY from the context menu.
- In the VIEW TOOLS toolbar, select the VIEW FROM +Z (TOP) orientation.
- Navigate to the SPECIFY ORIENTATION tab and select SIMPLE PLANE from the PICK menu.

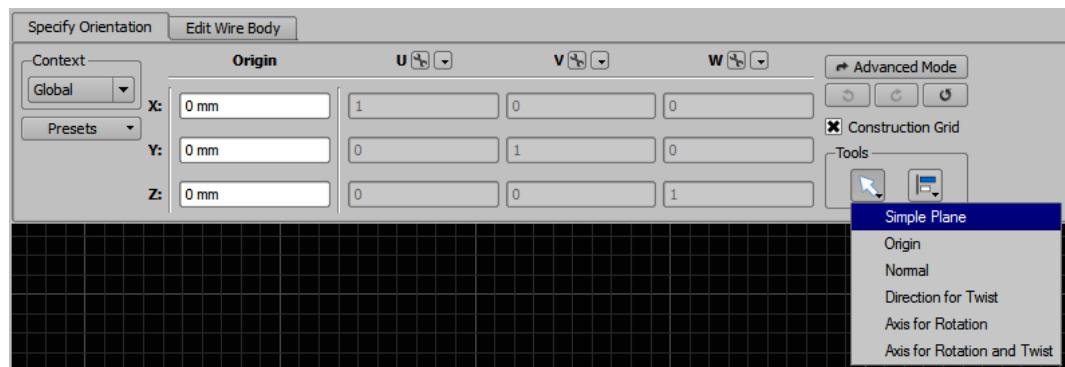


Figure 6.8: Setting the extrude options for the Monopole

- Hover the mouse over the face of the box and press C to center the ORIGIN. Click on this location to set the values.
- 💡 If the ORIGIN does not move to the center as expected, right-click in the geometry space to “activate” the window.

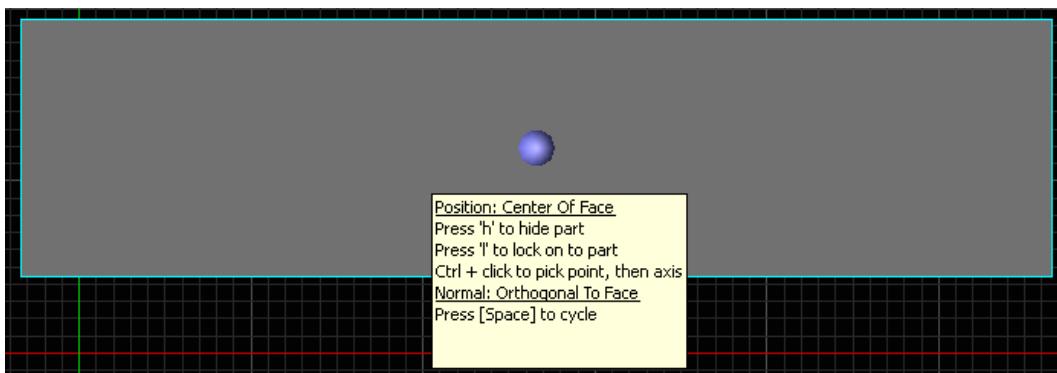


Figure 6.9: Placing the origin at the center of the Monopole

- Redefine the orientation of the sketching plane by selecting the ZX PLANE under the PRESETS drop-down.

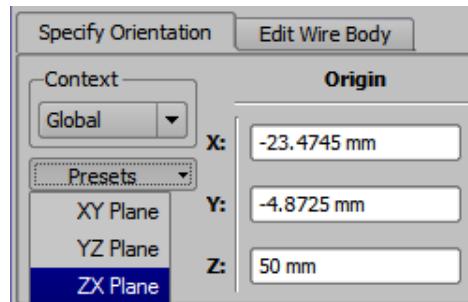


Figure 6.10: Adjusting the orientation plane to sketch the Monopole

- Navigate to the EDIT WIRE BODY tab. Type “Monopole” into the NAME box.
- In the VIEW TOOLS toolbar, select the VIEW FROM +X/ +Y / +Z orientation.



Figure 6.11: Selecting the correct orientation

- Select the STRAIGHT EDGE tool.
  - Click on the origin (where the green and red axes intersect) to place the first point of the wire antenna.
  - Click the second point anywhere along the axis directed normal to the plane of the box.

- Select the SELECT/MANIPULATE tool at the top left of the GEOMETRY workspace window. Right-click on the end of the wire at the origin and select LOCK POSITION.
- Select the DISTANCE constraint tool to constrain the length of the wire as “wireLength”.

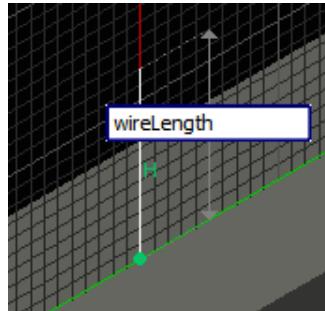


Figure 6.12: Constraining the length of the Monopole wire

- Click DONE to finish the MONPOLE geometry.

## 6.4 Creating Materials

### Define Material, PEC

- Right-click on the DEFINITIONS: MATERIALS branch of the PROJECT TREE and select NEW MATERIAL DEFINITION from the context menu.
- Set the perfect electric conductor material properties as follows:
  - NAME: PEC
  - ELECTRIC: Perfect Conductor
  - MAGNETIC: Freespace
- If desired, navigate to the APPEARANCE tab to set the PEC material’s display color.

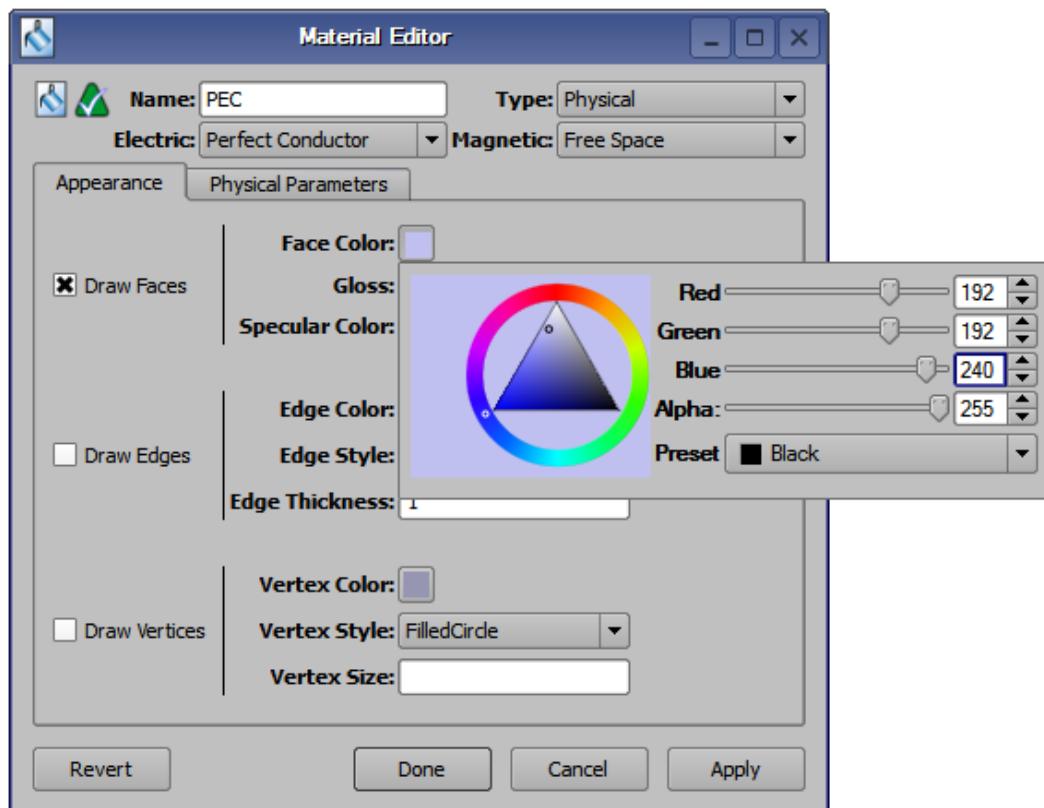


Figure 6.13: Editing the color of the PEC material

## 6.5 Assigning Materials

- Click-and-drag the PEC material object located in the PROJECT TREE and drop it on top of the MONOPOLE and BOX objects.

Below, the PROJECT TREE shows the geometry after materials have been applied.

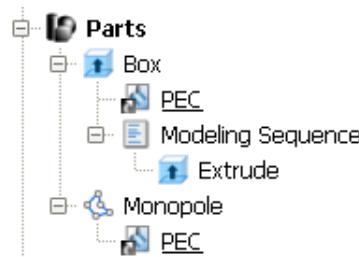


Figure 6.14: Assigning materials to the Monopole and Box parts

This image shows the monopole box geometry with materials applied and colors set for each.

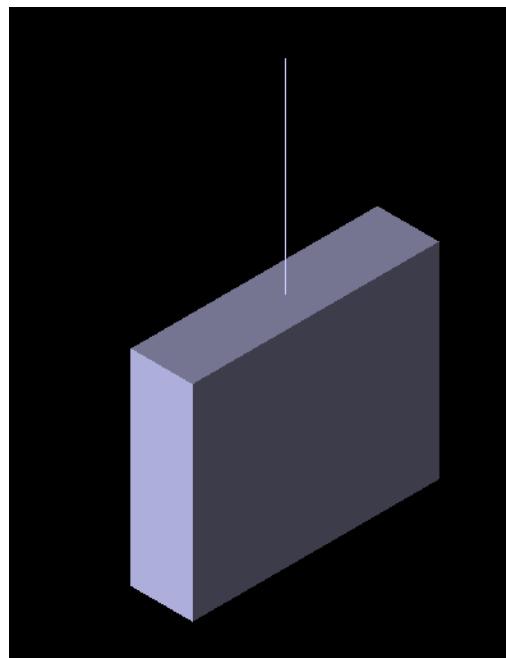


Figure 6.15: The profile of the Monopole Box geometry

## 6.6 Defining the Outer Boundary

- Double-click on the **OUTER BOUNDARY** branch of the **PROJECT TREE** to open the **OUTER BOUNDARY EDITOR**.
- Set the outer boundary properties as follows:
  - **BOUNDARY:** “Absorbing” for all boundaries
  - **ABSORPTION TYPE:** PML

→ LAYERS: 7

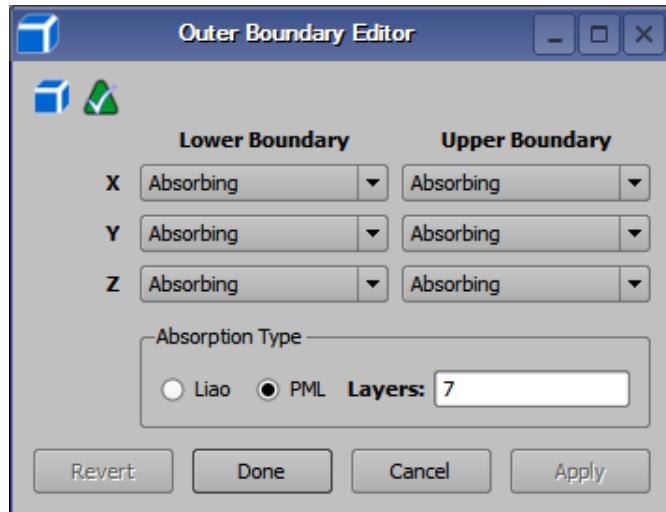


Figure 6.16: Defining the outer boundary for the monopole box project

→ Click DONE to apply the outer boundary settings.

## 6.7 Defining the Grid

Now we will define characteristics of the calculation grid.

- Double-click on the FDTD: GRID branch of the PROJECT TREE to open the GRID EDITOR.
- On the CELL SIZE tab (Figure 6.17), set MIN CELLS PER WAVELENGTH to “60”. This will increase the grid resolution (improving accuracy) without significantly increasing the amount of memory or runtime required for the simulation.

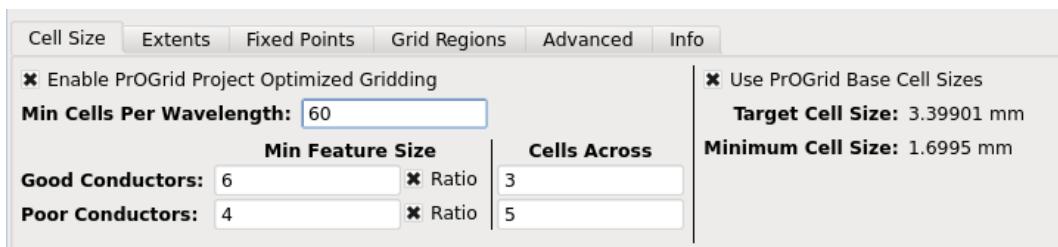


Figure 6.17: Defining cell size on the Grid Editor

→ Click DONE to apply the grid settings.

## Adding fixed points to the geometry

- In the PARTS branch of the PROJECT TREE, right-click on the Box object and select GRIDDING / MESHING > GRIDDING PROPERTIES to open the GRIDDING PROPERTIES EDITOR.
  - Check USE AUTOMATIC FIXED POINTS.
- Click DONE to close the editor.
- Repeat this process to apply fixed points to the MONPOLE part as well.

## 6.8 Adding a Feed

We will now add a FEED to the monopole geometry at the base of the MONPOLE antenna. The feed will consist of a voltage source and series  $50\Omega$  resistor connected at the base of the Monopole. We will then apply a sinusoidal excitation to the circuit through this feed.

- Right-click on the CIRCUIT COMPONENTS branch in the PROJECT TREE. Choose NEW CIRCUIT COMPONENT WITH > NEW FEED DEFINITION from the context menu.

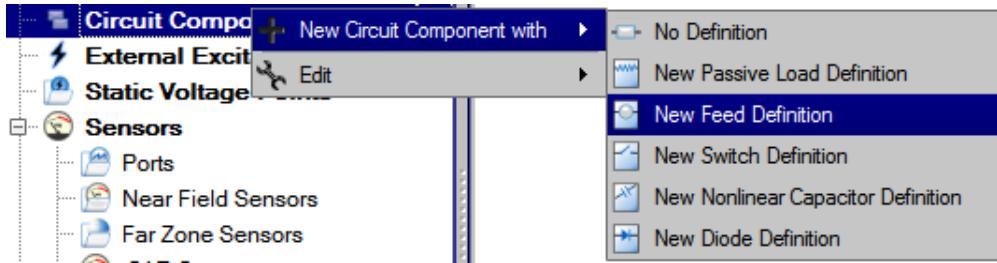


Figure 6.18: Adding a feed to the project

- Define the endpoints of the feed.
  - ENDPOINT 1: Zoom in to the area where the MONPOLE meets the Box. Using the PICK tool, click the point at the base of the wire. Then type in "height" for the ENDPOINT 1: Z: value.

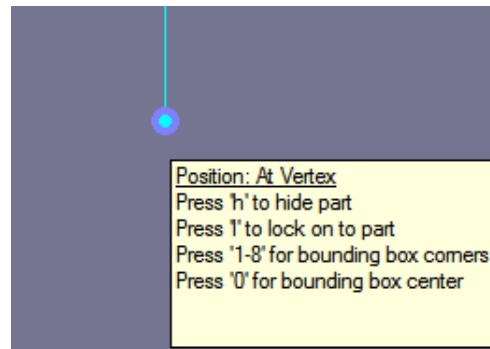


Figure 6.19: Using the Pick tool to place the first endpoint of the Feed

- ENDPOINT 2: Select the PICK tool under ENDPOINT 2, and click a higher location along the wire. Edit ENDPOINT 2: Z: to be “height + 1 mm”. The X: and Y: locations should be the same as the values for ENDPOINT 1.
- Your X: and Y: locations may differ from the figure since we began by drawing an arbitrary rectangular sketch in the *XY*-plane.

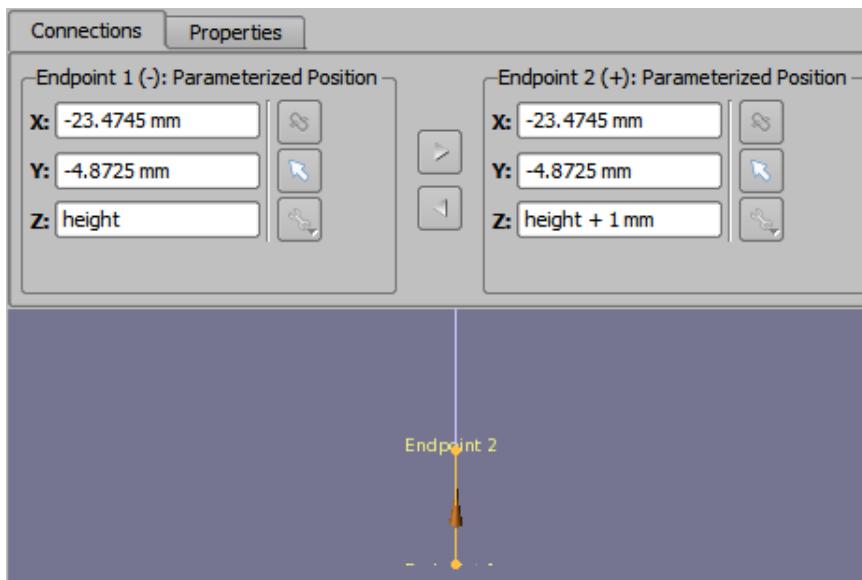


Figure 6.20: Defining the location the feed

→ Navigate to the PROPERTIES tab, and enter the following:

- NAME: Feed
- COMPONENT DEFINITION:  $50 \Omega$  Voltage Source
- POLARITY: Positive
- Check the box labeled THIS COMPONENT IS A PORT.

- Click DONE to add the FEED.

## 6.9 Viewing the Waveform

An associated waveform was automatically created for the feed definition.

- Navigate to the DEFINITIONS: WAVEFORMS branch of the PROJECT TREE. Double-click on the AUTOMATIC waveform to view its properties.
- Note that the AUTOMATIC waveform type selects a ramped sinusoid waveform when a single-valued FREQUENCY RANGE OF INTEREST is configured on the PROJECT PROPERTIES EDITOR as in Section 6.1.

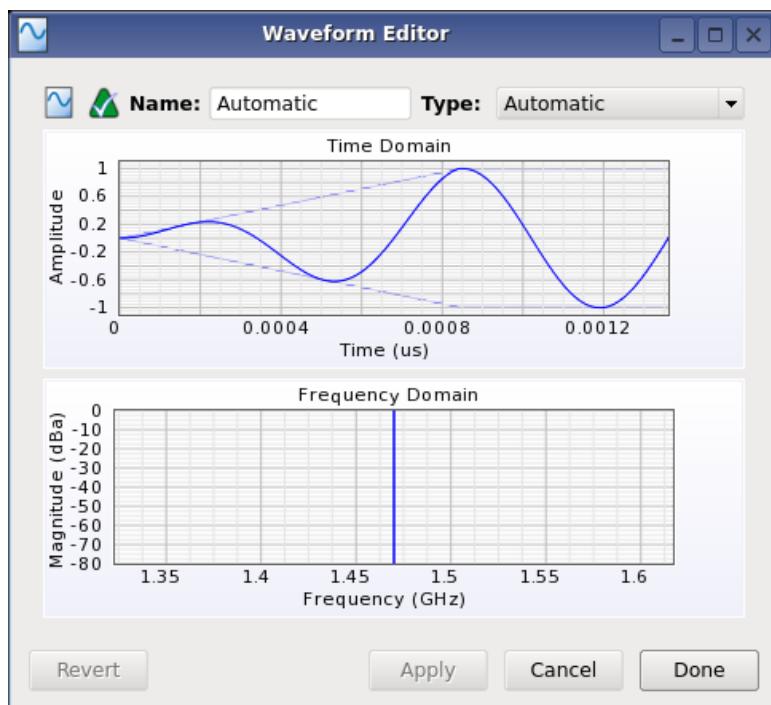


Figure 6.21: Viewing the sinusoidal waveform

- Click DONE to close the WAVEFORM EDITOR.

## 6.10 Requesting Output Data

Recall that the project already contains one port sensor named FEED that will request results. We also wish to collect field samplings at discrete frequencies throughout the calculation. To retrieve this data, add a SURFACE SENSOR.

## Adding a Surface Sensor Definition

- First, create the **SURFACE SENSOR DEFINITION**. Right-click on the **DEFINITIONS:SENSOR DATA DEFINITIONS** branch of the **PROJECT TREE**. Choose **NEW SURFACE SENSOR DEFINITION** from the context menu.
- Set the properties of the surface sensor definition as follows:
  - NAME: Field Sampling
  - FIELD VS. FREQUENCY: Steady E and Steady J

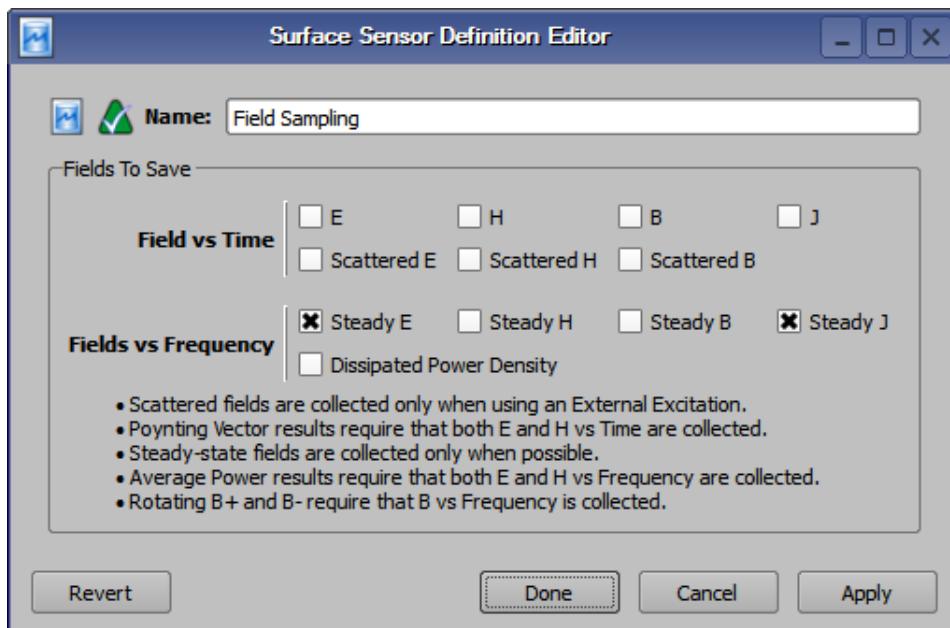


Figure 6.22: Adding the sensor definition

- Press **DONE** to finish editing the **FIELD SAMPLING** definition.

## Adding a Surface Sensor

- Right-click on the **SENSORS:NEAR FIELD SENSORS** branch of the **PROJECT TREE**. Select **NEW SENSOR ON MODEL SURFACE** from the context menu.
  - From the **SELECT MODEL** tab, use the **SELECT** tool (at the top of the **VIEW TOOLS** menu) and then click on the **Box**.
    - You will know that the box is selected when it changes color.
  - Under the **PROPERTIES** tab, enter the following:
    - \* NAME: Surface Sensor
    - \* SENSOR DEFINITION: Field Sampling

- \* SAMPLING METHOD: Snapped to E-Grid
- Press DONE to finish editing the  SURFACE SENSOR.

## 6.11 Running the Calculation

If you have not already saved your project, do so by selecting FILE > SAVE PROJECT. Once the project is saved, a new simulation can be created to send to the calculation engine.

- Open the SIMULATIONS workspace window. Click the FDTD button in the upper-left to set up a new simulation.
- Under FREQUENCIES OF INTEREST, check the COLLECT STEADY-STATE DATA box.
  - Under the FREQUENCIES tab, check USE WAVEFORM FREQUENCY.
- Select CREATE AND QUEUE SIMULATION to close the dialog and run the new simulation.

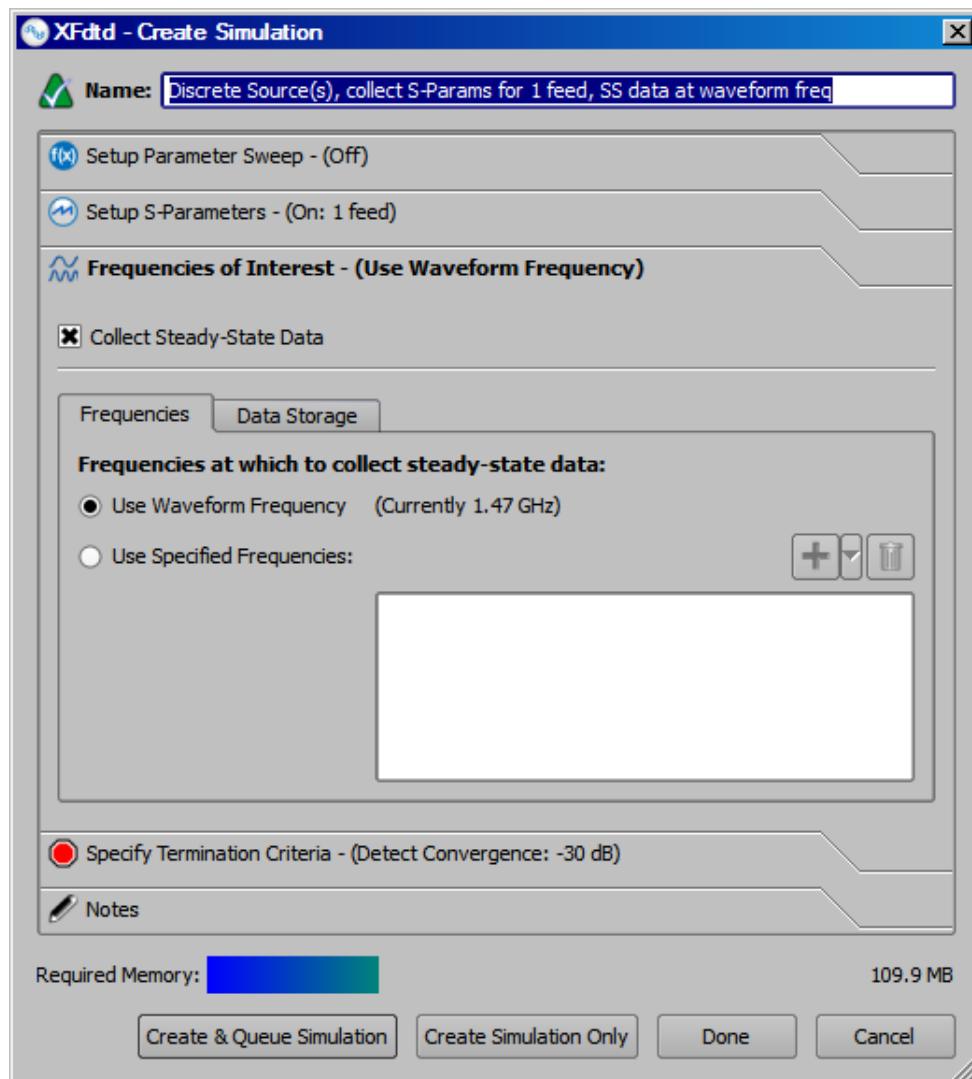


Figure 6.23: Setting up the simulation for the monopole box project

## 6.12 Viewing the Results

The OUTPUT tab of the SIMULATIONS workspace window displays the progress of the simulation. Once the STATUS column shows that the simulation has completed, we can view its results in the RESULTS workspace window.

### System Efficiency Results

First, we will view the SYSTEM results.

- To filter the list accordingly, select the following options in the columns in the top pane of the RESULTS window. (You may need to change your column headings first.)
  - SENSOR: System
  - RESULT TYPE: System Efficiency
- Double-click on the result. A list of power and efficiency results will appear in a dialog window.

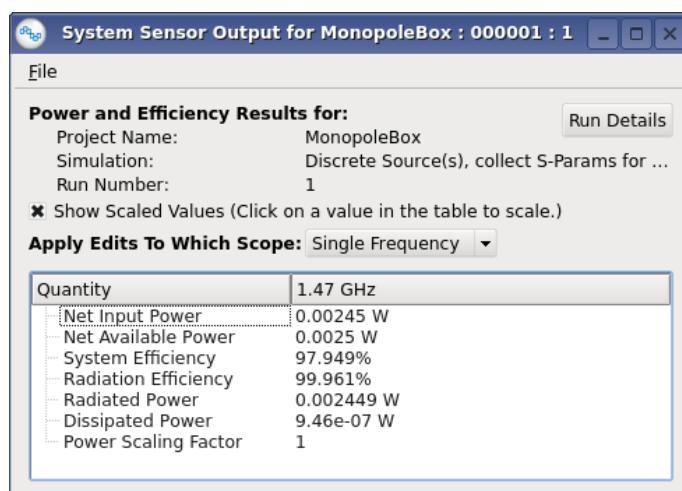


Figure 6.24: Viewing power and efficiency results

- You may close the window when you are finished viewing the results.

### Conduction Current Results from the Surface Sensor

Now we will view the conduction current results retrieved by the SURFACE SENSOR placed on the surface of the Box.

- Select the following options in the RESULTS window:
  - SENSOR: Surface Sensor
  - RESULT TYPE: Conduction Current ( $J_c$ )

- Double-click on the result. A plot will appear in the GEOMETRY workspace window.
- Below the plot, adjust the following settings on the SETUP tab:
  - SEQUENCE AXIS: Frequency
  - COMPLEX PART: Magnitude
- Press APPLY to finish editing the plot. You should see the conduction current data on the surface of the box, as seen in Figure 6.25.

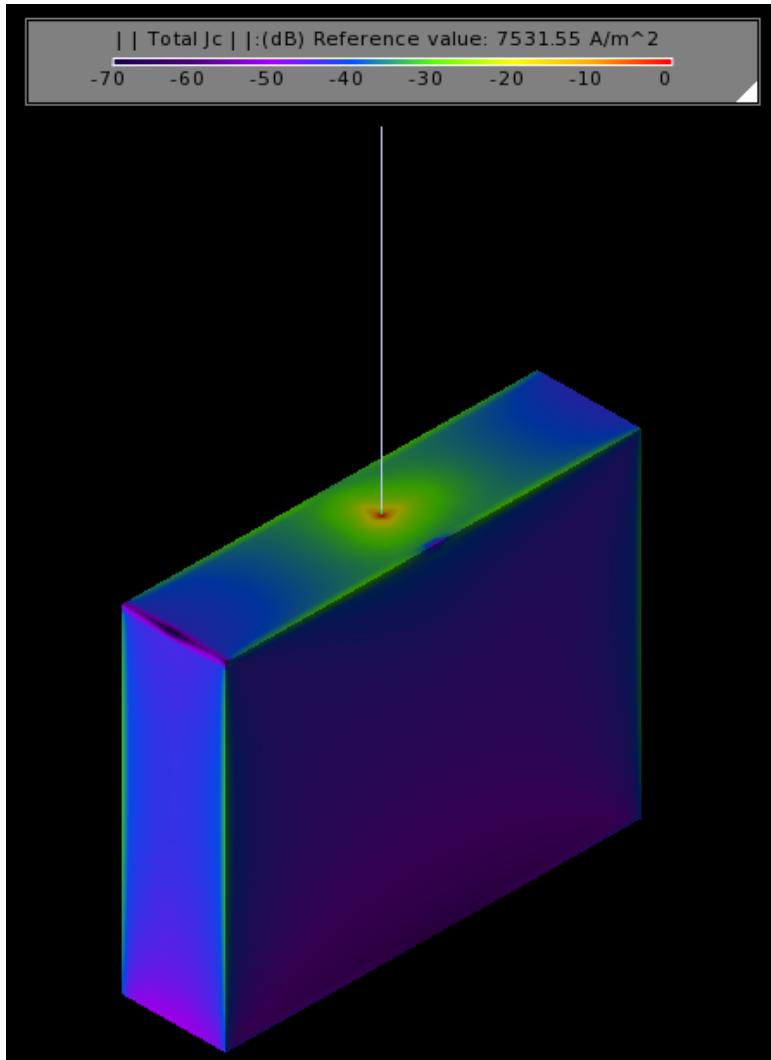


Figure 6.25: Viewing the conduction current plot for the monopole project

- When you are finished, press the UNLOAD button to close.

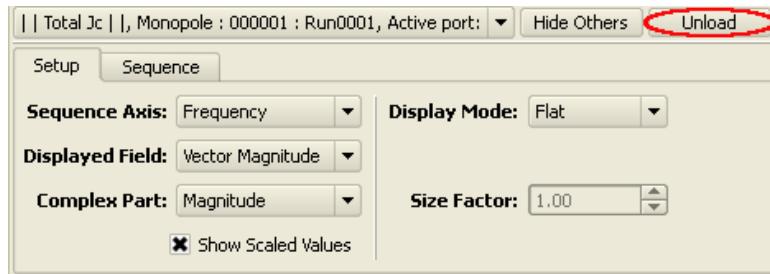


Figure 6.26: Unloading surface current results

## Far Zone Post-Processing

- To begin the far zone post-processing, select the following:
  - SENSOR: Raw Steady-State Far Zone Data
  - RESULT TYPE: E-Field (E)
- Right-click on the E-FIELD (E) result in the filtered list, and select POST-PROCESS RESULTS. The RESULTS workspace window will appear to set the properties of the FAR ZONE sensor.

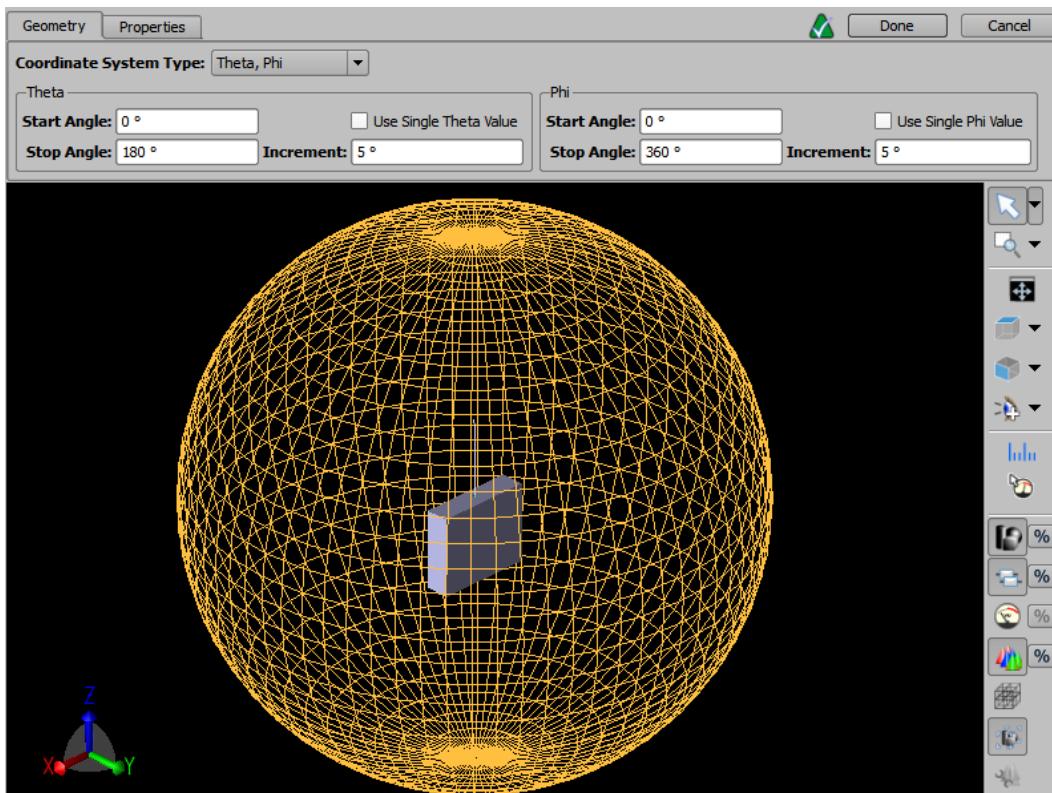


Figure 6.27: Setting up far zone post-processing

- The default definition is sufficient for this calculation. Press DONE.
- XFdtd will ask for a FAR ZONE POST-PROCESSOR EXECUTION MODE. Select SYNCRHONOUS and click OK to begin the steady state far zone data transform.
- When the computation is complete, select the following options in the RESULTS window:
  - SENSOR: Post Processed Far Zone
  - RESULT TYPE: Gain
- Double-click on the result. The plot will appear in the GEOMETRY workspace window.
- Right-click on the SCALE BAR at the top of the screen, and select PROPERTIES.
  - Under the SCALE section, select RELATIVE dB and check the AUTO REFERENCE VALUE box.
  - Under the LIMITS section, uncheck the AUTOMATIC RANGE box. Set the MINIMUM to “-70 dB” and the MAXIMUM to “0 dB”.
  - Press DONE to finish editing the SCALE BAR PROPERTIES.
- Far zone data similar to what is shown in Figure 6.28 will appear.

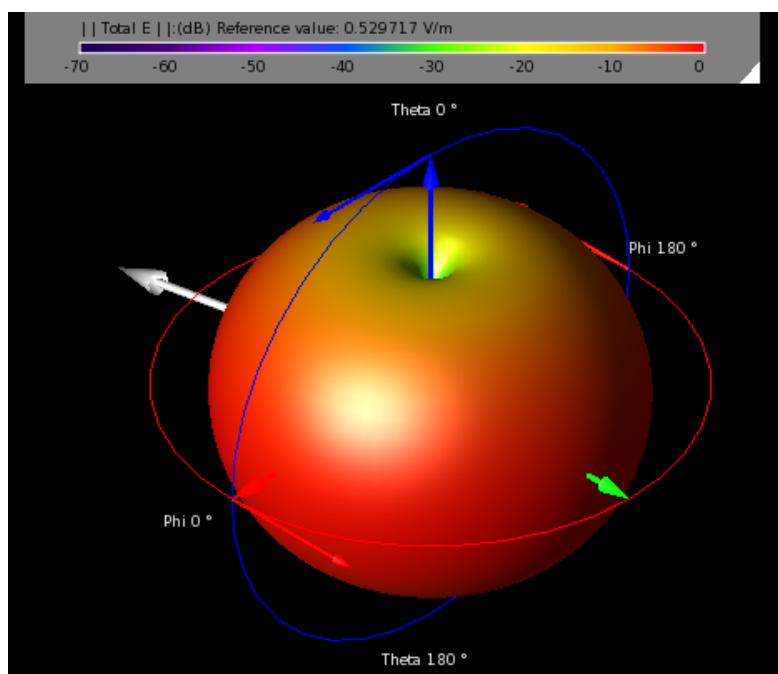


Figure 6.28: Far zone gain data for the monopole project



## Chapter 7

### Example: A Pyramidal Horn

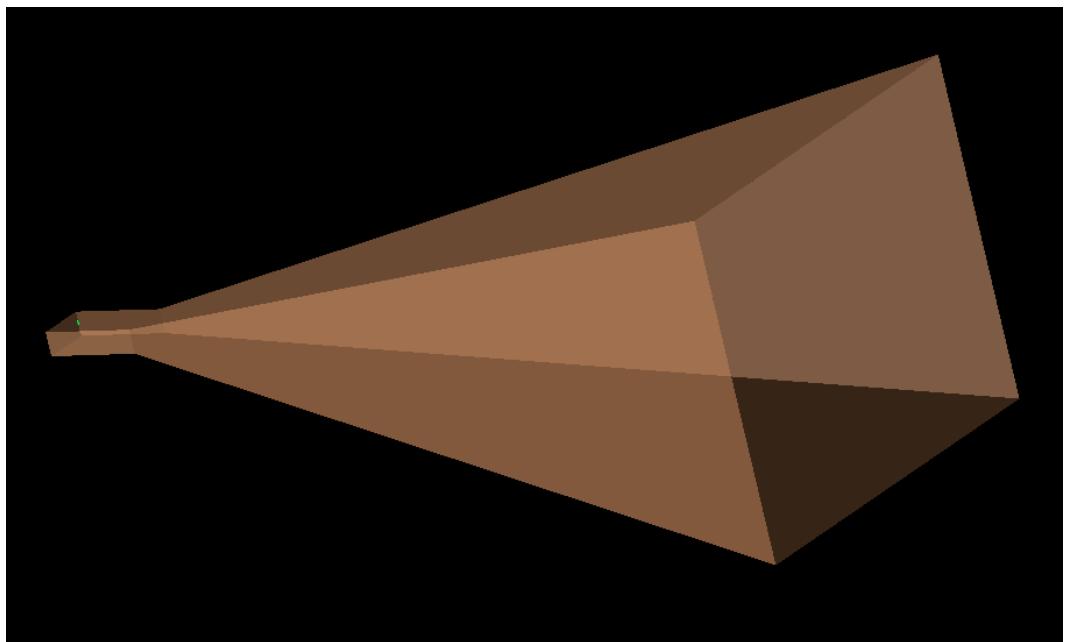


Figure 7.1: The pyramidal horn

Time to create: 35:00 (approx.)

In this chapter, you will learn how to...

- Import a CAD geometry object and add an antenna to it
- Add a feed to the antenna and simulate its effects
- Detect calculation convergence and view the gain of the antenna

This tutorial will demonstrate how to build a pyramidal horn waveguide antenna using an imported horn CAD object. The horn geometry is an optimum gain pyramidal horn antenna (see Antenna Theory and Design by W. Stutzman and G. Thiele, John Wiley and Sons, New York, 1981, pgs 413-415). The pyramidal horn aperture dimensions are 18.46 cm by 14.55 cm with a path length of the horn apex of 33.98 cm. The horn is fed by a WR-90 waveguide with an input signal of 9.3 GHz. The theoretical gain for this antenna is 22.1 dB with half-power beam widths of 12 degrees in the E-plane and 13.6 degrees in the H-plane.

## 7.1 Getting Started

First, a few Project Properties are set up for the pyramidal horn project.

- The PROJECT PROPERTIES EDITOR opens automatically whenever XFdtd is started or when a new project is created. If the editor is not currently visible, double-click on PROJECT at the top of the PROJECT TREE to open the editor.
- On the FREQUENCY RANGE OF INTEREST tab of the PROJECT PROPERTIES EDITOR, specify a single frequency of interest by setting both the MINIMUM and MAXIMUM controls to "9.3 GHz".
- Navigate to the DISPLAY UNITS tab, where a few relevant units should be set:
  - Set FREQUENCY to "gigahertz (GHz)"
  - Set LENGTH to "millimeters (mm)"
- Press DONE on the PROJECT PROPERTIES EDITOR.

## 7.2 Creating the Pyramidal Horn Geometry

Now we will create the pyramidal horn geometry out of two components: a HORN and a WIRE BODY. For this example, we will import the horn object from a CAD file, and we will use the GEOMETRY TOOLS interface to create the feed.

### 7.2.1 Modeling the Horn

- Navigate to **File > Import > CAD file(s)** and import the horn.sat file included with your XFdtd installation package.

- ▶ This object can be found in the doc/examples folder of your installation directory, depending on where you have XFDTD installed on your computer (e.g., C:\Program Files\Remcom\XFDTD 7.9.2 (64-bit)\doc\examples).
- In the IMPORT OPTIONS dialog, select INTERPRET UNITS AS: "centimeters".

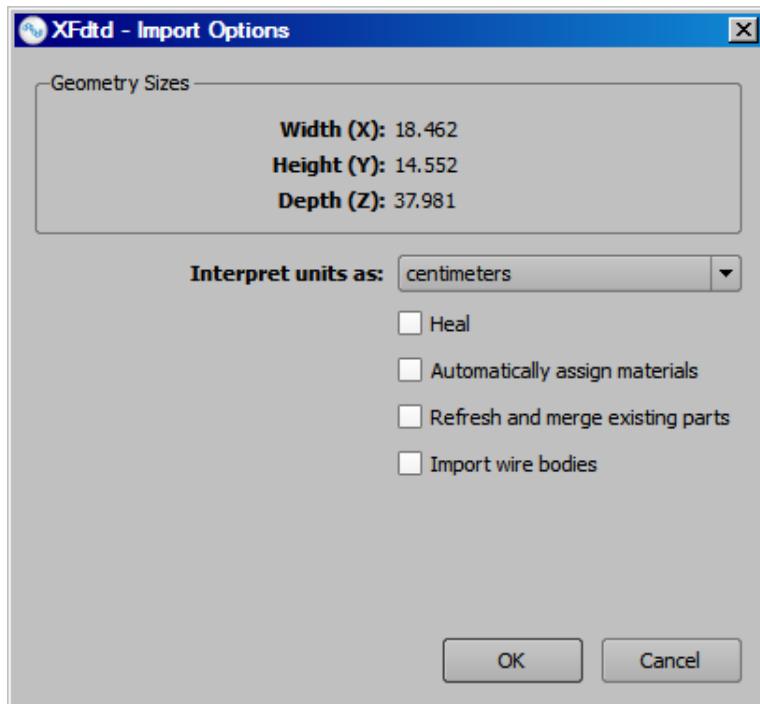


Figure 7.2: CAD import options for the horn

- Press OK to import the HORN object.

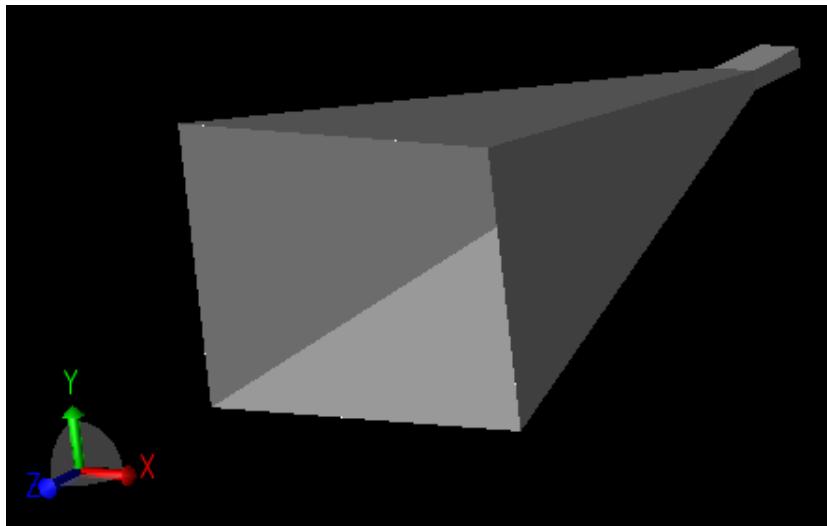


Figure 7.3: The imported horn geometry

### 7.2.2 Modeling the Excitation Wire

The Excitation Wire will be created with a *Y*-directed **WIRE BODY** object that is .508 cm long, located near the mid-back of the **HORN**.

- The Excitation Wire will be located inside the horn and thus obscured by the horn from most viewing angles. Reducing the geometry's opacity will help aid in the positioning of the wire. Next to the **TOGGLE PARTS VISIBILITY** button, click the **%** button and set the parts opacity to "25".
- Right-click on the **PARTS** branch of the **PROJECT TREE**. Choose **CREATE NEW > WIRE BODY** from the context menu.
- Under the **SPECIFY ORIENTATION** tab, enter the origin coordinates at (0, 0, -3.193 cm).

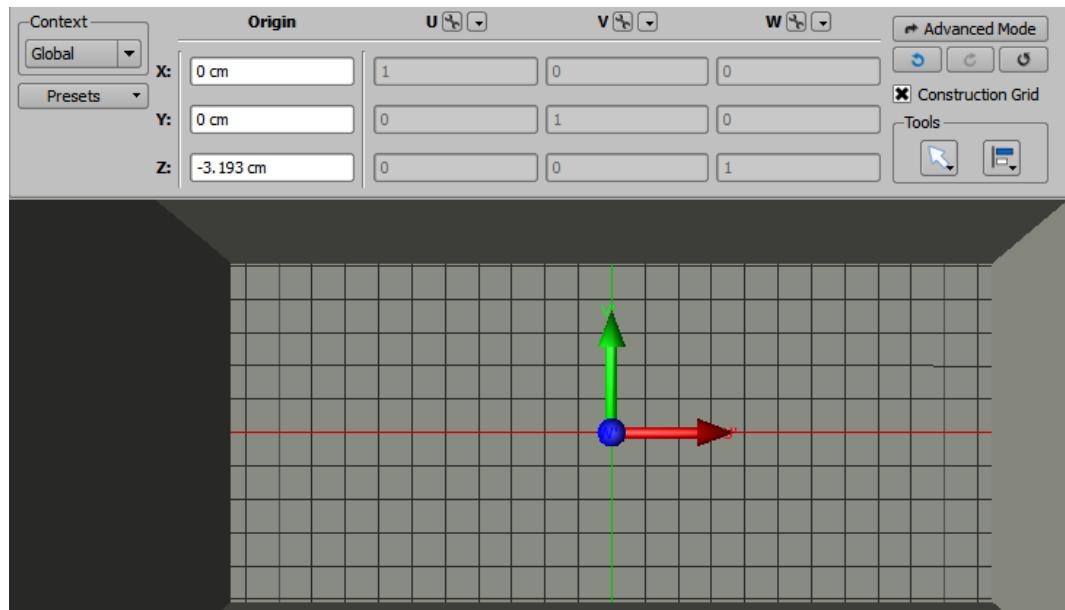


Figure 7.4: Specifying the orientation of the Excitation Wire inside the Horn

- Navigate to the EDIT WIRE BODY tab. Type “Excitation Wire” into the NAME box.
- Select the STRAIGHT EDGE tool.
  - Press to display the creation dialog for the first point. Enter (0, -0.508 cm) and press OK.
  - Press to display the creation dialog for the second point. Enter (0, 0) and press OK to complete the wire.
- Click DONE to finish the EXCITATION WIRE geometry.

## 7.3 Creating Materials

### Define material, PEC

- Right-click on the DEFINITIONS: MATERIALS branch of the PROJECT TREE. Choose NEW MATERIAL DEFINITION from the context menu.
- Double-click the new material to edit its properties. Set the perfect electric conductor material properties as follows:
  - NAME: PEC
  - ELECTRIC: Perfect Conductor
  - MAGNETIC: Freespace

→ If desired, navigate to the APPEARANCE tab to set the PEC material's display color.

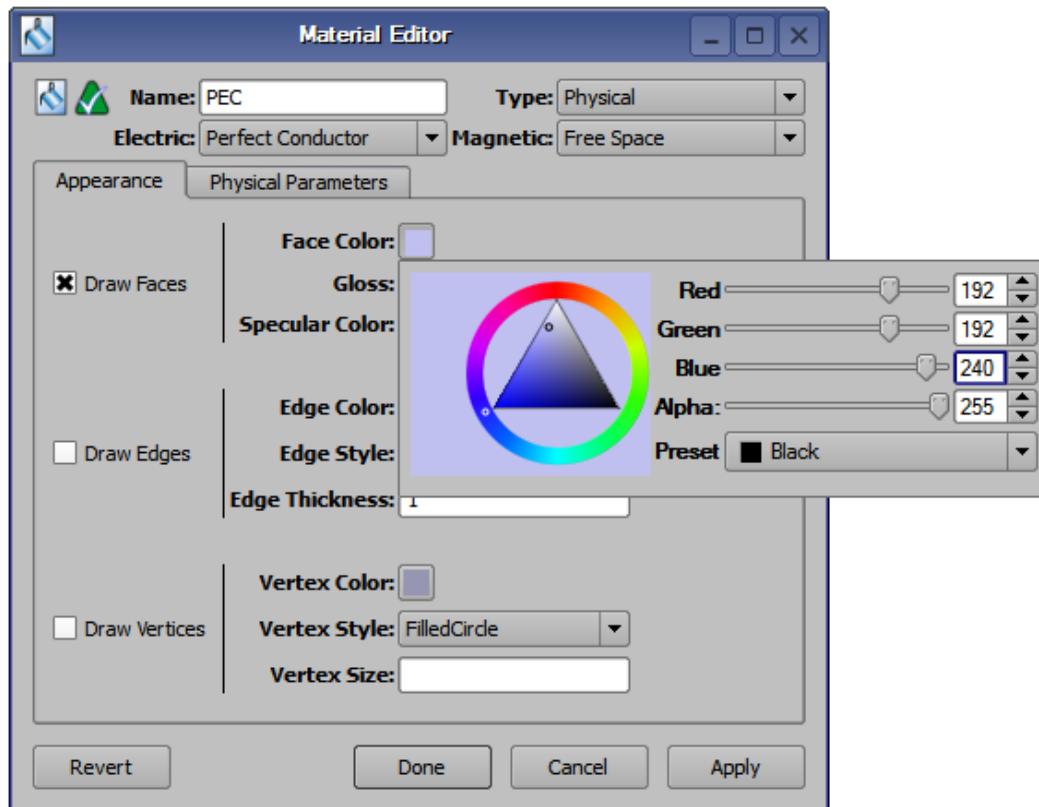


Figure 7.5: Editing the color of the PEC material

## Define material, COPPER

- Right-click on the DEFINITIONS: MATERIALS branch of the PROJECT TREE. Choose NEW MATERIAL DEFINITION from the context menu.
- Double-click the new material to edit its properties. Set the copper material properties as follows:
- NAME: Copper
  - ELECTRIC: Isotropic
  - MAGNETIC: Freespace
- Under the ELECTRIC tab:
- TYPE: Nondispersive
  - ENTRY METHOD: Normal
  - CONDUCTIVITY:  $5.8 \times 10^7$  S/m

- RELATIVE PERMITTIVITY: 1

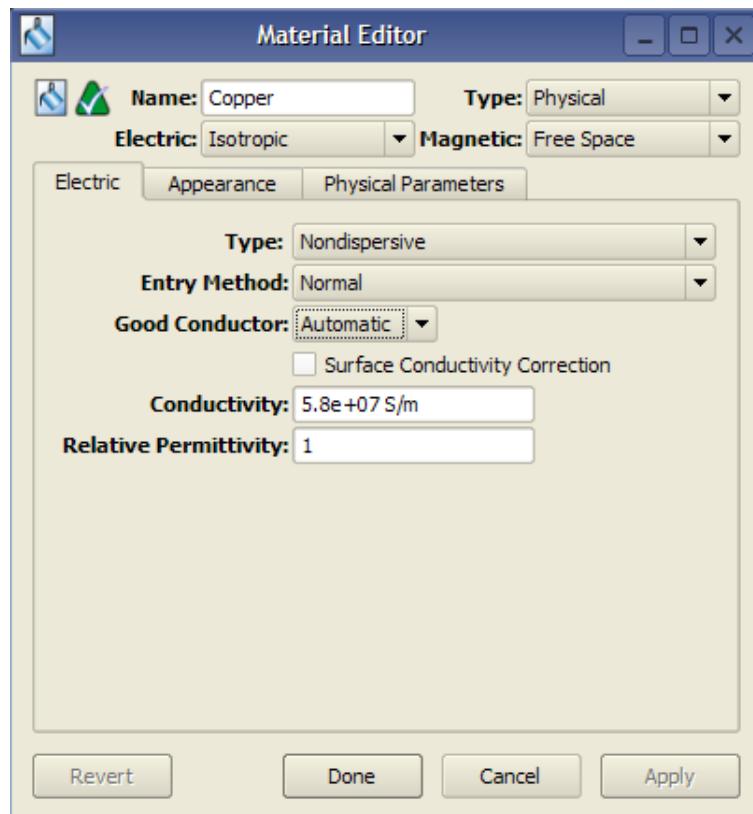


Figure 7.6: Editing the properties of the Copper material

→ If desired, navigate to the APPEARANCE tab to set the COPPER material's display color.

## 7.4 Assigning Materials

- Click-and-drag the PEC material object and drop it on top of the EXCITATION WIRE object in the PARTS branch of the tree.
- Assign the COPPER material to the HORN object using the same procedure.

This image shows the pyramidal horn geometry with materials applied and colors set for each.

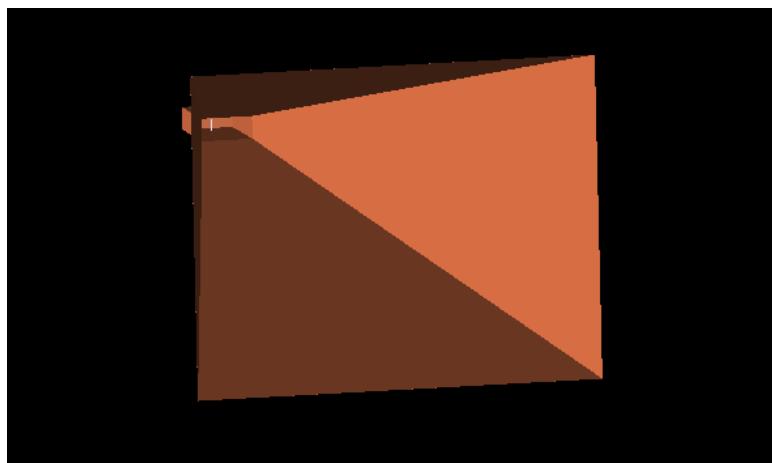


Figure 7.7: The profile of the pyramidal horn geometry

## 7.5 Defining the Outer Boundary

- Double-click on the **FDTD: OUTER BOUNDARY** branch of the **PROJECT TREE** to open the **OUTER BOUNDARY EDITOR**.
- Set the outer boundary properties as follows:
  - BOUNDARY: “Absorbing” for all boundaries
  - ABSORPTION TYPE: PML
  - LAYERS: 7

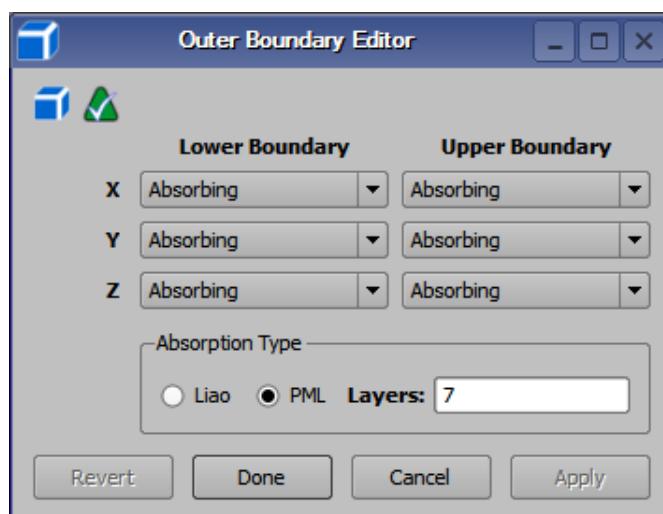


Figure 7.8: Defining the outer boundary for the pyramidal horn

- Click DONE to apply the outer boundary settings.

## 7.6 Defining the Grid

Now we will define characteristics of the cells in preparation to perform an accurate calculation.

The default of 15 cells per wavelength is sufficient for accuracy in this project, but PrOGrid boundary refinement introduces small cells near the horn sidewalls (where we do not expect field values to be rapidly changing) that unnecessarily increase the running time of the simulation. Disable PrOGrid boundary refinement as follows:

- Double-click on the FDTD:GRID branch of the PROJECT TREE to open the GRID EDITOR.
- Navigate to the Grid Editor's ADVANCED tab.
- Under PROGRID ADVANCED OPTIONS, set BOUNDARY REFINEMENT NUMBER OF CELLS to "0".

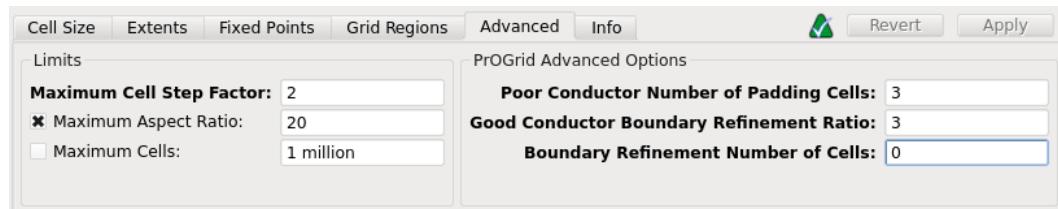


Figure 7.9: Disabling PrOGrid boundary refinement in the Grid Editor

- Click DONE to apply the grid settings.

## 7.7 Configuring the Mesh

This example will make use of the XACT meshing feature to perfectly match the grid to the geometry object. This feature is selected on a part-by-part basis.

- Right-click on the Horn Antenna object in the PARTS branch of the PROJECT TREE and choose GRIDDING / MESHING > MESHING PROPERTIES from the context menu.
- Click in the box labeled USE XACT MESH to activate XACT meshing for the Horn object.
- Click DONE to apply the changes to the Horn's meshing properties.

In the FDTD branch of the PROJECT TREE, double-click on the MESH icon. This will display the mesh and Mesh Viewing controls.

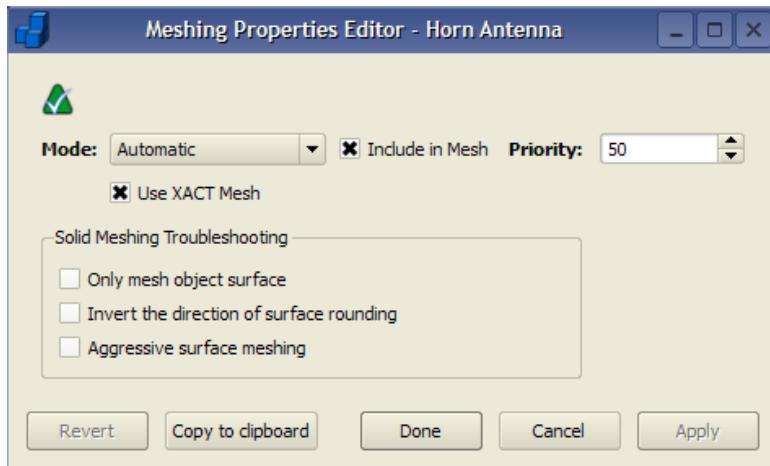


Figure 7.10: The Meshing Properties editor shown with the “Use XACT Mesh” box selected for the Horn object.

## 7.8 Adding a Feed

We will now add a FEED to the pyramidal horn geometry. The feed will consist of a voltage source and series  $50\Omega$  resistor connected at the base of the EXCITATION WIRE. We will then apply a 9.3 GHz sinusoidal excitation to the antenna through this feed.

- Right-click on the CIRCUIT COMPONENTS branch in the PROJECT TREE. Choose NEW CIRCUIT COMPONENT WITH > NEW FEED DEFINITION from the context menu.

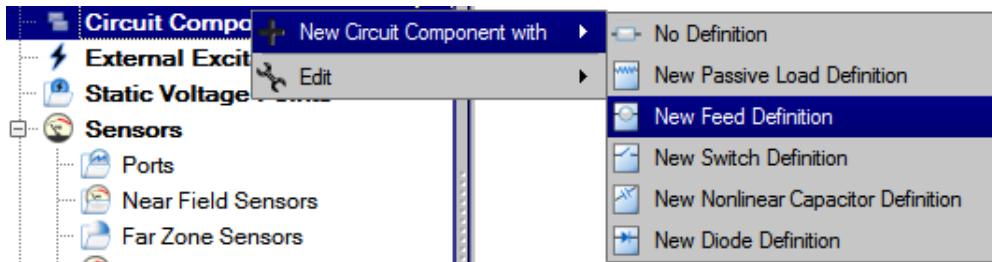


Figure 7.11: Adding a feed to the project

- Define the endpoints of the feed.
  - ENDPOINT 1: X: “0 cm”, Y: “-0.508 cm”, Z: “-3.193 cm”
  - ENDPOINT 2: X: “0 cm”, Y: “-0.308 cm”, Z: “-3.193 cm”
- Navigate to the PROPERTIES tab, and enter the following:
  - NAME: Feed
  - COMPONENT DEFINITION:  $50 \Omega$  Voltage Source
  - DIRECTION: Y

- POLARITY: Positive
  - Check the box labeled THIS COMPONENT IS A PORT.
  - Click the ADVANCED button and check the two boxes labeled ENABLE FIXED POINT ON ENDPOINT 1 and ENABLE FIXED POINT ON ENDPOINT 2.
- Click DONE to add the FEED.

An associated waveform was automatically created for the feed definition. Because a single-valued FREQUENCY RANGE OF INTEREST was configured on the PROJECT PROPERTIES EDITOR, the Automatic waveform type will use a ramped sinusoid waveform at the desired frequency of 9.3 GHz.

## 7.9 Running a Simulation

If you have not already saved your project, do so by selecting FILE > SAVE PROJECT. Once the project is saved, a new simulation can be created to send to the calculation engine.

- Open the SIMULATIONS workspace window. Click the FDTD button in the upper-left to set up a new simulation.
- Most of the default settings are sufficient. Navigate to the SPECIFY TERMINATION CRITERIA tab. Set up the termination criteria as follows:
- ANALYZE PROJECT CONTENTS: Unchecked
  - DETECT CONVERGENCE: Checked
  - THRESHOLD: -40 dB
  - MAXIMUM SIMULATION TIME: 10000 \* timestep

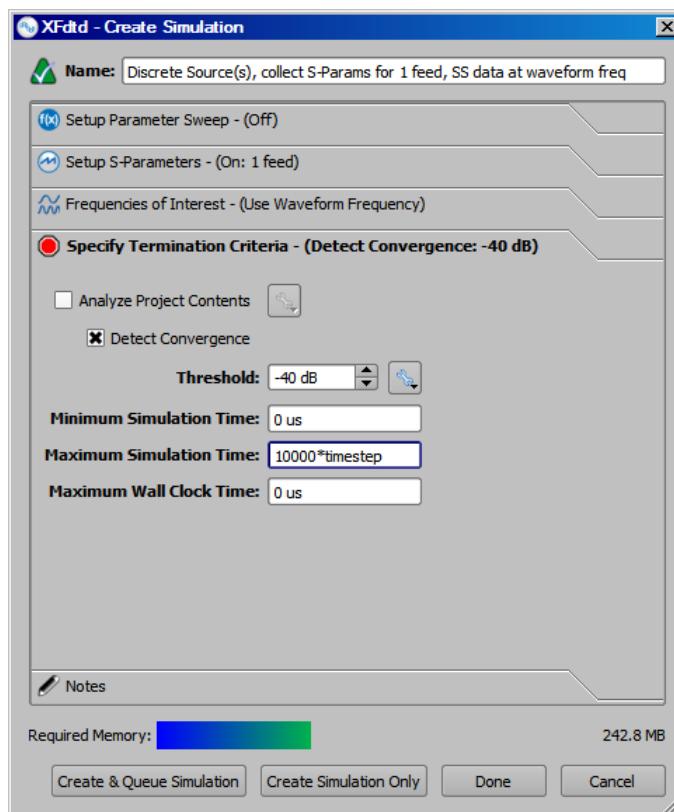


Figure 7.12: Adding a new simulation to the pyramidal horn project

→ Select CREATE AND QUEUE SIMULATION to close the dialog and run the new simulation.

## 7.10 Viewing the Results

The OUTPUT tab of the SIMULATIONS workspace window displays the progress of the simulation. Once the STATUS column shows that the simulation has completed, we can view its results in the RESULTS workspace window.

### Checking for convergence

Although detect convergence was selected during simulation setup, it is still good practice to check that convergence has been reached.

→ To filter the list accordingly, select the following options in the columns in the top pane of the RESULTS window. (You may need to change your column headings first.)

- SENSOR: Feed
  - SENSOR TYPE: Circuit Component
  - RESULT TYPE: Voltage
- Double-click on the result with a DOMAIN value of "Time" to view the voltage waveform. You can see by adjusting the time range that the source voltage has reached a steady-state sinusoidal waveform.

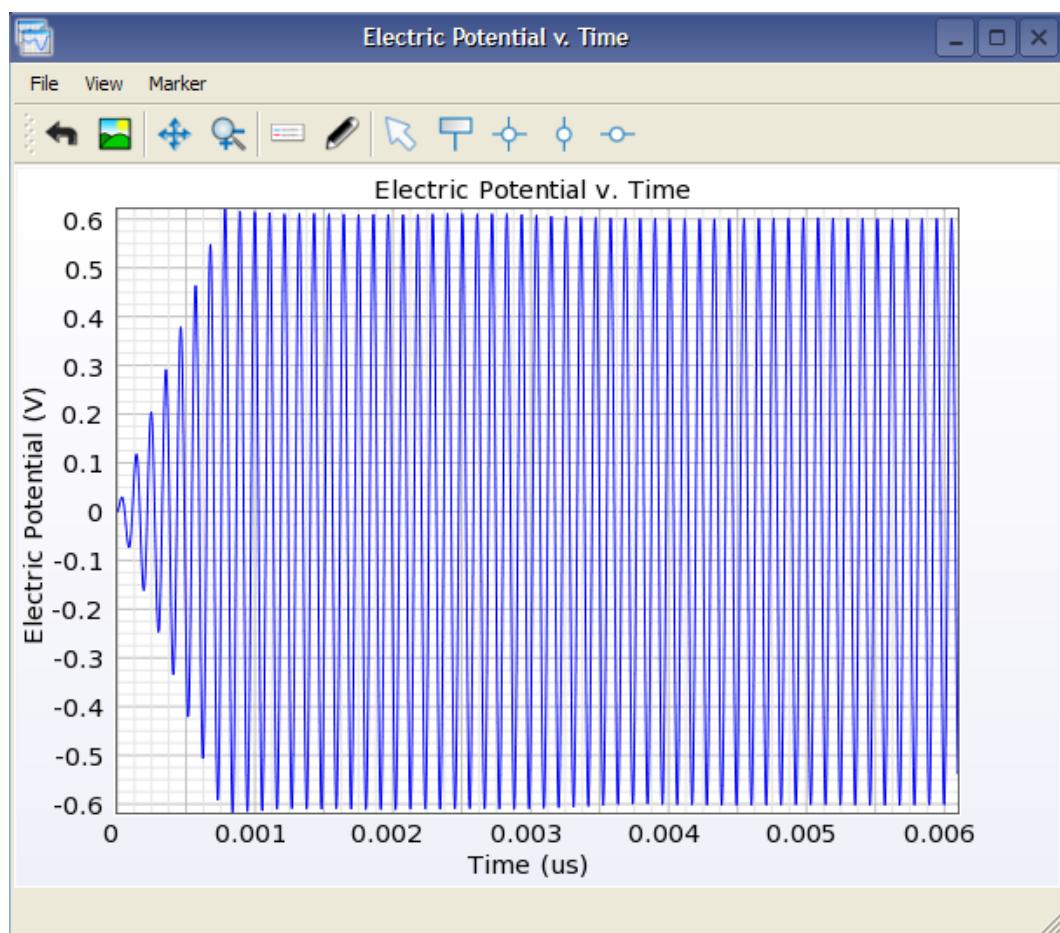


Figure 7.13: Checking the convergence of the voltage waveform

## Far Zone Post-Processing

- To begin the far zone post-processing, select the following:
- SENSOR: Raw Steady-State Far Zone Data
  - RESULT TYPE: E-Field (E)

- Right-click on the E-FIELD (E) result in the filtered list, and select POST-PROCESS RESULTS. The RESULTS workspace window will appear to set the properties of the FAR ZONE sensor.

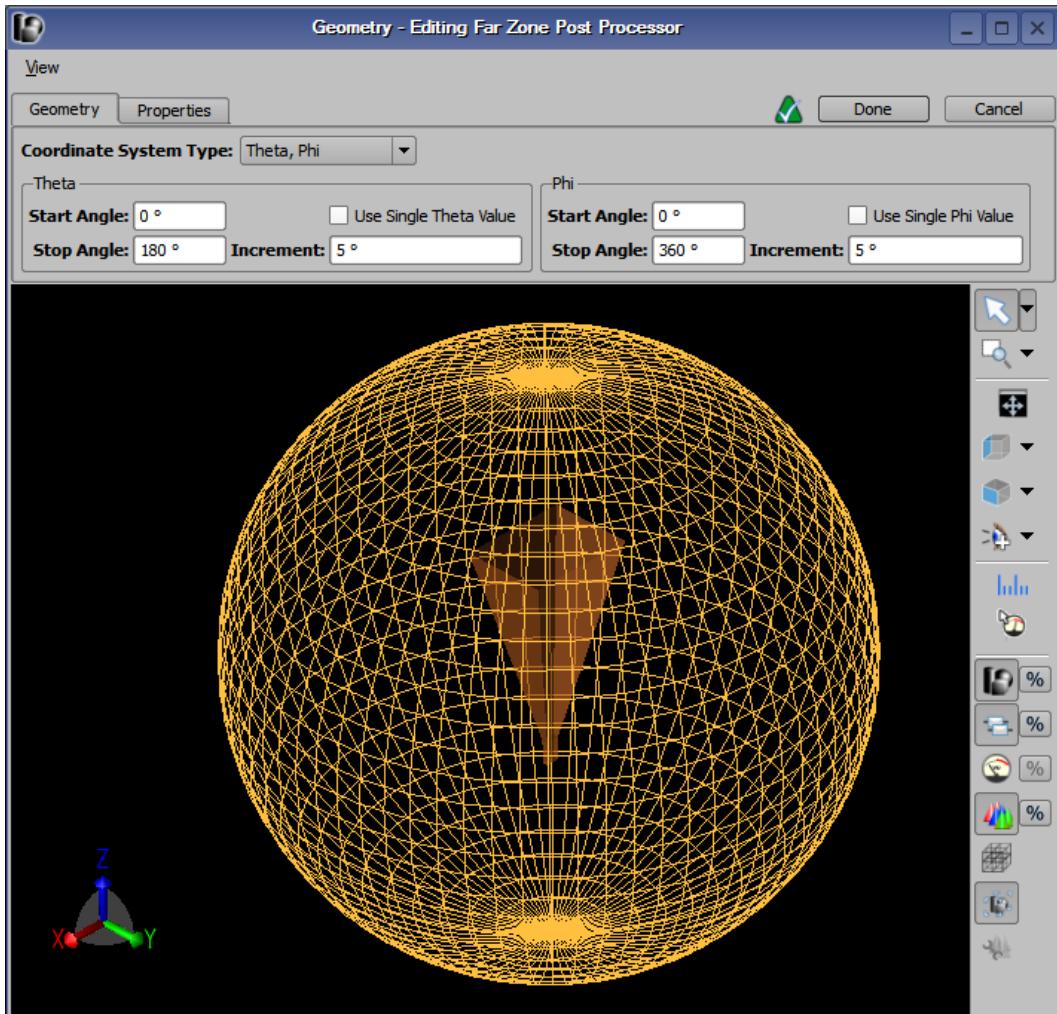


Figure 7.14: Setting up far zone post-processing for the horn

- The default definition is sufficient for this calculation. Press DONE.
- XFDTD will ask for a FAR ZONE POST-PROCESSOR EXECUTION MODE. Select SYNCRHONOUS and click OK to begin the steady state far zone data transform.
- Select the following options in the RESULTS window:
  - SENSOR: Post Processed Far Zone
  - RESULT TYPE: Gain
- Double-click on the result. The plot will appear in the GEOMETRY workspace window.
- Far zone data similar to what is shown in Figure 7.15 will appear.

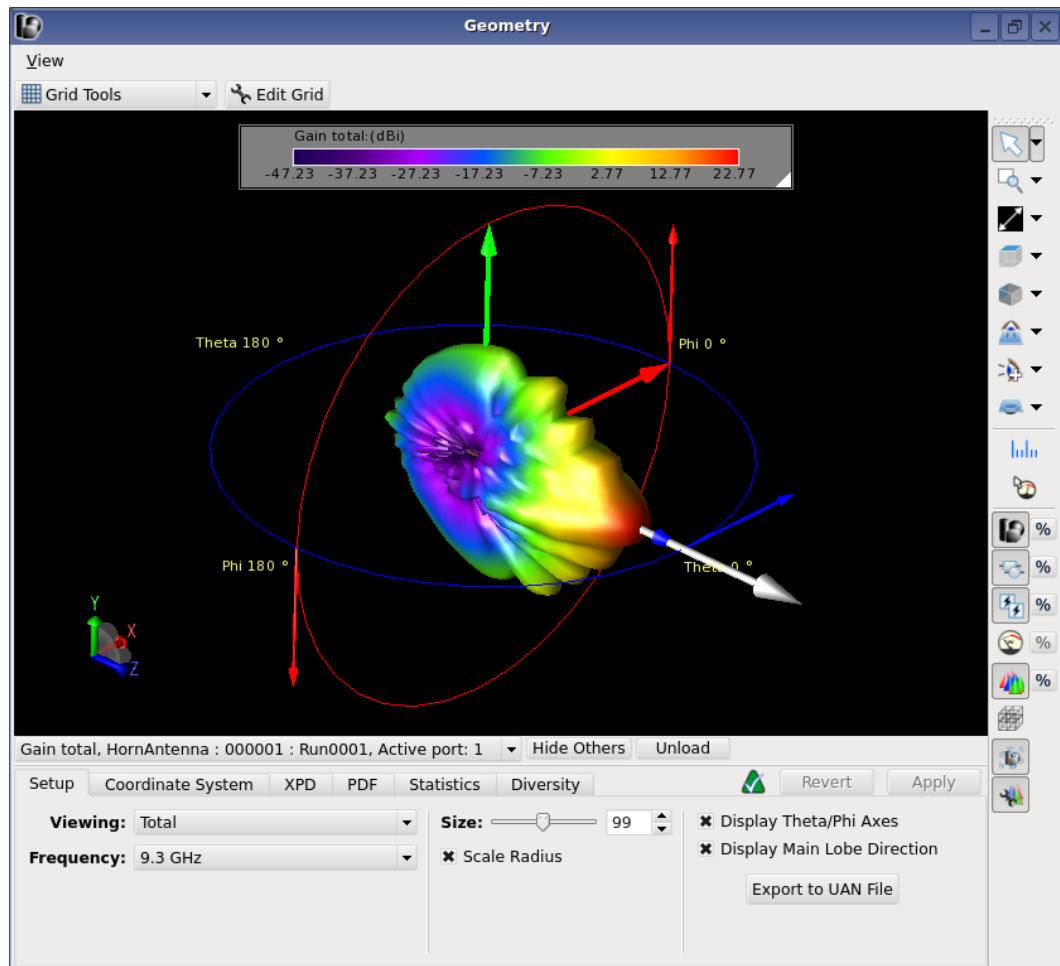


Figure 7.15: Far zone gain data for the horn antenna project



## Chapter 8

### Example: A Simple SAR Calculation

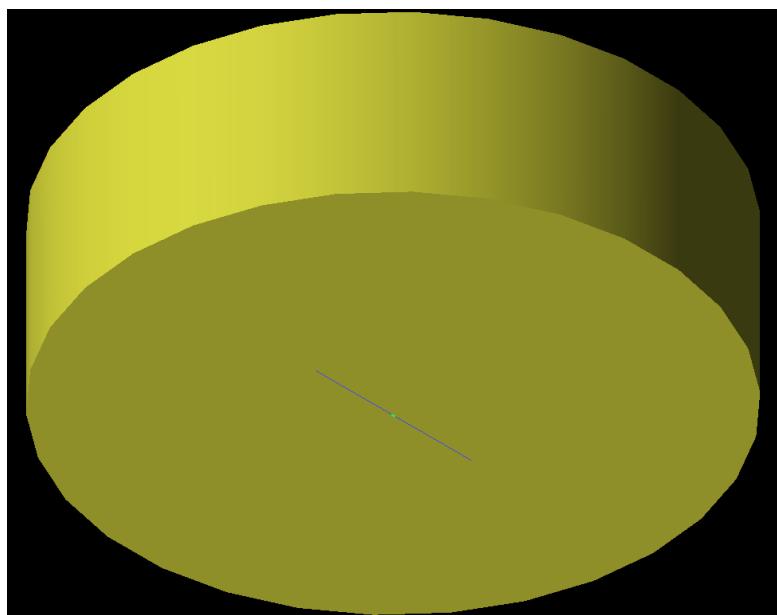


Figure 8.1: A Simple SAR example

Time to create: **25:00** (approx.)

In this chapter, you will learn how to...

- model a tissue with a dipole
- define the properties of the environment
- add a feed to the dipole and simulate its effects
- add a point sensor and measure E-field at the center of the tissue
- add an SAR sensor and retrieve SAR data

## 8.1 Getting Started

First, a few Project Properties are set up for the SAR project.

- The PROJECT PROPERTIES EDITOR opens automatically whenever XFDTD is started or when a new project is created. If the editor is not currently visible, double-click on PROJECT at the top of the PROJECT TREE to open the editor.
- On the FREQUENCY RANGE OF INTEREST tab of the PROJECT PROPERTIES EDITOR, specify a single frequency of interest by setting both the MINIMUM and MAXIMUM controls to "1 GHz".
- Navigate to the DISPLAY UNITS tab, where a few relevant units should be set:
  - Set FREQUENCY to "gigahertz (GHz)"
  - Set LENGTH to "millimeters (mm)"
- Press DONE on the PROJECT PROPERTIES EDITOR.

## 8.2 Creating the SAR Geometry

The geometry for this simple SAR calculation consists of a cylinder and a dipole antenna. The cylinder, named "Tissue", will be modeled with a simple EXTRUSION. The dipole antenna, named "Dipole", will be modeled with a WIRE BODY.

### 8.2.1 Modeling the Tissue

The Tissue will be created with a cylindrical EXTRUSION with a radius of 250 mm in the +Z direction.

- Right-click on the PARTS branch of the PROJECT TREE. Choose CREATE NEW > EXTRUDE from the context menu.

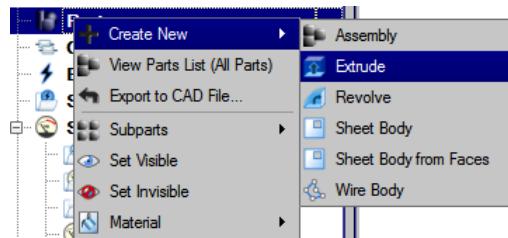


Figure 8.2: Selecting the geometry tool to perform an extrusion

- Enter “Tissue” as the NAME.
- Select the CONSTRUCTION GRID button.
  - Set the minor grid LINE SPACING to “10 mm”.
  - Set MOUSE SPACING to increments of “0.1 mm”.
  - Press OK.
- Choose the CIRCLE CENTER, RADIUS tool from the SHAPES toolbar.

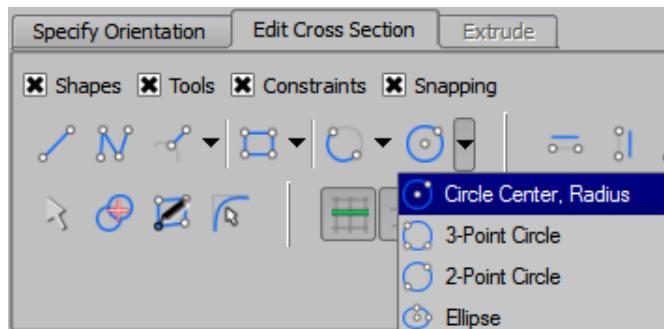


Figure 8.3: Selecting the circle tool

- Draw a circle by clicking the point (0, 0, 0) and then clicking on the point (250 mm, 0).

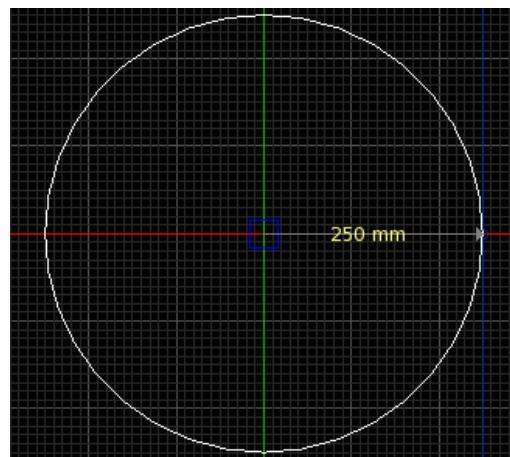


Figure 8.4: Sketching the circular cross-section of the cylinder

- Navigate to the EXTRUDE tab to extrude the cylindrical region. Enter “150 mm” as the distance, in the  $+Z$  direction.

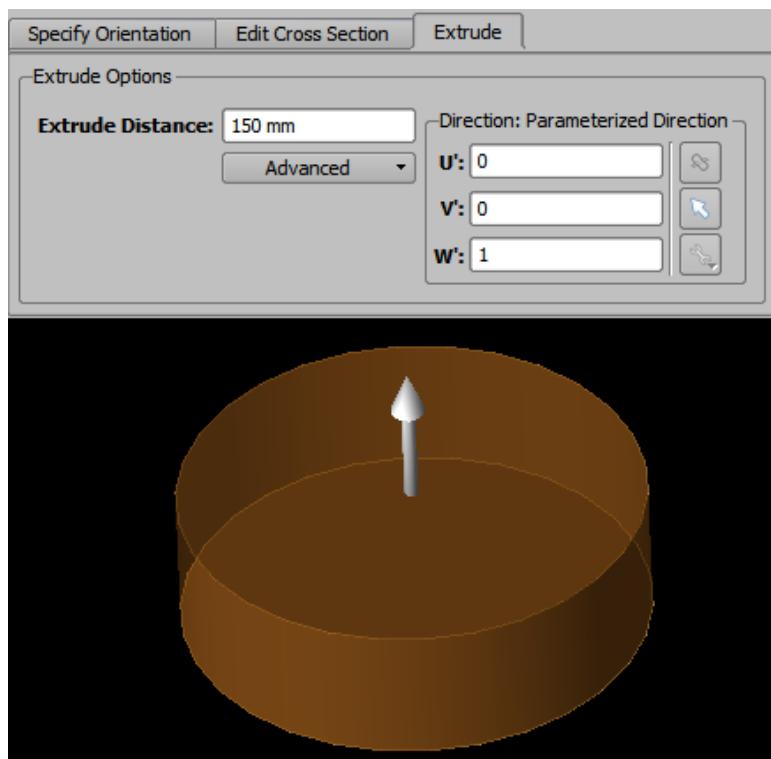


Figure 8.5: Sketching the circular cross-section of the cylinder

- Press DONE to complete the TISSUE geometry.

## 8.2.2 Modeling the Dipole

The Dipole will be created with a 100 mm WIRE BODY object that is centrally located just under the TISSUE cylinder.

- Right-click on the PARTS branch of the PROJECT TREE. Choose CREATE NEW > WIRE BODY from the context menu.
- Under the SPECIFY ORIENTATION tab, set the origin to (0, 0, -10 mm).
- Navigate to the EDIT WIRE BODY tab. In the NAME box, type “Dipole”.
- Select the STRAIGHT EDGE tool.
  - Press TAB to display the creation dialog for the first point. Enter (-50 mm, 0 mm) and press OK.
  - Press TAB to display the creation dialog for the second point. Enter (U: 1, V: 0, Length: 100 mm) and press OK to complete the Dipole.
- Click DONE to finish the DIPOLE geometry.

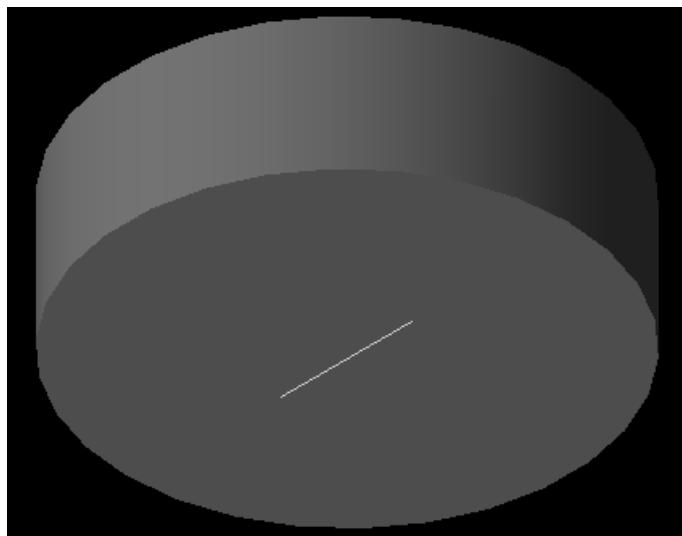


Figure 8.6: The finished SAR geometry

## 8.3 Creating Materials

### Define material, PEC

- Right-click on the DEFINITIONS: MATERIALS branch of the PROJECT TREE. Choose NEW MATERIAL DEFINITION from the context menu.
- Set the perfect electric conductor material properties as follows:

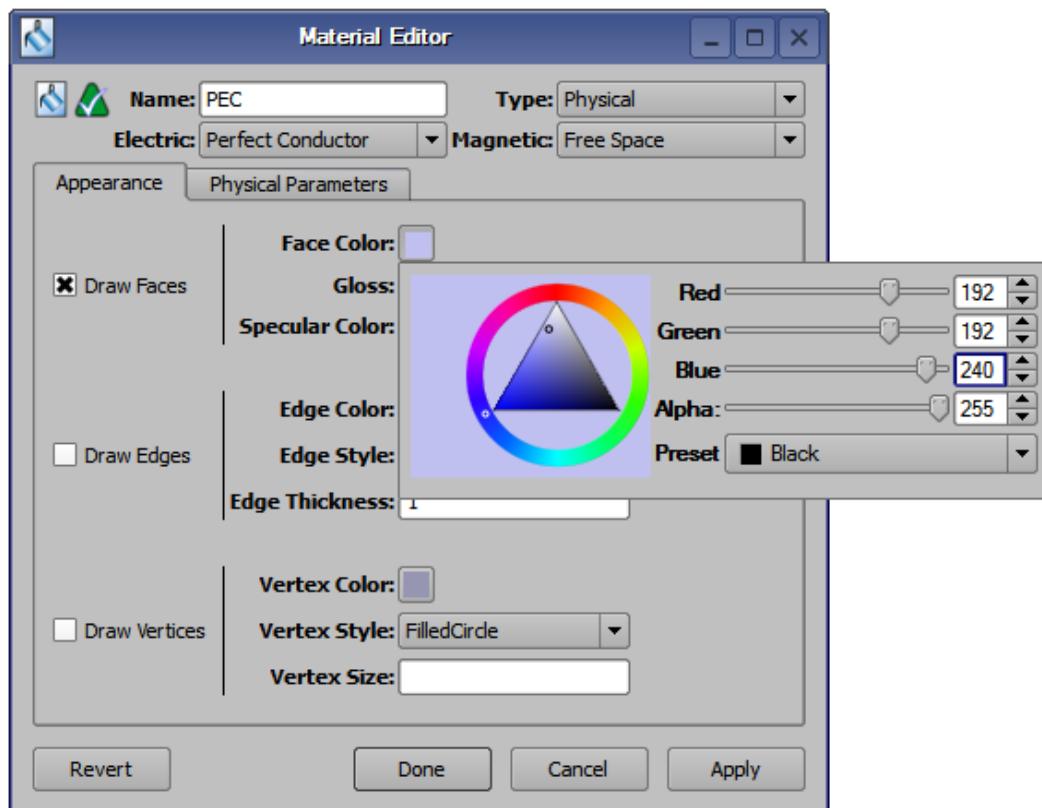


Figure 8.7: Editing the color of the PEC material

- NAME: PEC
- ELECTRIC: Perfect Conductor
- MAGNETIC: Freespace

→ If desired, navigate to the APPEARANCE tab to set the PEC material's display color.

#### Define Material, FAT, YELLOW MARROW

When a SAR sensor is enabled, XFDTD calculates SAR for all materials which are designated as “Tissue” and which have non-zero density. We will define a tissue material below.

- Right-click on the DEFINITIONS: MATERIALS branch of the PROJECT TREE. Choose NEW MATERIAL DEFINITION from the context menu.
- Set the material properties as follows:
- NAME: Fat, Yellow Marrow
  - ELECTRIC: Isotropic

- MAGNETIC: Freespace

Under the ELECTRIC tab:

- TYPE: Nondispersive
- ENTRY METHOD: Normal
- CONDUCTIVITY: "0.054069 S/m"
- RELATIVE PERMITTIVITY: "4.72728"

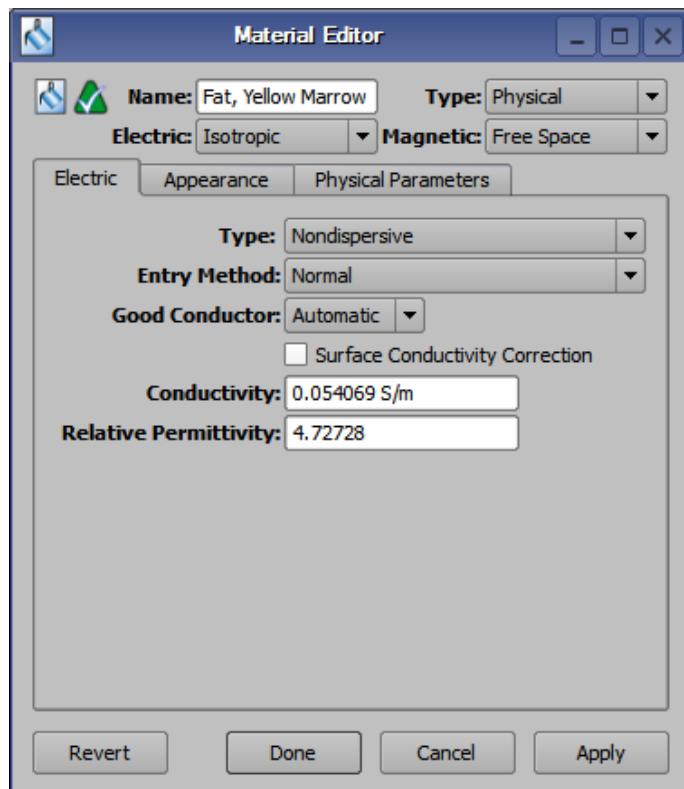


Figure 8.8: Editing the Fat, Yellow Marrow material

→ Navigate to the PHYSICAL PARAMETERS tab to set parameters used for the SAR computation.

- Check the TISSUE MATERIAL checkbox so that SAR is computed where this material is present.
- Enter "943 kg/m<sup>3</sup>" as the DENSITY of the material.

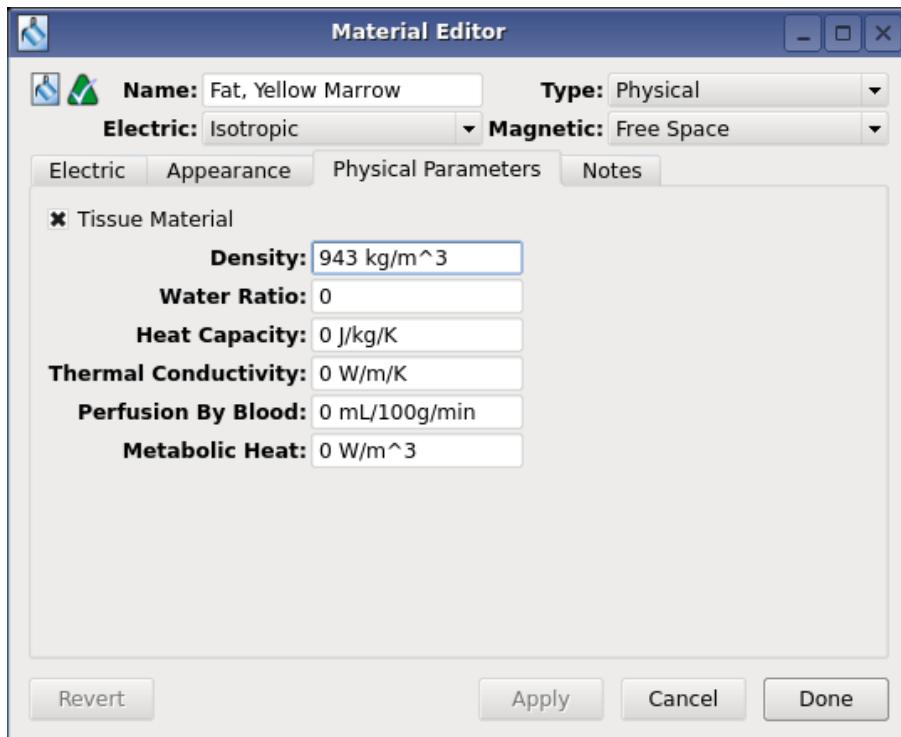


Figure 8.9: Assigning Physical Parameters to Material

- Navigate to the APPEARANCE tab and assign the FAT, YELLOW MARROW material a new color to distinguish it from PEC.
- Click DONE to add the new material, FAT, YELLOW MARROW.

## 8.4 Assigning Materials

- Click-and-drag the PEC material object located in the PROJECT TREE and drop it on top of the DIPOLE object in the PARTS branch of the tree.
- Assign the FAT, YELLOW MARROW material to the TISSUE object.

The finished geometry with applied materials is seen in Figure 8.10.

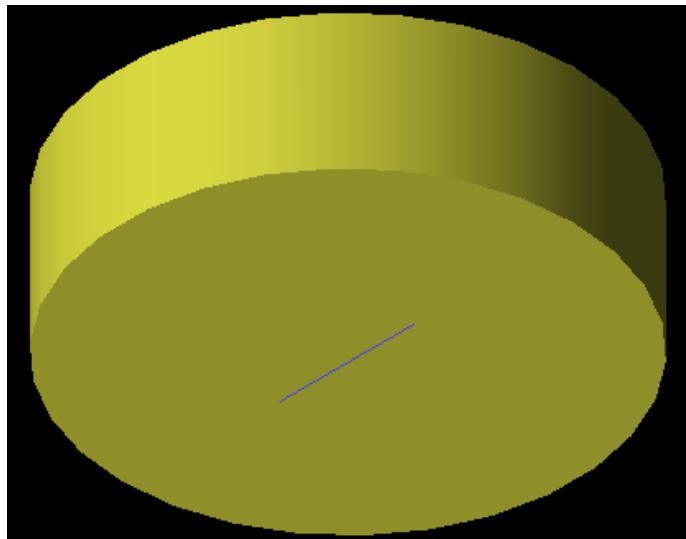


Figure 8.10: Finished geometry with applied material definitions

## 8.5 Defining the Outer Boundary

- Double-click on the FDTD: OUTER BOUNDARY branch of the PROJECT TREE to open the OUTER BOUNDARY EDITOR.
- Set the outer boundary properties as follows:
  - BOUNDARY: “Absorbing” for all boundaries
  - ABSORPTION TYPE: PML
  - LAYERS: 7

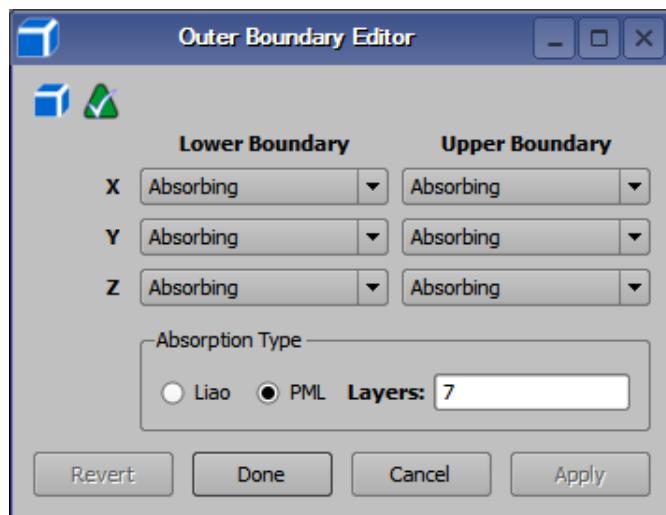


Figure 8.11: Defining the outer boundary for the SAR calculation

- Click DONE to apply the outer boundary settings.

## 8.6 Defining the Grid

Now we will define characteristics of the calculation grid.

- Double-click on the FDTD: GRID branch of the PROJECT TREE to open the GRID EDITOR.
- On the CELL SIZE tab (Figure 8.12), set MIN CELLS PER WAVELENGTH to “60”. This will increase the grid resolution, allowing the spatial distribution of SAR to be observed with a higher level of detail.

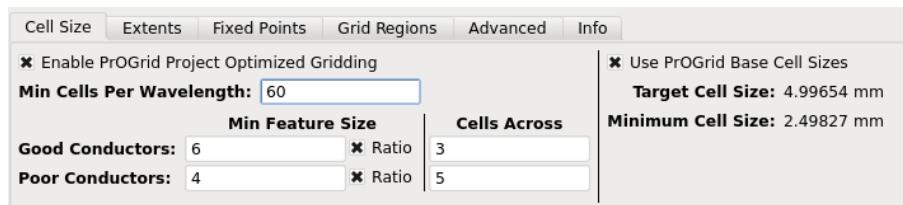


Figure 8.12: Defining cell size on the Grid Editor

- Click DONE to apply the grid settings.

### Adding fixed points to the geometry

In order to ensure that the distance between the dipole and the cylinder of tissue is accurately modeled, we now apply fixed points to both parts.

- In the PARTS branch of the PROJECT TREE, right-click on the TISSUE object and select GRIDDING / MESHING > GRIDDING PROPERTIES to open the GRIDDING PROPERTIES EDITOR.
  - Check USE AUTOMATIC FIXED POINTS.
- Click DONE to close the editor.
- Repeat this process to apply fixed points to the DIPOLE part as well.

## 8.7 Adding a Feed to the Dipole Wire

- Right-click on the CIRCUIT COMPONENTS branch of the PROJECT TREE. Choose NEW CIRCUIT COMPONENT WITH > NEW FEED DEFINITION from the context menu.

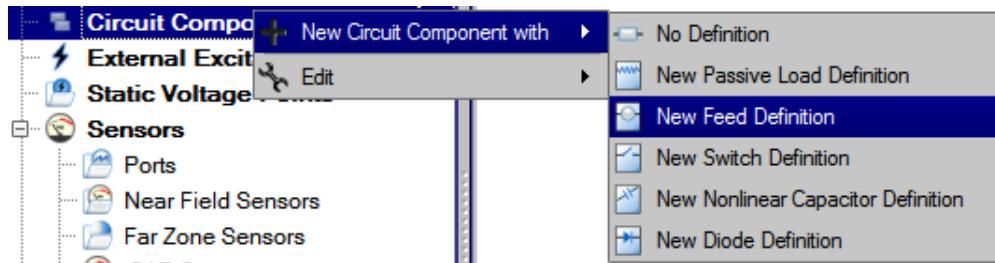


Figure 8.13: Adding a feed to the project

- Define the endpoints of the feed.
  - ENDPOINT 1: X: “-1 mm”, Y: “0 mm”, Z: “-10 mm”
  - ENDPOINT 2: X: “1 mm”, Y: “0 mm”, Z: “-10 mm”
- Navigate to the PROPERTIES tab, and enter the following:
  - NAME: Feed
  - COMPONENT DEFINITION: 50 Ω Voltage Source
  - DIRECTION: Auto
  - POLARITY: Positive
  - Check the box labeled THIS COMPONENT IS A PORT.
- Click DONE to add the FEED.

An associated waveform was automatically created for the feed definition. Because a single-valued FREQUENCY RANGE OF INTEREST was configured on the PROJECT PROPERTIES EDITOR, the Automatic waveform type will use a ramped sinusoid waveform at the desired frequency of 1 GHz.

## 8.8 Requesting Output Data

Recall that the project already contains one port sensor named FEED that will request results. We also wish to collect SAR results by adding an SAR SENSOR.

### Adding an SAR Sensor

→ Right-click on the SENSORS: SAR SENSOR branch of the PROJECT TREE. Select PROPERTIES from the context menu.

- Check the ENABLE COLLECTION OF RAW SAR DATA radio button.
- Select the Box radio button, and enter the following coordinates:
  - \* CORNER 1: (-250 mm, -250 mm, 0 mm)
  - \* CORNER 2: (250 mm, 250 mm, 150 mm)

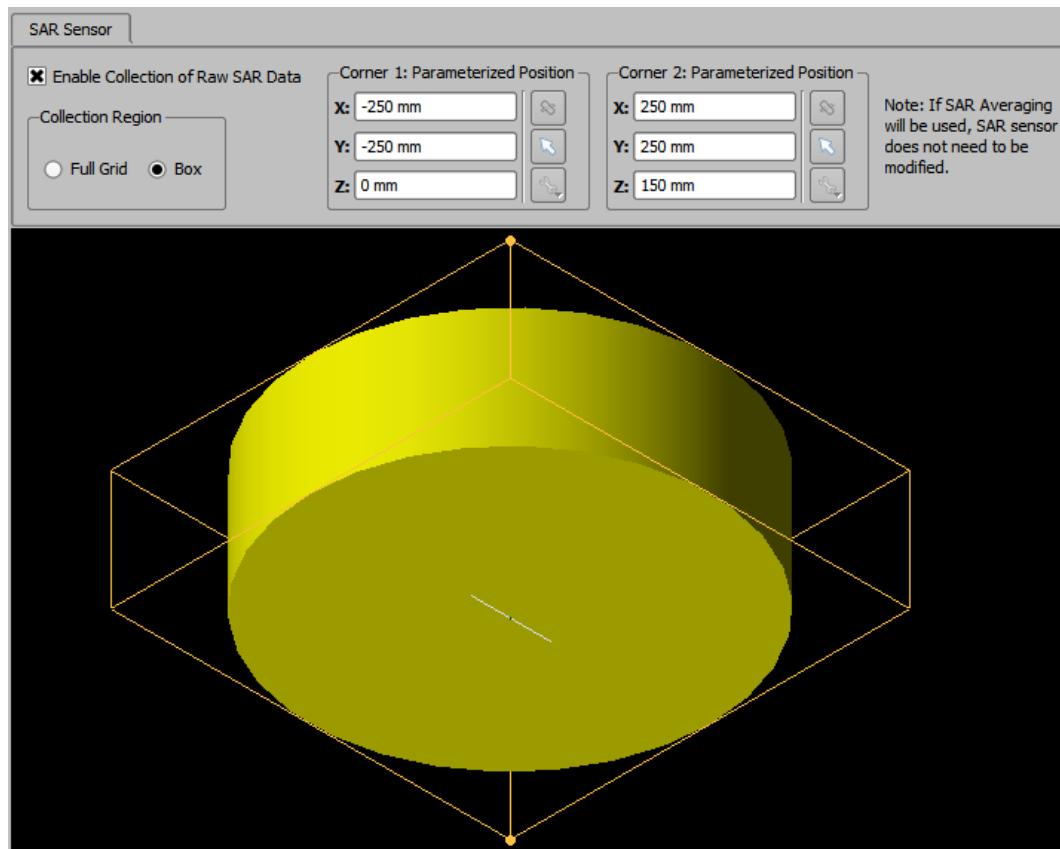


Figure 8.14: Adding the SAR sensor definition

→ Press DONE to finish editing the SAR SENSOR.

## Adding a Point Sensor Definition

A POINT SENSOR may be saved inside the TISSUE object to monitor the convergence of the fields during the calculation. First, we will create its definition.

- Right-click on the DEFINITIONS: SENSOR DATA DEFINITIONS branch of the PROJECT TREE. Choose NEW POINT SENSOR DEFINITION from the context menu.
- Set the properties of the surface sensor definition as follows:
  - NAME: E-field vs. Time
  - FIELD VS. TIME: E
  - SAMPLING INTERVAL: timestep

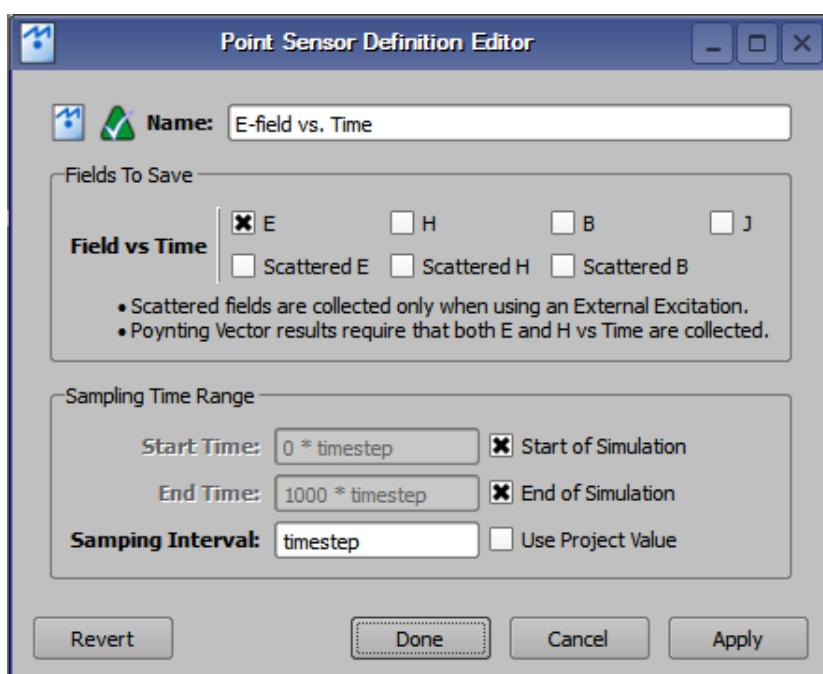


Figure 8.15: Adding the sensor definition

- Press DONE to finish editing the E-FIELDS VS. TIME definition.

## Adding a Point Sensor

- Right-click on the SENSORS: NEAR FIELD SENSORS branch of the PROJECT TREE. Select NEW POINT SENSOR from the context menu.
  - Enter its LOCATION as (0, 0, 75 mm).
  - Under the PROPERTIES tab, enter the following:

- \* NAME: E-field at Tissue Center
- \* SENSOR DEFINITION: E-field vs. Time
- \* SAMPLING METHOD: Snapped to E-Grid

→ Press DONE to finish editing the E-FIELD AT TISSUE CENTER Sensor.

## 8.9 Running the Calculation

If you have not already saved your project, do so by selecting FILE > SAVE PROJECT. Once the project is saved, a new simulation can be created to send to the calculation engine.

- Open the SIMULATIONS workspace window. Click the FDTD button in the upper-left to set up a new simulation.
- Type in a descriptive NAME for the simulation, such as “Tissue Cylinder exposed to 1 GHz Dipole, SAR Saved”.
- Under FREQUENCIES OF INTEREST, check the COLLECT STEADY-STATE DATA box.
- Under the FREQUENCIES tab, check USE WAVEFORM FREQUENCY.
- Most of the default settings are sufficient. Navigate to the SPECIFY TERMINATION CRITERIA tab. Set up the termination criteria as follows:
  - ANALYZE PROJECT CONTENTS: Unchecked
  - DETECT CONVERGENCE: Checked
  - THRESHOLD: -20 dB
  - MAXIMUM SIMULATION TIME: 10000 \* timestep
- Select CREATE AND QUEUE SIMULATION to close the dialog and run the new simulation.

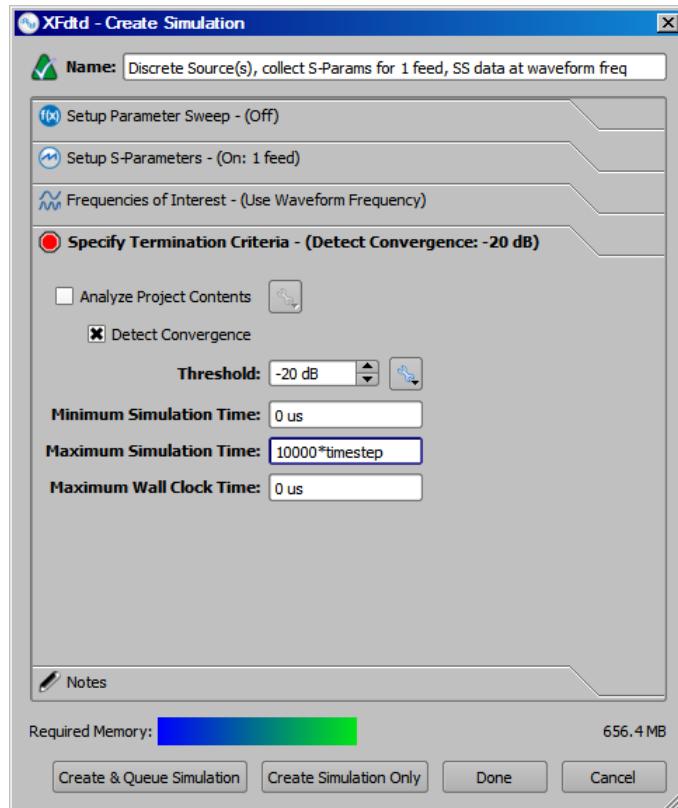


Figure 8.16: Setting up the simulation for the SAR project

## 8.10 Viewing the Results

The OUTPUT tab of the SIMULATIONS workspace window displays the progress of the simulation. Once the STATUS column shows that the simulation has completed, we can view its results in the RESULTS workspace window.

### E-field Results

Now we will view the E-field results retrieved from the center of the TISSUE.

- To filter the list accordingly, select the following options in the columns in the top pane of the RESULTS window. (You may need to change your column headings first.)
  - SENSOR: E-field at Tissue Center
  - RESULT TYPE: E-field (E)
- Right-click on the result and select CREATE LINE GRAPH.

- Select “X” as the COMPONENT, and press VIEW. The plot of the E-field at the center of the TISSUE object will appear.

- ✓ It is possible to view the data before the simulation is complete. The plot will update automatically as more data is computed.

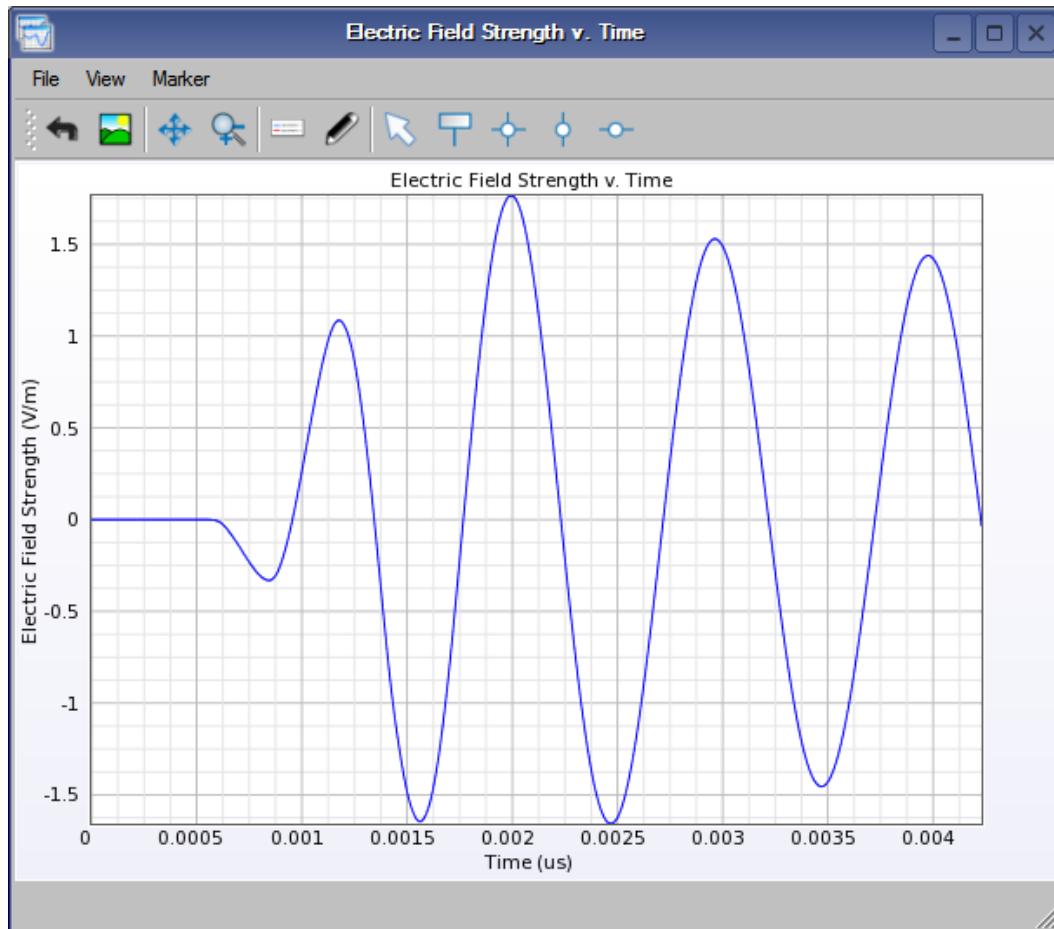


Figure 8.17: Viewing the E-field at the center of the Tissue

- You may close the window when you are finished viewing the results.

## Steady State Feed Results

Now we will view steady state result data from the feed.

- To view the FEED results, select the following:
  - SENSOR: Feed
  - RESULT TYPE: S-Parameters

- Double-click on the result under the “Discrete Frequencies” domain. The following results will appear showing the impedance at the feed, the input power delivered, and the return loss.

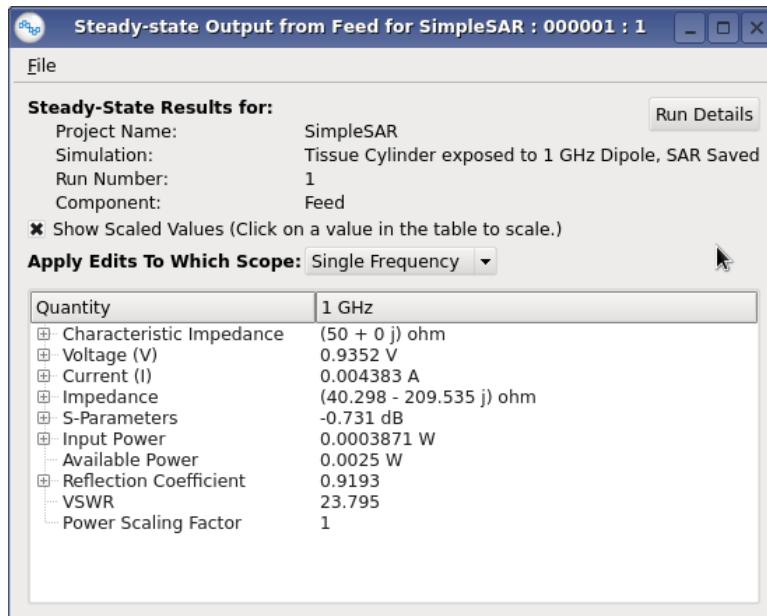


Figure 8.18: Viewing SAR results at the Feed

- You may close the window when you are finished viewing the results.

## SAR Sensor Data

Now we will load the SAR data into the field viewer.

- To view the SAR sensor data, select the following:
- SENSOR: SAR Sensor (Raw)
  - RESULT TYPE: SAR (Specific Absorption Rate)
- Double-click on the result in the filtered list. The plot will appear in the GEOMETRY workspace window.
- Under the SETUP tab, as seen in Figure 8.19, adjust the following settings:
- SEQUENCE AXIS: Z
  - DECIMATION: Finest
  - The axis range controls for the independent variables Frequency, X, and Y are sufficient as-is:
    - \* FREQUENCY: Only one steady-state data collection frequency was defined for this run (1 GHz).
    - \* X RANGE: The full range of X (from -249.55 mm to 249.55 mm) is selected by default.

- \* Y RANGE: The full range of Y (from -247.82 mm to 247.82 mm) is selected by default.
- \* To change which values are displayed for any axis, click the MODIFY TOOL button next to that axis display.

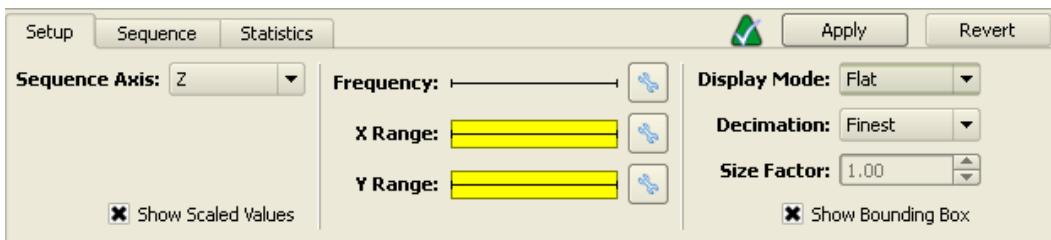


Figure 8.19: Setting the view of the SAR sensor data

- Press APPLY to finish editing the SAR sensor setup.
- Toggle the PARTS VISIBILITY to turn off the display of the geometry, and select the TOP (+Z) orientation. The SAR image shown in Figure 8.20 should appear.
- Click the SEQUENCE tab and PLAY button, to view a movie of the SAR slices. To increase the speed of the movie, change the DECIMATION on the SETUP tab to NORMAL.
- To review the SAR statistics of the peak and average SAR, press the Statistics tab.

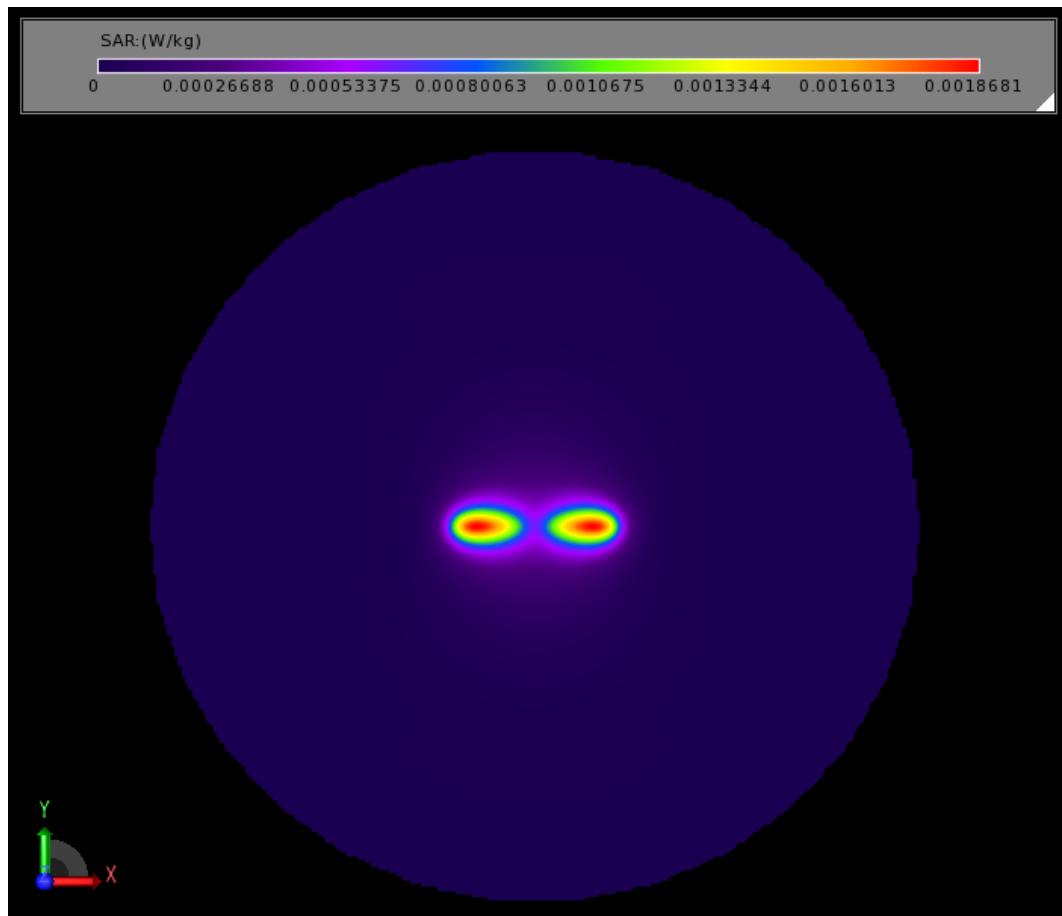


Figure 8.20: SAR sensor data



## Chapter 9

### Example: A Validation of SAR Calculations

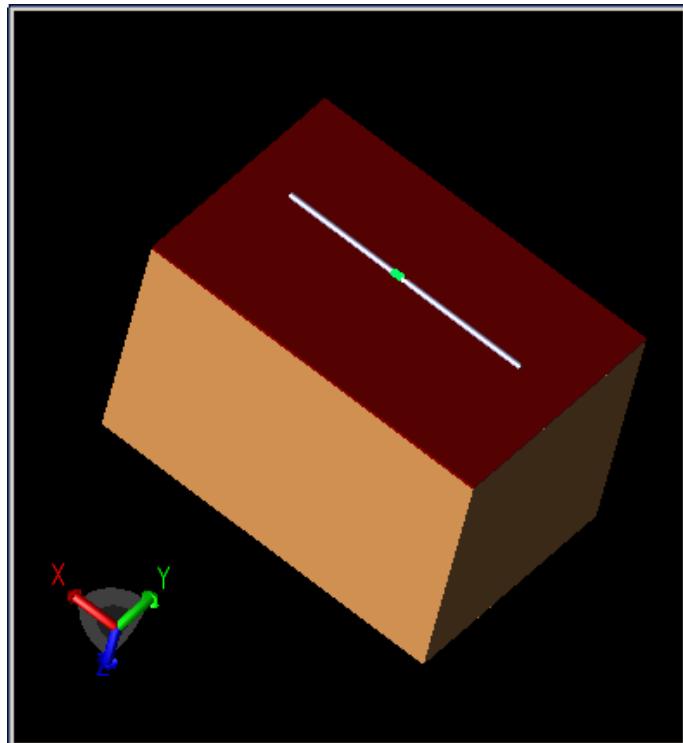


Figure 9.1: The Flat Phantom

Time to create: 25:00 (approx.)

In this chapter, you will learn how to...

- model a tissue-simulating liquid with a dipole
- define the properties of the environment
- add a feed to the dipole and simulate its effects
- add a point sensor and measure E-field at the center of the liquid
- add SAR sensors and retrieve raw and averaged SAR data

This example simulates an experimental setup for calibrating SAR measurement apparatus at 835 MHz. The subject under test is a flat phantom comprising a plastic shell and a tissue-equivalent liquid. The phantom is exposed to fields generated by a dipole located 15 mm from the shell/liquid interface. Raw SAR values as well as 1 g and 10 g averaged SAR quantities are then observed relative to specified input power or feed current quantities.

## 9.1 Getting Started

First, a few Project Properties are set up for the SAR Validation project.

- The PROJECT PROPERTIES EDITOR opens automatically whenever XFDTD is started or when a new project is created. If the editor is not currently visible, double-click on PROJECT at the top of the PROJECT TREE to open the editor.
- On the FREQUENCY RANGE OF INTEREST tab of the PROJECT PROPERTIES EDITOR, specify a single frequency of interest by setting both the MINIMUM and MAXIMUM controls to "835 MHz".
- Navigate to the DISPLAY UNITS tab, where a few relevant units should be set:
  - Set FREQUENCY to "megahertz (MHz)"
  - Set LENGTH to "millimeters (mm)"
  - Set CURRENT to "milliamperes (mA)"
- Press DONE on the PROJECT PROPERTIES EDITOR.

## 9.2 Creating the Geometry

The geometry for this example consists of a FLAT PHANTOM, PHANTOM SHELL, and a dipole made of two CYLINDERS.

### 9.2.1 Modeling the Flat Phantom

First, we will create the rectangular extrusion named "Flat Phantom" which represents the tissue-simulating liquid used for SAR measurements. Phantom dimensions appropriate for simulation at 835 MHz are 220 x 150 mm with an depth of 150 mm.

- Right-click on the PARTS branch of the PROJECT TREE. Choose +CREATE NEW > EXTRUDE from the context menu.

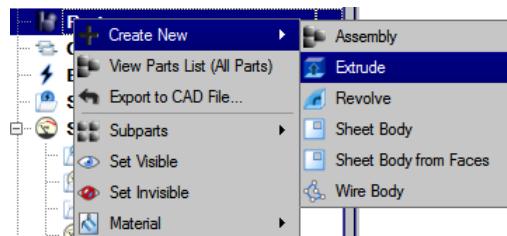


Figure 9.2: Selecting the geometry tool to perform an extrusion

- Name the substrate by typing “Flat Phantom” in the NAME box.
- In the VIEW TOOLS toolbar, select the VIEW FROM +Z (TOP) orientation.
- Choose the RECTANGLE tool from the SHAPES toolbar.

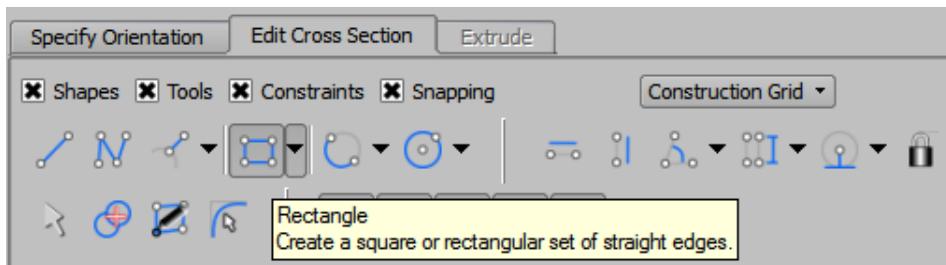


Figure 9.3: Selecting the rectangle tool

- Click the mouse on the origin of the coordinate system.
- Press **TAB** to display the creation dialog for the second point. Enter (225 mm, 150 mm) and press OK to complete the rectangle.
- Navigate to the EXTRUDE tab to extrude the rectangular region. Enter a distance of 150 mm.

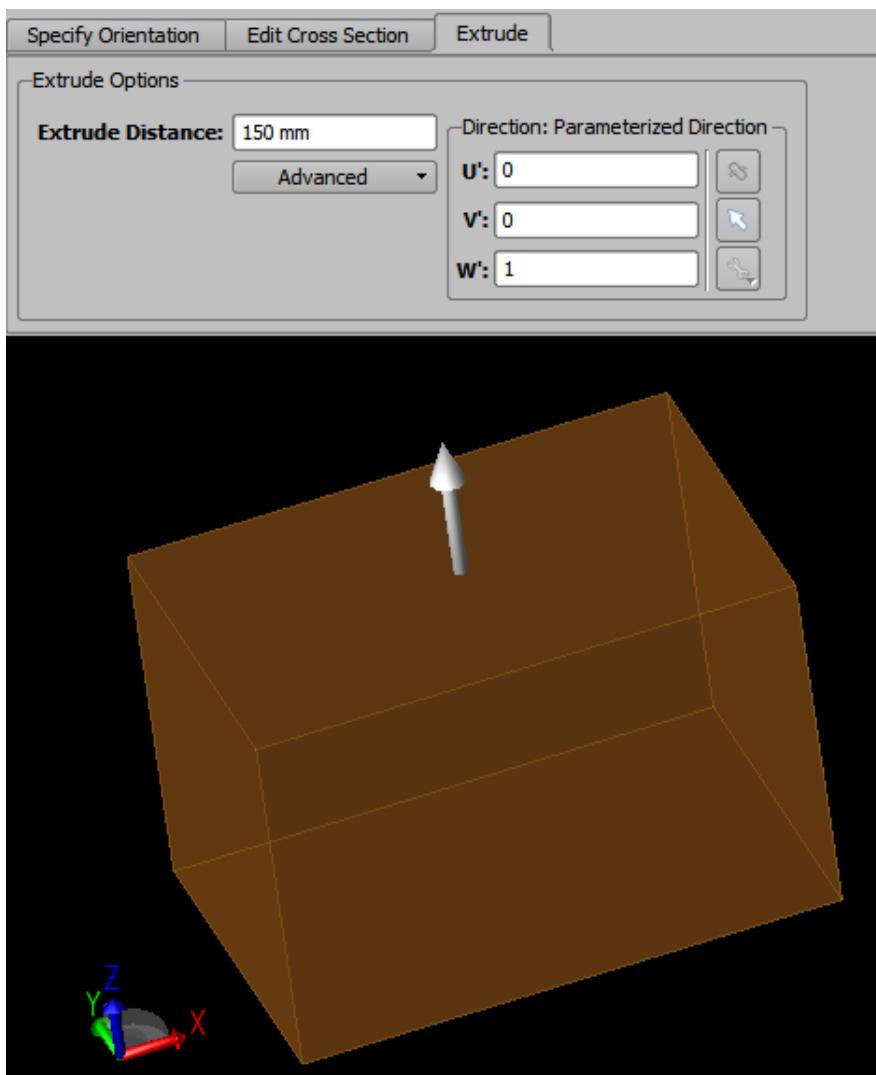


Figure 9.4: Specifying the extrude distance

→ Press DONE to finish the FLAT PHANTOM geometry.

### 9.2.2 Modeling the Phantom Shell

Next, we will create the rectangular extrusion named “Phantom Shell”. This shell is a plastic vessel that will hold the simulating liquid. For our simulation, we need only add the bottom of the vessel that separates the liquid from the dipole source. This shell size will match the phantom size in  $X$  and  $Y$ , and have a thickness of 2 mm.

→ Right-click on the PARTS branch and choose CREATE NEW > EXTRUDE from the context

menu.

- Under the EDIT CROSS SECTION tab, type “Phantom Shell” in the NAME box.
- In the VIEW TOOLS toolbar, select the VIEW FROM -Z (BOTTOM) orientation.
- Choose the RECTANGLE tool from the SHAPES toolbar.
- Trace the new cross-section over the existing cross-section (of the flat phantom) since they are of equal width and length.

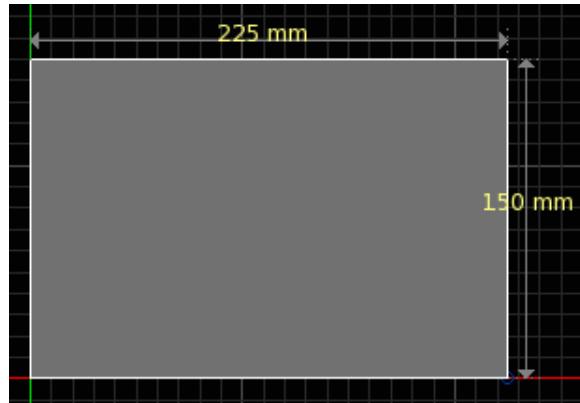


Figure 9.5: Specifying the phantom shell size

- Navigate to the EXTRUDE tab to extrude the rectangular region a distance of -2 mm. Extruding in the negative direction here (when the FLAT PHANTOM was extruded in the positive direction) prevents the Shell and Phantom from overlapping in space.
- Press DONE to finish the FLAT PHANTOM geometry.

### 9.2.3 Modeling the Dipole

Now we will create the dipole geometry, which is comprised of two cylindrical extrusions. Typically the dipole will have a balun structure as well, but we will omit that for simplicity in this example. The dipole will have a radius of 1.8 mm, a length of 161 mm, and will be centered at  $Z = -15$  mm (giving the desired spacing of 15 mm between the dipole and the shell/liquid interface, which is at  $Z = 0$ ).

- Right-click on the PARTS branch and choose CREATE NEW > EXTRUDE from the context menu.
- Under the SPECIFY ORIENTATION tab, define the origin at (113 mm, 75 mm, -15 mm).
  - Redefine the orientation of the sketching plane by selecting the YZ PLANE under the PRESETS drop-down.

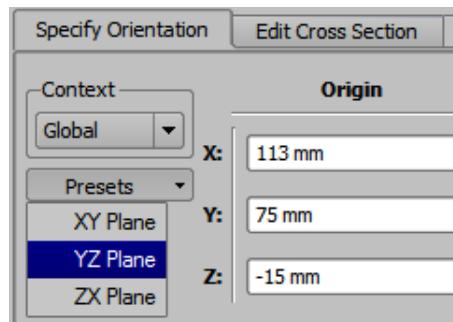


Figure 9.6: Specifying the cylinder sketching plane orientation

- Under the EDIT CROSS SECTION tab, type “Cylinder1” in the NAME box.
- In the VIEW TOOLS toolbar, select the VIEW FROM -X (LEFT) orientation.
- Choose the CIRCLE CENTER, RADIUS tool from the SHAPES toolbar.
  - Click the mouse on the origin of the coordinate system.
  - Press TAB to display the creation dialog for the radius. Enter 1.8 mm and press OK.

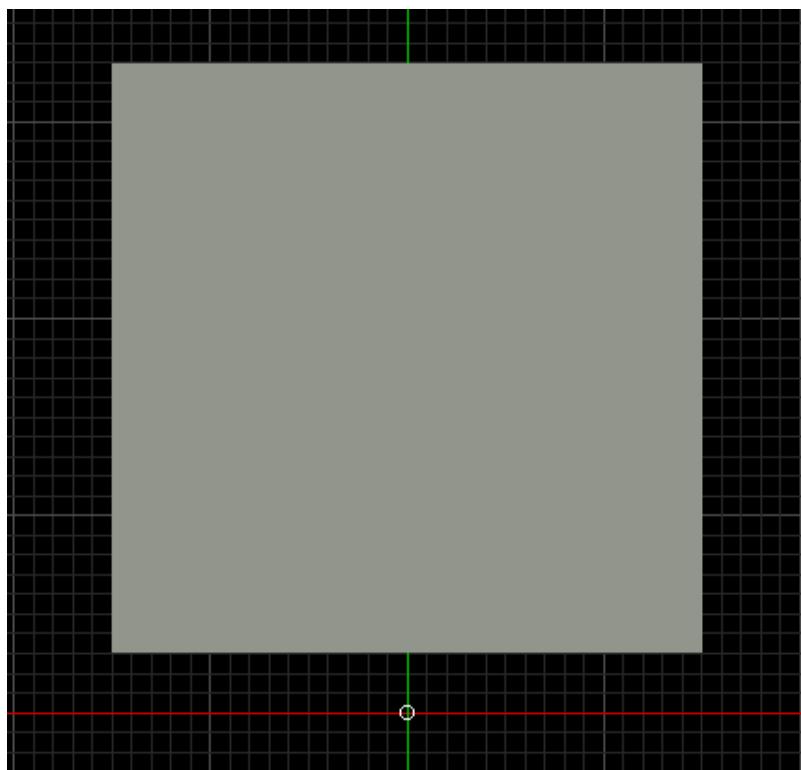


Figure 9.7: A larger view of the cross-section in relation to the phantom geometry

- Navigate to the EXTRUDE tab to extrude the cylinder. Enter a distance of 80 mm.
- Press DONE to finish the CYLINDER1 geometry.

### Create the second extrusion

Now we will create the second part of the dipole, CYLINDER2.

- Right-click on the PARTS branch and choose CREATE NEW > EXTRUDE.
- Under the SPECIFY ORIENTATION tab, define the origin at (32 mm, 75 mm, -15 mm).
  - Redefine the orientation of the sketching plane by selecting the YZ PLANE under the PRESETS drop-down.
- Under the EDIT CROSS SECTION tab, type “Cylinder2” in the NAME box.
- In the VIEW TOOLS toolbar, select the VIEW FROM +X (RIGHT) orientation.
- Choose the CIRCLE CENTER, RADIUS tool from the SHAPES toolbar.
  - Click the mouse on the origin of the coordinate system.
  - Press TAB to display the creation dialog for the radius. Enter 1.8 mm and press OK.
- Navigate to the EXTRUDE tab to extrude the cylinder. Enter a distance of 80 mm.
- Press DONE to finish the CYLINDER2 geometry.

Figure 9.8 shows a view of the finished geometry.

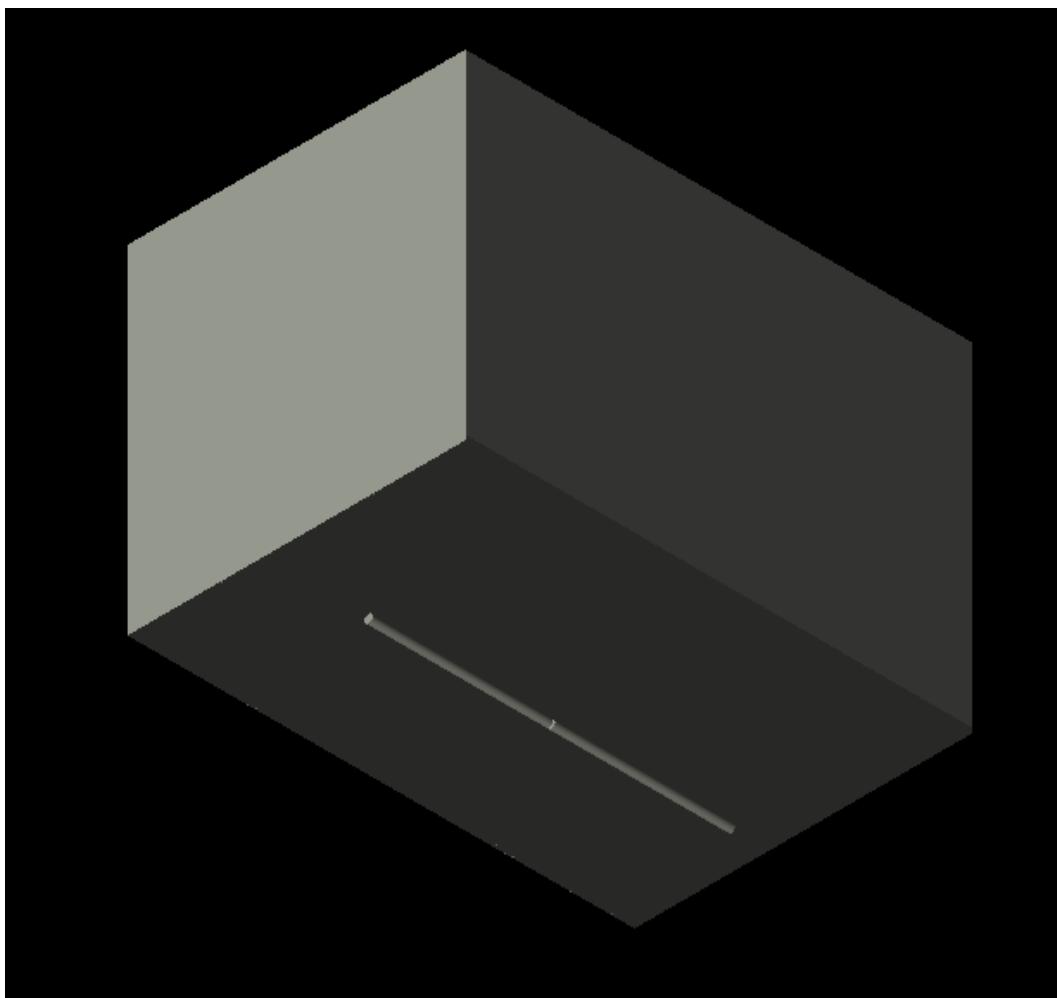


Figure 9.8: The finished Phantom geometry

### 9.3 Creating Materials

#### Define material, PEC

- Right-click on the DEFINITIONS: MATERIALS branch of the PROJECT TREE. Choose NEW MATERIAL DEFINITION from the context menu.
- Set the perfect electric conductor material properties as follows:
  - NAME: PEC
  - ELECTRIC: Perfect Conductor
  - MAGNETIC: Freespace

- If desired, navigate to the APPEARANCE tab to set the PEC material's display color.

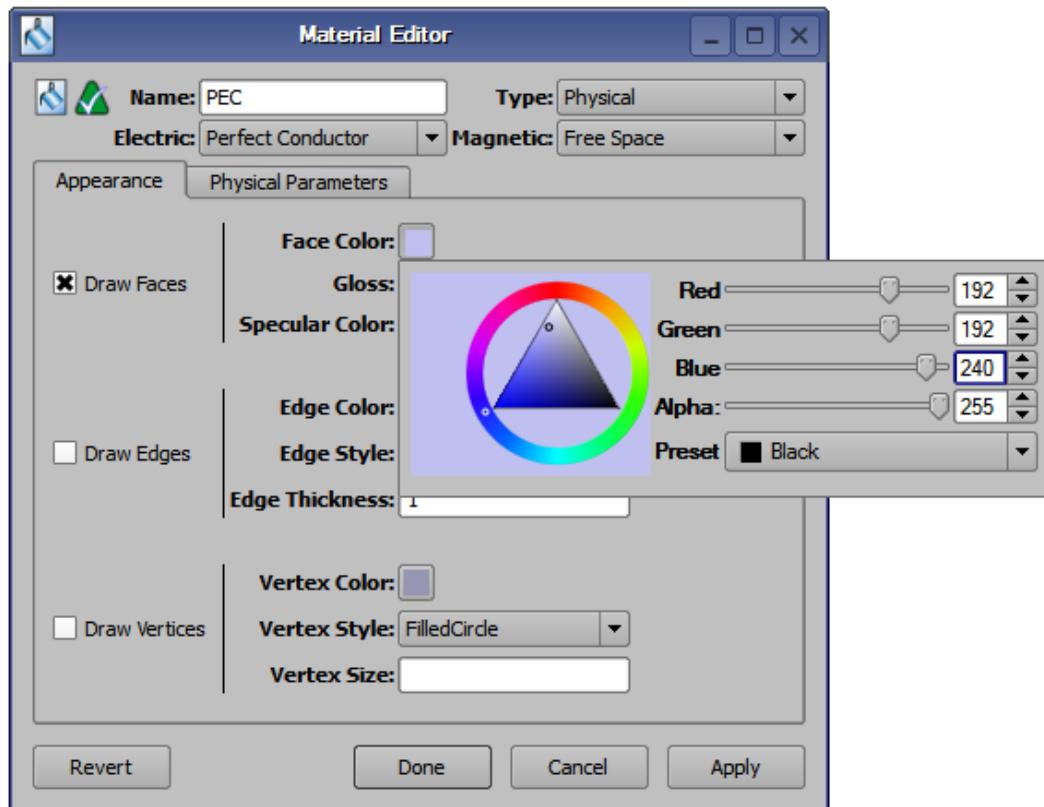


Figure 9.9: Editing the color of the PEC material

### Define Material, PHANTOM LIQUID

- Right-click on the DEFINITIONS: MATERIALS branch of the PROJECT TREE and select NEW MATERIAL DEFINITION.
- Set the material properties as follows:
- NAME: Phantom Liquid
  - ELECTRIC: Isotropic
  - MAGNETIC: Freespace
- Under the ELECTRIC tab:
- TYPE: Nondispersive
  - ENTRY METHOD: Normal
  - CONDUCTIVITY: “0.9 S/m”

- RELATIVE PERMITTIVITY: “41.5”

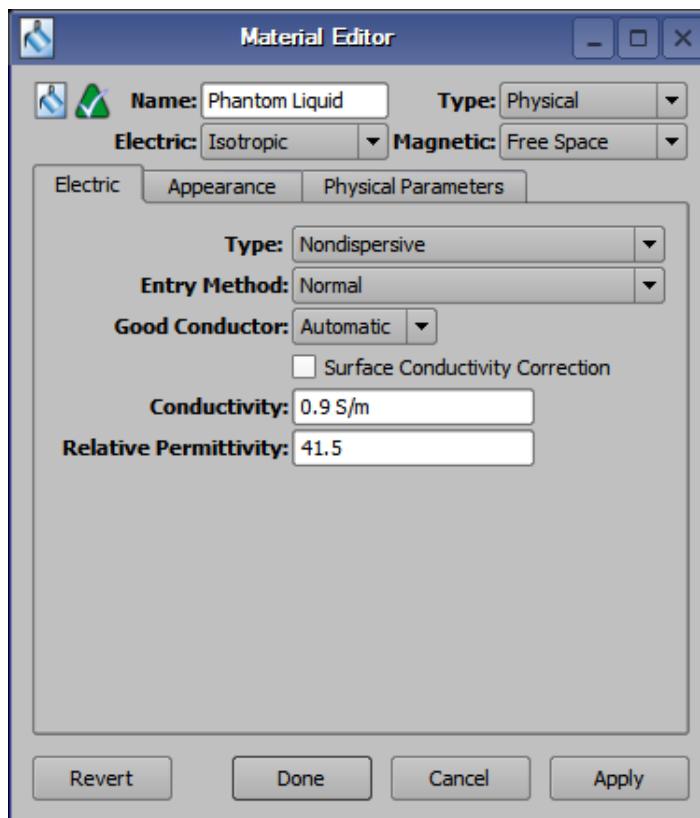


Figure 9.10: Editing the electric properties of the Phantom Liquid material

- Navigate to the PHYSICAL PARAMETERS tab to set parameters used for the SAR computation.
- Check the TISSUE MATERIAL checkbox so that SAR is computed where this material is present.
  - Enter “1000 kg/m<sup>3</sup>” as the DENSITY of the material.

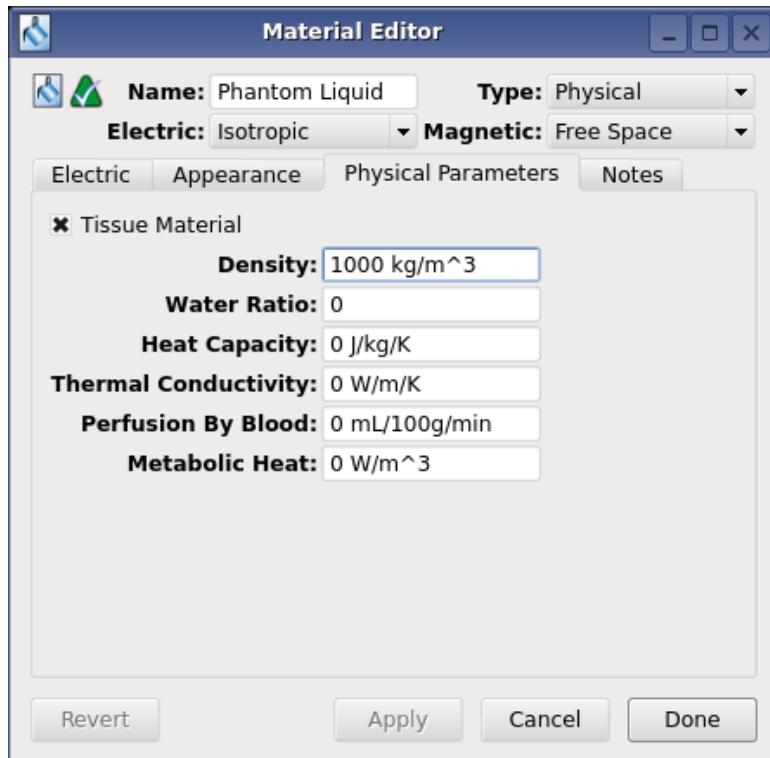


Figure 9.11: Editing the physical parameters Phantom Liquid material

- Navigate to the APPEARANCE tab and assign the PHANTOM LIQUID material a new color to distinguish it from PEC.
- Click DONE to add the new material, PHANTOM LIQUID.

#### Define material, PHANTOM SHELL

- Right-click on the DEFINITIONS: MATERIALS branch of the PROJECT TREE and select NEW MATERIAL DEFINITION.
- Set the material properties as follows:
  - NAME: Phantom Shell
  - ELECTRIC: Isotropic
  - MAGNETIC: Freespace
 Under the ELECTRIC tab:
  - TYPE: Nondispersive
  - ENTRY METHOD: Normal

- CONDUCTIVITY: “0 S/m”
- RELATIVE PERMITTIVITY: “3.7”

- Navigate to the APPEARANCE tab and assign the PHANTOM SHELL material a new color to distinguish it from PEC.
- Click DONE to add the new material, PHANTOM SHELL.

## 9.4 Assigning Materials

- Click-and-drag the PEC material object located in the PROJECT TREE and drop it on top of CYLINDER1 and CYLINDER2.
- Assign the PHANTOM LIQUID material to the FLAT PHANTOM object.
- Assign the PHANTOM SHELL material to the PHANTOM SHELL object.

## 9.5 Defining the Outer Boundary

- Double-click on the FDTD: OUTER BOUNDARY branch of the PROJECT TREE to open the OUTER BOUNDARY EDITOR.
- Set the outer boundary properties as follows:
- BOUNDARY: “Absorbing” for all boundaries
  - ABSORPTION TYPE: PML
  - LAYERS: 7

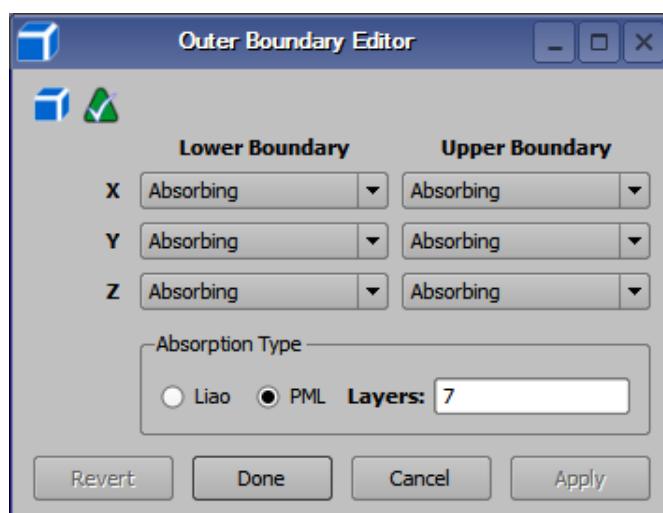


Figure 9.12: Defining the outer boundary for the SAR Phantom calculation

- Click DONE to apply the outer boundary settings.

## 9.6 Defining the Grid

Now we will define characteristics of the calculation grid.

- Open the **GEOMETRY** browser window, select **GRID TOOLS** and click on **EDIT GRID**.
- On the **SIZE** tab, set **MIN CELLS PER WAVELENGTH** to “55”. This will increase the grid resolution, allowing the spatial distribution of SAR to be observed with a higher level of detail.
- The default **MIN FEATURE SIZE** for **POOR CONDUCTORS** will cause the grid to resolve the 2 mm-thick Phantom Shell with at least 5 cells. As we don't expect fields to be rapidly changing across that distance, we can increase the **MIN FEATURE SIZE** giving fewer cells in the Shell and a faster runtime.
  - Uncheck the **POOR CONDUCTORS MIN FEATURE SIZE RATIO** checkbox to allow an absolute distance to be entered.
  - Enter a value in **POOR CONDUCTORS MIN FEATURE SIZE** that is larger than the shell thickness (e.g. “2 mm”).

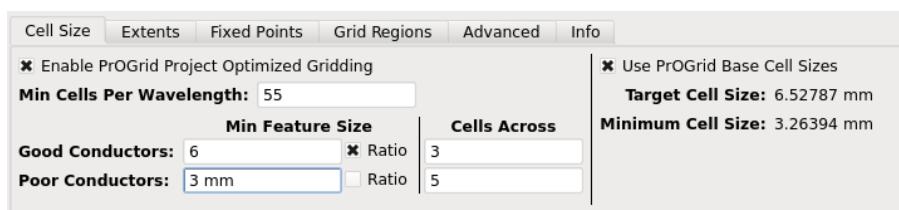


Figure 9.13: Defining cell size and min feature size on the Grid Editor

- Click **DONE** to apply the grid settings.

## Adding fixed points to the geometry

In order to ensure that the distance between the dipole and the tissue/shell interface is accurately represented in the mesh, we now apply fixed points to those parts.

- In the **PARTS** branch of the **PROJECT TREE**, right-click on the **FLAT PHANTOM** object and select **GRIDDING / MESHING** > **GRIDDING PROPERTIES** to open the **GRIDDING PROPERTIES EDITOR**.
  - Check **USE AUTOMATIC FIXED POINTS**.
  - Press **COPY TO CLIPBOARD** to save these settings.
- Click **DONE** to close the editor.
- Select both **CYLINDER1** and **CYLINDER2** from the **PARTS** branch. Right-click and select **EDIT > PASTE** to copy the clipboard contents to these two objects. This will turn on fixed points for the dipole as well.

## 9.7 Adding a Feed

Now, we will add a FEED to the geometry. We want to place the feed in the gap between the two cylinders made of PEC materials.

- Right-click on the CIRCUIT COMPONENTS branch of the PROJECT TREE. Choose NEW CIRCUIT COMPONENT WITH > NEW FEED DEFINITION from the context menu.

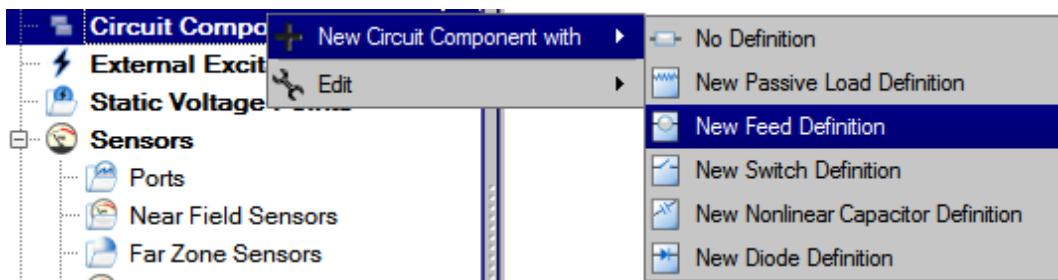


Figure 9.14: Adding a feed to the project

- Define the endpoints of the feed.
  - ENDPOINT 1: X: "113 mm", Y: "75 mm", Z: "-15 mm"
  - ENDPOINT 2: X: "112 mm", Y: "75 mm", Z: "-15 mm"
- Navigate to the PROPERTIES tab, and enter the following:
  - NAME: Feed
  - COMPONENT DEFINITION: 50  $\Omega$  Voltage Source
  - DIRECTION: Auto
  - POLARITY: Positive
  - Check the box labeled THIS COMPONENT IS A PORT.
  - Press the ADVANCED button to expose additional grid-related settings. Check the ENABLE FIXED POINT ON ENDPOINT 1 and ENABLE FIXED POINT ON ENDPOINT 2 checkboxes as shown in Figure 9.15. These additional fixed points will make sure that the Yee cell edge representing the Feed is centered within the circular cross-section of the dipole.
- Click DONE to add the FEED.

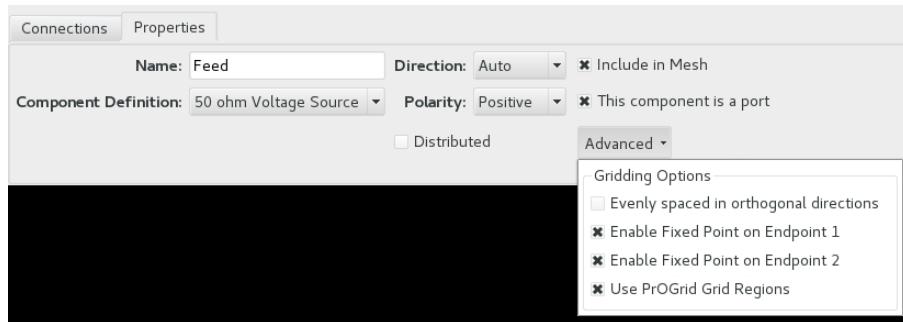


Figure 9.15: Defining Feed Properties on the Circuit Component Editor

An associated waveform was automatically created for the feed definition. Because a single-valued FREQUENCY RANGE OF INTEREST was configured on the PROJECT PROPERTIES EDITOR, the Automatic waveform type will use a ramped sinusoid waveform at the desired frequency of 835 MHz.

## 9.8 Requesting Output Data

Recall that the project already contains one port sensor named FEED that will request results. We also wish to collect SAR results by adding an SAR SENSOR.

### Adding an SAR Sensor

- Right-click on the SENSORS:SAR SENSOR branch of the PROJECT TREE. Select PROPERTIES from the context menu.
  - Check the ENABLE COLLECTION OF RAW SAR DATA radio button.
  - Select the FULL GRID radio button. XFdtd requires that the data be saved over the full grid if Averaged SAR values will be computed.
- Press DONE to finish editing the SAR SENSOR.

To collect averaged SAR data, we must define a sensor for it as well.

- Right-click on the SENSORS:SAR AVERAGING SENSOR branch of the PROJECT TREE. Select PROPERTIES from the context menu.
  - Check the ENABLE SAR AVERAGING box.
  - Check the COLLECT 1-GRAM AVG. SAR DATA and COLLECT 10-GRAM AVG. SAR DATA boxes.
  - Select the Box radio button, and enter the following coordinates:
    - \* CORNER 1: (0 mm, 0 mm, 0 mm)
    - \* CORNER 2: (225 mm, 150 mm, 150 mm)

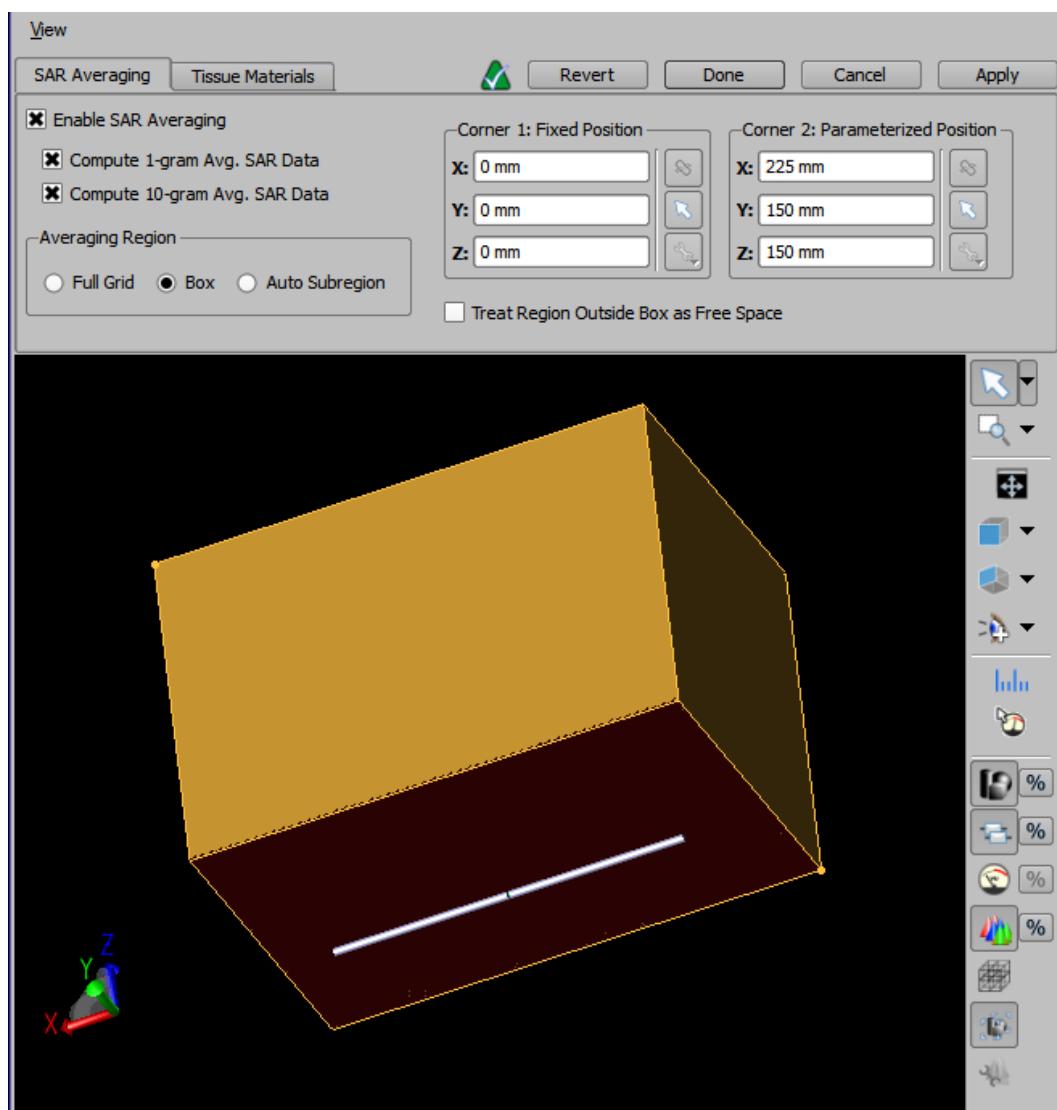


Figure 9.16: Adding the SAR Averaging sensor definition

- Press DONE to finish editing the SAR AVERAGING SENSOR.

## Adding a Point Sensor Definition

A POINT SENSOR may be saved inside the FLAT PHANTOM object to monitor the convergence of the fields during the calculation. First, we will create its definition.

- Right-click on the DEFINITIONS: SENSOR DATA DEFINITIONS branch of the PROJECT TREE. Choose NEW POINT SENSOR DEFINITION from the context menu.
- Set the properties of the surface sensor definition as follows:
- NAME: E-field vs. Time
  - FIELD VS. TIME: E
  - SAMPLING INTERVAL: timestep

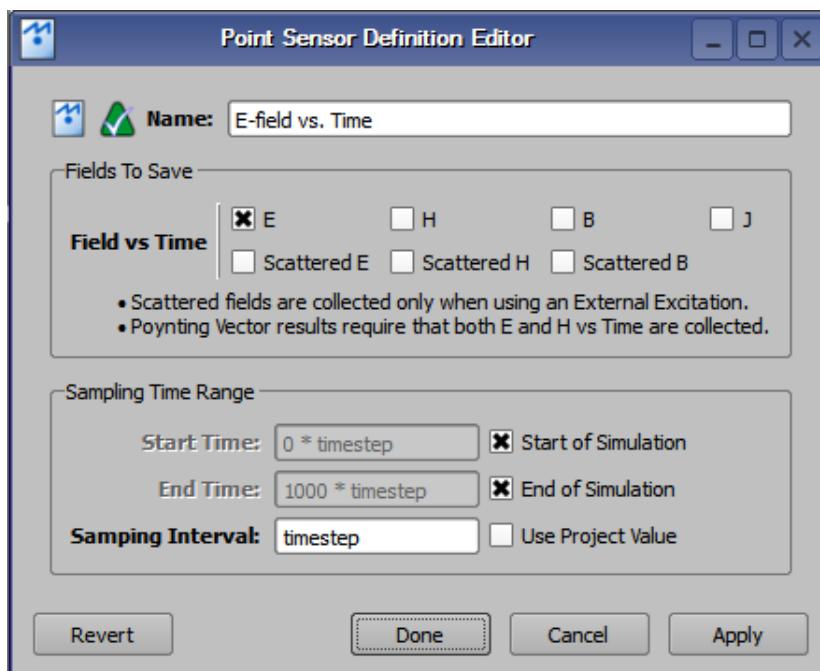


Figure 9.17: Adding the sensor definition

- Press DONE to finish editing the E-FIELD VS. TIME definition.

## Adding a Point Sensor

- Right-click on the SENSORS: NEAR FIELD SENSORS branch of the PROJECT TREE. Select NEW POINT SENSOR from the context menu.
- Enter its LOCATION as (112.5 mm, 75 mm, 15 mm).

- Under the PROPERTIES tab, enter the following:

- \* NAME: E-field
- \* SENSOR DEFINITION: E-field vs. Time
- \* SAMPLING METHOD: Snapped to E-Grid

→ Press DONE to finish editing the E-FIELD Sensor.

## 9.9 Running the Calculation

If you have not already saved your project, do so by selecting FILE > SAVE PROJECT. Once the project is saved, a new simulation can be created to send to the calculation engine.

- Open the SIMULATIONS workspace window. Click the FDTD button in the upper-left to set up a new simulation.
- Type in a descriptive NAME for the simulation, such as “Flat Phantom at 835 MHz”.
- Most of the default settings are sufficient. Navigate to the SPECIFY TERMINATION CRITERIA tab. Set up the termination criteria as follows:
  - ANALYZE PROJECT CONTENTS: Unchecked
  - DETECT CONVERGENCE: Checked
  - THRESHOLD: -30 dB
  - MAXIMUM SIMULATION TIME: 10000 \* timestep
- Select CREATE AND QUEUE SIMULATION to close the dialog and run the new simulation.

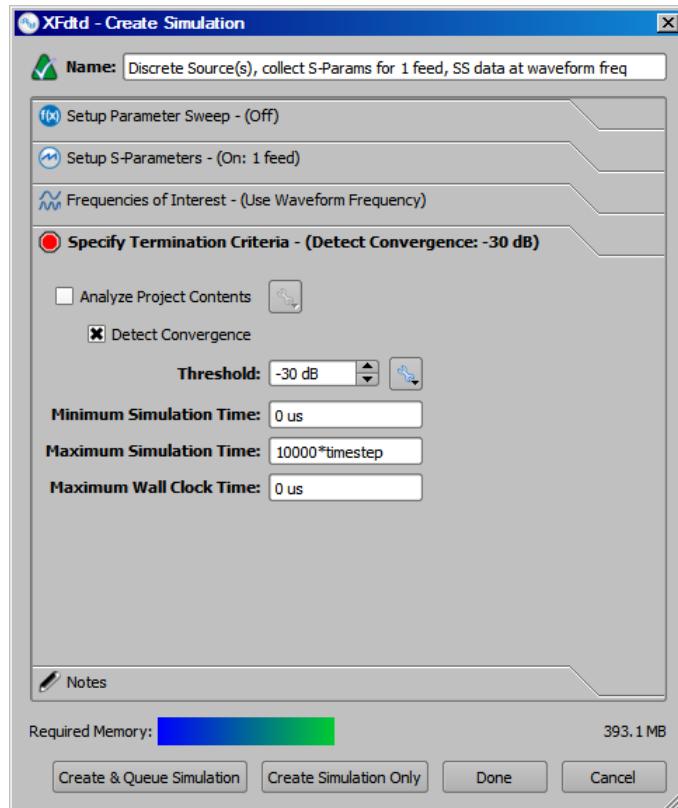


Figure 9.18: Setting up the simulation for the SAR Phantom project

## 9.10 Viewing the Results

The OUTPUT tab of the SIMULATIONS workspace window displays the progress of the simulation. Once the STATUS column shows that the simulation has completed, we can view its results in the RESULTS workspace window.

### E-field Results

Now we will view the E-field results retrieved from the center of the TISSUE.

- To filter the list accordingly, select the following options in the columns in the top pane of the RESULTS window. (You may need to change your column headings first.)
  - SENSOR: E-field
  - RESULT TYPE: E-field (E)
- Right-click on the result and select CREATE LINE GRAPH.

— Select “X” as the COMPONENT, and press VIEW. The plot of the E-field at the center of the FLAT PHANTOM object will appear.

- ✓ It is possible to view the data before the simulation is complete. The plot will update automatically as more data is computed.

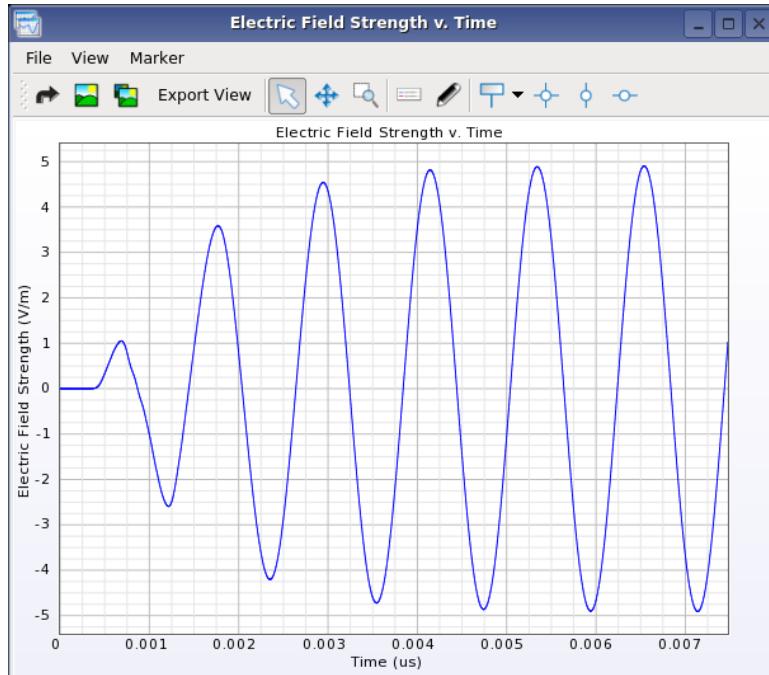


Figure 9.19: Viewing the E-field at the center of the Flat Phantom

The resulting plot indicates that the fields inside the phantom are at steady-state as a smooth sine wave is visible. This confirms our convergence condition of -30 dB that was set during the simulation setup.

- You may close the window when you are finished viewing the results.

## System Sensor Results

Now we will view System sensor data.

- To view the system efficiency results, select the following:
  - SENSOR: System
  - RESULT TYPE: Net Input Power
- Double-click on the result under the “Discrete Frequencies” domain. The following results will appear showing the Net Input Power, System Efficiency, Radiated Power, and other quantities.

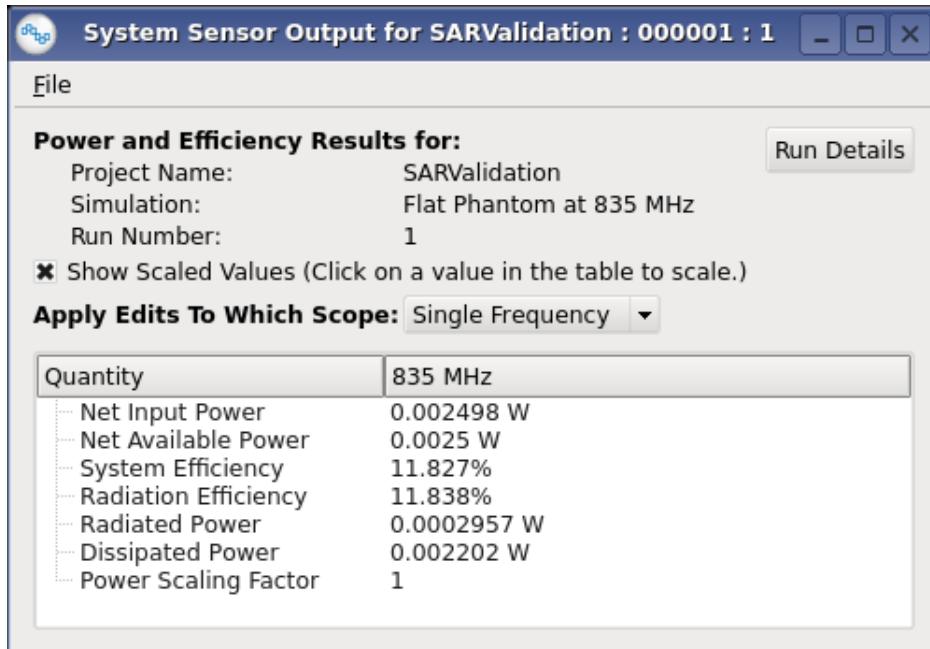


Figure 9.20: The power table from the SAR Phantom simulation, including input and dissipated powers

The powers in the simulation are displayed. As we can see, the power delivered to the antenna is relatively small, just under 2.5 mW. For many SAR analyses, the power is adjusted to a value such as 1 W to normalize all results. We can do this by clicking on the System Sensor Output window.

- Click on the power value to the right of Net Input Power (0.002498 W in Figure 9.20).
- Type in a value of “1 W” and press [ENTER]. The powers should now scale to the 1 W input. This will also scale the SAR values, as we will see shortly.

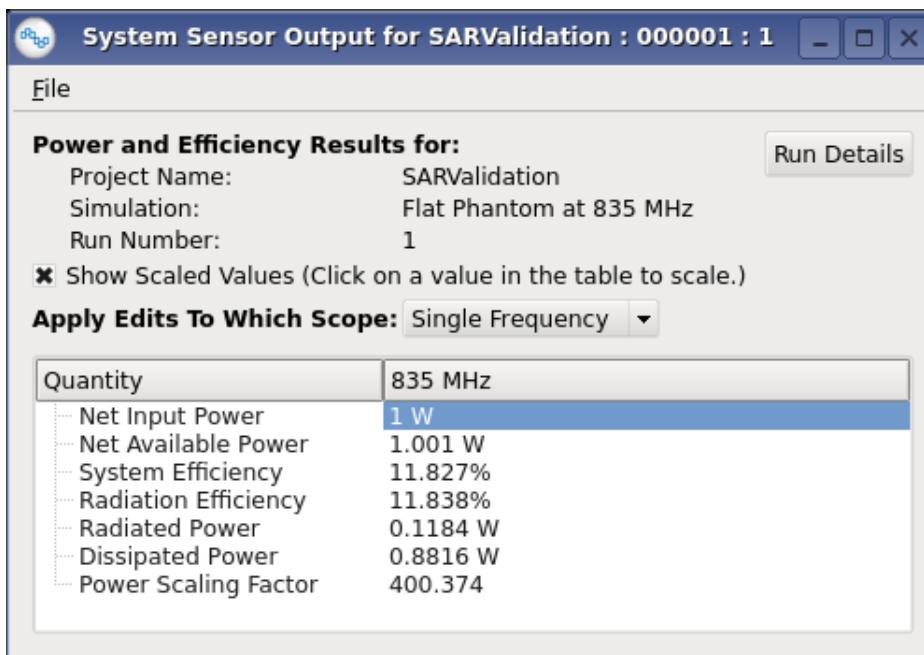


Figure 9.21: The power table after the input power has been adjusted to 1 W

→ You may close the window when you are finished viewing the results.

To view the FEED results:

→ In the RESULTS workspace window, select:

- SENSOR: Feed
- RESULT TYPE: S-Parameters

→ Double-click on the result under the “Discrete” domain. The following results will appear showing the impedance at the feed, the input power delivered, and the return loss.

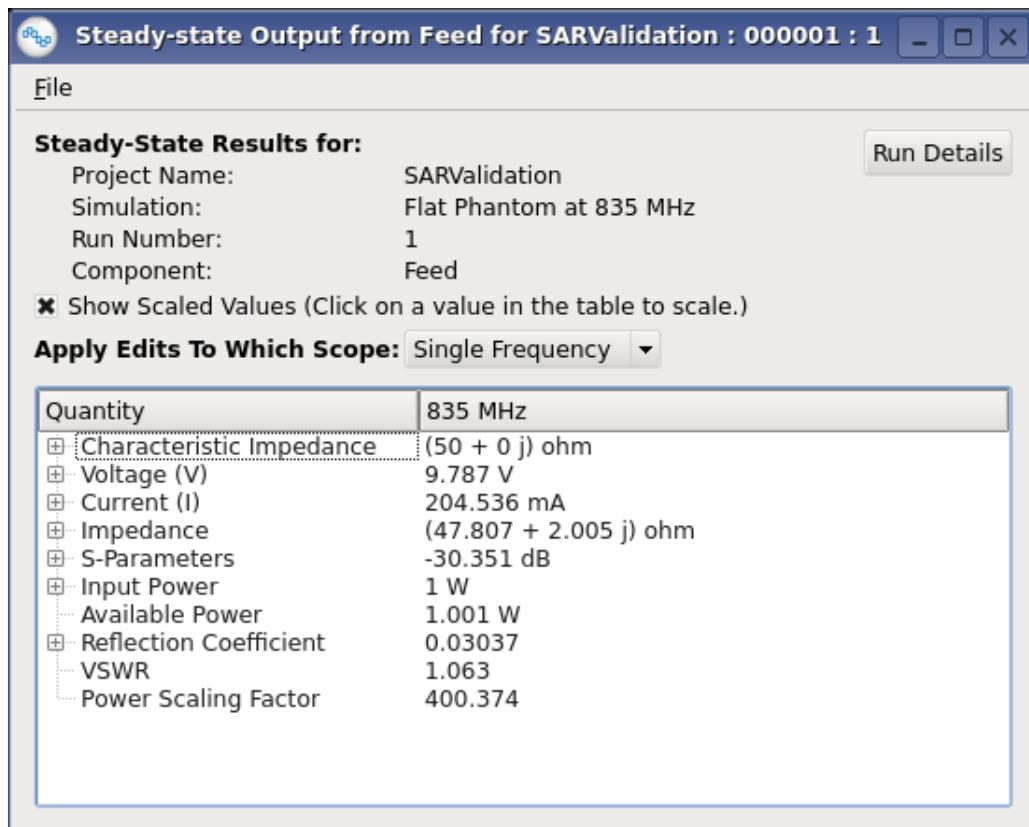


Figure 9.22: The steady-state output data for the feed

We can see from the table that our return loss is below -30 dB, so we have a very good match at our frequency of interest.

- You may close the window when you are finished viewing the results.

## SAR Sensor Data

Now we will load the SAR data into the field viewer.

- To view the SAR sensor data, select the following:
  - SENSOR: SAR Sensor (Raw)
  - RESULT TYPE: SAR (Specific Absorption Rate)
- Double-click on the result in the filtered list. The plot will appear in the GEOMETRY workspace window.
- Under the SETUP tab, adjust the following settings:
  - SEQUENCE AXIS: X

- DECIMATION: Finest
- The axis range controls for the independent variables Frequency, Y, and Z are:
  - \* FREQUENCY: No change- Only one steady-state data collection frequency was defined for this run (0.835 GHz).
  - \* Y RANGE: Change the minimum to 0 and the maximum to 149 to plot the SAR data over the full available range of Y.
  - \* Z RANGE: Change the minimum to 0 and the maximum to 148 to plot the SAR data over the full available range of Z.
  - \* To change which values are displayed for any axis, click the  MODIFY TOOL button next to that axis display.
- Toggle the  PARTS VISIBILITY to turn off the display of the geometry, and select the  LEFT (-X) orientation.
- Under the  SEQUENCE tab, set the SHOWING editor to a value of 111 to plot the SAR data slice at  $X = 112.5$  mm. The SAR image shown in Figure 9.23 should appear.

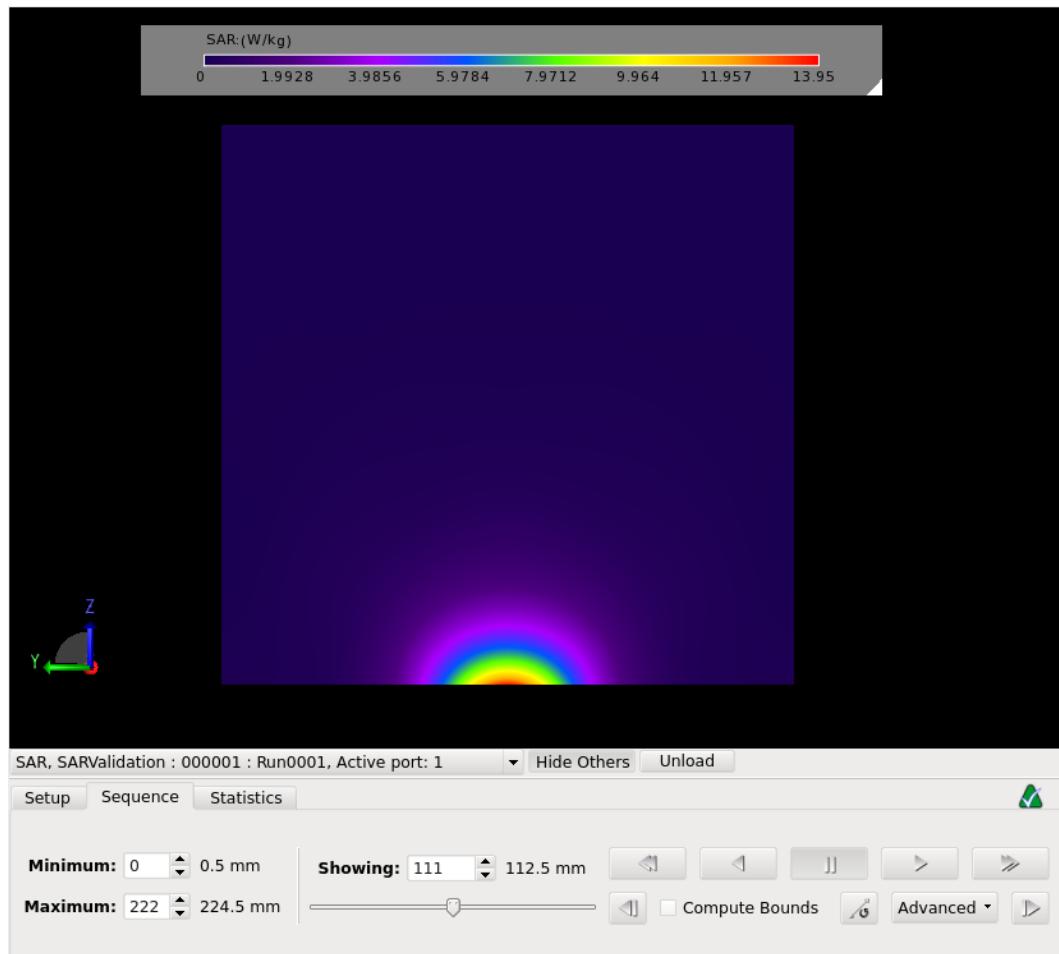


Figure 9.23: Setting the view of the SAR sensor data

- Under the STATISTICS tab, press VIEW ALL SAR STATS. A summary table of the SAR values will appear, as shown in Figure 9.24.

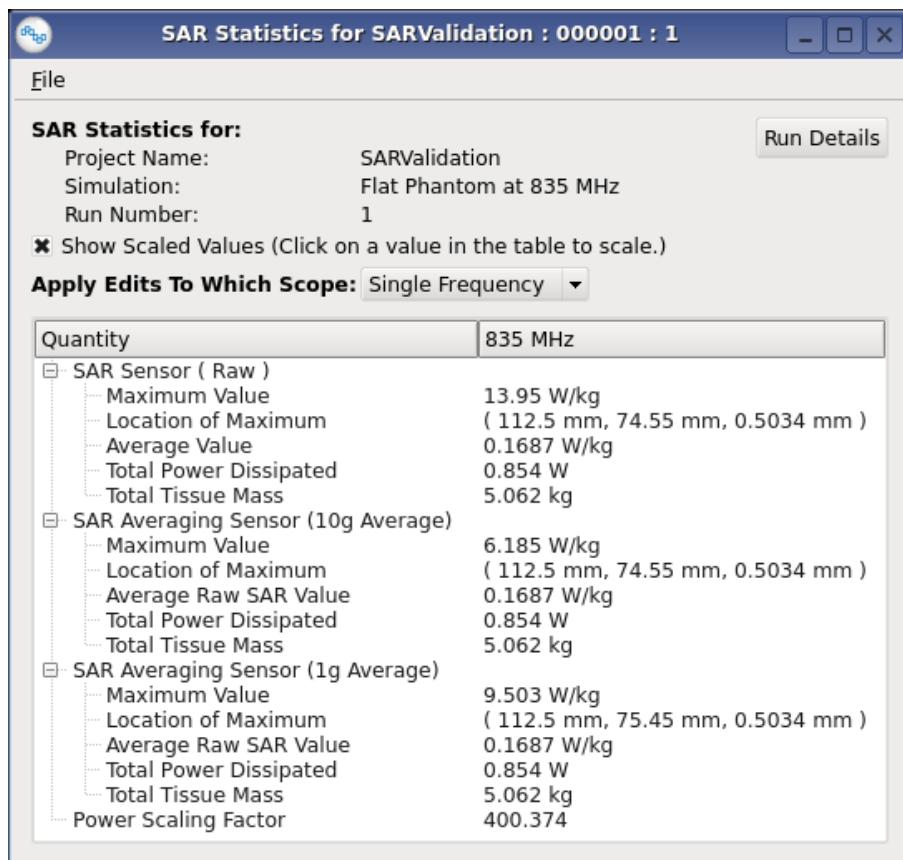


Figure 9.24: The SAR statistics for the simulation showing the peak values and locations

For some situations, the SAR results should be normalized to the feed point current rather than the forward power.

- To make this adjustment, return to the RESULTS workspace window and select:
  - SENSOR: Feed
  - RESULT TYPE: Current
- Double-click on the result under the “Discrete Frequencies” domain.
- On this screen, the current value may be edited by clicking on the numerical value. Type in a value of “200 mA” and all results, including the SAR values, will be adjusted to a feed point current of 200 mA.

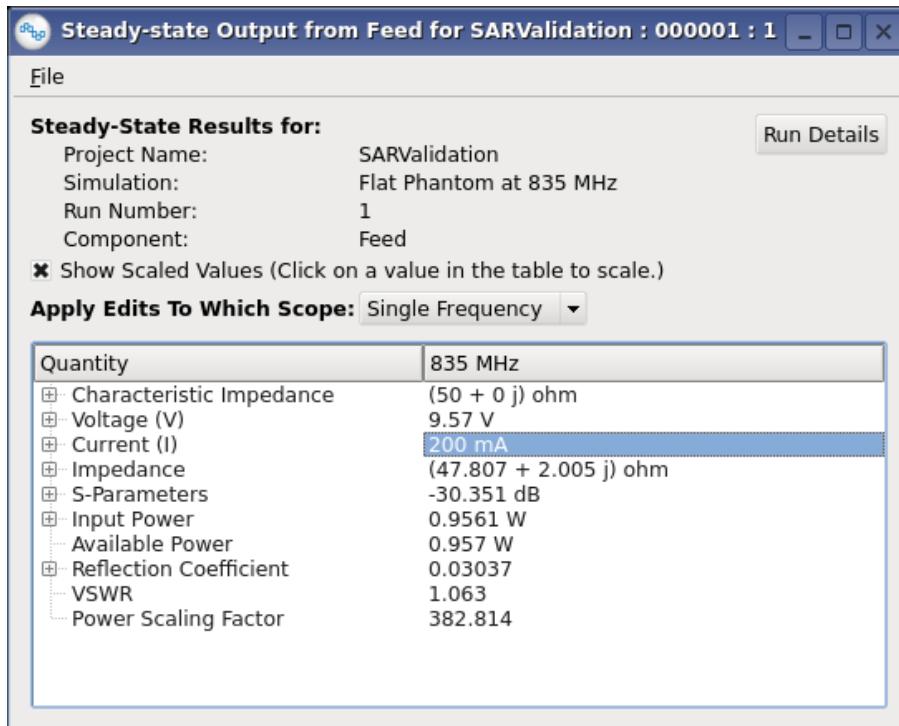


Figure 9.25: The display of the feed statistics after scaling the feed point current to 200 mA

- You may close the window when you are finished viewing the results.

It may be of interest to plot the SAR as a function of distance along a line extending above the feed point.

- In the RESULTS workspace window, select the SAR data.
- Right-click on the result and select CREATE LINE GRAPH.
  - INDEPENDENT AXIS: Z
  - X: 111
  - Y: 74

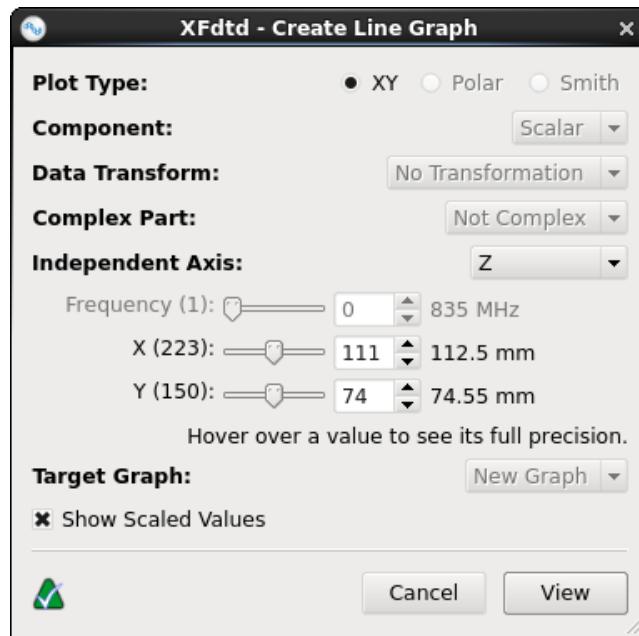


Figure 9.26: Creating a line plot of the SAR above the feed point

- Press VIEW to see a line plot of the SAR as a function of distance from the feed point in the center of the phantom, as shown in Figure 9.27.

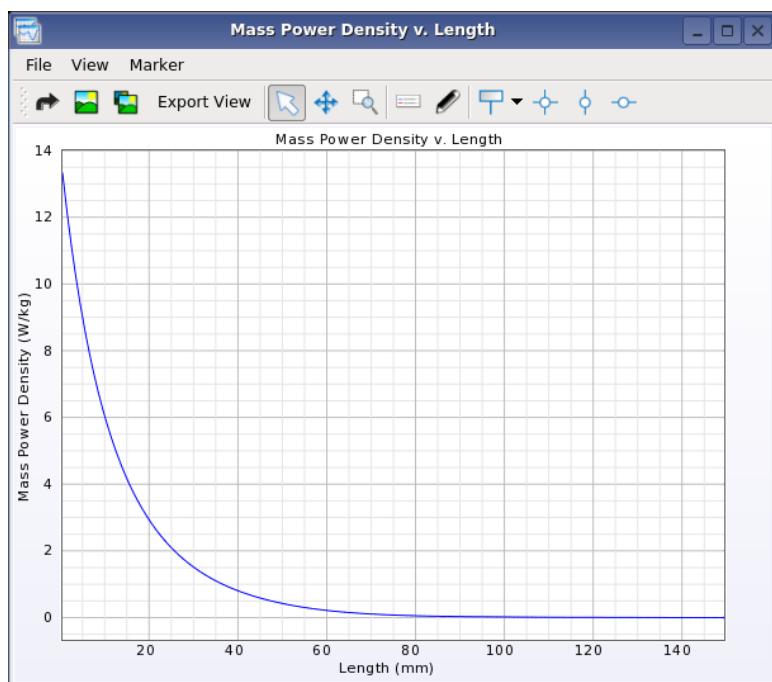


Figure 9.27: A line plot of the SAR as a function of distance above the feed point



## Chapter 10

# Appendix: Application Preferences

This section describes how to set your preferences for the XFDTD GUI - from the display of specific buttons to the color scheme of your materials. Here you will also learn how to set the display units and initialize a universal system of measurement for your project.

Once XFDTD opens, begin by defining your project preferences. Select **Edit > Application Preferences** (above the PROJECT TREE) to bring up the APPLICATION PREFERENCES editor.

- ▶ For a thorough description of each setting in this dialog, see the “Application Menu Bar and Toolbar” section in “The XFDTD Interface” chapter of the reference manual.

### Setting Project Preferences

For all examples in this user guide, the following settings will initialize XFDTD in a way that makes navigating the interface easy and intuitive.

Navigate to the INTERFACE tab:

- Define NEW ITEM ACTION as EDIT PROPERTIES.
- These definitions will automatically prompt a definition editor window to open when you add a new item to the PROJECT TREE. (Alternatively, the EDIT NAME preference will simply add the new item to the PROJECT TREE and prompt you to name it.)
- Check the box titled OPEN PROPERTIES IN NEW WINDOW WHEN POSSIBLE.
- In the WORKSPACE section, select SHOW ALL TABS so that all browser window tabs are visible whether they are active or inactive.

Navigate to the MODELING tab:

- Select your preferred color scheme for parts and materials if you wish to change it from XFDTD’s default scheme.
- Press OK to apply the changes and close the editor.

## Set display units

At the top of the PROJECT TREE, double-click on PROJECT branch to bring up the PROJECT PROPERTIES EDITOR window.

Navigate to the DISPLAY UNITS tab, as seen in figure 10.1:

- Set the units as specified by the example that you are working on.
- If the unit you need to change is not shown, check the SHOW ALL UNITS checkbox.
- Press DONE

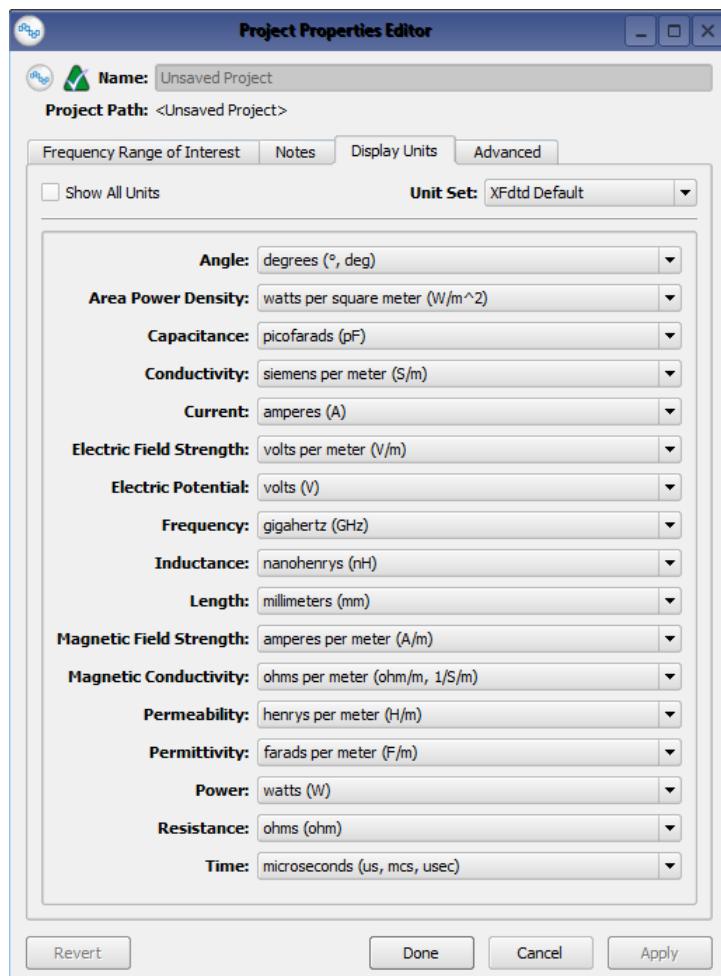


Figure 10.1: Setting the project display units

# Bibliography

- [1] C. A. Balanis, *Antenna Theory Analysis and Design*. New York: John Wiley and Sons, third ed., 2005.
- [2] D. M. Sheen, S. M. Ali, M. D. Abouzahra, and J. A. Kong, “Application of the three-dimensional finite-difference time-domain method to the analysis of planar microstrip circuits,” *IEEE Transactions on Microwave Theory and Techniques*, vol. 38, pp. 849–857, July 1990.





Remcom provides electromagnetic simulation software for analyzing complex EM problems and antenna propagation. We empower design engineers with unique solutions for navigating today's rapidly changing technologies.

**Remcom's products simplify EM analysis for a wide variety of applications, including:**

- ▶ Antenna Design and Placement
- ▶ Mobile Device Design
- ▶ 5G MIMO
- ▶ Automotive Radar
- ▶ Biomedical
- ▶ Microwave Devices
- ▶ Radar and Scattering
- ▶ Outdoor and Indoor mmWave
- ▶ WiFi Device Performance
- ▶ Metamaterials

**■ Remcom Professional Support**

One year of Remcom Professional Support is included with each product you purchase. Maintaining your license ensures that you receive important updates and support services. To contact Remcom Support, call **814.861.1299** or email [support@remcom.com](mailto:support@remcom.com).

**■ [www.remcom.com](http://www.remcom.com)**

Remcom's website offers many resources for users of our software, including application examples, videos, whitepapers, and on-demand webinars.

**■ Subscribe to Remcom Communications**

Stay informed of product announcements, new examples, upcoming training and webinars, and more. Subscribe at [www.remcom.com/communications](http://www.remcom.com/communications).

