A Full 3-D Parallel EM Simulation Software and System

GEMS

Quick Start Guide

Version 7.8

2COMU, Inc.
4031 University Dr. Suite 200
Fairfax, VA, 22030 USA
www.2comu.com

Notice

The information contained in this document is subject to change without notice.

2COMU, Inc. makes no warranty of any kind with regard to this material, including, but not limited to, the implied warranties of merchantability and fitness for a particular purpose. 2COMU shall not be liable for errors contained herein or for incidental or consequential damages in connection with the furnishing, performance, or the use of this material.

2COMU, Inc.
4031 University Dr. Suite 200
Fairfax, VA 22030
Mobile: (814) 441 9403
Fax: (703) 277-7730

Table of Contents

1

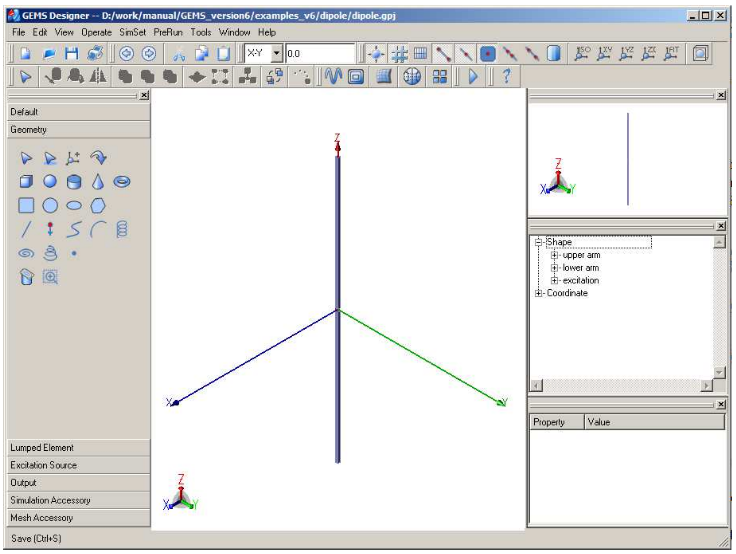
Example 1. Dipole Antenna

Description: A dipole antenna is excited by using a lumped port with an internal resistance. The output parameters include return loss, impedance and far field pattern.

Keywords: Open Boundary (add space), lumped port, impedance, return loss, and far field pattern.

1.1 Dipole Antenna Configuration

We simulate a half wavelength dipole antenna with a rectangular cross section. The antenna is excited by using a lumped port (voltage with an internal resistance). The output parameters are time domain port voltage and current, port impedance, return loss, and far field pattern. The dipole configuration is shown in figure below.



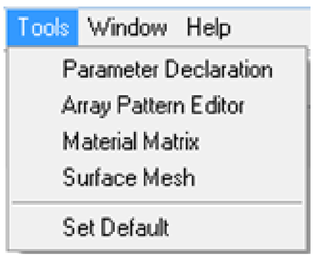
Suppose that the working frequency is 1.5GHz, the length of half wavelength dipole is 100mm. The feed gap is taken to be 1mm, and hence, the length of each arm is 49.5mm. The material of dipole arms is selected to be PEC (perfect electric conductor) or copper. The lumped port excitation with 50 Ohms internal resistance is located in the feed gap from the lower to upper arm.

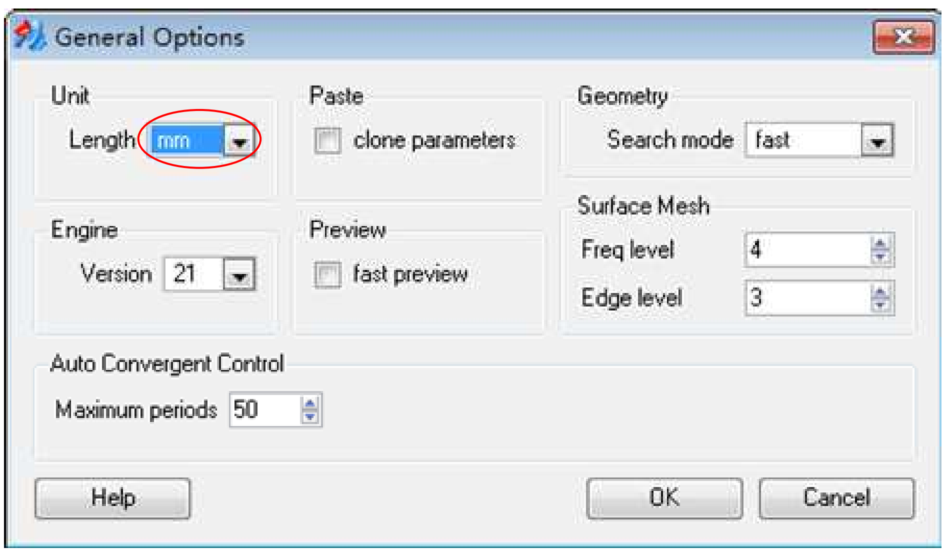
1.2 Create Dipole Model

Follow the steps below to create a dipole model:

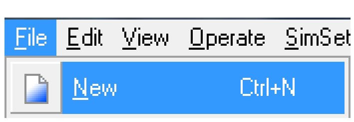
(1) Open the GEMS **Designer** in the **GEMS Simulator** folder

(2) Select the **Set Default** option in the **Tools** menu, and then select "mm" in the **Unit->Length** box as the project unit, click on the **OK** button to close the window.

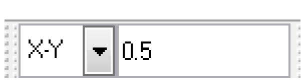




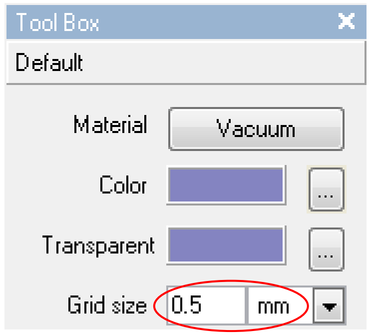
(3) Click on the **New** button in the toolbar or select the **New** option in the **File** menu.



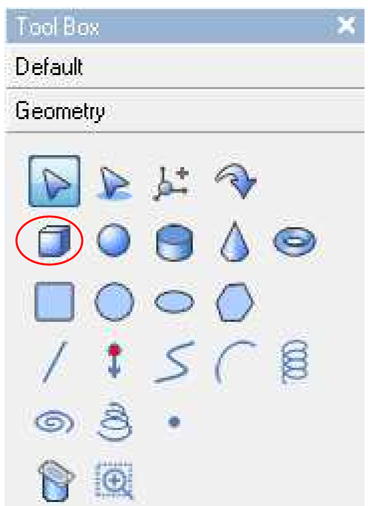
(4) Select the drawing plane (X-Y) and height (0.5mm)

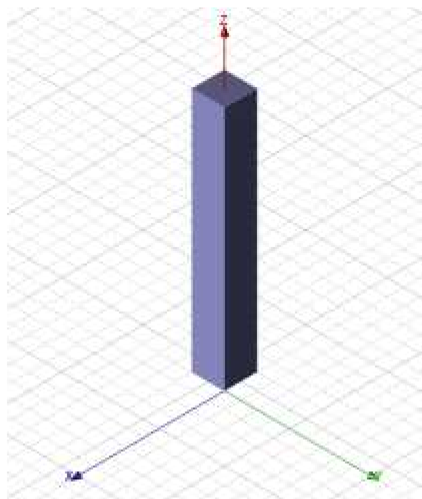


(5) Specify the background resolution (0.5mm). This resolution can be adjusted at any time. It is only for modeling and no relation with the simulation mesh.



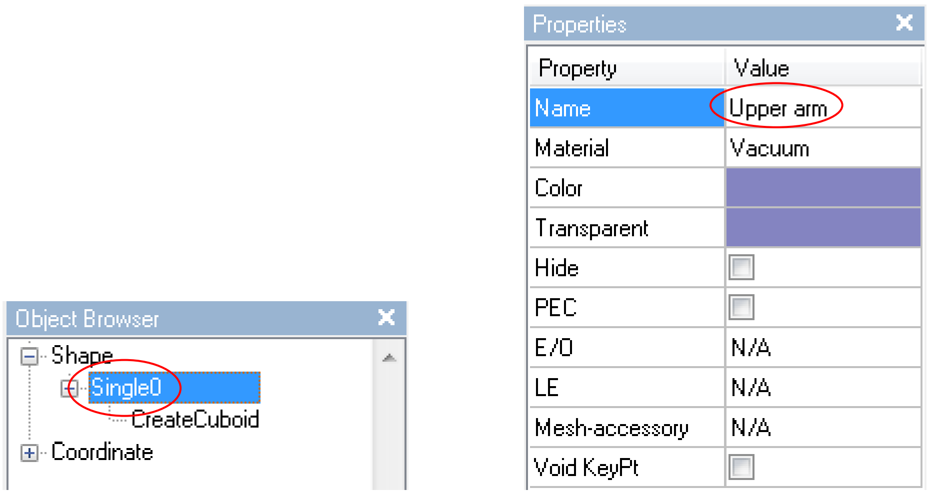
(6) Click on the **Cuboid** icon in the **Tool Box->Geometry** box and draw a box in the figure region.



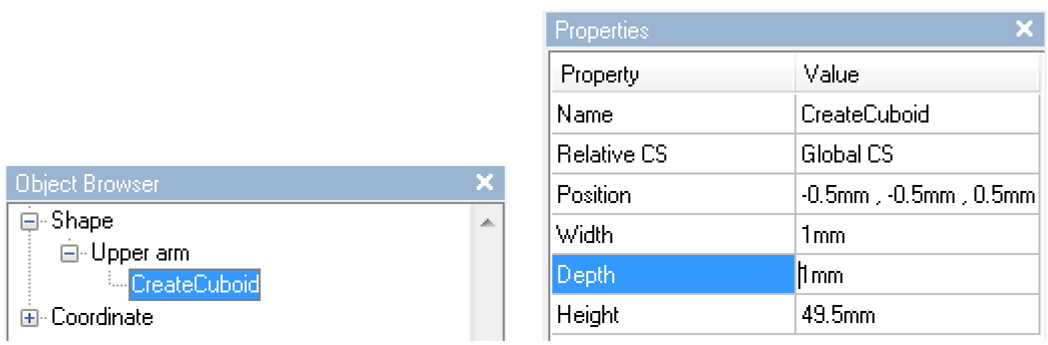


The coordinates of mouse icon are shown at the bottom left corner of figure region when the mouse icon moves in the figure region. You can also draw a shape using this coordinate dialog.

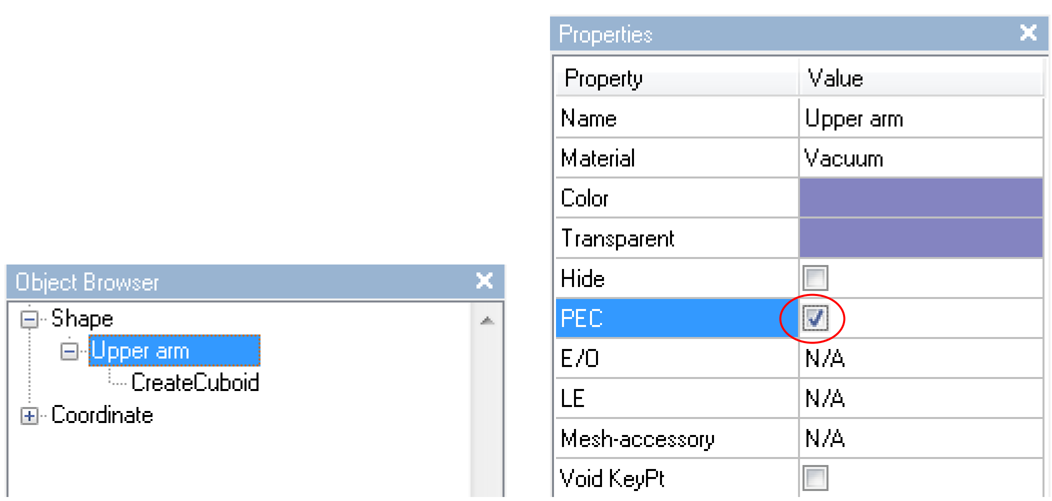
(7) Select the “Single0” option in the **Object Browser** box and change its name to “Upper arm”. Change its display color, transparency by clicking the **Color** and **Transparent** boxes in the **Properties** box.



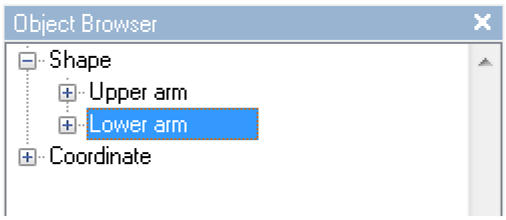
(8) Select the **CreateCuboid** option in the "Upper arm" folder and change its dimensions in the **Properties** box if necessary.



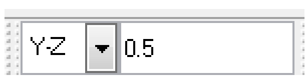
(9) Select the "Upper arm" option in the **Object Browser** box, and then check **PEC** box in the **Properties** box.



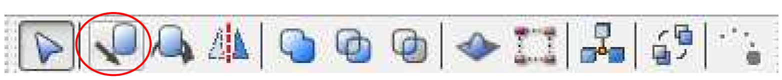
(10) Duplicate the lower arm from the upper arm by selecting the “Upper arm” in the **Object Browser** box, and pressing “Ctrl + c” and then “Ctrl + v” to copy and paste the upper arm. Select the pasted object in the **Object Browser** box and change its name to “Lower arm”.

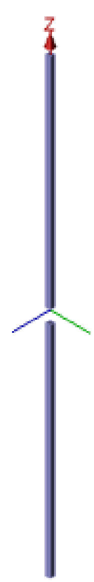


Select the “Lower arm” option in the **Object Browser** box and change the drawing plane to “Y-Z”.

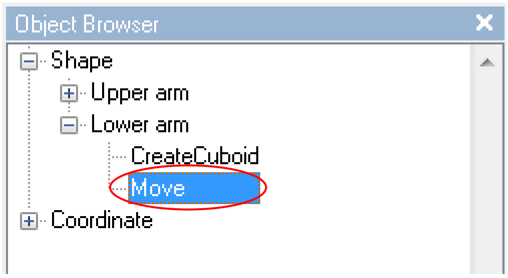


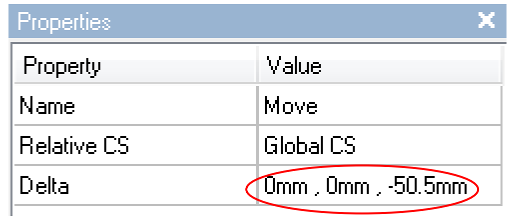
Click on the **Move** button in the toolbar, and then move the mouse icon into the picture region. Select a point on the lower arm and move the mouse icon along the –z-direction.





Select the **Move** option in the “Lower arm” folder, and adjust the moving parameters in the **Properties** box.

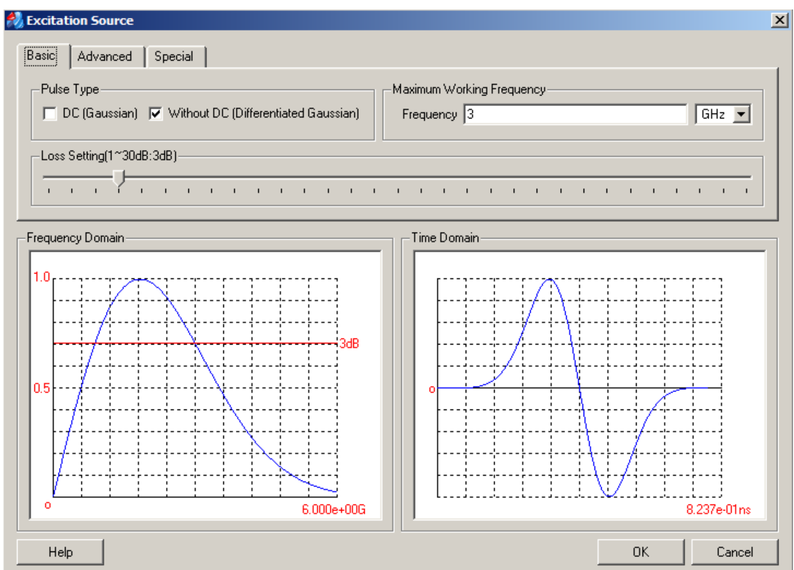




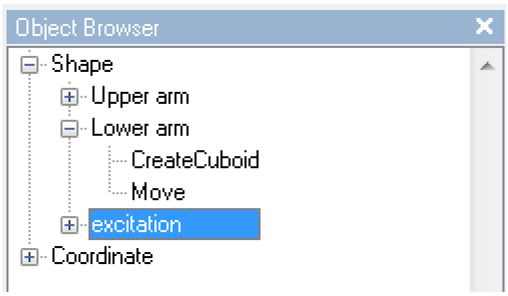
(11) Click on the **FIT** button in the toolbar to fit the dipole into the figure region.

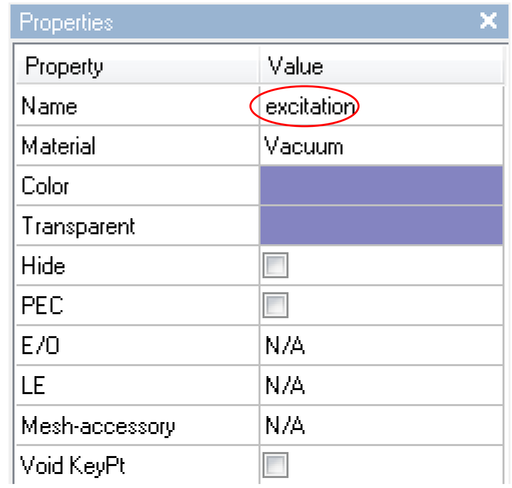
1.3 Specify Excitation

Click on the **Set excitation source** button in the toolbar to set the excitation pulse. Type “3” in the **Frequency** box, which is the maximum frequency of interest. Click on the **OK** button to confirm the excitation pulse setting.

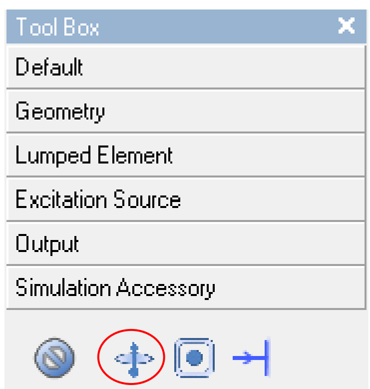


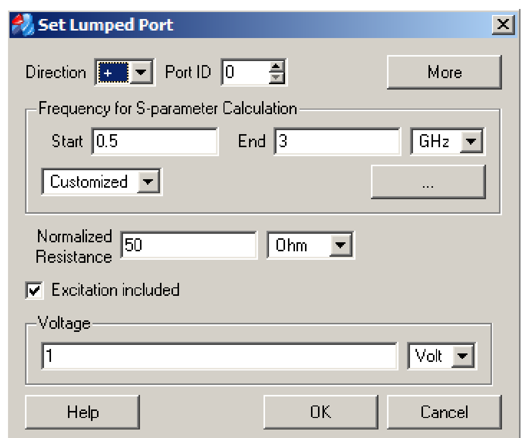
Click on the **Line** icon in the **Tool Box->Geometry** box and draw a line in the feed gap of dipole antenna from the lower to upper arm. Move the mouse icon to the center of the top cross section of lower arm and click on the left mouse button to select the start point. Release the mouse icon and move the mouse icon to the center of the low cross section of upper arm and click on the left mouse button to select the end point.





Select the **excitation** option in the **Object Browser** box, and then click on the **Lumped port** icon in the **Tool Box->Simulation Accessory** box.





Specify the frequency band of interest in the *Frequency for S-parameter Calculation* box.

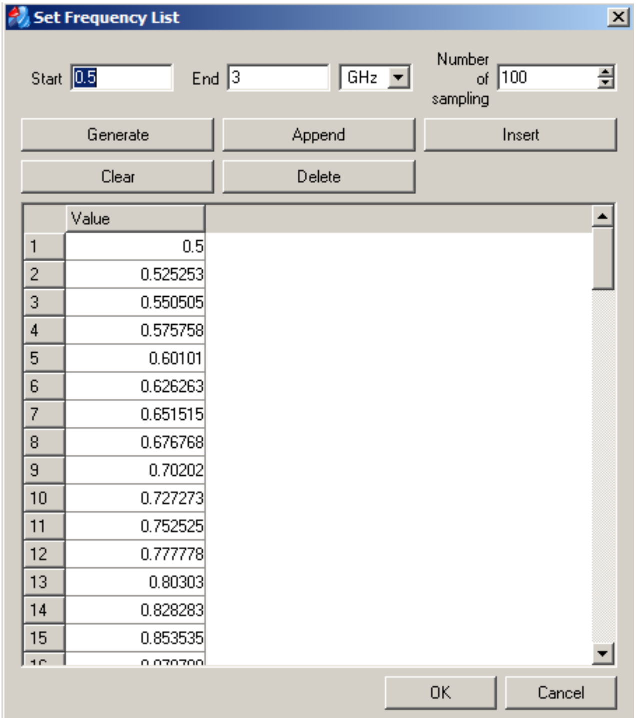
There are two meanings for the **Normalized Resistance** value:

(i) The internal resistance of the excitation source;
(ii) The return loss is normalized to this value.

The port ID is designed for the multiple port excitation problems. The S-parameter matrix can be obtained by switching the port excitation. The excitation sources are added independently each time during the simulation if they have different port IDs.

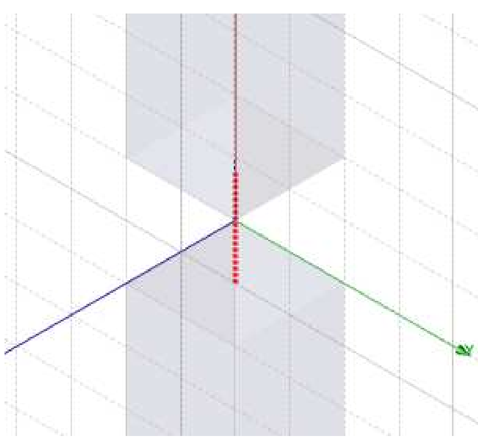
Only if the ***Excitation included*** *box is checked, the voltage excitation will be added into the lumped port, and otherwise, it will serve as an output port.*

Select the “Customized” option in the *Frequency for S-parameter Calculation* box to specify the number of sample points for the S-parameter output in the frequency domain. You can select a proper number according to the problem property.



Select the “Adaptive” option in the *Frequency for S-parameter Calculation* box to allow GEMS to decide the frequency resolution according to the width of simulation time window.

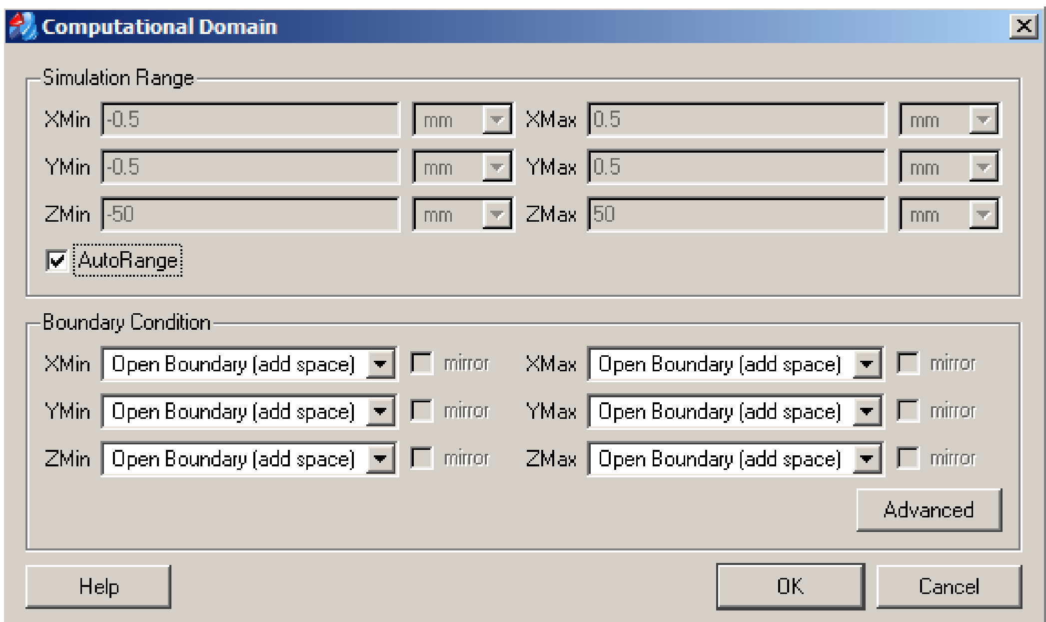
Click on the **OK** button in the *Set Lumped Port* window to confirm the port setting, and the lumped port will appear in the style as shown in the figure below. The black end indicates the higher potential of the excitation port.



This excitation port provides an electric field source along the port path and an internal resistance. It will measure the port voltage and current, and, calculate the port impedance and the return loss using the port voltage and current at the specified frequencies. The return loss is normalized to the “Normalized resistance” value specified in the *Set Lumped Port* window.

1.4 Boundary and Domain Specifications

For an open space problem, the absorbing boundary is needed to truncate the simulation domain. Click on the **Set boundary condition** button in the toolbar to specify both the domain size and the boundary condition.



In the default case, the **AutoRange** box is checked, which means that the domain is the minimum size enclosing all the objects and options. You can set the different domain size by unchecking the **AutoRange** box.

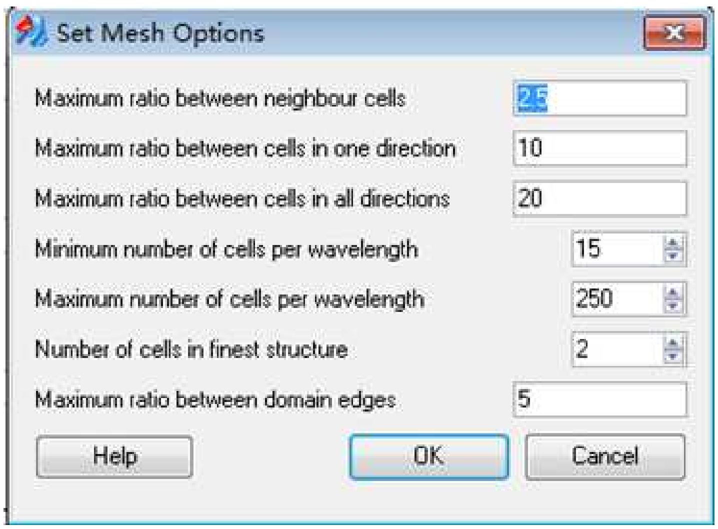
The distance between the dipole and absorbing boundary in the horizontal directions should be large enough to ensure the accurate results. GEMS can select it according to the height in the vertical direction.

Click on the **OK** button to confirm the domain size and boundary condition settings.

1.5 Mesh Design

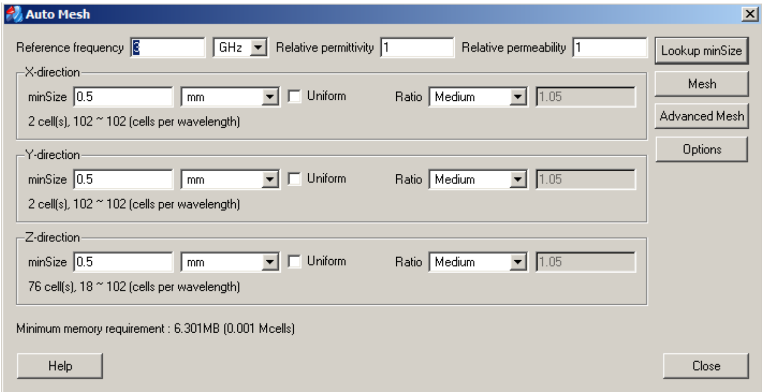
Click on the **Auto mesh** button in the toolbar to design the mesh distribution.

GEMS provides two ways to generate the adaptive mesh distribution: (1) GEMS searches and finds the minimum cell size based on the presetting information, which are located in the **Options** window. For example, the default settings are as follows:



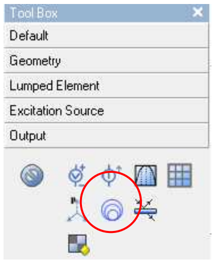
For the simple problem, the two factors in the window above are important. (1) The number in the "Number of cells in finest structure" box tells the software how many cells are in the smallest gap. For example, the width of dipole cross section is 1mm, if we want to put two cells inside the dipole in the x- or y-direction, the number should be "2" in the box. (2) The number "5" in the "Maximum ratio between domain edge" box means that the selection of the domain size in the horizontal directions should be larger than the one-fifth dimension in the vertical direction to ensure the simulation accuracy. Clicking the **OK** button to confirm the mesh setting options.

Clicking on the **Lookup minSize** button in the *Auto Mesh* window. For the complex problems, the recommended way is that typing the half of fine structure dimension in the **minSize** box in each direction, and then pressing **Mesh** button to generate a good mesh distribution. Increasing the ratio value will reduce the memory requirement and simulation time, however, the local maximum cell sizes should be sufficient to describe the geometry features and smaller than one-tenth wavelength of the highest frequency of interest. *Proper adjusting the minimum cell sizes in each direction and the corresponding ratio will achieve the better results using the minimum computing resource.* Click on the **Close** button to generate the mesh distribution based on the parameter settings.

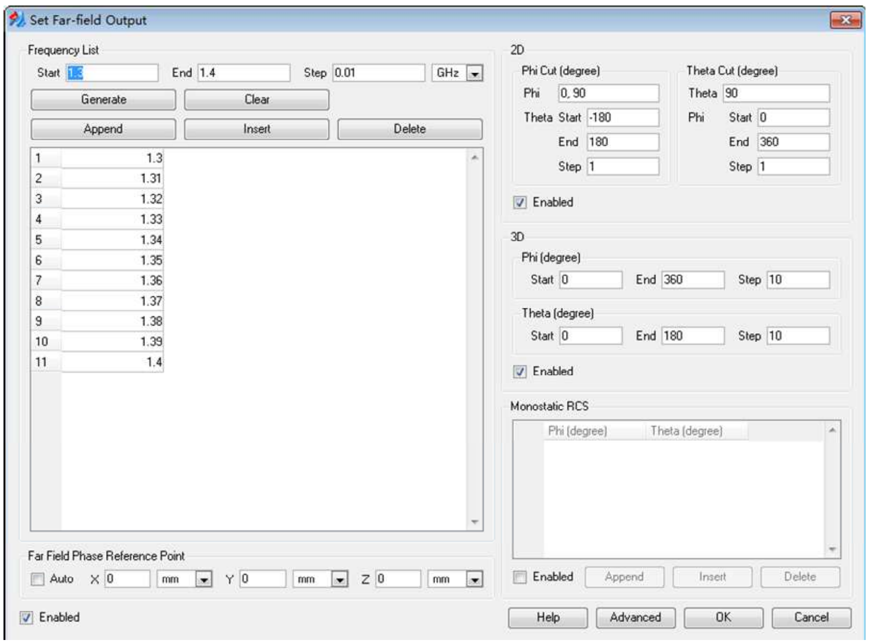


1.6 Far Field Output Setting

Besides the port voltage, current, impedance and return loss, another important output parameter in this project is the far field pattern. To add the far field output in the project, click on the **Far-field** icon in the **Tool Box -> Output** box.



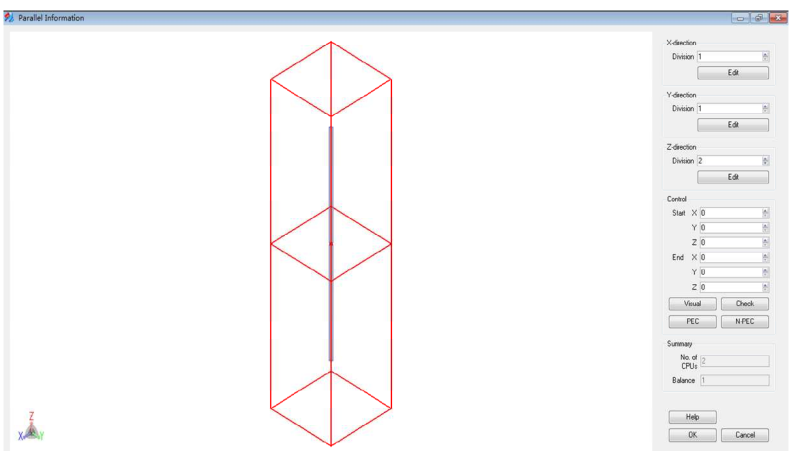
Check the **Enabled** box to enable the far field output parameter settings.



Specify a frequency list for the far field pattern, and insert a frequency if it is not generated by the **Generate** button. Check the **3D->Enabled** box to enable 3-D far field pattern output. Once you select the far field pattern output, GEMS will generate all output parameters associated with the far field such as directivity, gain, axial ratio, radiation power, and radiation efficiency, etc.. Click on the **OK** button to confirm the far field output setting. Check the “Auto” checkbox, the center of the far field pattern plot will be at the excitation source when the far field pattern is plotted together with the project model; otherwise, it will be at the origin of the coordinate system.

1.7 Parallel Processing Setting

If your computer includes two or more CPUs, you can speed up the simulation by splitting the computational domain to the subdomains, whose number should be equal the number of CPUs. Clicking the “Set parallel info” button in the toolbar and put a “2” in the Z-direction->Division box, namely, one CPU simulate the lower half space and another CPU will simulate the upper half space. Skip this step for a single CPU computer.



1.8 Save Project

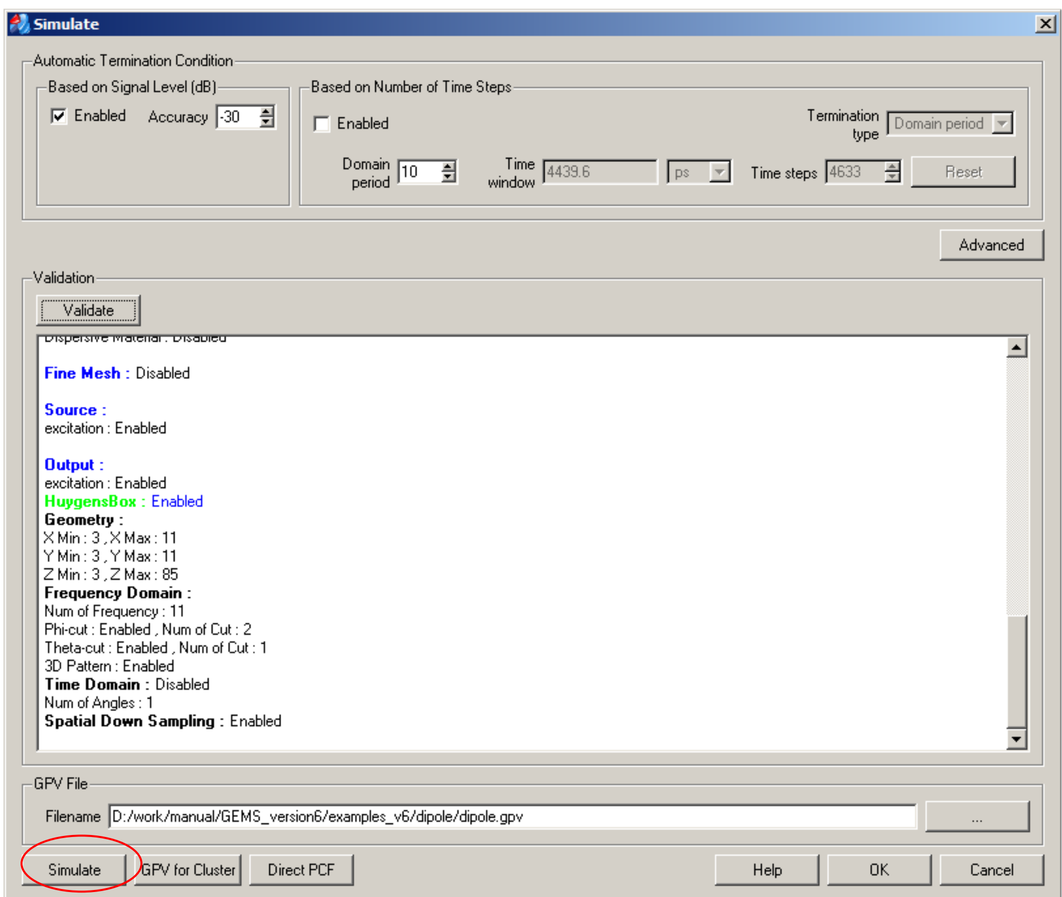
Click on the **Save** button in the toolbar to save the project design. The saved project has a default extension name “gpj”. It includes all the project information, and can be loaded to the GEMS Designer later.

1.9 Generate Simulation File

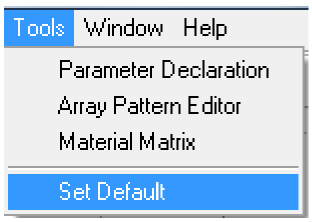
Click on the **Precalculate** button in the toolbar to generate a simulation file with an extension name "gpv".

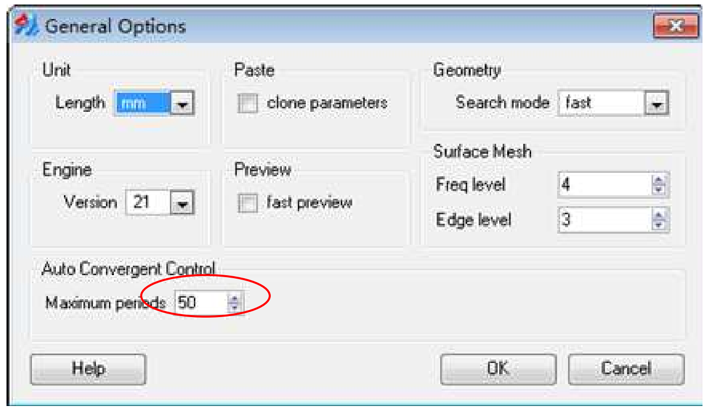
The simulation will stop when the results in both the time and frequency domains are convergent to the convergence criterion (say, -30 dB). Using the convergence criterion option, we cannot predict the total simulation time before the simulation is completed because the width of the time window varies from case to case.

Click on the **Validate** button to validate the project settings. If there is no message in red, click on the **Simulate** button to start the project simulation.



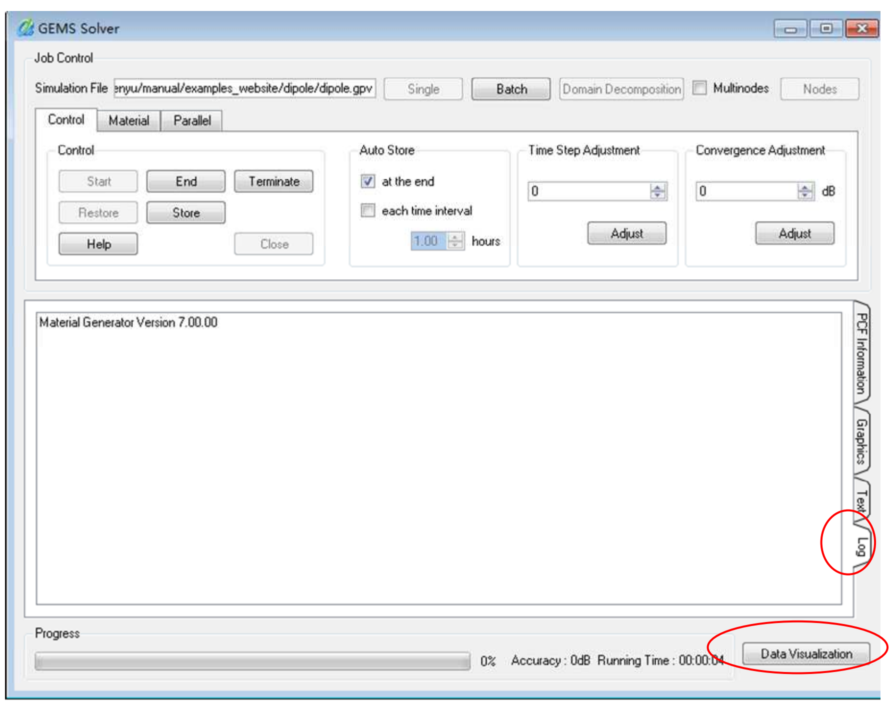
GEMS will force the simulation to stop when is not completed after 50 periods. Select the **Set Default** option in the **Tools** menu to open the *General Options* window. You can adjust number (the default number is 50) of the maximum domain periods for the simulation, namely, the simulation stops after finishing 50 time periods. One domain period equals to the time when the signal propagates along the longest side of the computational domain.





1.10 Project Simulation

Click on the **Start** button to start the simulation, and GEMS will use all the cores in your PC for the simulation. Click on the **Log** label to view the detailed simulation status. The simulation status is shown in the progress bar.



After the simulation is completed, click on the **Data Visualization** button to open the *GEMS display* window for the result visualization. The simulation time and project simulation information are in the **Log** window, and also stored in the **GEMSSummary.txt** file in the project folder.

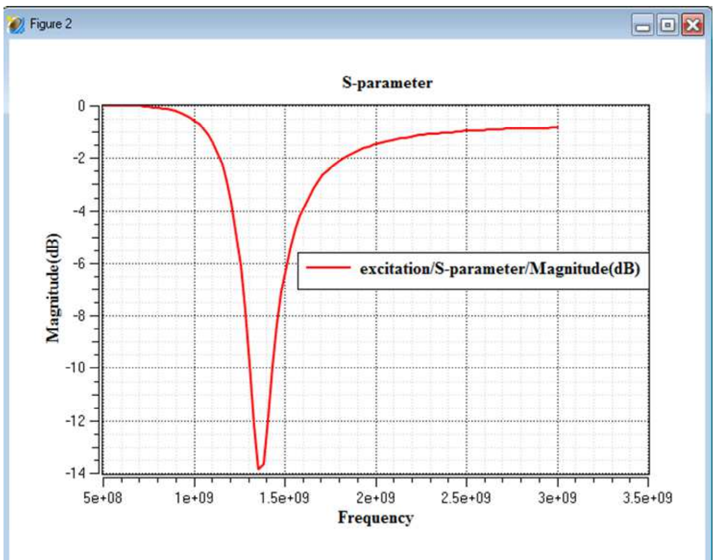
*In case something happens and the **Start** button is not active (**Terminate** and **End** buttons are not active at the same time), you can click on the **Single** button to select and load the "*.gpv" file again.*

When you click the **Data Visualization** button to launch the GEMS display window, GEMS will generate the intermediate frequency domain result.

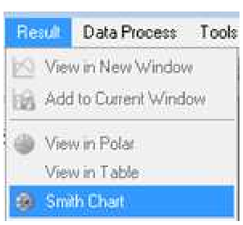
You can also open the **GEMS Solver** window by selecting the **Start->GEMS Simulator->Solver** option.

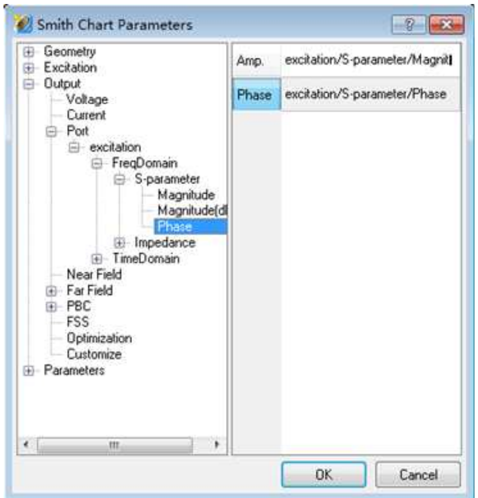
1.11 Result Visualization

The simulation results are listed in the result tree in the *GEMS Display* window. To view the return loss, double-click on the **Output->Port->Excitation->FreqDomain->S-parameter->Magnitude(dB)** option.

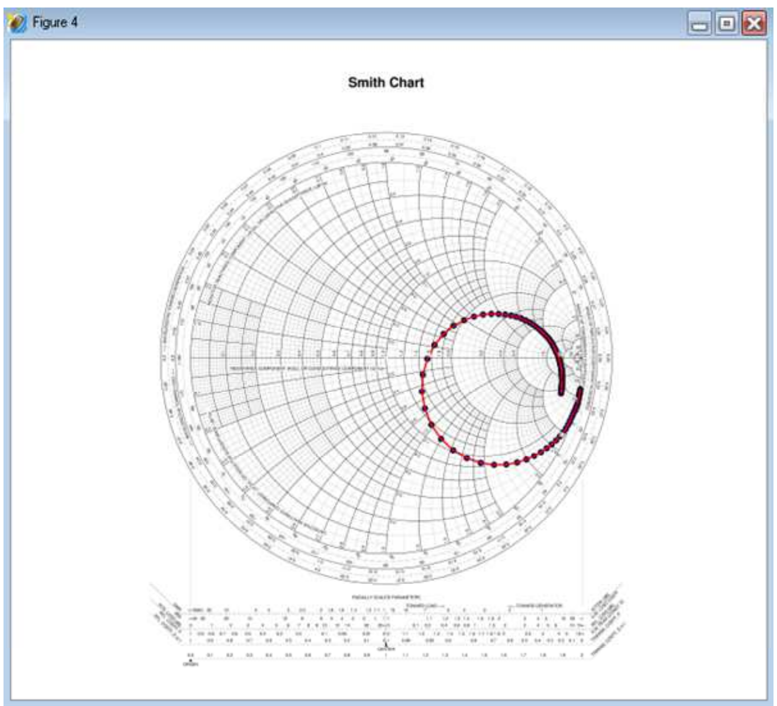


Double-click the left mouse button in the figure region, and you can edit the figure option in the popup window. You can change the axis labels, curve type and color, figure title, axis unit, and so on. The Smith chart can be plotted by selecting the "Result->Smith Chart" option, and select the input data, amplitude (liner) and phase from the result tree in the new window.

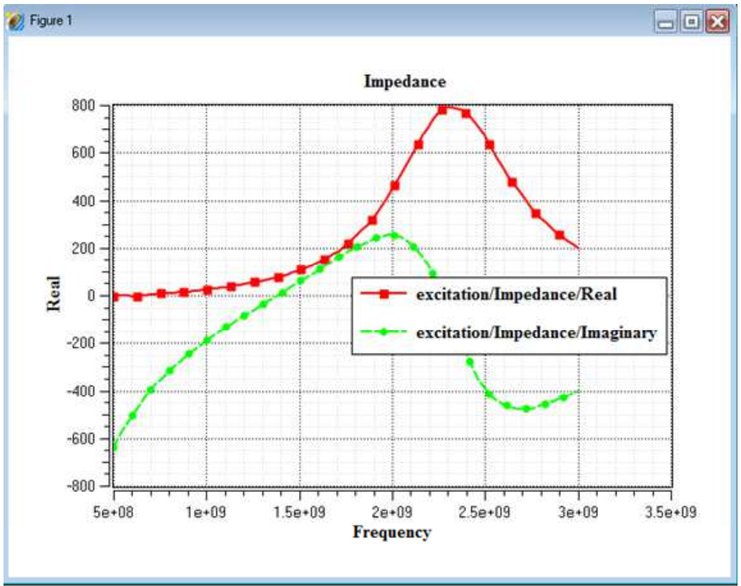




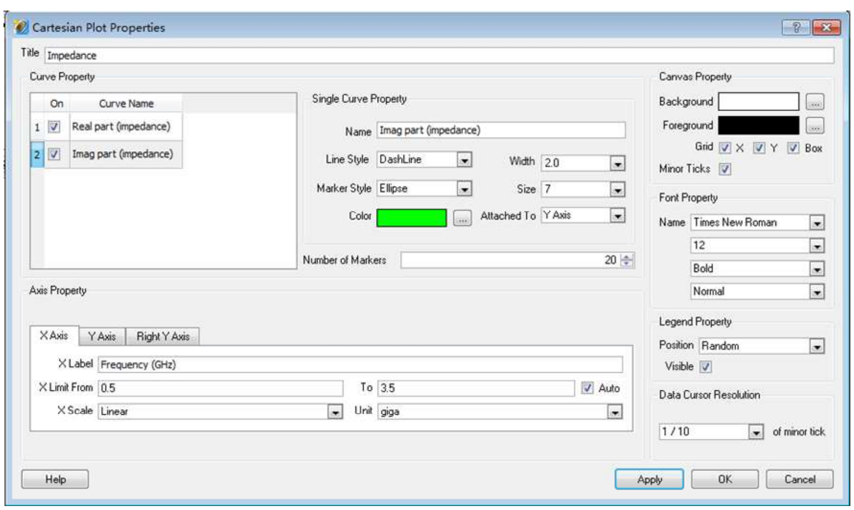
Click the OK button to plot the curve in the Smith Chart.



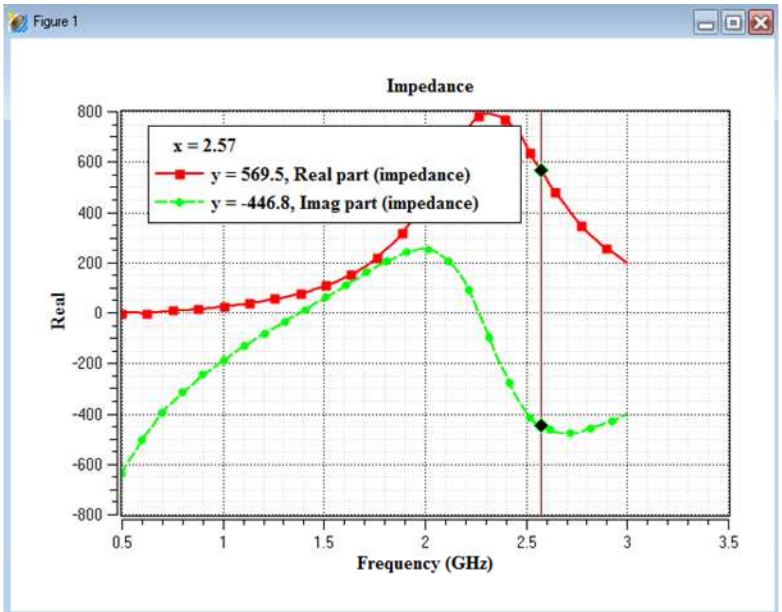
To view the impedance, click on the **Output->Port->Excitation->FreqDomain->Impedance->Real** option. And then, select **->Imaginary** and then click on the “Add to current window” button in the toolbar.



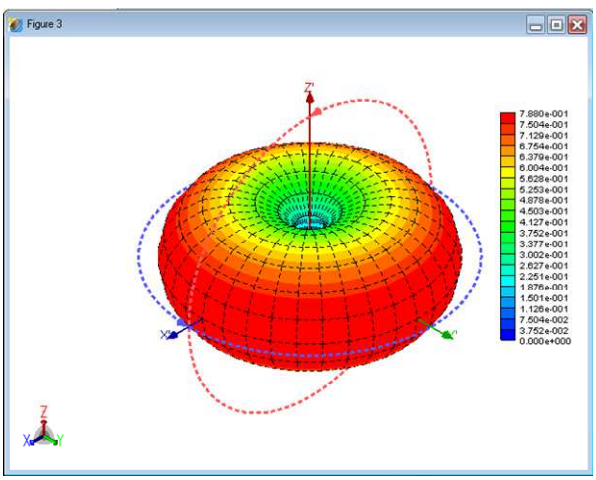
Double-click the left mouse button in the figure region, and you can edit the figure option in the popup window. You can change the axis labels, curve type and color, figure title, axis unit, and so on. For example, change the name of the real part to "Real part (Impedance)", and the imaginary part to "Imag part (impedance)", and the axis unit to "Frequency (GHz)", as shown in the figure below.



Click on the "Data cursor" button in the toolbar and mouse the mouse icon to the figure region and then click the left mouse button to select the start point. Move the vertical marker line by using the arrow keys in the keyboard to read the data on the curves. The values on the curves will appear inside the legend box.

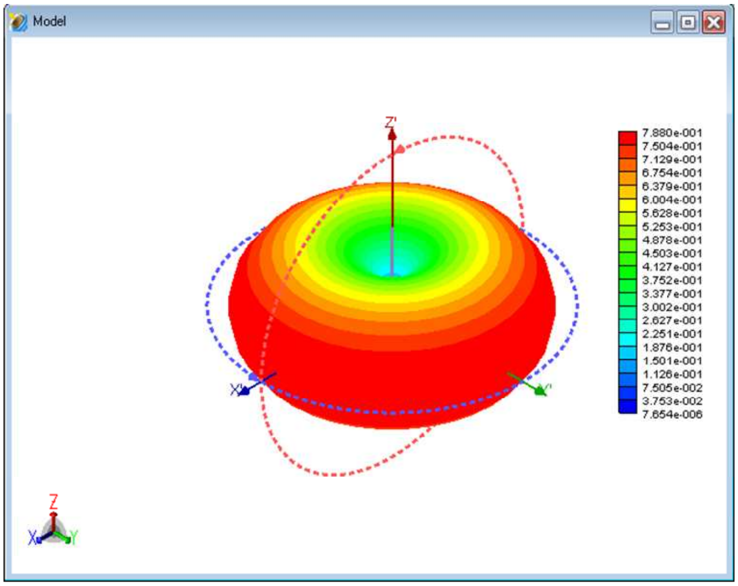


To view the 3-D far field pattern, click on the **Output->Far Field->FreqDomain->3D Pattern->1.5GHz->ETotal->Amp/Ein** option. The far field is $r \times E$ not only E. You can calculate the E far field for a given distance r.

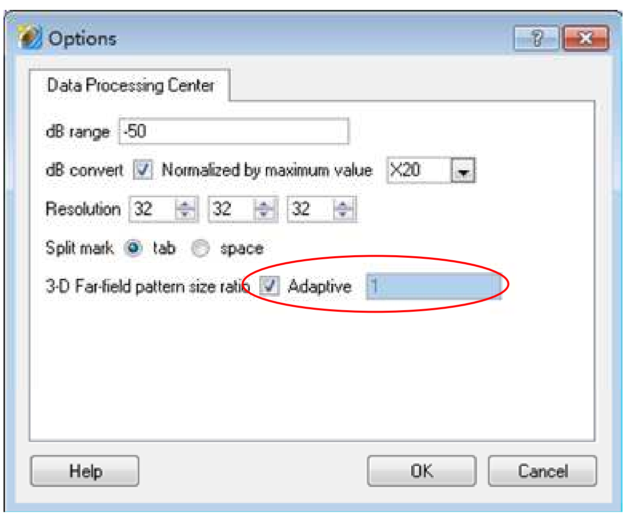


The far field is normalized to the voltage that is measured at the excitation port, namely, the far field is taken to be the value when the distance from the antenna to the observation point is 1 meter. If there is no port voltage is defined in some cases, the far field is normalized to the incident power. The unit of far field is V/m.

You can plot the far field pattern with the project model. To this end, click on the “View model” button in the toolbar, and then select the far field option in the result tree, then click the “Add to current window” button in the toolbar.

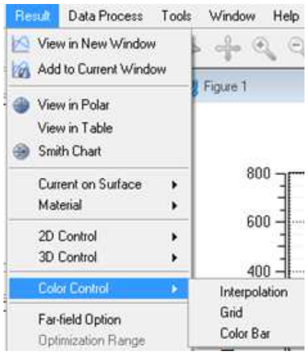


The plot range of 3-D far field pattern related to the project model is controlled by the "Tool->Set Default" option.

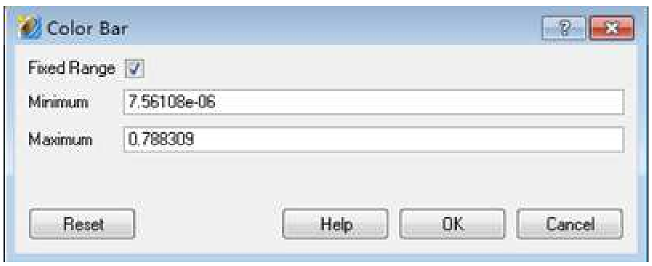


For the complex model, you can select the model to be transparent by clicking the "Transparent model" button in the toolbar. The far field pattern has two types of appearances: "Interpolation" and "Grid", which are located in the "Result->Color Control" option.

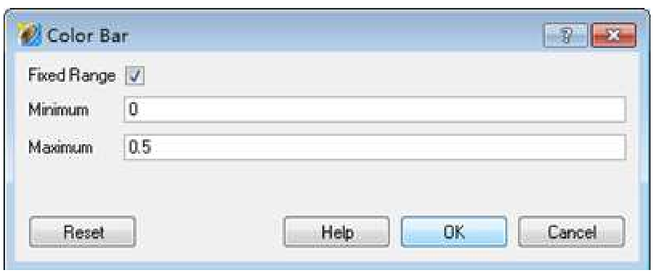
You can adjust the color level for the 3-D figures by selecting the "Result->Color Control->Color bar" option. This option is used to display appearance when the values at some positions are extremely small or large.

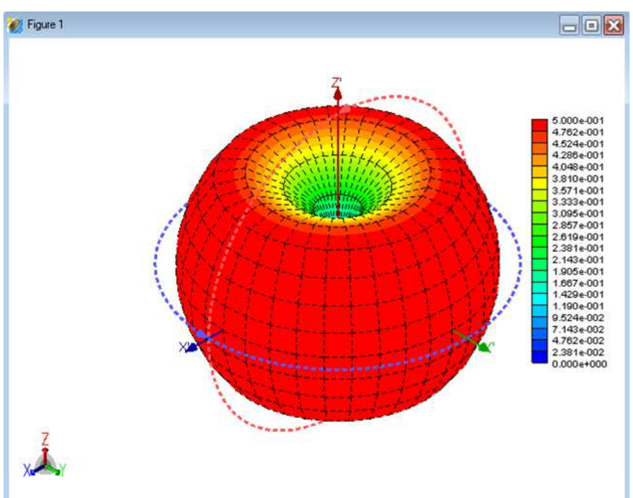


The default value range is following:



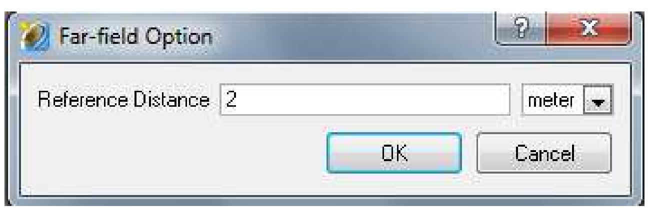
We can adjust it as the follows:





To plot the far field value at the arbitrary distance from the antenna, select the **Far-field Option** option in the **Result** menu.

Input the distance in the ***Reference Distance*** *box where you like to calculate the field value. The far field in the 2D and 3D folders are normalized to the specified distance. For example, you input a number "2", the far field value in the result tree will be at 2 meter away from the antenna.*



You can also open the *GEMS Display* window by selecting the **Start->GEMS Simulator->Display** option.

You can load more than one project in the *GEMS Display* window at the same time.

2

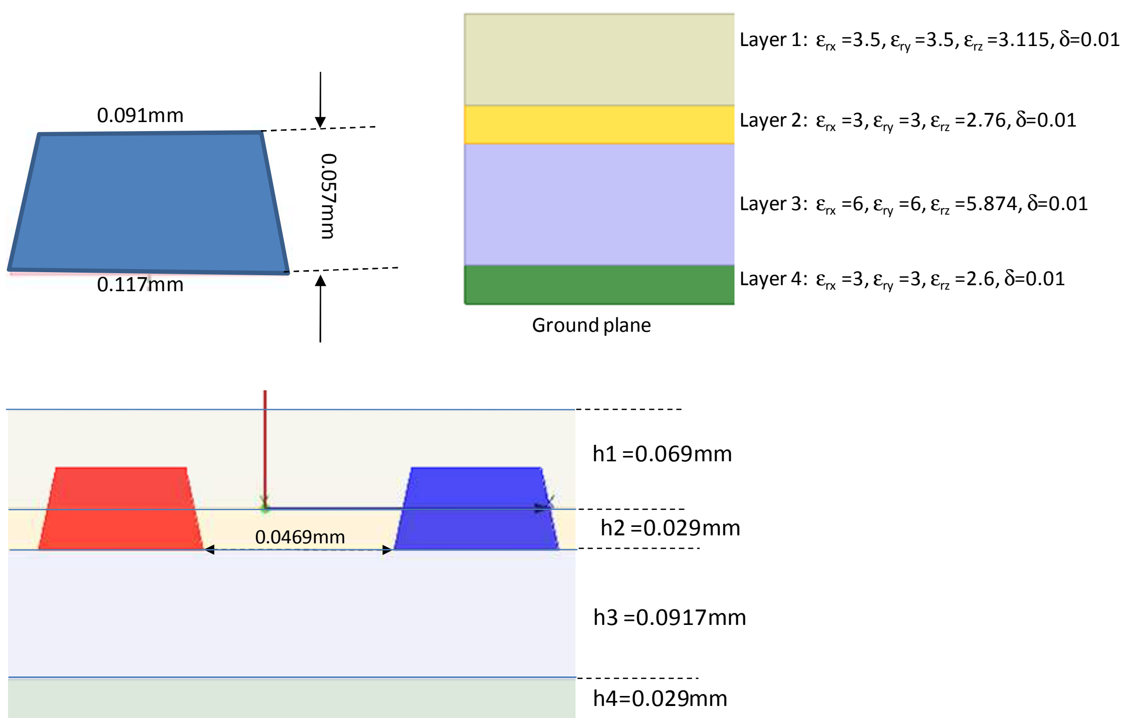
Example 2. Parallel Signal Lines

Description: A pair of striplines is embedded in the four anisotropic dielectric layers, which are backed by a PEC ground plane. Both the dielectric layers and ground plane have the finite size. The output parameters include S-parameters, field distributions in both the time and frequency domains, and far field pattern.

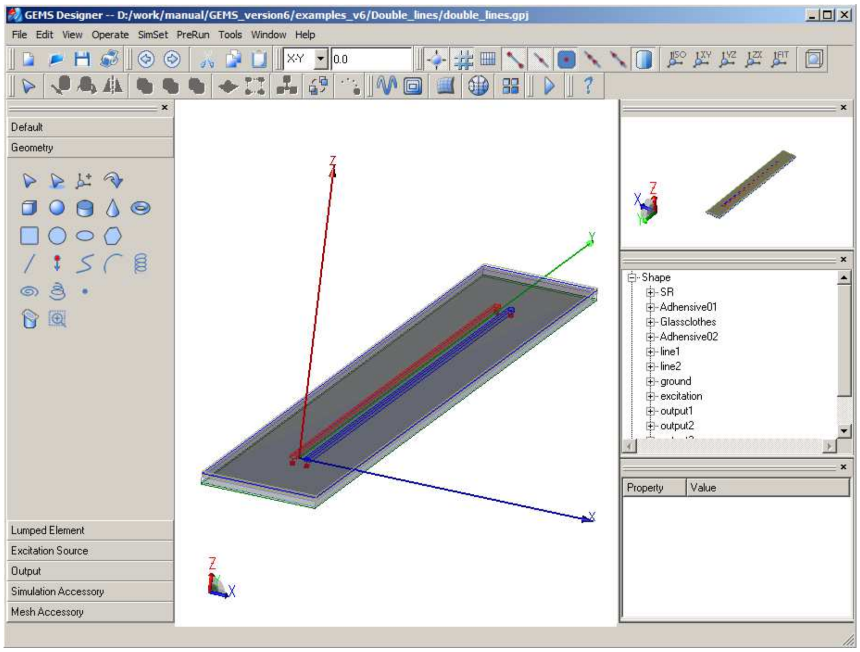
Keywords: Anisotropic material, dispersive medium, open Boundary (add space), lumped port, S-parameters, field distribution, and far field pattern.

2.1 Problem Configuration

The cross section of the two striplines, and, the parameters and dimensions of dielectric layers are shown in the figure below:



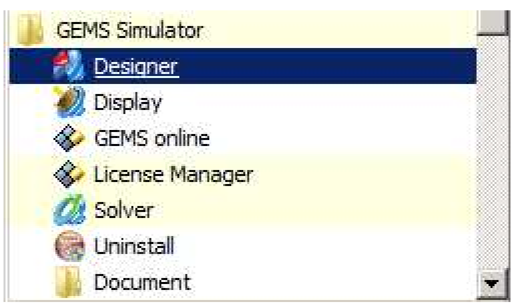
The problem configuration is shown in the figure below:



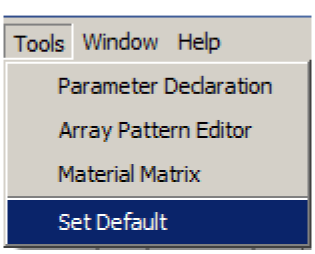
2.2 Create Problem Model

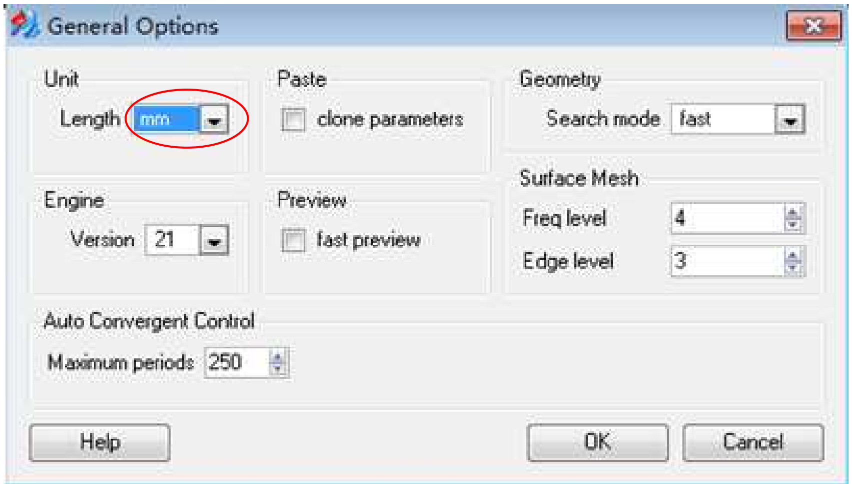
Follow the steps below to start a new project:

(1) Open the GEMS **Designer** in the **GEMS Simulator** folder

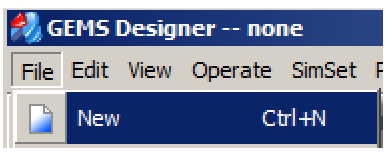


(2) Select the **Set Default** option in the **Tools** menu, and then select “mm” in the **Unit->Length** box as the project unit, click on the **OK** button to close the window.



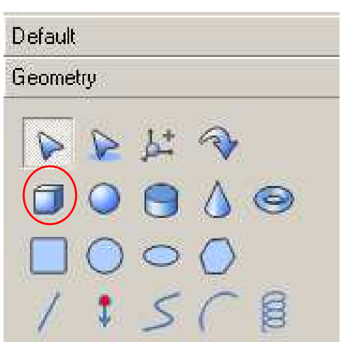


(3) Click on the **New** button in the toolbar, or, select the **New** option in the **File** menu.



Follow the steps below to draw the dielectric layers:

(1) Click on the **Cuboid** icon in the **Tool Box->Geometry** box



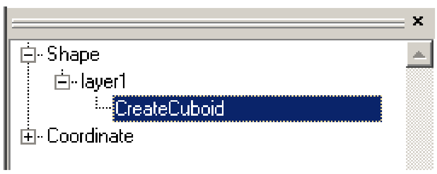
(2) Change the drawing plane to the “X-Y” plane.

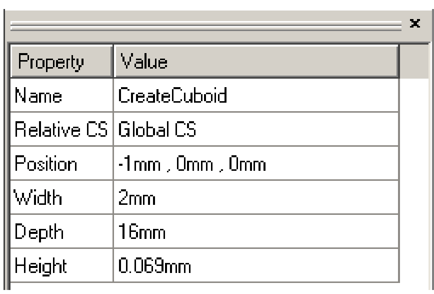
(3) Move the mouse icon to (-1mm, 0mm, 0mm) and press the left mouse button. Move the mouse icon to (1mm, 16mm, 0mm) and press the left mouse button. Move the mouse icon along the z-direction to 0.069mm and press the left mouse button.
You may not capture the exact position using the mouse, however, you can use the mouse coordinates at the bottom of the figure region to specify the exact coordinates.

Or , draw a box and then change its coordinates in the ***Properties*** *box.*

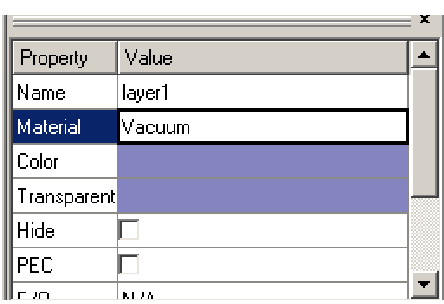
For example, draw a box in any dimensions, and select the object in the **Object Browser** box and change its name to “layer1” in the **Properties** box.

Select the **CreateCuboid** option in the “layer1” folder in the **Object Browser** box and change its coordinates in the **Properties** box.

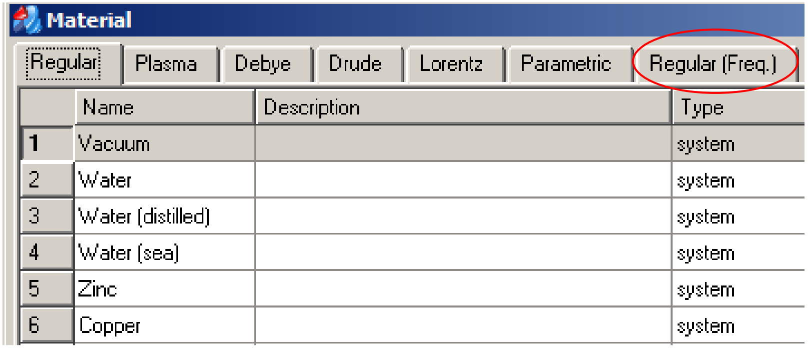


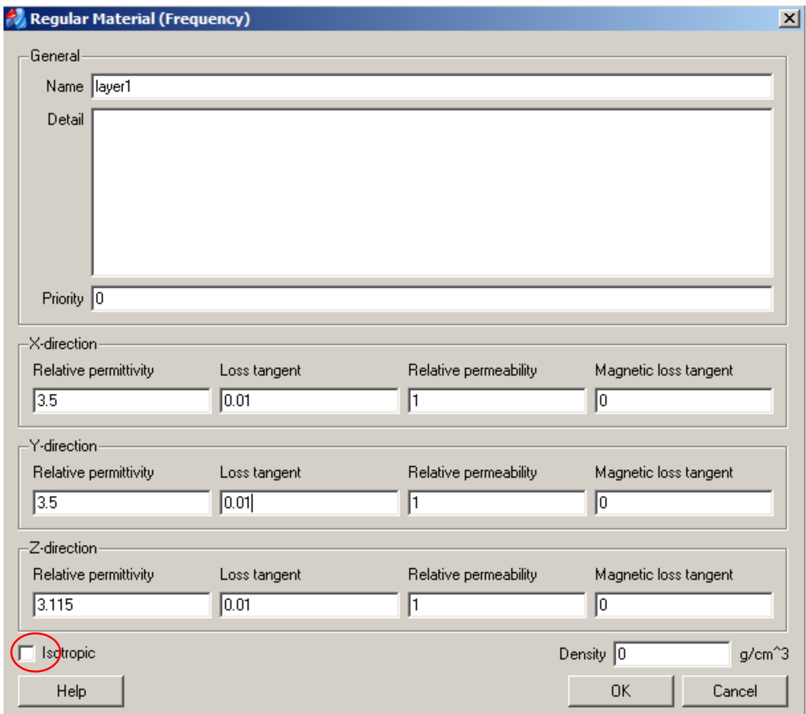


(4) Select the “layer1” option in the **Object Browser** box and click on the **Material** box in the **Properties** box.



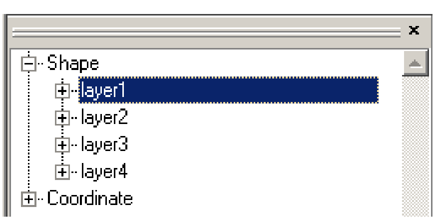
Click on the **Regular(Freq.)** label and specify the material parameters for the “layer1” dielectric.



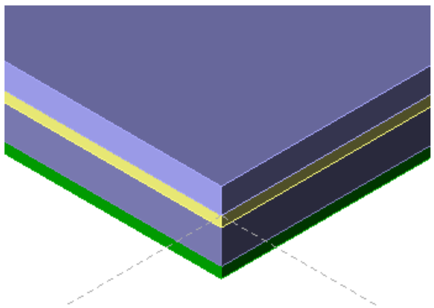


Click on the **OK** button in the **Regular Material (frequency)** window, and then on the **OK** button in the **Material** window to confirm the material specifications for the “layer1” dielectric.

(5) Follow the same procedure to add the dielectric layers 2, 3 and 4.

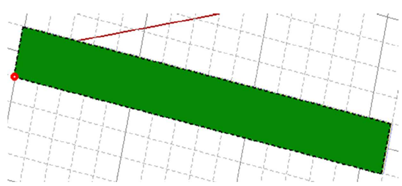


The order of the dielectric layers does not affect the simulation results since they are dielectric layers and their thickness is finite.

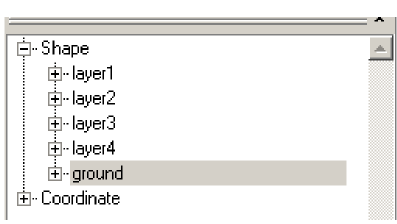


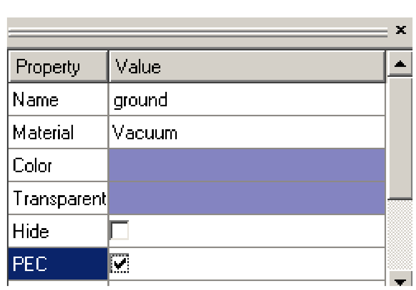
Follow the steps below to draw the PEC ground plane:

(1) Press and hold the left mouse button to rotate the model so that we can see the bottom of the "layer4" dielectric.
(2) Select the **Rectangle** icon in the **Tool Box->Geometry** box
(3) Move the mouse icon to select one corner of the bottom surface of the "layer4" dielectric, and then press the left mouse button.
(4) Move the mouse icon to the cross corner of the bottom surface of the "layer4" dielectric, and then press the left mouse button.



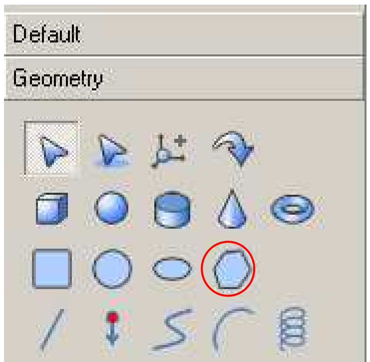
(5) Select the rectangle option in the **Object Browser** box, change its name to "ground" and its color to red in the **Properties** box, and check the **PEC** box.



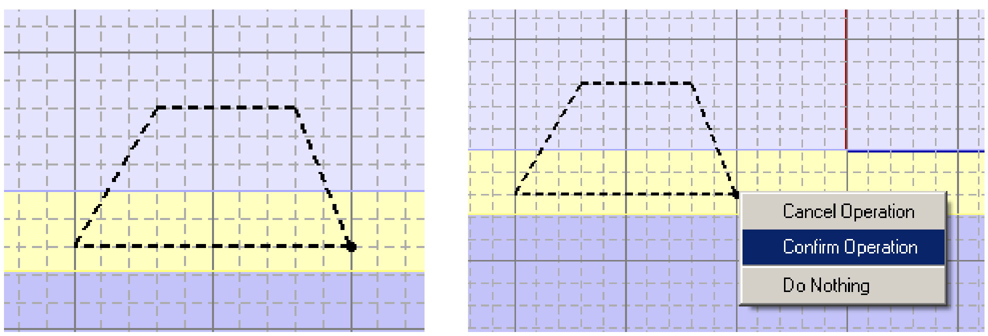


Follow the steps below to draw the signal lines:

(1) Select the **Polygon** icon in the **Tool Box->Geometry** box



(2) Draw a quadrangle in the figure region. At the last point, press the right mouse and select the **Confirm operation** option.



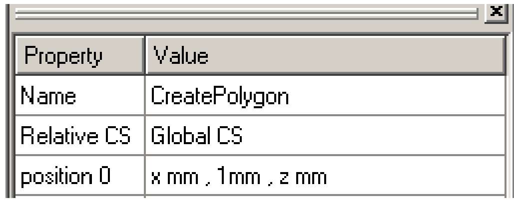
(3) Modify the coordinates using the following two ways:

(i) Select the polygon option in the quadrangle folder in the **Object Browser** box, and change the coordinates of points 1, 2, 3 and 4. You can use “-“, “+”, “*” , “/”, “sqrt”, and “^” operations in the coordinate expression.

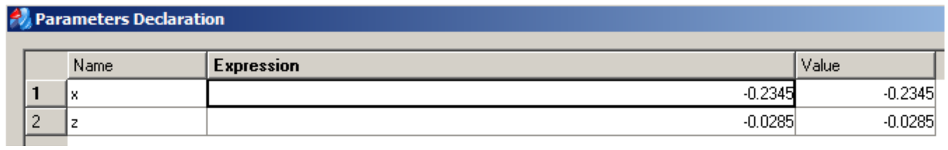
Property	Value
Name	CreatePolygon
Relative CS	Global CS
position 0	-0.02345mm , 1mm , -0.0285mm
position 1	-0.02345-0.117mm ,1mm , -0.0285mm
position 2	-0.02345-0.117+0.013mm ,1mm , 0.0285mm
position 3	-0.02345-0.117+0.013+0.091mm ,1mm , 0.0285mm

(ii) If you use a variable in the coordinate in the **Properties** box, the **Parameter Declaration** window will popup, and the variables will be listed in the variable list.

Click on the **Expression** column to input the variable value.



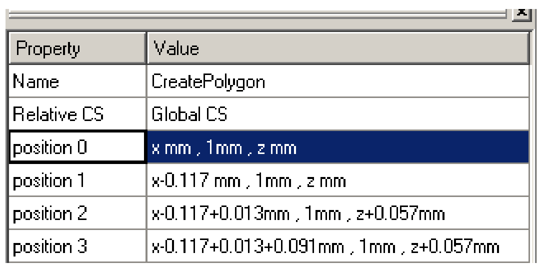
Property	Value
Name	CreatePolygon
Relative CS	Global CS
position 0	x mm , 1mm , z mm



Parameters Declaration

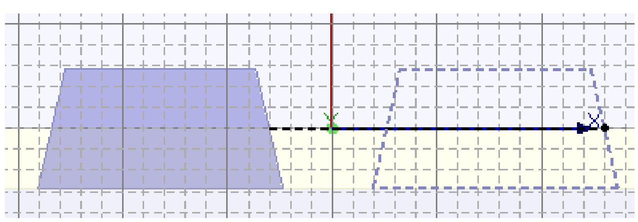
	Name	Expression	Value
1	x	-0.2345	-0.2345
2	z	-0.0285	-0.0285

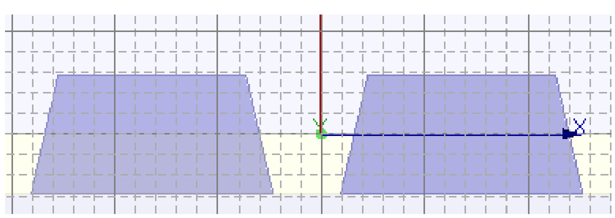
You can use the variables in the variable list to define a new variable in the **Properties** box.



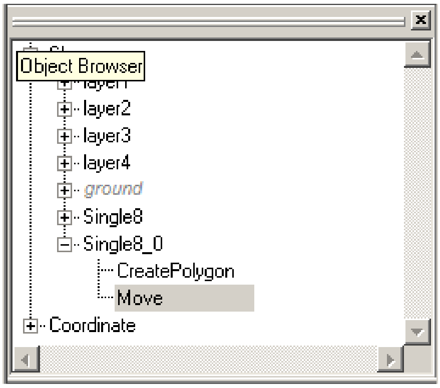
Property	Value
Name	CreatePolygon
Relative CS	Global CS
position 0	x mm , 1mm , z mm
position 1	x-0.117 mm , 1mm , z mm
position 2	x-0.117+0.013mm , 1mm , z+0.057mm
position 3	x-0.117+0.013+0.091mm , 1mm , z+0.057mm

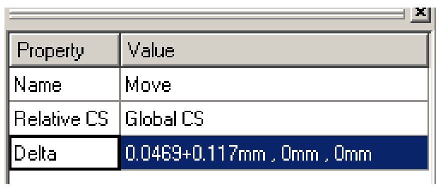
(4) Select the quadrangle option in the **Object Browser** box and then press the "Ctrl + c" and "Ctrl + v" keys. Select the pasted object in the **Object Browser** box, click on the **Move** button in the toolbar and then select a point on the pasted object and move the mouse icon along the x-direction. Click on the left mouse button to confirm the movement.





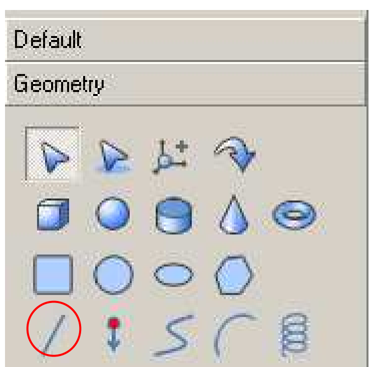
(5) Select the **Move** option in the second quadrangle folder in the **Object Browser** box and then change the coordinates in the Delta box (0.0469+0.117, 0, 0).





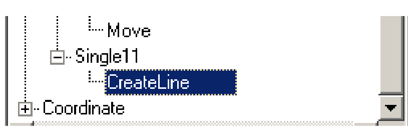
Select the “Single8” and “Single8_0” options in the **Object Browser** box and change the “Single8” and “Single8_0” to “line1” and “line2” in the **Properties** box, respectively.

(6) Select the **Line** icon in the **Tool Box->Geometry** box



Snap the mouse icon to one point on the second quadrangle, and draw a line along the y-direction. Ensure the drawing plane to be the either “X-Y” or “Y-Z” plane.

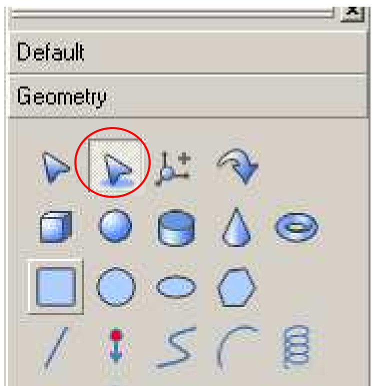
Select the **CreateLine** option in the **Object Browser** box



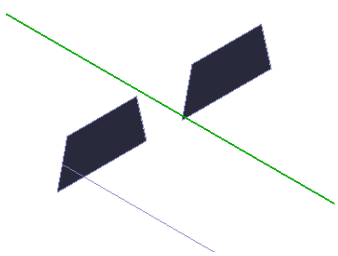
Change the x- and z-coordinates of **Position1** to be same as those of **Positon 0**. Change the y-coordinate of **Position 1** to “15”.

Property	Value
Name	CreateLine
Relative CS	Global CS
Position 0	0.13395mm , 1mm , 0mm
Position 1	0.13395mm , 15mm , 0mm

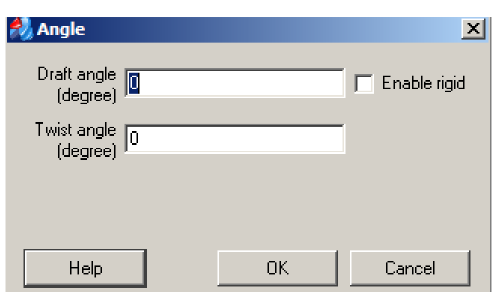
Click on the **Select face** icon in the **Object Browser** box.

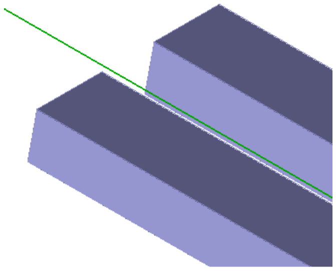


Select the “line1” object using the mouse icon and then press and hold the “Ctrl” or “Shift” button to select the “line2” and then the sweep line.

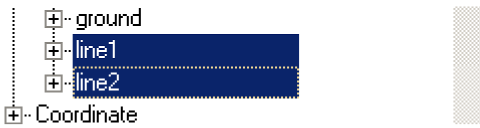


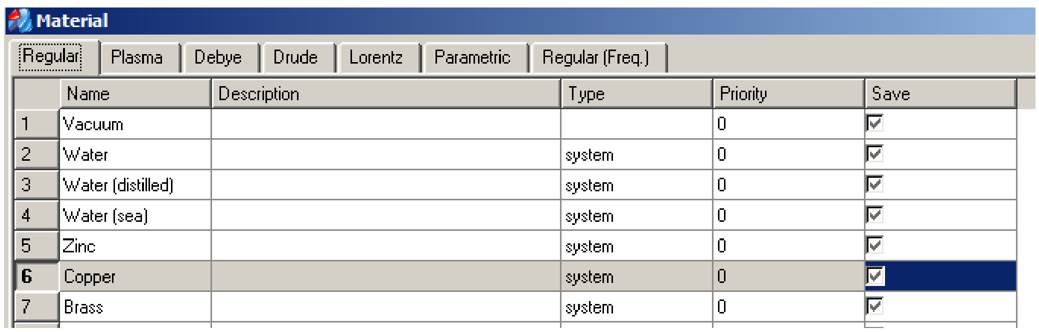
Click on the **Sweep** button in the toolbar. Click on the **OK** button in the **Angle** window to draw two signal lines.





(7) Select the **line1** and **line2** options in the **Object Browser** box, and then click on the **Material** box in the **Properties** box.

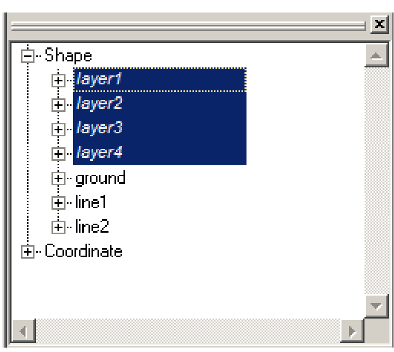




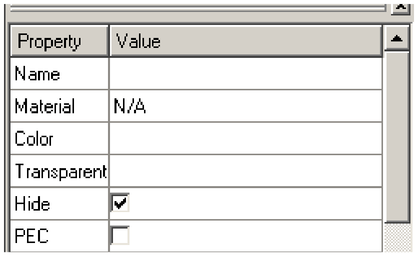
2.3 Add Excitation and Output Ports

Follow the steps below to add excitation and output ports:

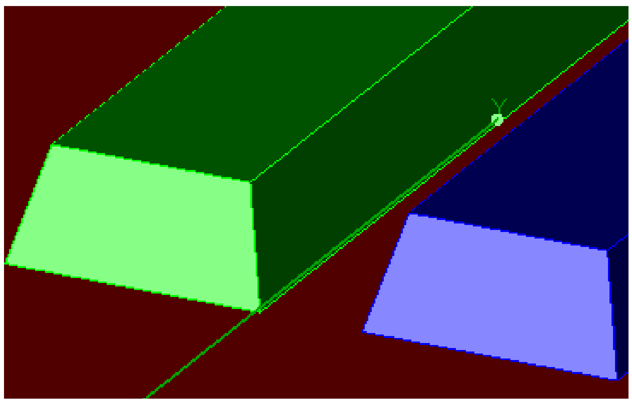
(1) Select the options “layer1”, “layer2”, “layer3”, and, “layer4” in the **Object Browser** box.



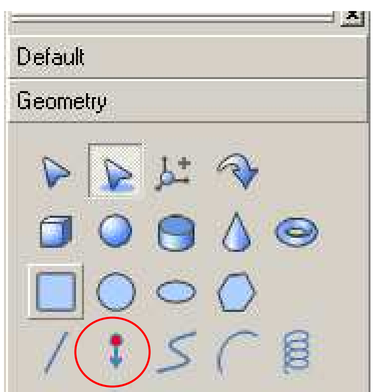
Check the **Hide** box in the **Properties** box



Adjust the color of “ground”, “line1” and “line2” for a better view.



(2) Click on the **Plumb line** icon in the **Object Browser** box.



(3) Draw two lines from the bottom center of signal lines to the ground plane.

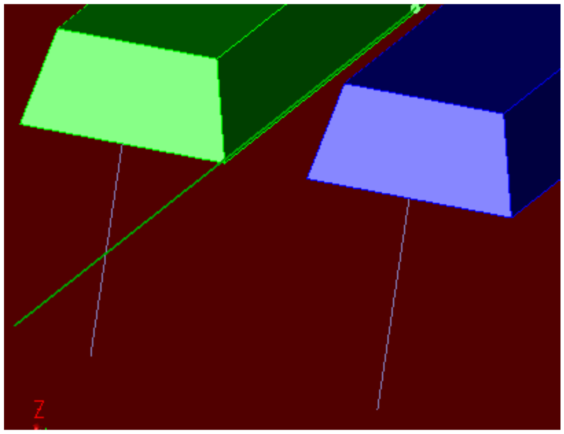
Change the drawing plane to the “X-Y” plane.

Snap to the bottom center of “line1” object and move the mouse along the –z-direction, and stop at the ground plane.

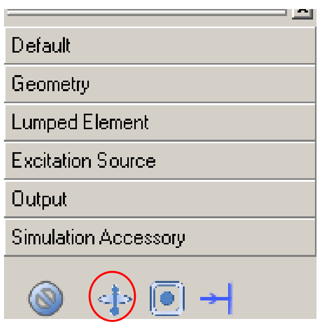
Change the line name to “port_excitation” in the **Properties** box.

Snap to the bottom center of “line2” object and move the mouse along the –z-direction, and stop at the ground plane.

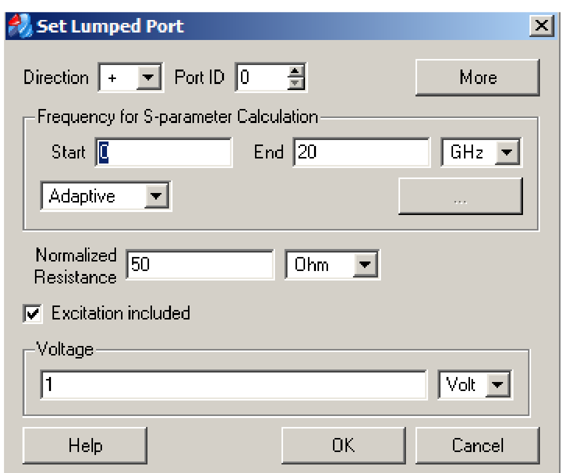
Change the line name to "port_output" in the **Properties** box.



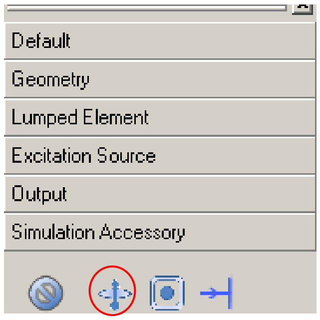
Select the "port_excitation" in the **Object Browser** box, and click on the **Lumped port** icon in the **Tool Box->Simulation Accessory** box.



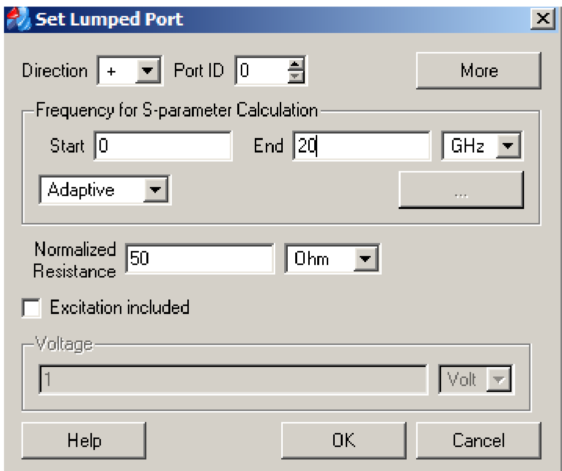
Specify the frequency range of interest and click on the **OK** button to confirm the setting.



Select the "port_output" in the **Object Browser** box, and click on the **Lumped port** icon in the **Tool Box->Simulation Accessory** box.

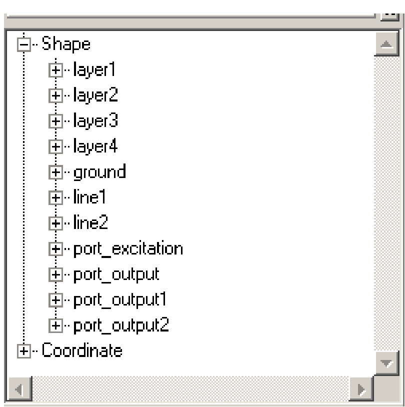


Specify the frequency range of interest and click on the **OK** button to confirm the setting.



Follow the similar procedure to add two more output ports at the other side of the signal lines.

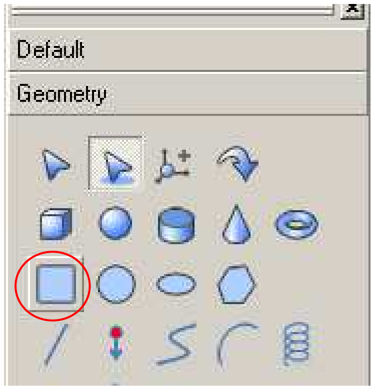
We have four dielectric layers, ground plane, two signal lines, one excitation port, and three output ports so far in the project model.



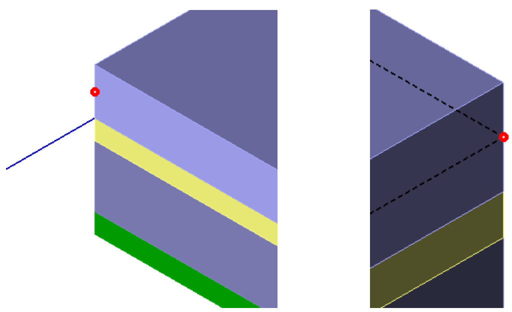
2.4 Add Field Distribution Output

Follow the steps below to add field distribution output:

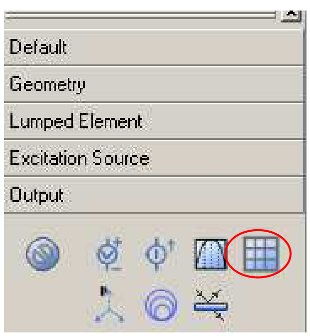
(1) Click on the **Rectangle** icon in the **Object Browser** box.



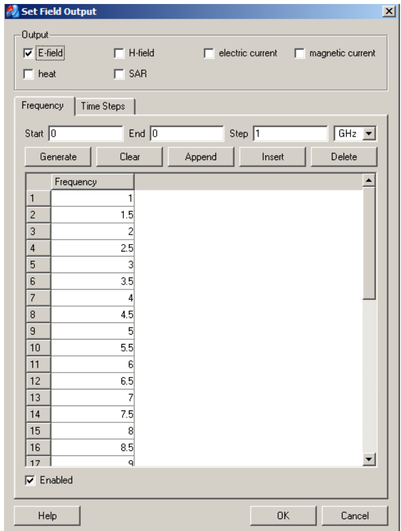
(2) Snap the mouse icon to the middle of the first dielectric layer, draw a rectangle with the same size as the dielectric layer.

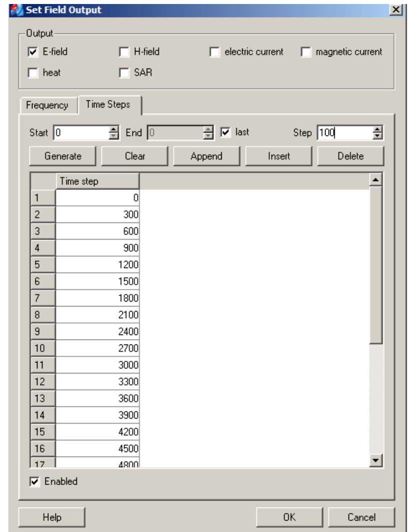


(3) Select the rectangle option in the **Object Browser** box and change the rectangle name to “field_output” in the **Properties** box, and then click on the **Field** icon in the **Tool Box->Geometry** box.



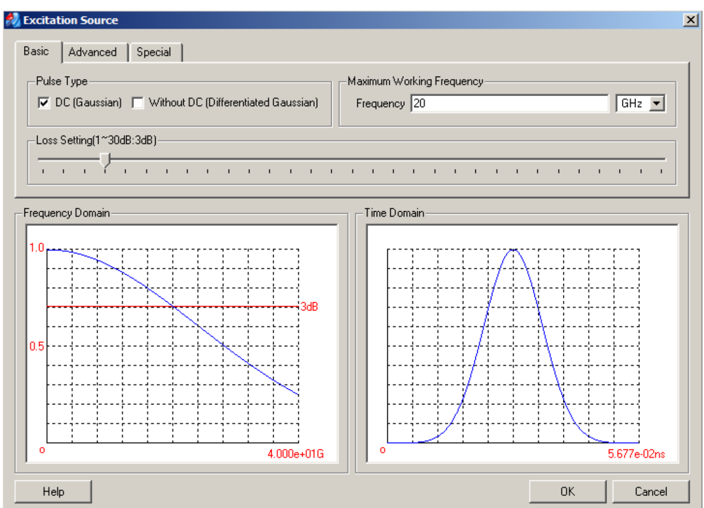
Specify the frequency range of interest, and click on the **Generate** button to generate the frequency list. Click on the **Time Steps** button and on the **Enabled** button, Specify the number of time steps for the field animation. Click on the **OK** button to confirm the settings.





2.5 Set Excitation Pulse

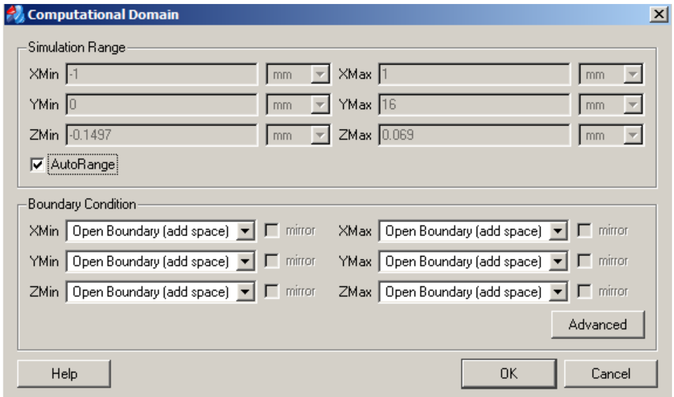
Click on the **Set excitation source** button in the toolbar. Specify the highest frequency of interest, “20” in the Frequency box.



2.6 Set Domain and Boundary Condition

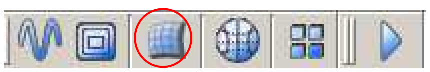
Click on the **Set boundary condition** button in the toolbar.

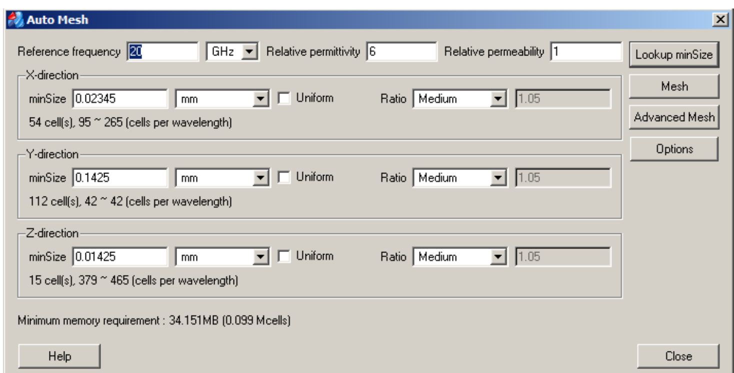
Use the **Open Boundary(add space)** in all six directions for the finite structure. Click on the **OK** button to confirm the setting.

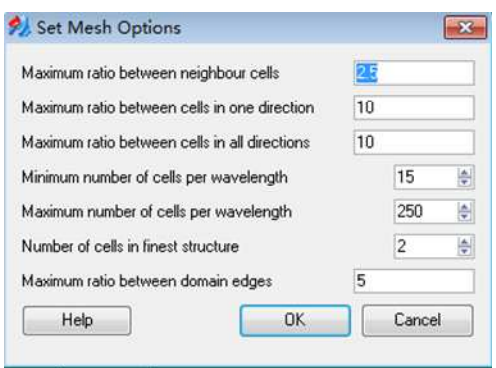


2.7 Design Mesh Distribution

Click on the **Auto Mesh** button in the toolbar. Click on the **Options** button and change the number in the **maximum ratio between cells in all directions** box from “20” to “10”. Otherwise, the mesh ratio is too large in some area; in turn, it may generate the error in the results.



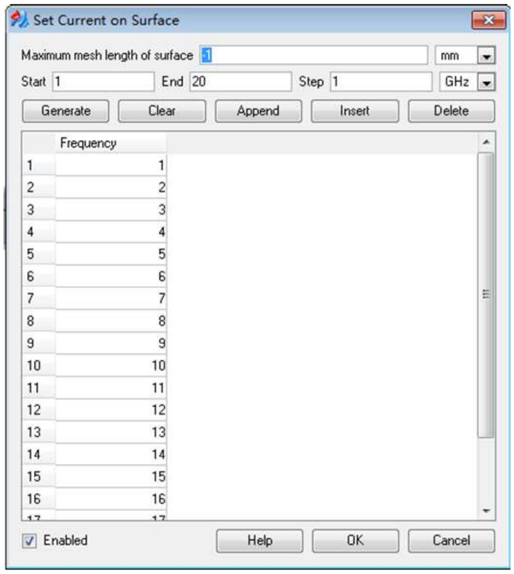




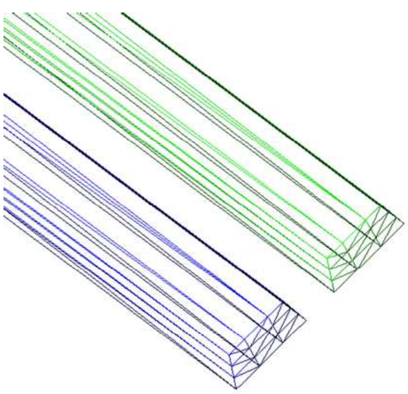
2.8 Set Conformal Current Output

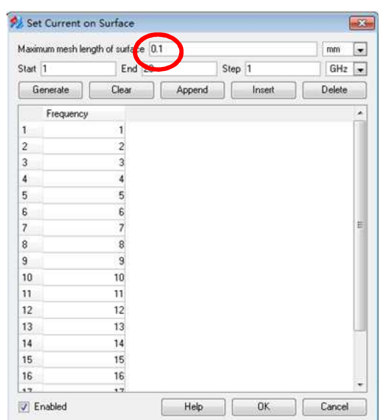
Select the two parallel signal lines in the **Object browser** box, and click on the “Disabled” option with label “Surf-current” in the **Properties** box. Set the frequency range and press the

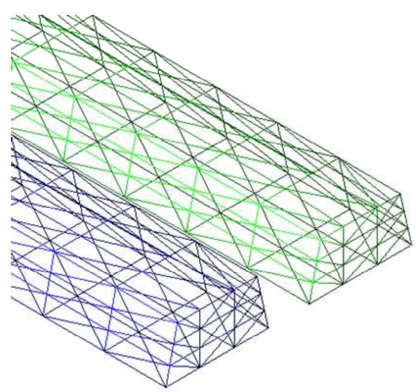
Generate button in the popup window. The number “-1” means that GEMS will use the default surface mesh for surface current calculation.



Select the two parallel signal lines in the **Object browser** box, and select the “Tools->Surface mesh” option to view the mesh for surface current distribution. It is evident that the mesh is too coarse for the surface current. To increase the resolution, click the “Enabled” in the Surf_current option in the **Properties** box. Change the number from “-1” to “0.1” (mm) in the “Maximum mesh length of surface” box. Select the “Tools->Surface mesh” option to view the improved mesh for surface current distribution





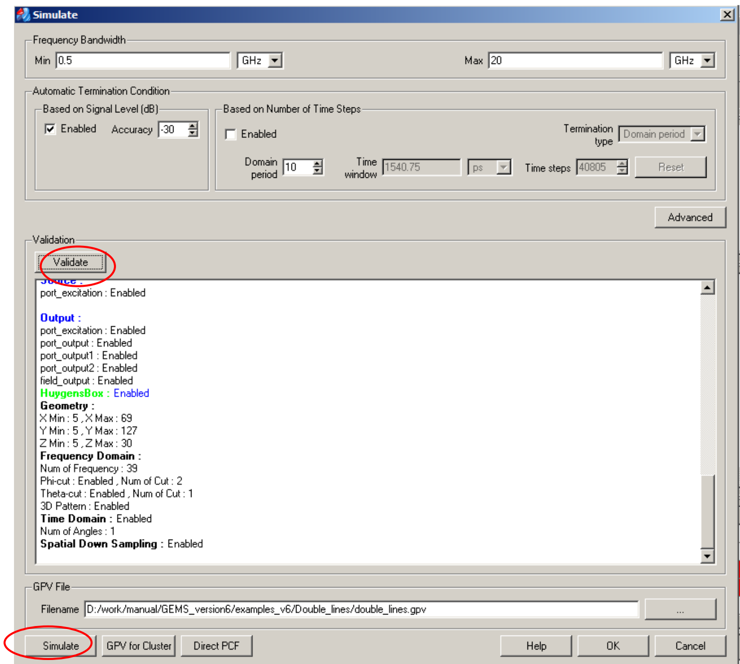


2.9 Save Project

Click on the **Save** button in the toolbar.

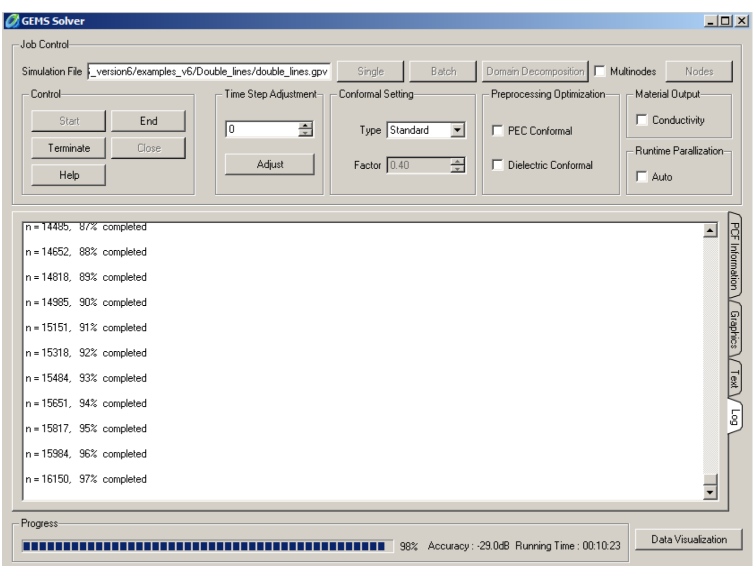
2.10 Generate Simulation File

Click on the **PreCalculate** button in the toolbar.



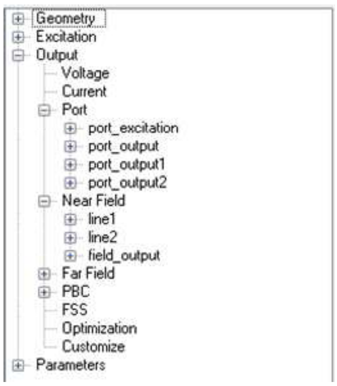
2.11 Simulate Project

Click on the **Simulate** button in the *Simulate* window.



2.12 Result Visualization

Click on the **Data Visualization** button in the *GEMS Solver* window. The outputs include (i) Time domain voltage and current at four ports; (ii) S-parameters; (iii) Port impedance; (iv) Field distribution on a selected surface; (v) Far fields.



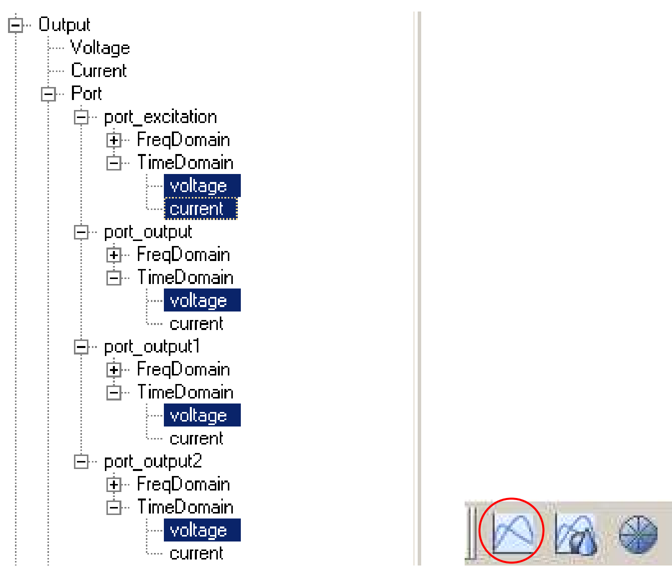
Follow the steps below to draw four voltages in the same figure:

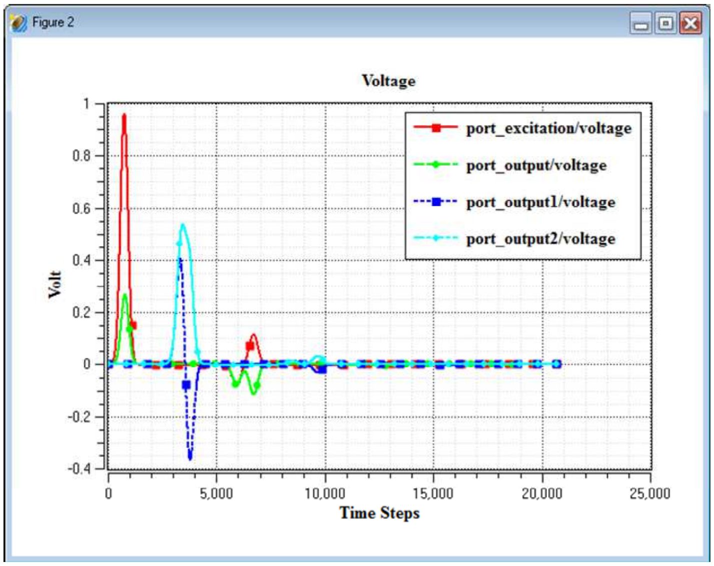
(i) Double-click on the voltage at the "port_excitation" in the result tree

(ii) Select the "port_output" option and click on the **Add to current window** button in the toolbar

(iii) Select the "port_output1" option and click on the **Add to current window** button in the toolbar

(iv) Select the "port_output2" option and click on the **Add to current window** button in the toolbar





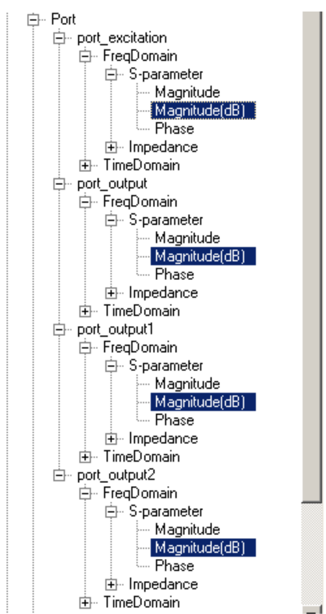
Follow the steps below to draw four S-parameters in the same figure:

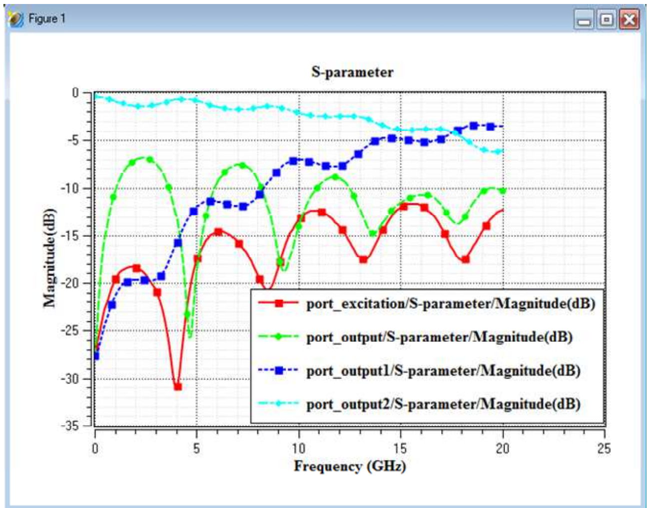
(i) Double-click on the voltage at the “Magnitude(dB)” in the “port_excitation” folder

(ii) Select the “Magnitude(dB)” option in the “port_output” folder and click on the **Add to current window** button in the toolbar

(iii) Select the “Magnitude(dB)” option in the “port_output1” folder and click on the **Add to current window** button in the toolbar

(iv) Select the “Magnitude(dB)” option in the “port_output2” folder and click on the **Add to current window** button in the toolbar

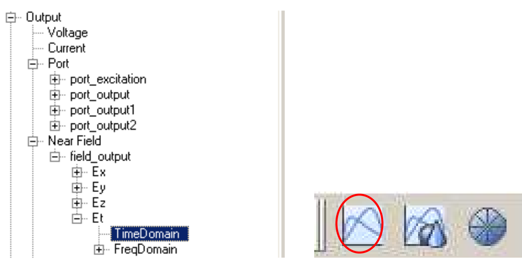




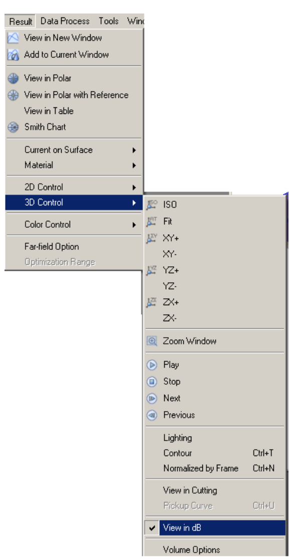
Follow the steps below to plot the field distribution on the selected surface at the different time:

(i) Click on the **View model** button in the toolbar, and then on **Transparent mode** button.

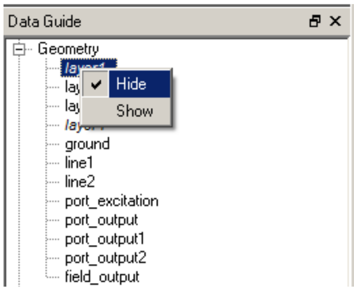
(ii) Select the "TimeDomain" option in the "field_output" folder and click on the **Add to current window** button in the toolbar



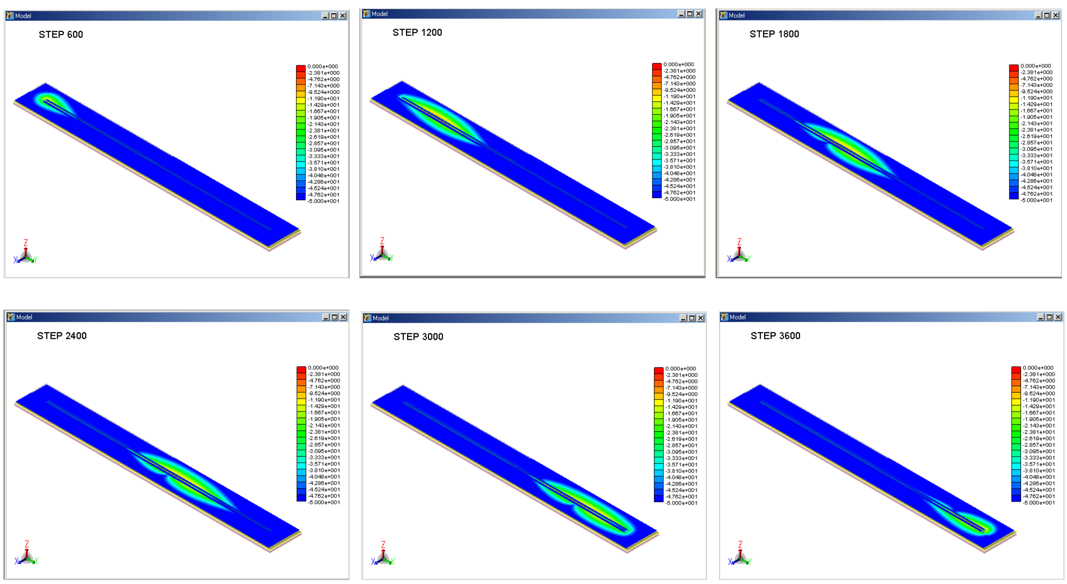
(iii) Check the "View in dB" option for a better view



(iv) Click on the **Geometry** entrance in the result tree, and select the "layer1" dielectric option and then press the right mouse button and select the "Hide" option for a better view.

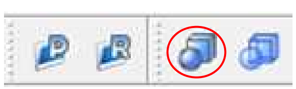


(v) Click on the **Play** button in the toolbar to view the field animation.

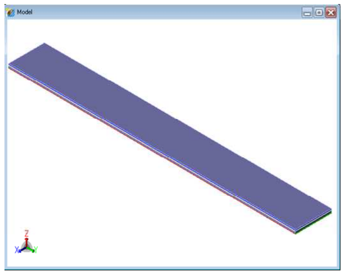


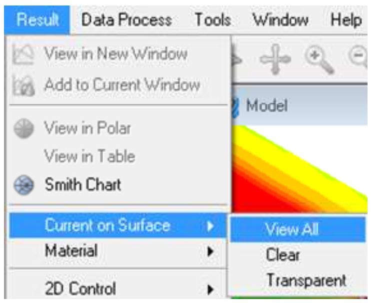
Follow the steps below to plot the 3-D surface current distribution at the different frequencies:

(i) Click on the **View model** button in the toolbar.



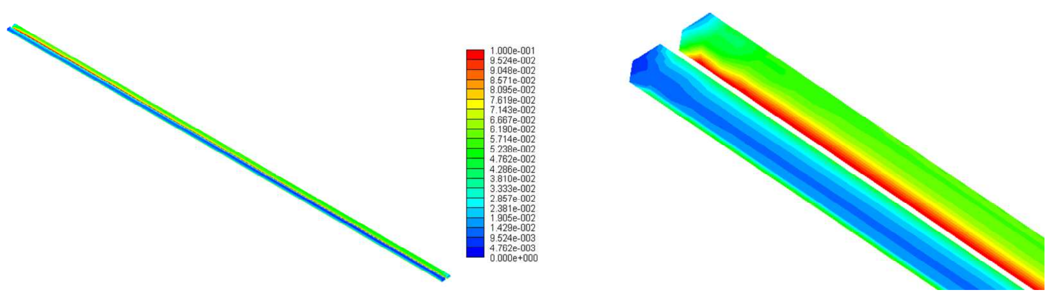
(ii) Move the mouse icon to the Geometry in the result tree, and click the right mouse button and select the "Hide" option. Select the Result->Current on Surface->ViewAll" option and select the frequency you like to view.



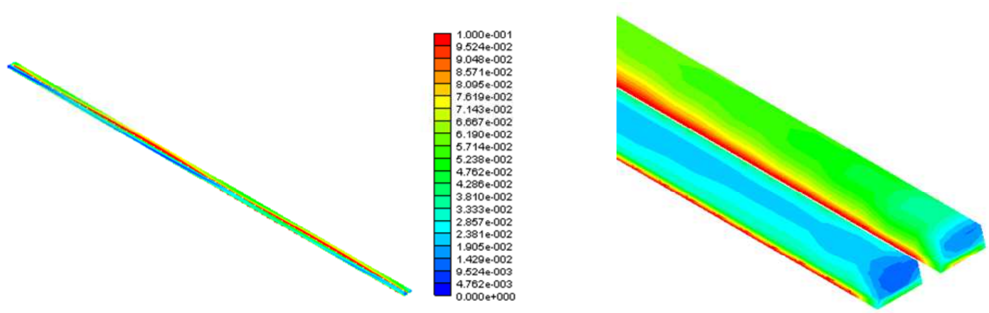


(iii) Select the "Result->Color Control->Color bar" option and select the color display range in the popup window by check the "Fixed range" box and adjust the minimum and maximum values.

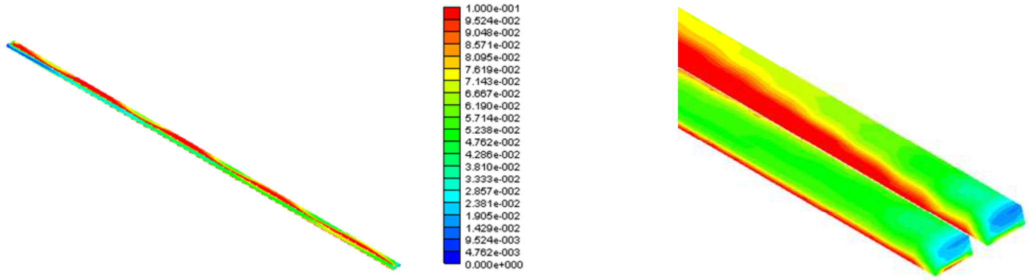
Frequency = 1GHz



Frequency = 10GHz

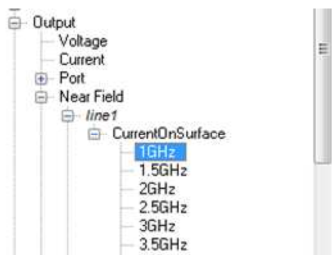


Frequency = 20GHz

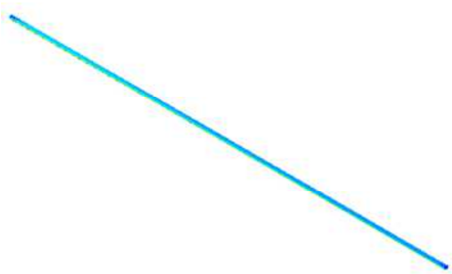


(iv) You can select one signal line to view the surface current following the steps below:

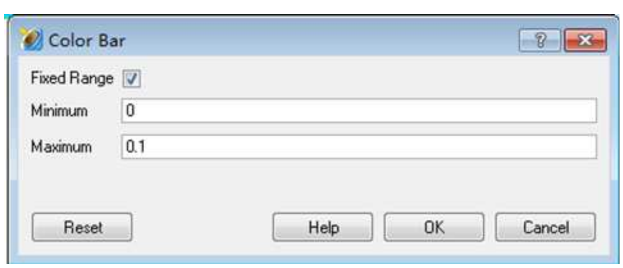
(a) Open the sub-folder in the result tree;
(b) Select the frequency of interest;

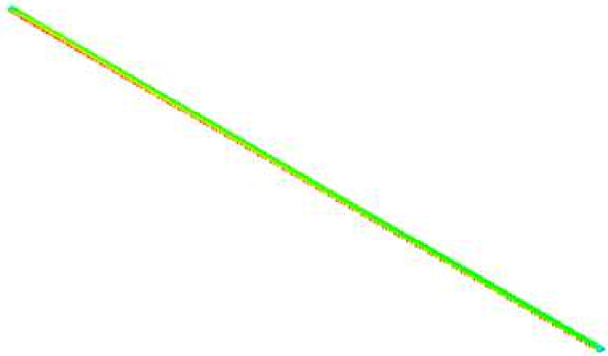


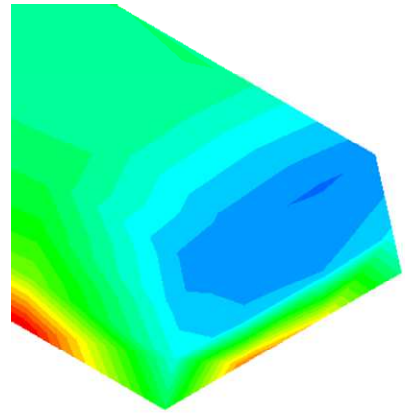
(c) Double-click the frequency “1GHz”.



(d) Adjust the color range by selecting the “Result->Color Control->Color bar” option.







3

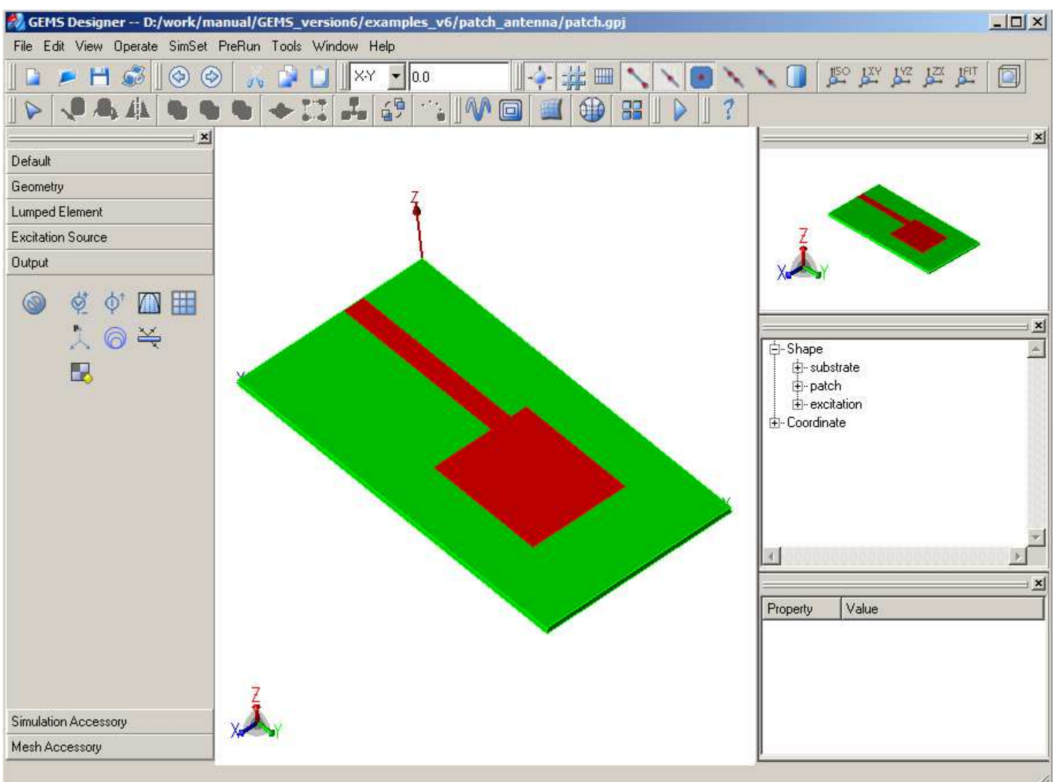
Example 3. Patch Antenna

Description: A rectangular patch antenna with an infinitely large ground plane and substrate is excited by a wave port. The feed microstripline is infinitely long and is truncated at the domain boundary. The output parameters include return loss and far field pattern.

Keywords: Open Boundary (add space), open Boundary (touched), perfect E boundary, wave port, return loss, and far field pattern.

3.1 Patch Antenna Configuration

The patch antenna includes a rectangular patch (thin PEC), microstrip feed line (thin PEC), substrate (relative permittivity=2.2), and ground plane (PEC). The mrcorstrip line is infinitely and terminated by the absorbing boundary.



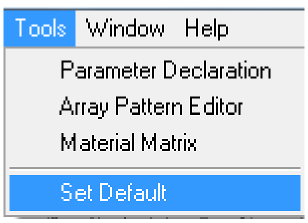
The difference of the wave port from the lumped port is that the wave port is used to excite a matched port. The lumped port is used to excite an open port.

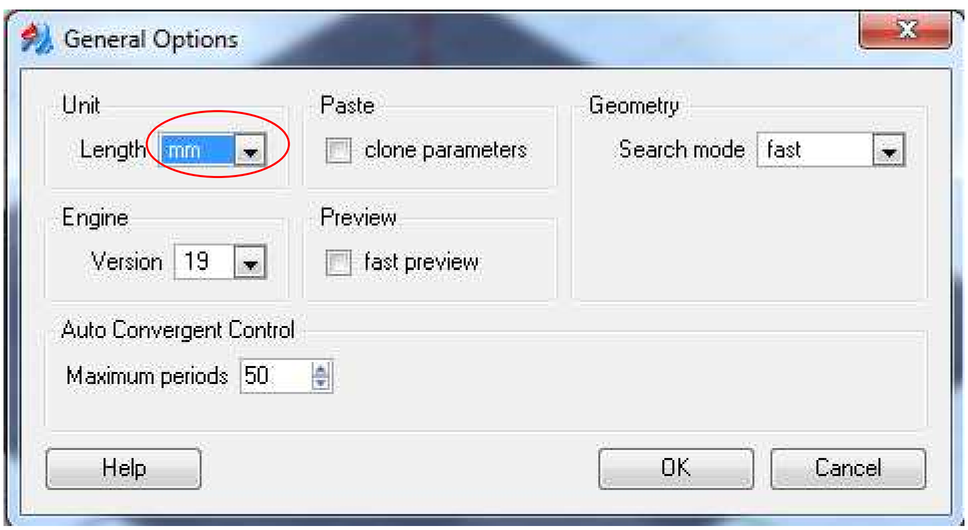
The frequency band of interest is from very low frequency to 20GHz. But the far field patterns are only at the several selected frequencies. Since the ground plane is infinitely large; hence, we can use the PEC boundary of computational domain to serve as the ground plane.

3.1 Create Patch Antenna Model

Follow the steps below to create the patch antenna model:

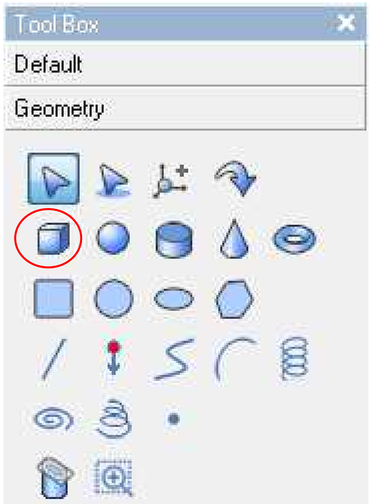
(1) Open the GEMS Designer
(2) Select the **Set Default** option in the **Tools** menu

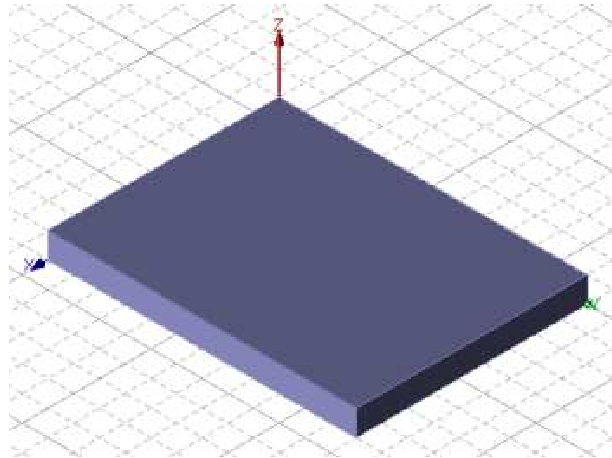




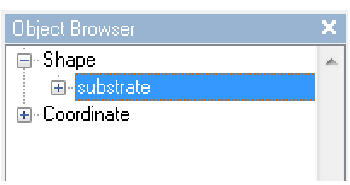
(3) Click on the **New** button in the toolbar to create a new project

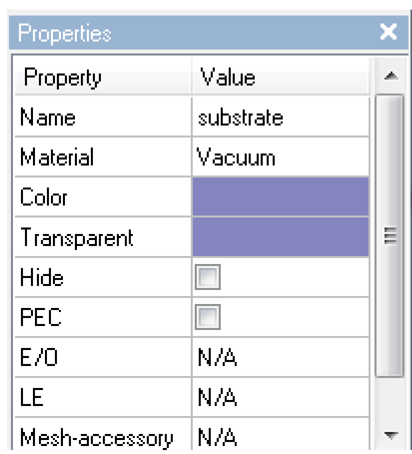
(4) Select the **Cuboid** icon in the **Tool Box->Geometry** box, and draw a box in the figure region.



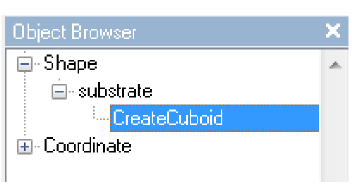


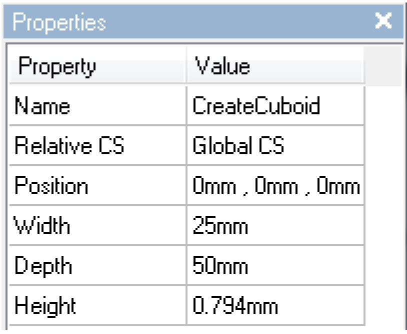
(5) Select the "Single0" in the **Object Browser** box, and change its name to "substrate" in the **Properties** box. Click on the **Color** bar in the **Properties** box and set the substrate color to be green.





(6) Select the **CreateCuboid** option in the **substrate** folder, and then change the dimensions of the substrate, the reference point (0, 0, 0), width=25mm, length=50mm, and thickness=0.794mm.

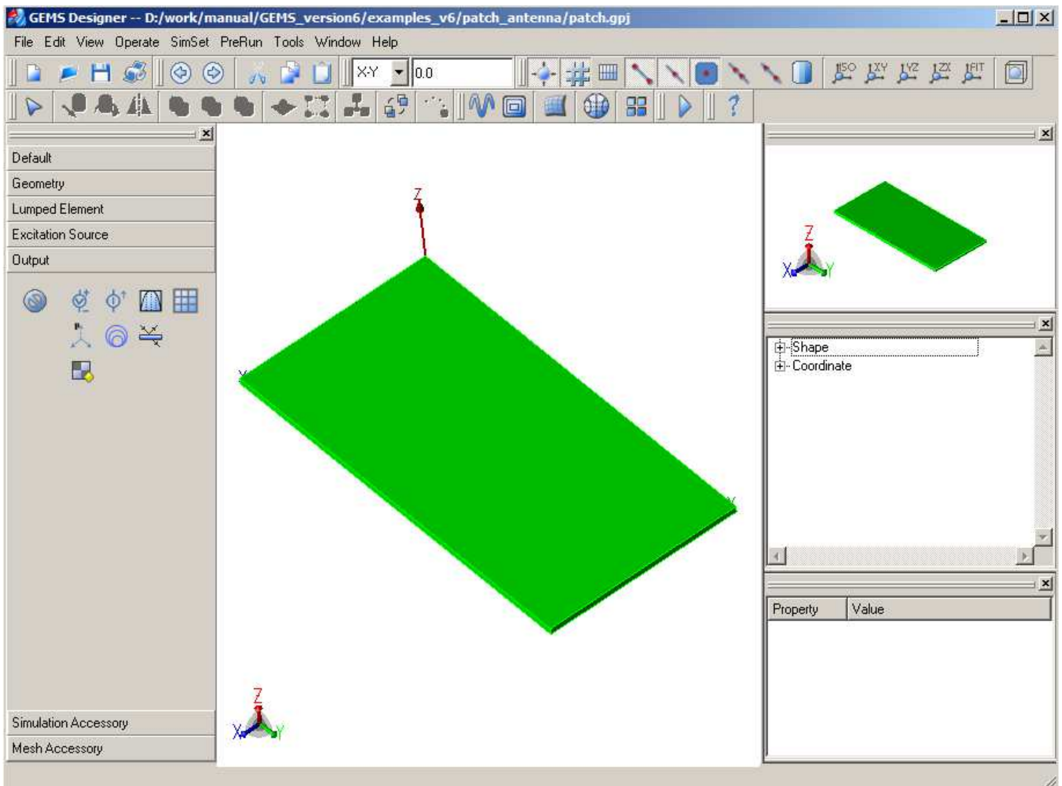




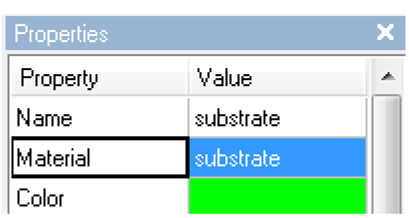
Properties

Property	Value
Name	CreateCuboid
Relative CS	Global CS
Position	0mm , 0mm , 0mm
Width	25mm
Depth	50mm
Height	0.794mm

Click on the **FIT** button in the toolbar to fit the substrate inside the figure region.



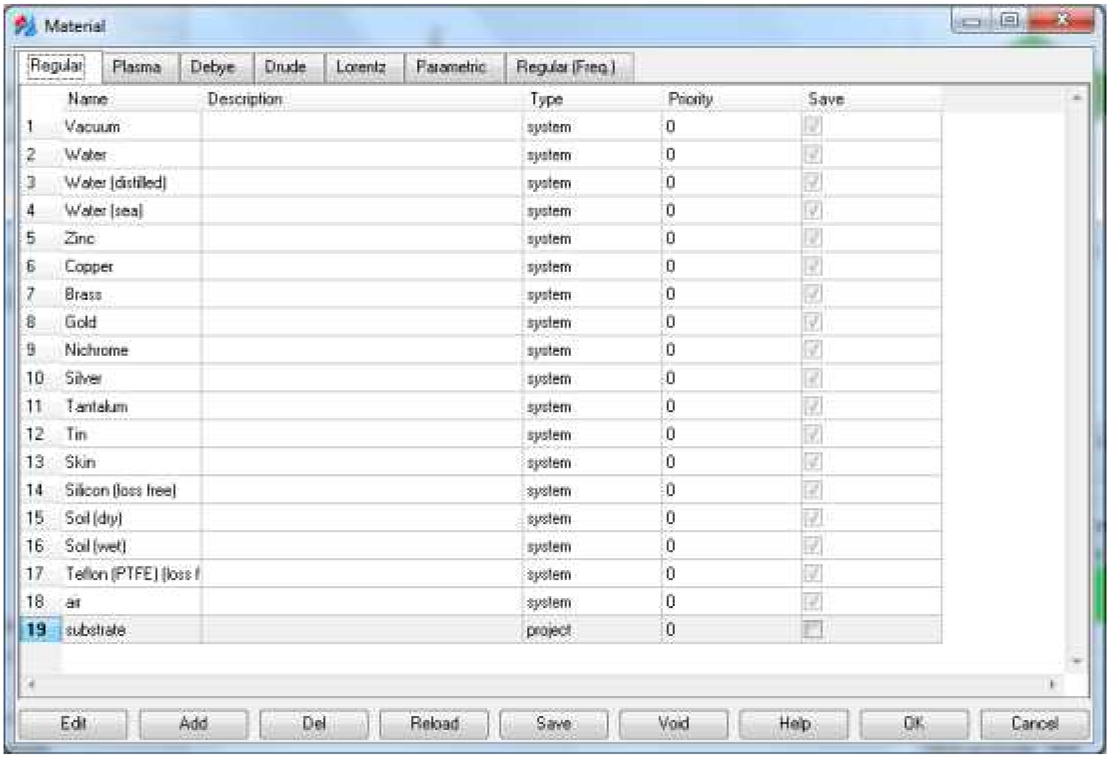
(7) Select the **substrate** option in the **Object Browser** box and click on the **Material** box in the **Properties** box. In this example, the substrate is lossless material, namely, its conductivity is zero.

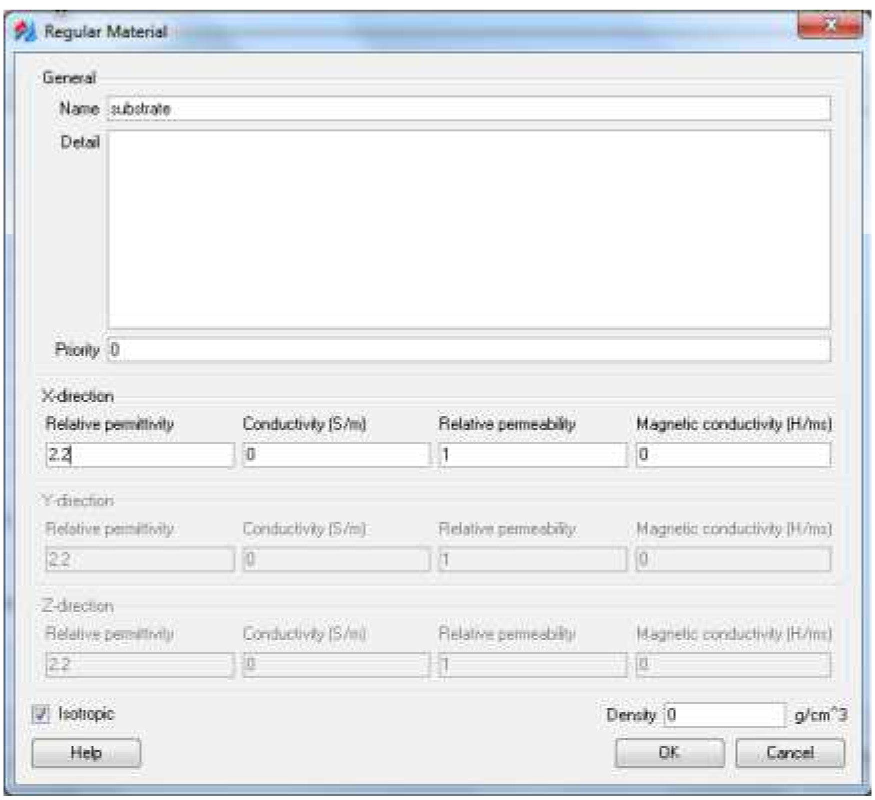


Properties

Property	Value
Name	substrate
Material	substrate
Color	

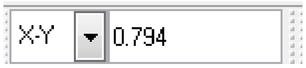
Click on the **Add** button to add a new material for this substrate. Specify a name in the **Name** box, and input the dielectric constant (2.2) in the **Relative permittivity** box.





Click on the **OK** button to confirm the dielectric setting.

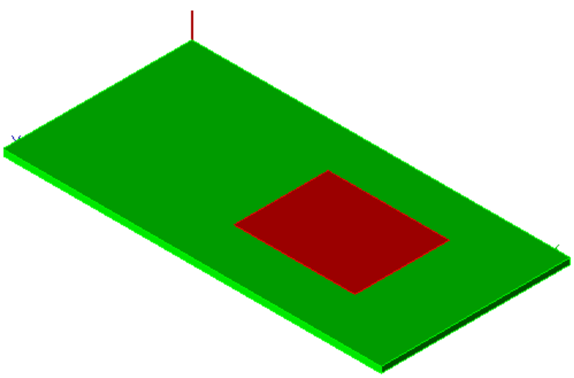
(8) Change the height of the drawing plane to be 0.794.



Click on the **Rectangle** icon in the **Tool Box->Geometry** box, and draw a rectangle on the surface of substrate.

Select the patch option in the **Object Browser** box, and change its name to “patch” and color to red, and check the **PEC** box in the **Properties** box.

Select the **CreateRectangle** option in the **patch** folder in the **Object Browser** box, and change its coordinates to (6mm , 24mm , 0.794m), width=12.45mm, and length=16mm in the **Properties** box.

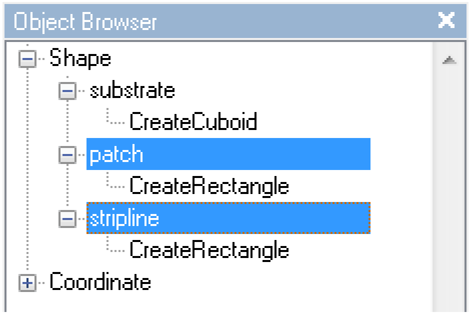


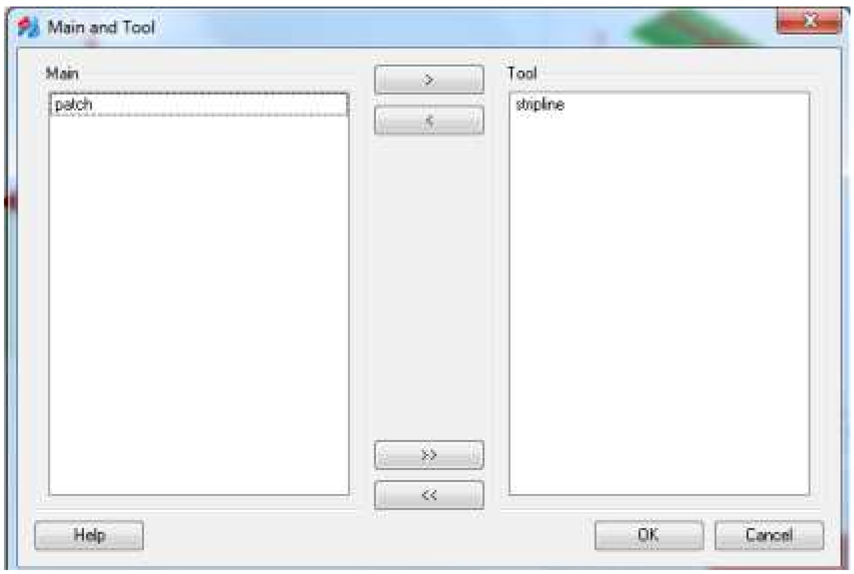
(9) Click on the **Rectangle** icon in the **Tool Box->Geometry** box, and draw a microstrip line on the surface of substrate.

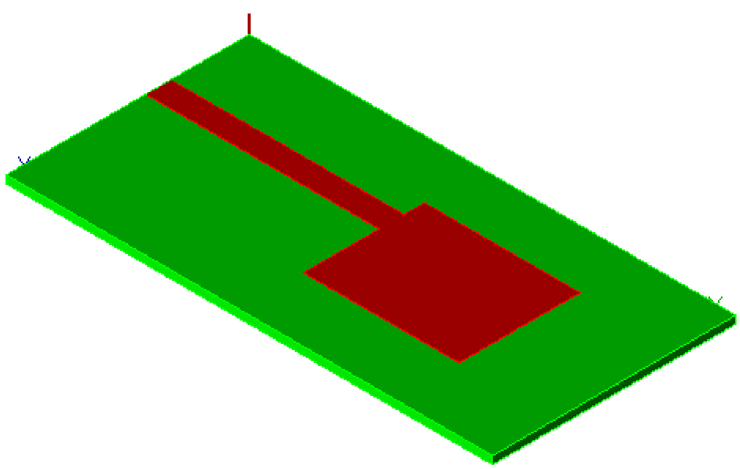
Select the microstrip line option in the **Object Browser** box, and change its name to “stripline” and color to red, and check the **PEC** box in the **Properties** box.

Select the **CreateRectangle** option in the **stripline** folder in the **Object Browser** box, and change its coordinates to (8.09mm , 0mm , 0.794m), width=2.56mm, and length=-24mm in the **Properties** box.

Select both the **patch** and **stripline** options in the **Object Browser** box, and click on the **Unite** button in the toolbar. The stripline and patch will be united as a single object.



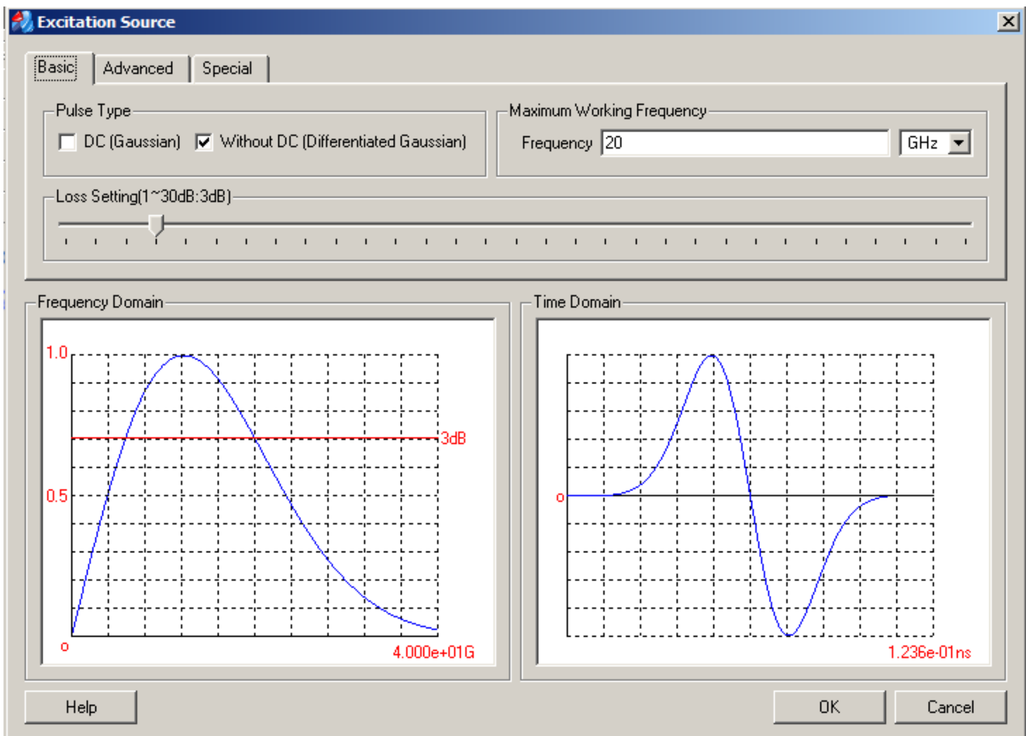




The order is very important for an object without thickness and it touches other objects. The wrong object order will erase the infinitely thin structure; in turn, the simulation result may be incorrect.

3.2 Set Excitation Pulse

Click on the **Set excitation source** button in the toolbar. Since the maximum frequency of interest is 20GHz, so we type "20" in the **Frequency** box (unit is GHz). Click on the **OK** button to confirm the excitation pulse setting.

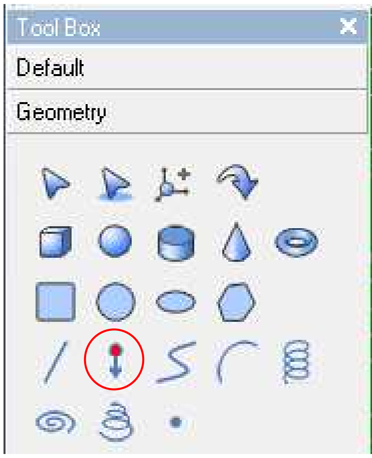


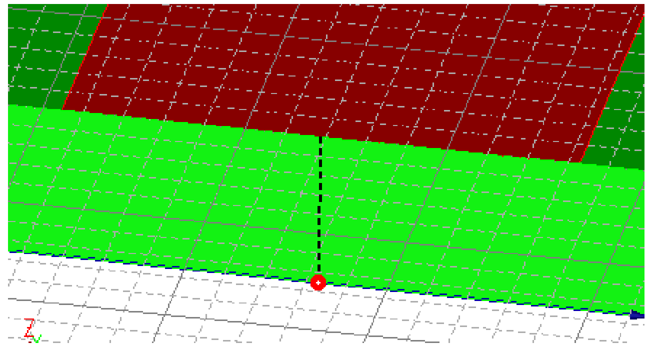
3.3 Create Wave Port Excitation

Since the feed line is terminated at the domain boundary, it is perfectly matched at the domain boundary. The excitation wave port is not only an excitation source, but also an output port that returns the time domain port voltage and current, port impedance, and return loss.

Click on the **Plumb line** icon in the **Tool Box->Geometry** box, and move the mouse icon to the center of the stripline, press the left mouse button, move the mouse icon along the vertical direction (the drawing plane is "X-Y"), stop when it meets the bottom of the substrate.

Select the **Plumb line** option in the **Object Browser** box and change the name of the plumb line to be "excitation" in the **Properties** box.

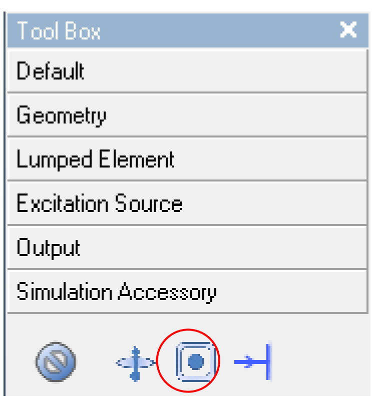




For an infinitely large ground and dielectric substrate, we can use the wave port to excite the antenna. The ground plane and substrate is truncated by PML boundary (Open Boundary (touch)) in the horizontal directions. Since there is no field below the ground, the perfect E boundary is used to truncate the domain in the domain bottom.

For a finite ground and dielectric substrate, we can use the lumped port to excite the antenna. There is a gap between the ground plane and the PML boundary (Open Boundary (Add Space)) in the horizontal directions. The Open Boundary (Add space) is used to truncate the domain in the domain bottom and top.

In this example, the ground and substrate are infinitely large, so we use the wave port to excite the patch antenna. Select the "excitation" option in the **Object Browser** box and click on the **Wave port** icon in the **Tool Box->Simulation accessory** box.

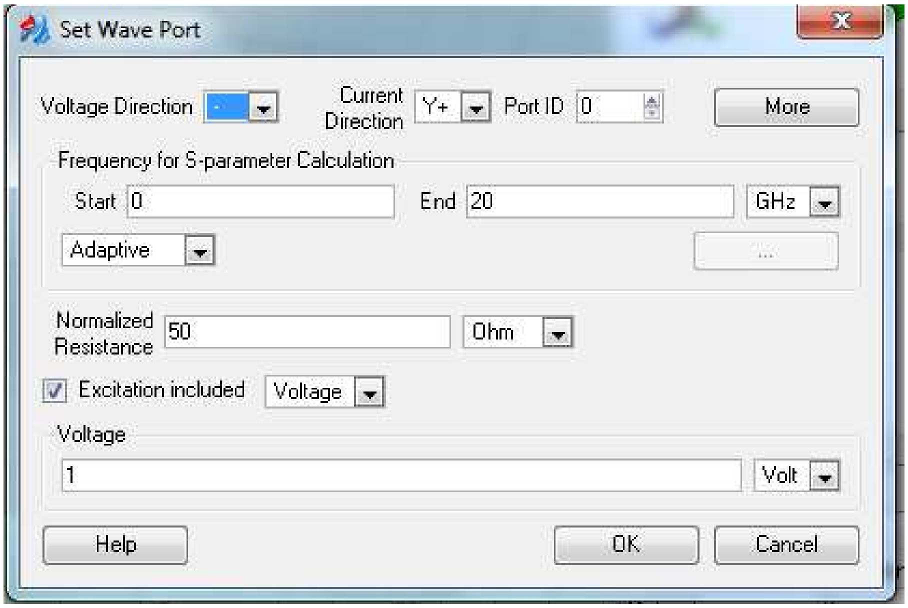


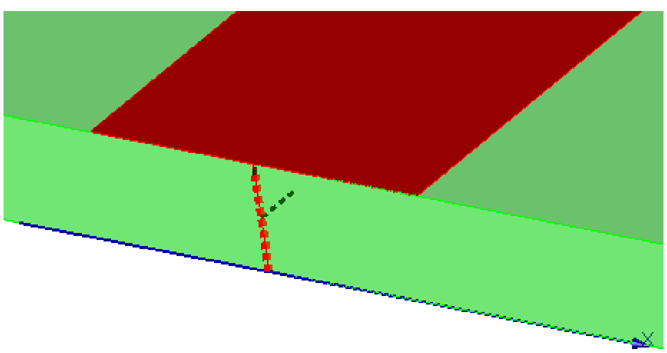
Two things are important when the wave port is used as an excitation:

(i) The black end should touch the signal conductor (higher potential). In this example, it should touch the stripline. It can be adjusted by the sign in the **Voltage Direction** box.

(ii) The black dash line should point to the power propagation direction. In this case, it points to the +y-direction.

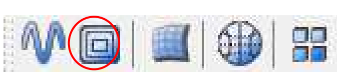
The value in the **Normalized Resistance** box is a reference for the return loss calculation. The wave port will have a different ID when the project includes more than one excitation sources and they are excited independently.



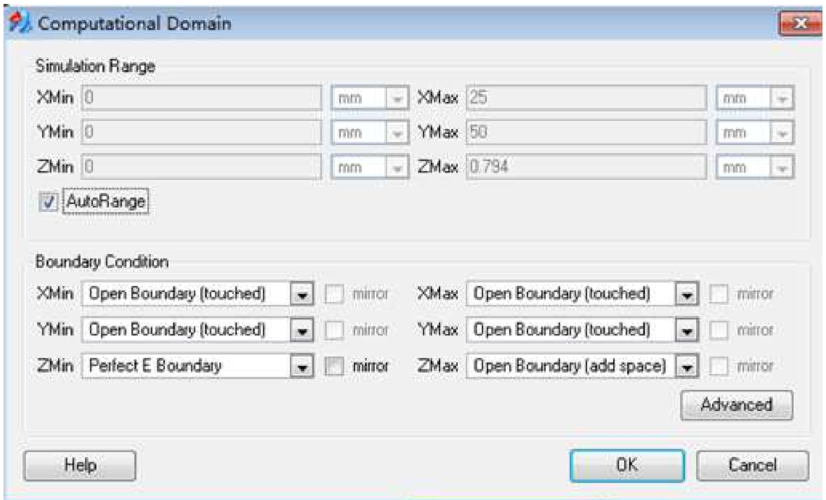


3.4 Domain and Boundary Settings

Click on the **Set boundary condition** button in the toolbar.

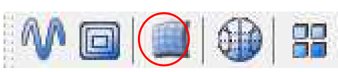


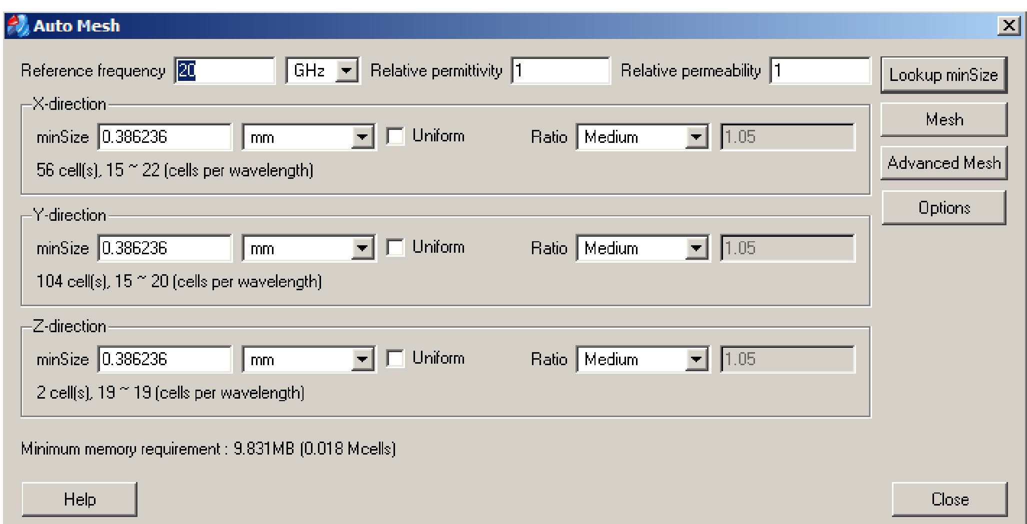
We need to understand the boundary type used in this project. (i) Both the substrate and ground plane are infinite in the horizontal directions. The “Open Boundary (touched)” should be selected to truncate the horizontal directions so that there is no reflection from the edge of the substrate and ground plane. (ii) The “Perfect E Boundary” at the “Zmin” side serves as the boundary and ground plane at the same time. (iii) The “Open Boundary (add space)” should be used to truncate the “Zmax” direction, which allows a white space between the absorbing boundary of domain and patch. It means that the patch antenna has an open space in the upper hemisphere.



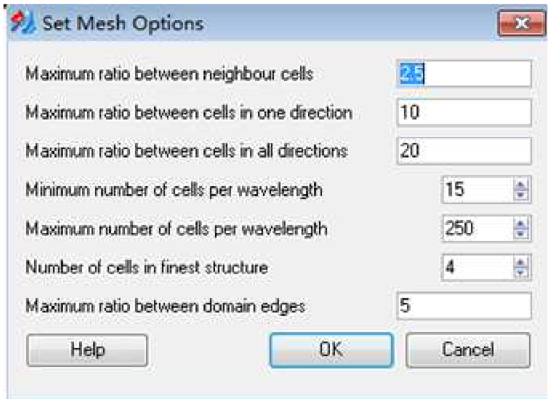
3.5 Mesh Generation

Click on the **Auto mesh** button in the toolbar.

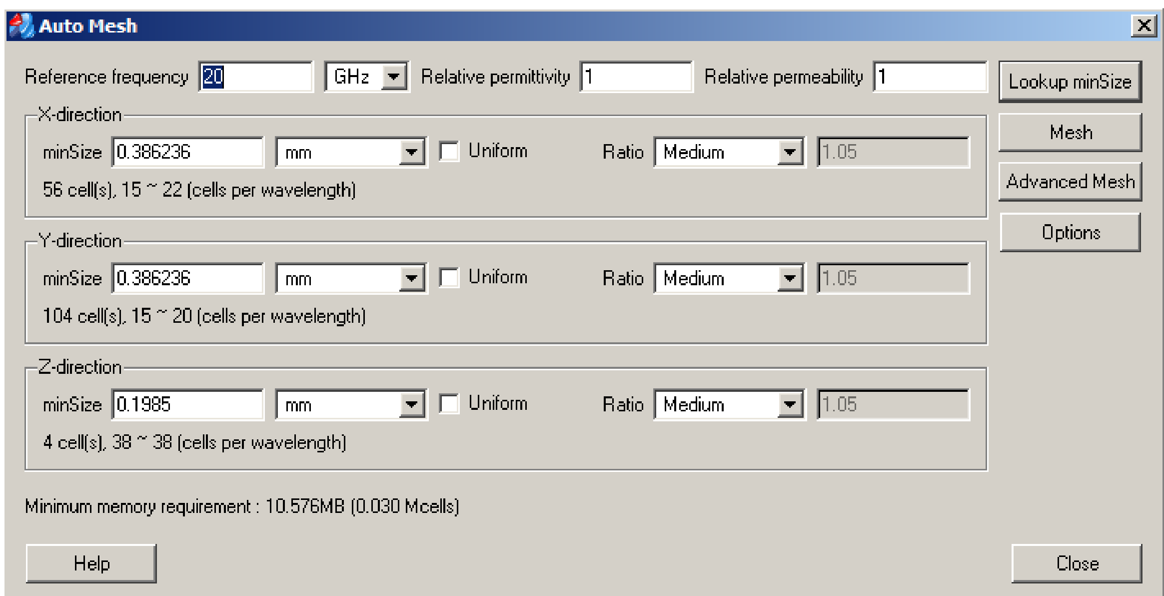




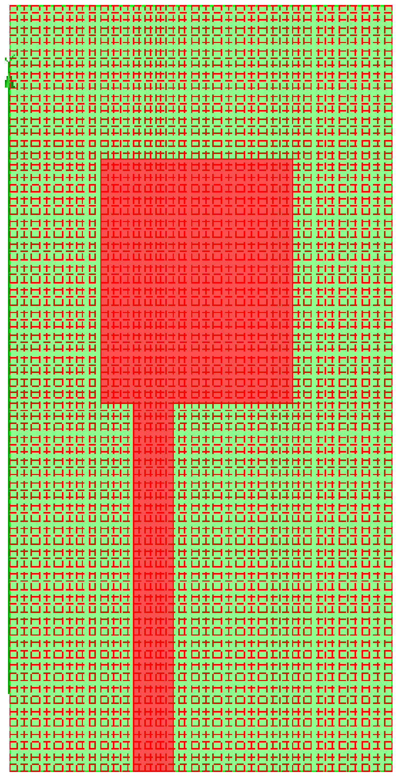
For the simple case, you can adjust the parameters in the **Options** window to find the proper mesh size and ratio, and then generate a good mesh distribution. For example, if two cells are not sufficient to describe the field variation inside the substrate in the vertical direction, you can change the number from "2" to "4" in the **Number of cells in finest structure** box.



Press the **Lookup minSize** button again, and there will be 4 cells in the vertical direction. Click on the **Close** button to confirm the mesh design.



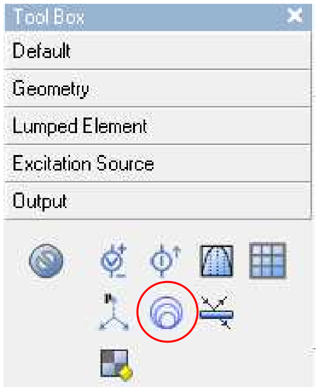
Click on the **Mesh mode** button in the toolbar to view the mesh distribution, and the **View grid** button must be pushed down. You can switch the mesh view plane by clicking on the **X-Y**, **Y-Z**, or **Z-X** button in the toolbar. If the mesh is embedded inside the model, to view the mesh distribution, you can set the model to be transparent or change the height of the display plane.

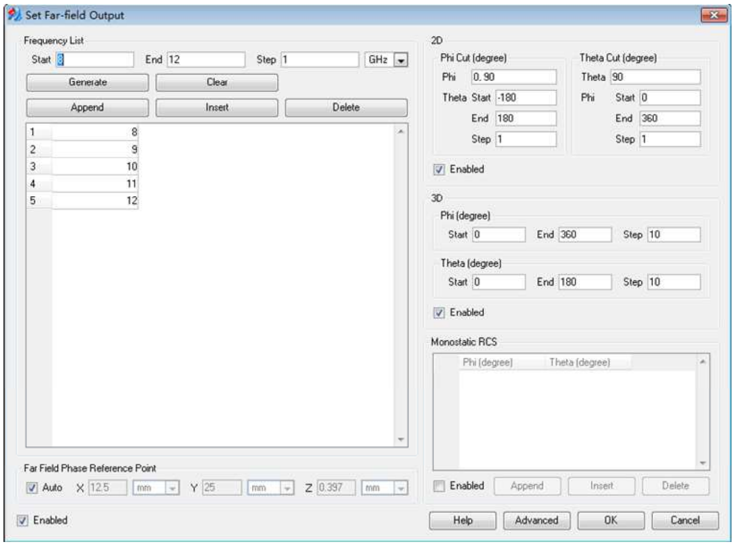


3.6 Far Field Output Setting

In this example, we cannot use a closed Huygens' surface for the far field pattern calculation since the size of substrate and ground plane is infinite. Because we are only interested in the far field in the upper hemisphere, the Huygens' surface will be degenerated into one surface at z=Zmax.

Click on the *Far-field* icon in the **Tool Box -> Output** box. Check the **Enabled** box to enable the far field output parameter settings. Specify a frequency list and enable the 2D and 3D far field options. Click on the **OK** button to confirm the far field parameter settings.



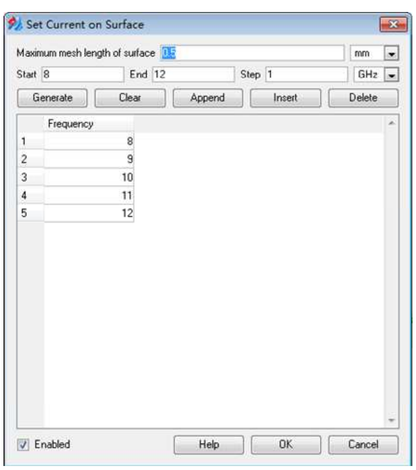


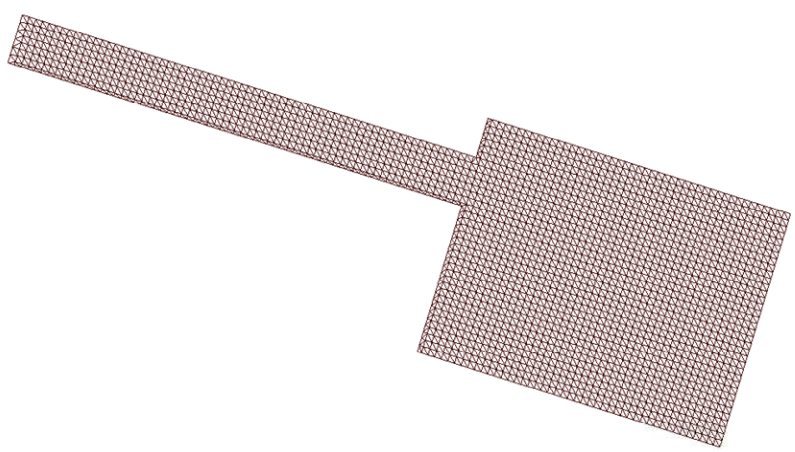
Check the “Auto” box and the plot of far field pattern will set the position of excitation source as the reference when the far field pattern is plotted together with the project model.

3.7 Surface Current Output on the Patch

GEMS allows you to calculate the 3-D surface current distribution on the arbitrary 3-D object.

Select the patch option in the **Object Browser** box, and click on then “Disabled” in the **Surf_current** option in the **Properties** box. Check the “Enabled” box at the bottom left corner in the popup window, and specify the frequency information. Input the “0.5” (mm) in the “Maximum mesh length of surface” box. Select the “Tools->Surface Mesh” option to view the mesh distribution.



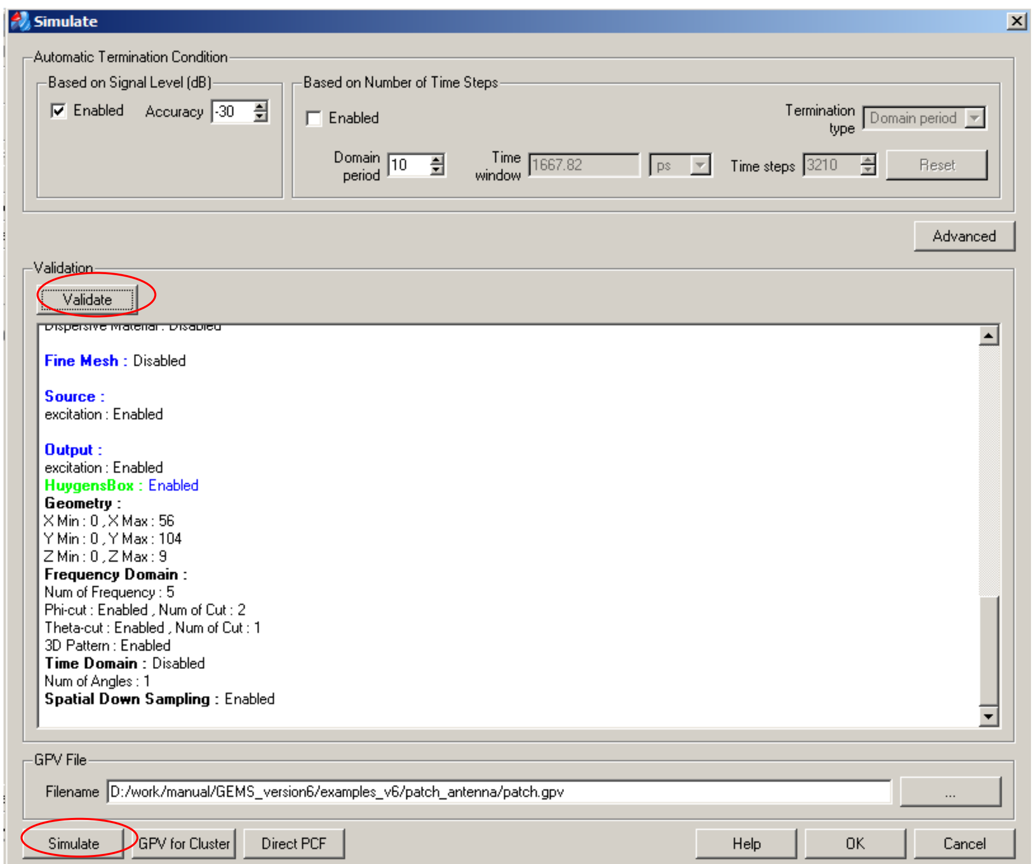


3.7 Save Project

Click on the **Save** button in the toolbar to save the project. The saved project has a default extension name “gpj”. It includes all the project information, and it can be loaded to the GEMS Designer later.

3.8 Generate Simulation File

Click on the **Precalculate** button in the toolbar to generate the simulation file.



Check the ***Enabled*** *box and select a convergence criterion in the* ***Based on Signal level(dB)*** *box. GEMS simulation stops when the convergence criterion is achieved.*

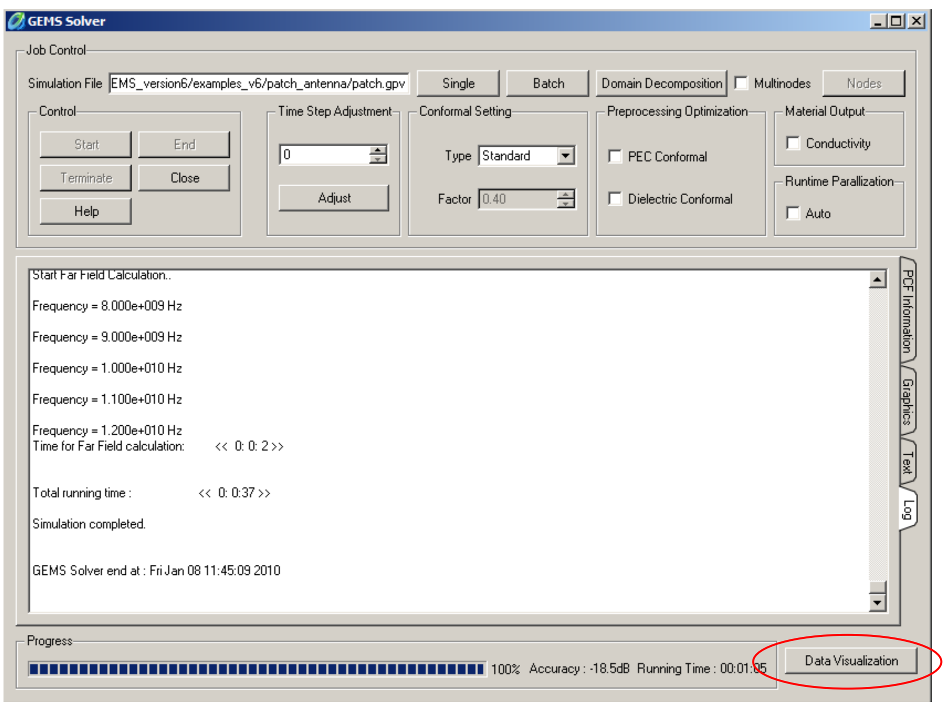
*Check the **Enabled** box in the **Based on Number of time steps** box, and check one of the convergence options. We can estimate the total simulation time from the time spent on the first few percent.*

*A time window function, Hamming window, is an optional in the simulation. In the default case, the **Automatic window function** box is checked and the Hamming window function is automatically added to the time domain signature during the simulation.*

Click on the **Validate** button to validate the project settings. If there is no message in red, click on the **Simulate** button to start the project simulation.

3.9 Simulate Project

Click on the **Simulate** button in the **Simulate** window to open the **GEMS Solver** window. Click on the **Start** button to start the simulation. The simulation status will also be displayed in the progress bar. The simulation information is summarized and stored in the **GEMSSummary.txt** file. Click on the **Data Visualization** button to open the display window.

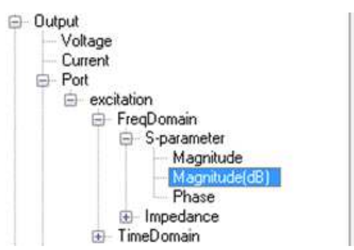


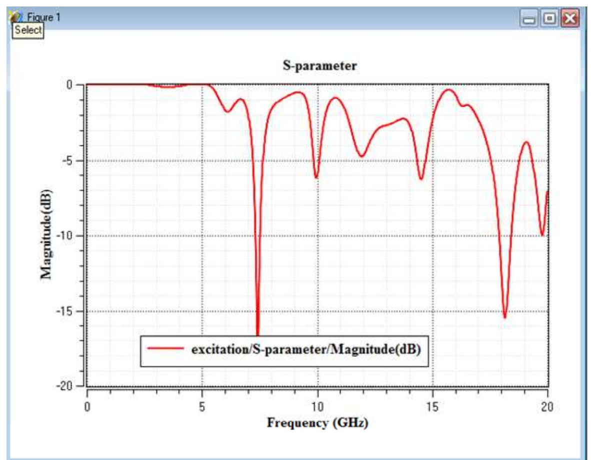
3.10 Result Visualization

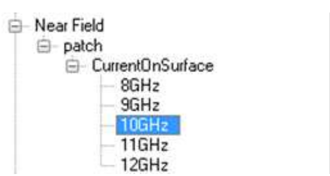
Click on the **Data Visualization** button to open the *GEMS display* window. The direct simulation results are listed inside the result tree. Select an option in the result tree, and then double-click on it to view the result. You can also select the option and select the **Export** option in the **File** menu to export the result into a text file.

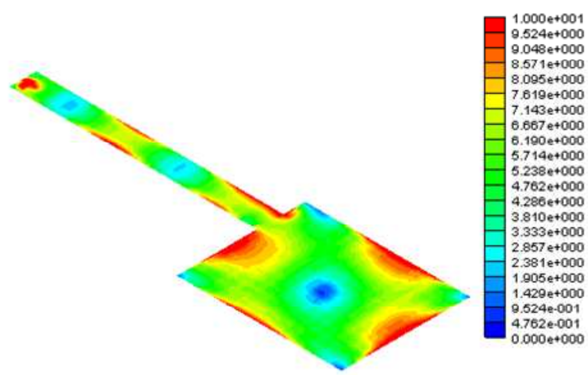
The far field value is normalized the port voltage, namely, the far field is the value generated by 1V voltage at the one meter away from the antenna.

For example, click on the “Magnetude(dB)” option to view the return loss. Click on the “10GHz” to view the surface current distribution. The color display level of the surface current can be adjusted by selecting the “Result->Color control->Color bar” option, check the “Fixed range” box and then adjust the display range.

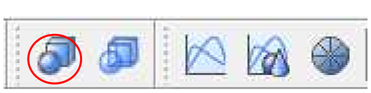


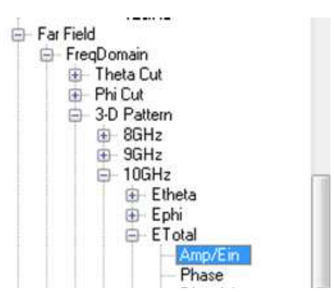


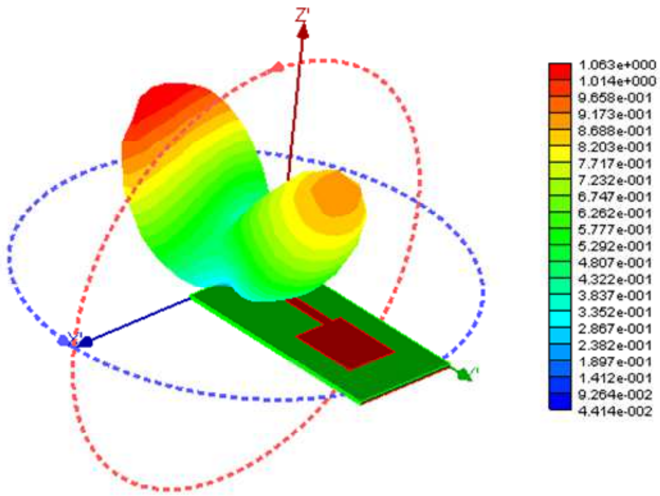




Click the “View model” button in the toolbar, and select one far field option. Click the Play button in the toolbar to view the far field pattern with the project model.







Z'
X'
Y'
1.063e+000
1.014e+000
9.658e-001
9.173e-001
8.688e-001
8.203e-001
7.717e-001
7.232e-001
6.747e-001
6.262e-001
5.777e-001
5.292e-001
4.807e-001
4.322e-001
3.837e-001
3.352e-001
2.867e-001
2.382e-001
1.897e-001
1.412e-001
9.264e-002
4.414e-002

4

Example 4. Lowpass Filter

Description: A two-port microwave circuit with a finite ground plane and substrate is excited by using the lumped ports. The output parameter is S-parameter matrix.

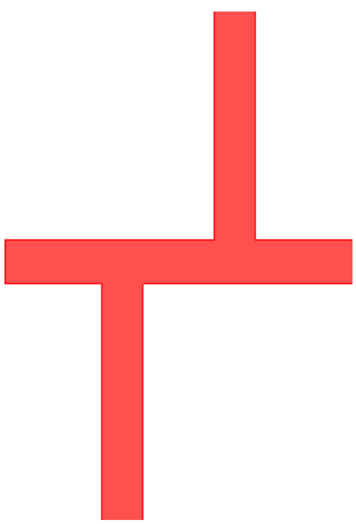
Keywords: Microwave circuit, S-parameter matrix, lumped ports, SAT file, and port scan.

4.1 Filter Configuration

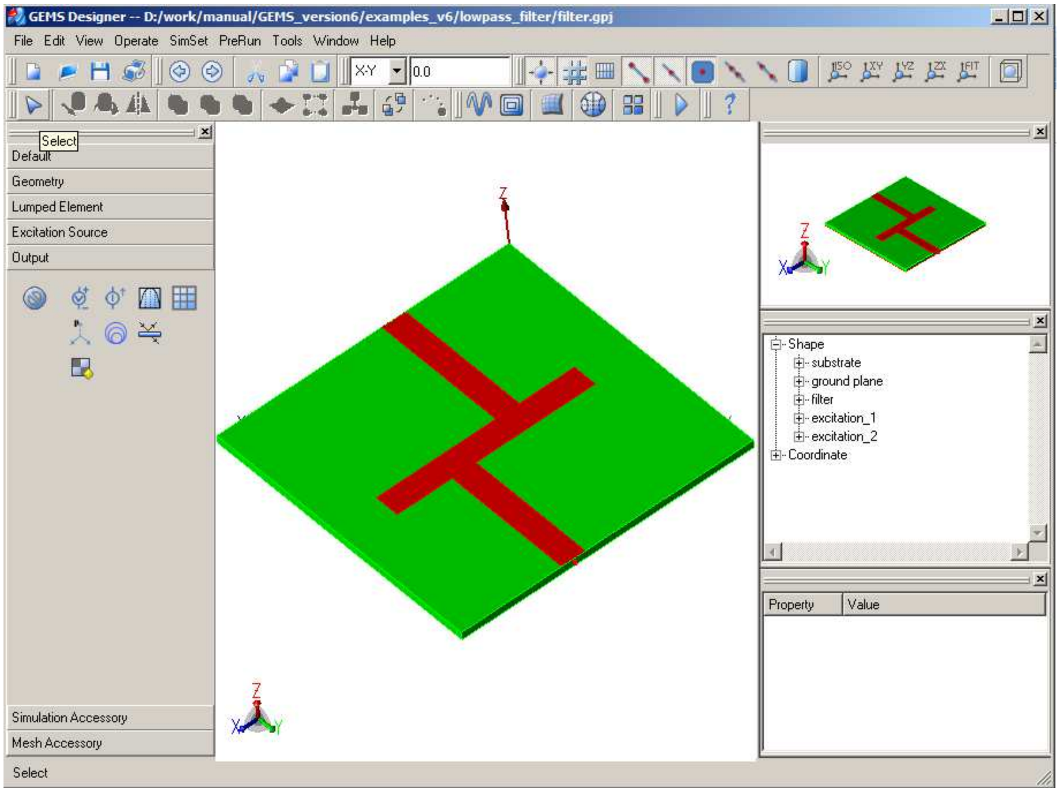
If the substrate and ground plane are finite in the horizontal directions, and the two feed lines are stop at the edge of the substrate (open port), the lumped port (with a match load) should be selected to excite the two ports. In this example, we consider that the structure is finite, therefore, the lumped port is used to excite the feed lines.

Since the two ports are excited independently, namely, when one port is excited, another port is just an output port. GEMS allows you to get the S-parameter matrix in the single simulation.

Suppose the filter is designed and saved in the SAT format. We can import it directly to GEMS interface and then assign it as the PEC material.



The low pass filter configuration is shown in the figure below:



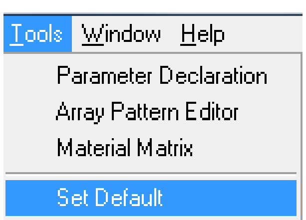
4.2 Create Filter Model

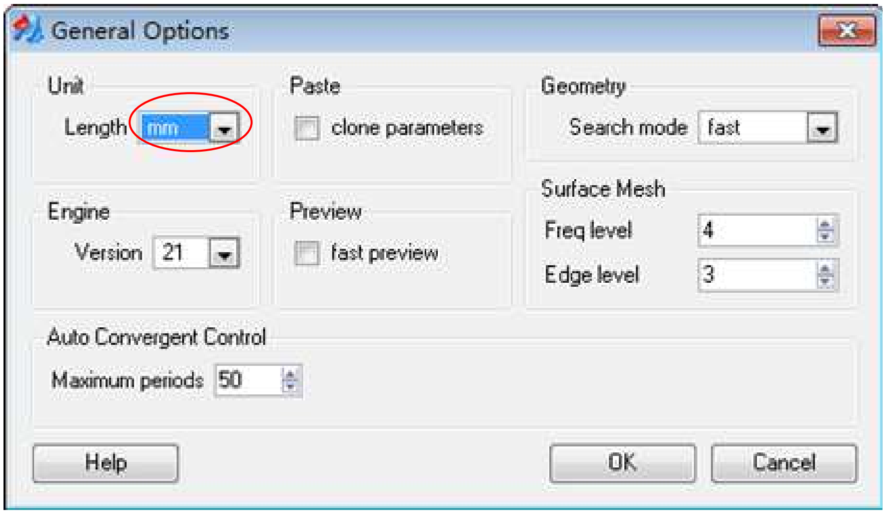
Follow the steps below to create the low pass filter model:

(1) Open the GEMS designer

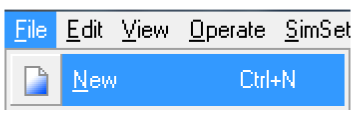
(2) Specify the project unit, which cannot be changed during the project modeling though you can input a variable in any units; however, the default unit can be only specified once at the beginning.

Select the **Tools->Set Default** option, and then select "mm" in the **Unit->Length** box as the project unit, click on the **OK** button to close the window.



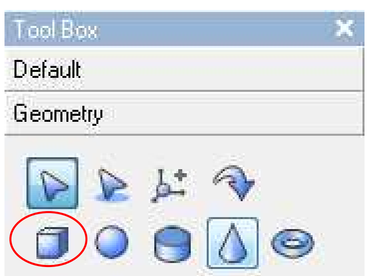


(3) Click on the **New** button in the toolbar or select the **New** option in the **File** menu.



(4) Use the default drawing plane (X-Y) and height (0.0mm)

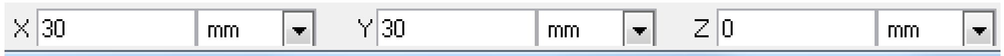
Click on the **Cuboid** icon in the **Tool Box->Geometry** box.



There are two ways to draw a substrate model:

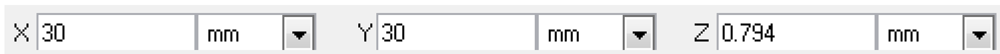
(i) Use the dialog box

- Move the mouse icon to the figure region, press the left mouse button at the start point (0mm, 0mm, 0mm).
- Move the mouse icon to the **X** box at the bottom and type 30, and then move the mouse icon to the **Y** box and type "30", and then press the **Enter** key.

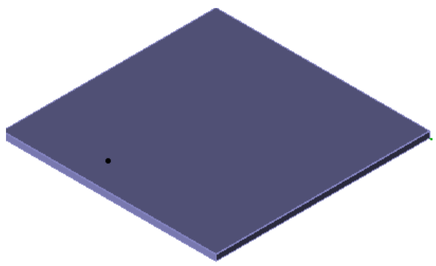


A rectangle with 30mmx30mm size will be drawn in the figure region.

- Move the mouse icon to the **Z** box at the bottom and type 0.794, and then press the **Enter** key.

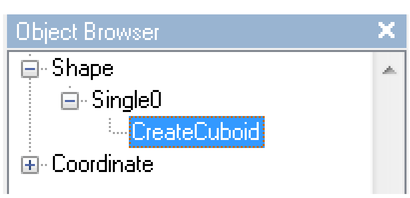


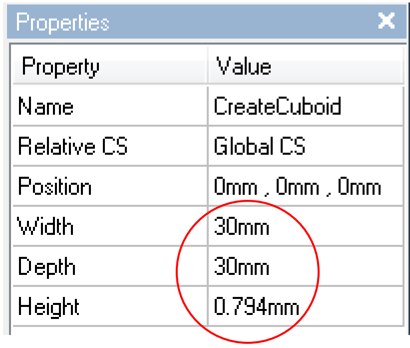
A substrate box (30mm x 30mm x 0.794 mm) is drawn in the figure region.



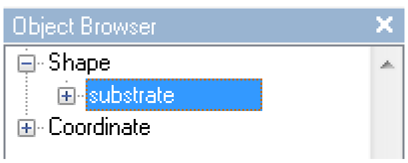
(ii) Use mouse only

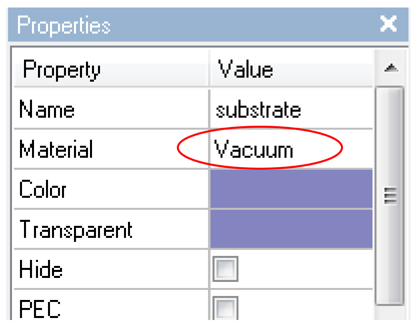
- Move the mouse icon to the figure region, press the left mouse button at the start point (0mm, 0mm, 0mm).
- Move the mouse icon in the X-Y plane to select the size of rectangle. To zoom out the figure region you need to roll the mouse wheel (middle button). You may not get the exact position though mouse. Draw a rectangle which is closer to the actual dimensions. Press the left mouse button.
- Move the mouse icon in the vertical direction, and select the height of the substrate, and get a box which is closer to the actual dimensions. Press the left mouse button to finish the box drawing.
- Select the **CreateCuboid** option under the substrate entrance, and adjust the dimensions to the exact numbers in the **Properties** box.

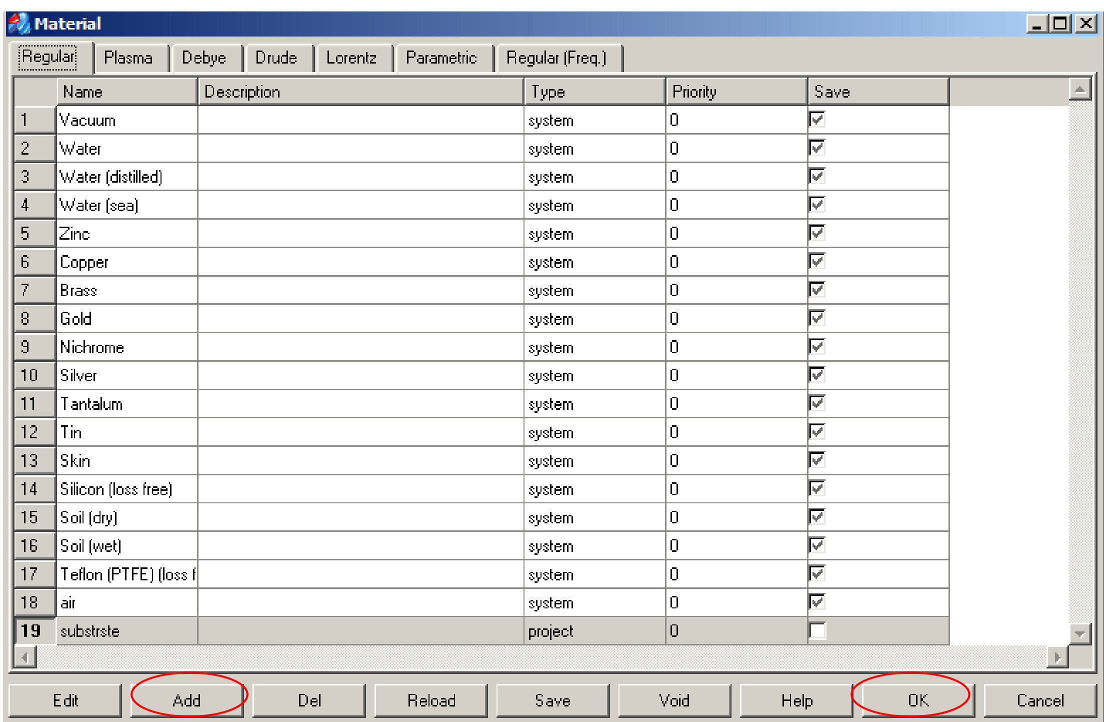


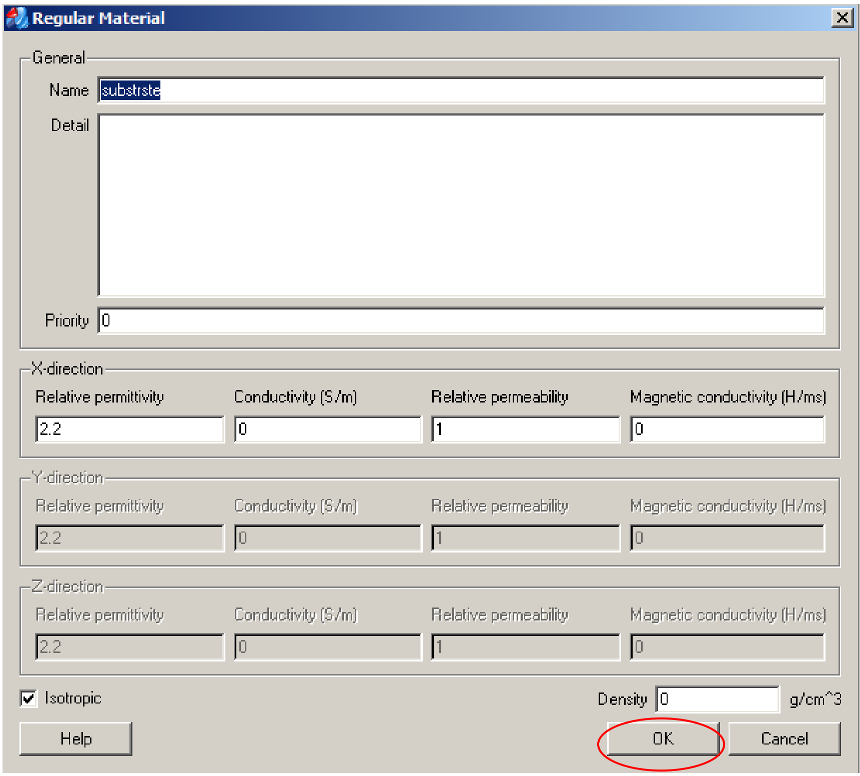


(5) Select the entrance that corresponds to the substrate in the **Object Browser** box, and change its name to “substrate”, change its color to the desired one, and then click on the **Material** box.









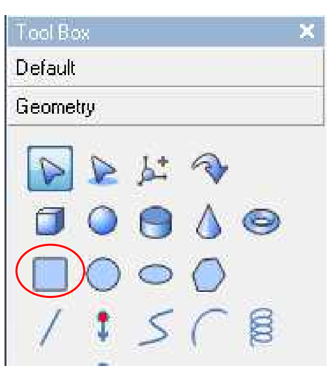
(6) Draw the ground plane

There are two ways to draw a ground plane:

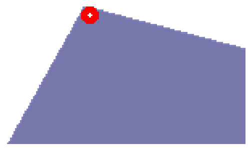
(i) Draw a ground plane

- Select the drawing plane and height

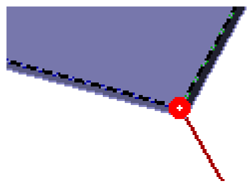
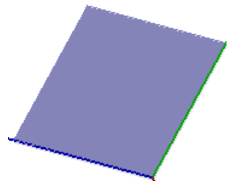
- Click on the *Rectangle* icon in the **Tool Box->Geometry** box.



- Select one corner of substrate as the start point, press the left mouse button.



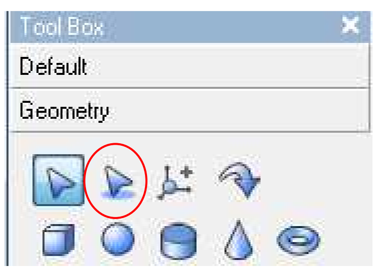
- Move the mouse icon in the X-Y plane and let mouse icon snap to other corner of the substrate in the X-Y plane. Press the left mouse button to finish the drawing.

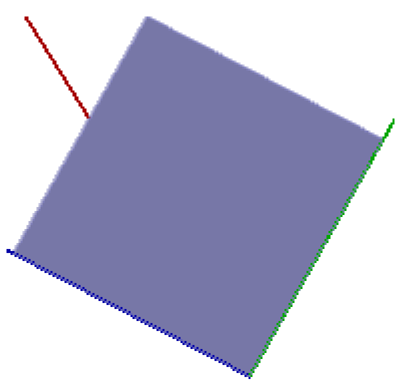
- Change its name to “ground” and check the **PEC** box in the **Properties** box.

(ii) Copy a surface from the substrate

- Click on the **Select face** icon in **the Tool Box->Geometry** box



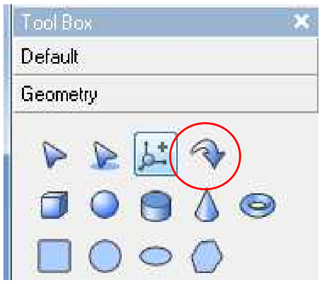
- Use the mouse to select the bottom surface of substrate.



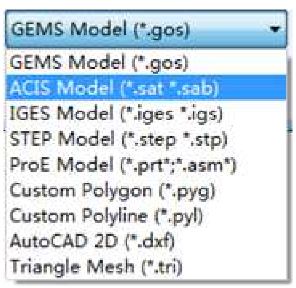
- Press "Ctrl + c" and then "ctrl + v" to get a copy of the bottom surface of the substrate.
- Change the name to "ground plane" and check the **PEC** box in the **Properties** box.

(7) Import the filter

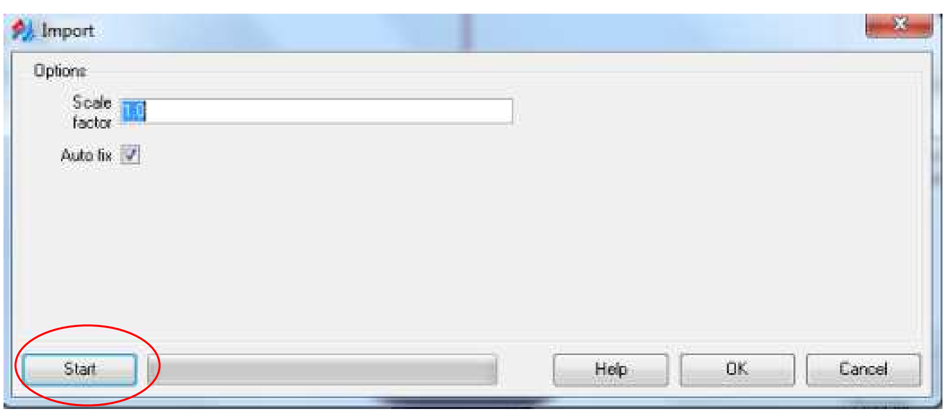
Click on the *Import model* icon in **the Tool Box->Geometry** box.

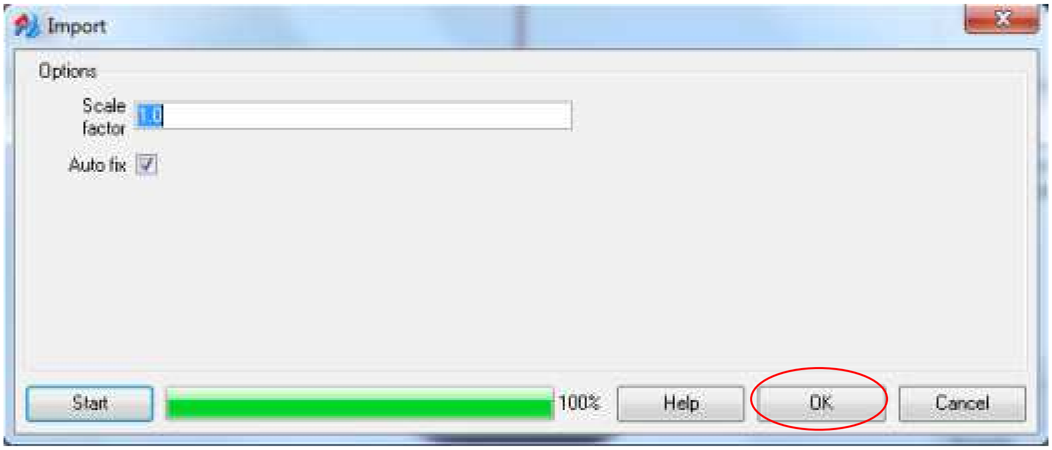


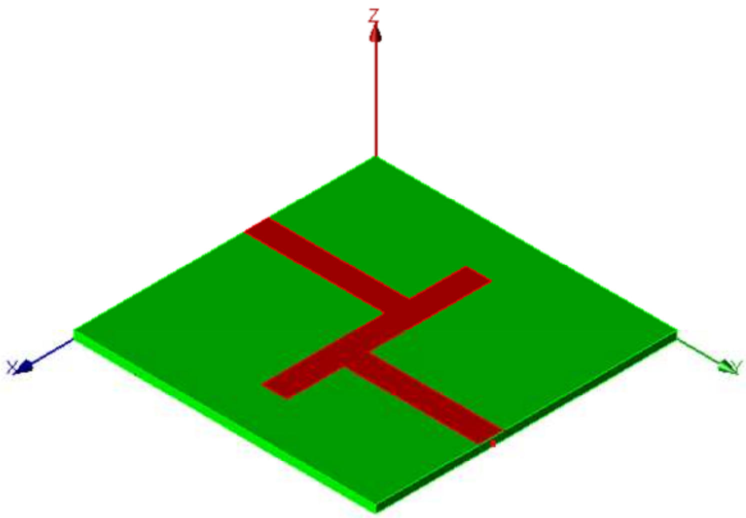
Select the file format in the format list, and search the "*.sat" file.



Click on the **Start** button to start the processing. You can adjust the model size through the scale factor. The **Auto fix** box is designed to fix some minor problems in the model. For the complex model, checking the **Auto fix** box will take extra time. If the model is correct, you can uncheck this option.





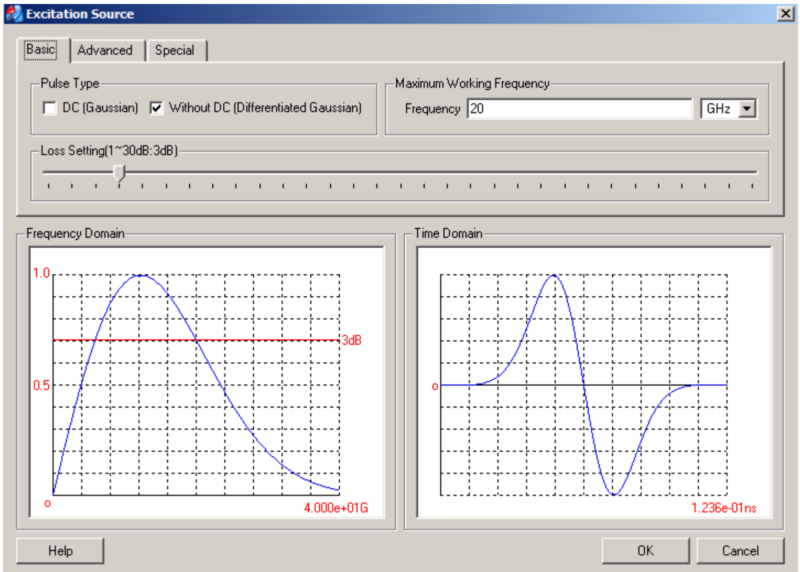


You can adjust the location and orientation of the filter in the domain through the moving and rotation options.

Select the filter option in the **Object Browser** box, and then change the name to "filter", change its color to red, and check the **PEC** box in the **Properties** box.

4.3 Set Excitation Pulse

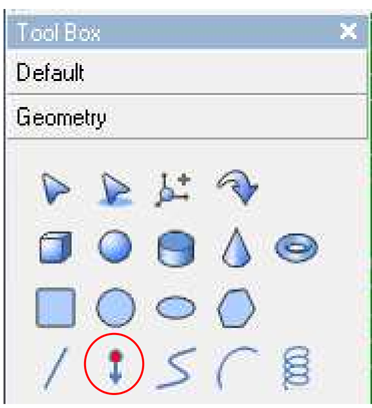
Click on the **Excitation source** button in the toolbar. Since the maximum frequency of interest is 20GHz, so we type "20" in the **Frequency** box (unit is GHz). Click on the **OK** button to confirm the excitation pulse setting.

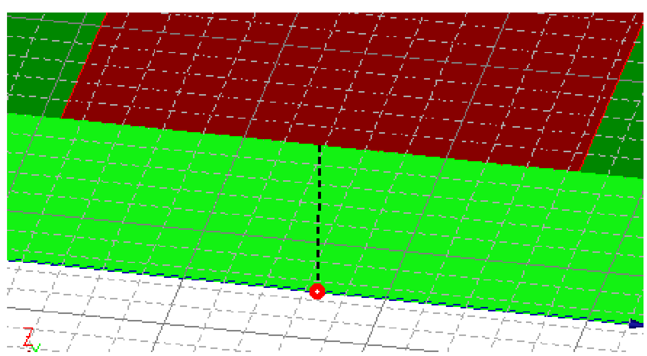


4.4 Create Lumped Port Excitation

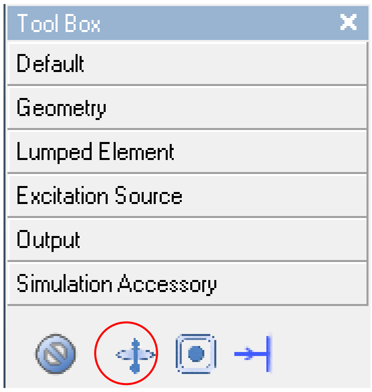
Click on the **Plumb line** icon in the **Tool Box->Geometry** box, and move the mouse icon to the center of the stripline, press the left mouse button, move the mouse icon along the vertical direction, stop when it meets the bottom of the substrate. This line will be defined as the wave port.

Select the **Plumb line** option in the **Object Browser** box and change the name of the plumb line to be "excitation_1" in the **Properties** box.



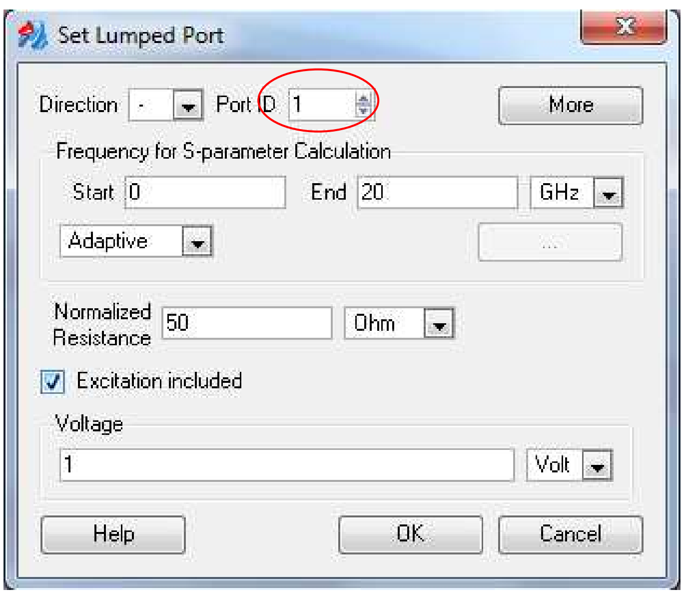


Select the "excitation_1" option in the **Object Browser** box and click on the **Lumped port** icon in the **Tool Box->Simulation accessory** box.



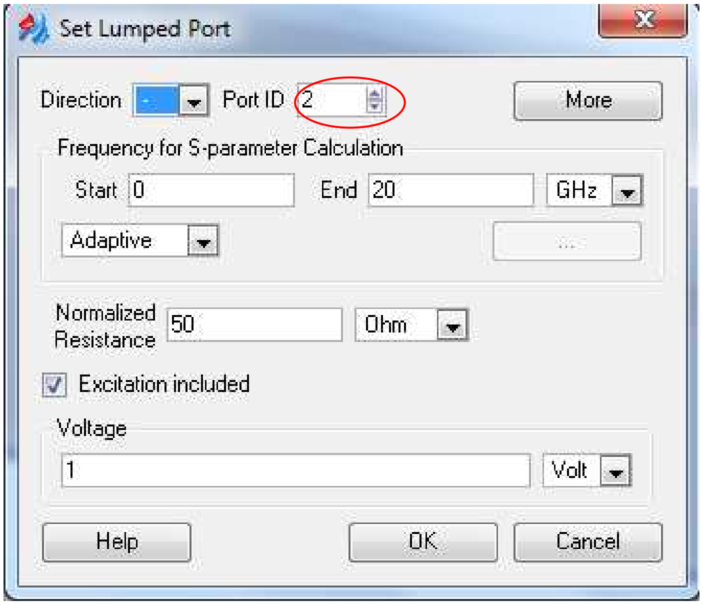
The black end should touch the signal line (feed line), which can be adjusted by the sign in the **Direction** box.

The port ID is selected to be "1" for the excitation_1. The "Adaptive" option allows GEMS to decide the frequency resolution in the S-parameter and impedance outputs. The value in the **Normalized Resistance** box is a reference for the return loss calculation.



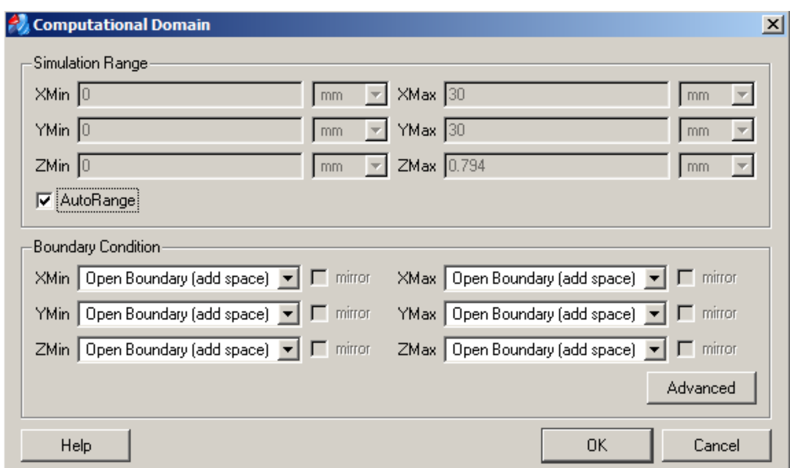
Check the **Excitation included** box to ensure this is an excitation port.

Follow the same procedure, select the "excitation_2" option in the **Object Browser** box and click on the **Lumped port** icon in the **Tool Box->Simulation accessory** box. The port ID is selected to be "2" for the "excitation_2" port.



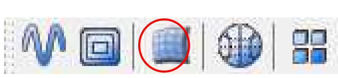
4.5 Domain and Boundary Settings

Click on the **Set boundary** button in the toolbar. The “Open Boundary (add space)” is used to truncate all the six walls.

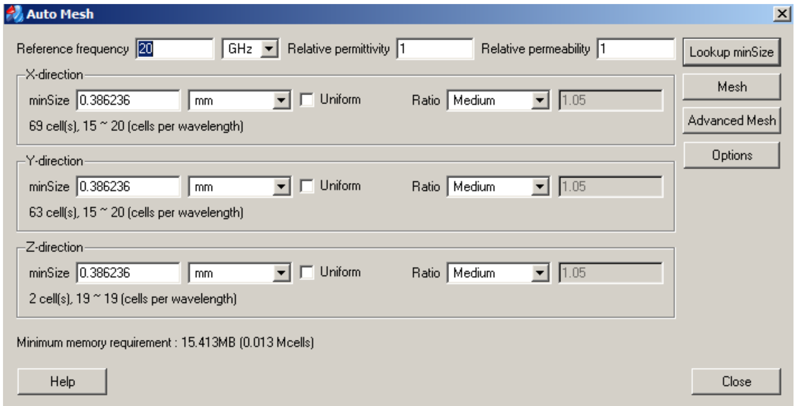


4.6 Mesh Generation

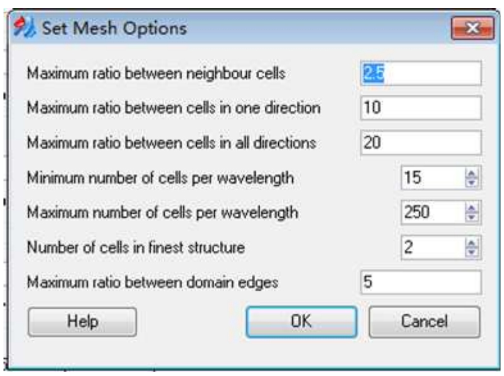
Click on the **Auto mesh** button in the toolbar.



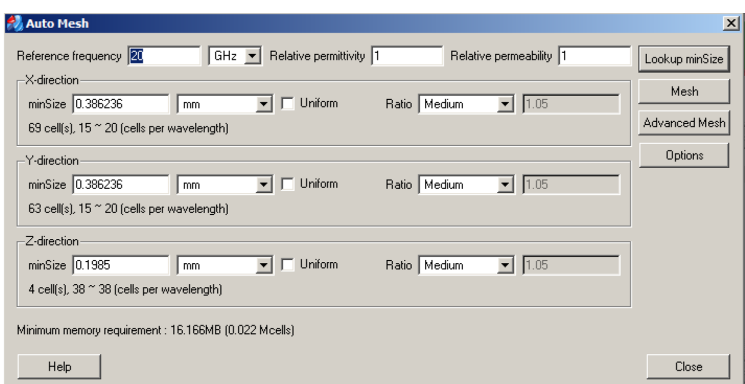
Click on the **Lookup minSize** button to find the minimum cell size in each direction. And then click on the **Mesh** button to generate the adaptive mesh.



For the simple case, you can adjust the parameters in the **Options** window to search and find the proper mesh size and ratio, and then generate a good mesh distribution. For example, if two cells are not sufficient to describe the field variation inside the substrate in the vertical direction, you can change the number from "2" to "4" in the **Number of cells in finest structure** box.



Press the **Lookup minSize** button again, there will be 4 cells in the vertical direction. Click on the **Close** button to confirm the mesh design.

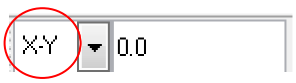


Click on the **Mesh mode** button in the toolbar to view the mesh distribution, and the **View grid** button must be pushed down. You can switch the mesh view plane by clicking on the **X-Y**, **Y-Z**, or **Z-X** button. If the mesh is embedded inside the model, to view the mesh distribution, you can set the model to be transparent or change the height of the drawing plane.

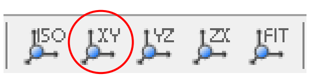
4.7 Mesh View

Push the **View grid** button down in the toolbar if it is up. Click on the **Mesh mode** button in the toolbar.

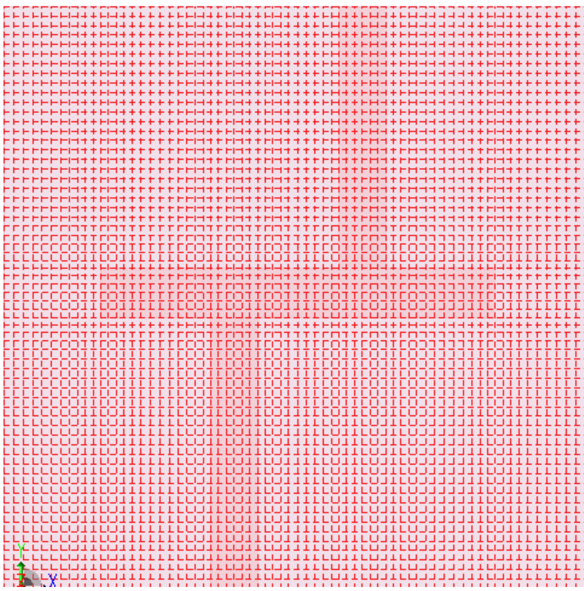
Select the display plane (say, X-Y plane). You can adjust the height if the mesh is embedded inside the model.



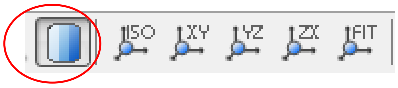
Click on the **XY** plane button in the toolbar.



The mesh distribution will be displayed in the figure region.



You can set the model to be transparent if the mesh is embedded inside the model and blocked by the objects.

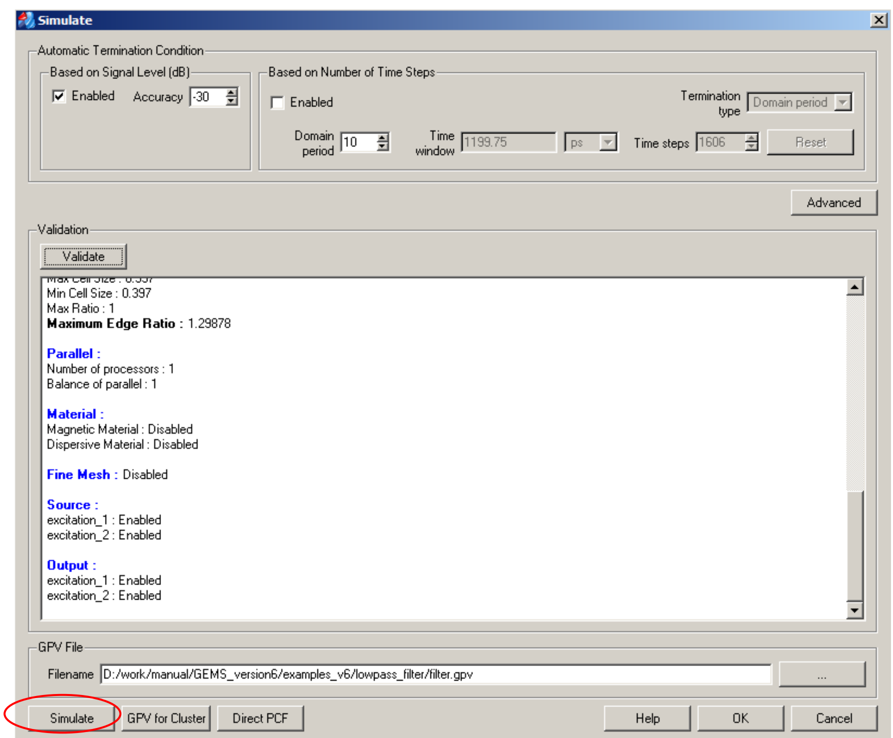


4.8 Save Project

Click on the **Save** button in the toolbar to save the project. The saved project has a default extension name “gpj”.

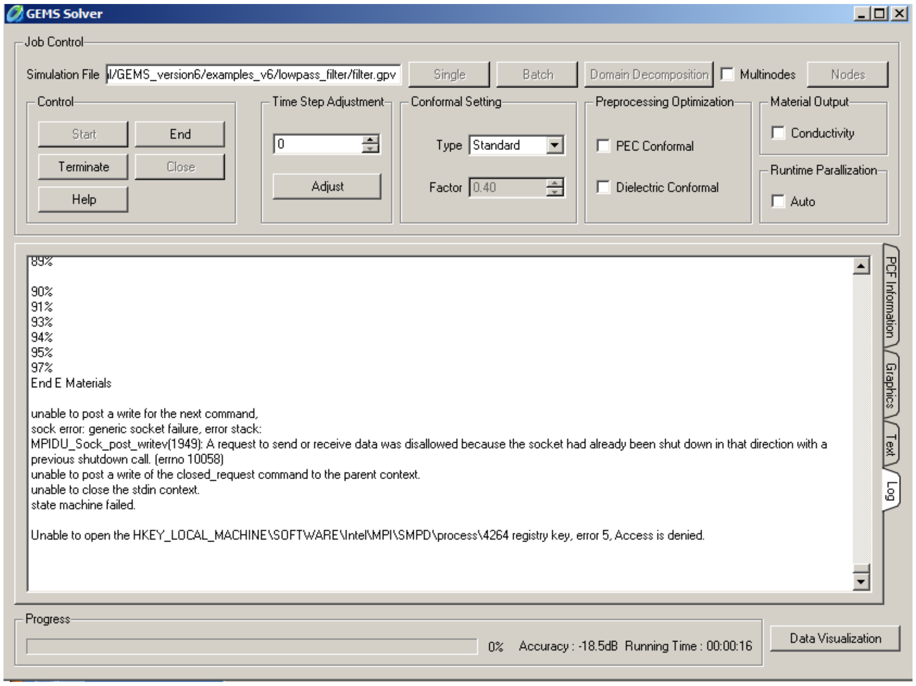
4.9 Generate Simulation File

Click on the **Precalculate** button in the toolbar to generate a simulation file with an extension name “gpv”.



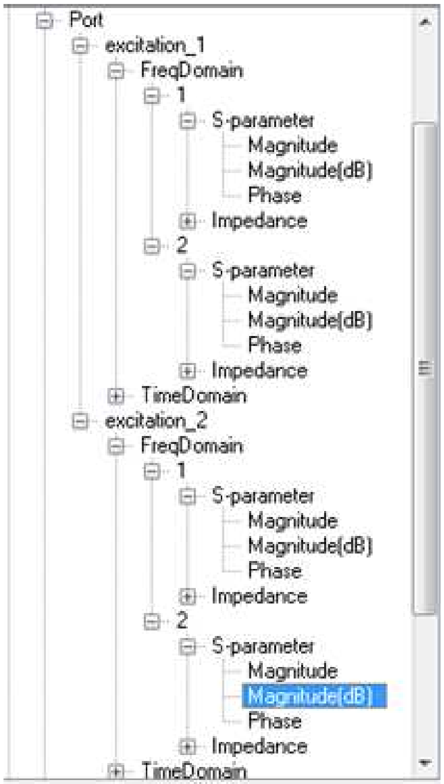
4.10 Simulate Project

Click on the **Simulate** button in the *Simulate* window to open the simulation window. Click on the **Start** button to start the project simulation.



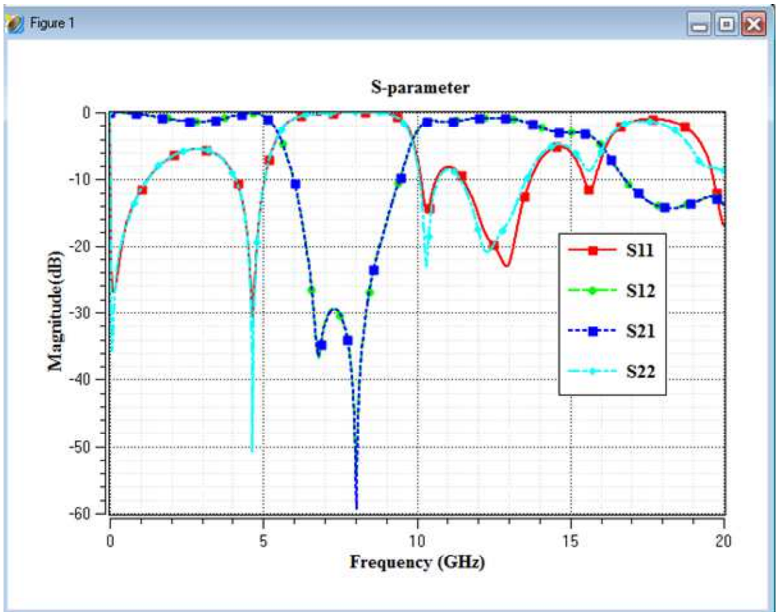
4.11 Result Visualization

Click on the **Data Visualization** button in the *GEMS Solver* window to open the *GEMS Display* window. GEMS direct results are listed in the result tree.

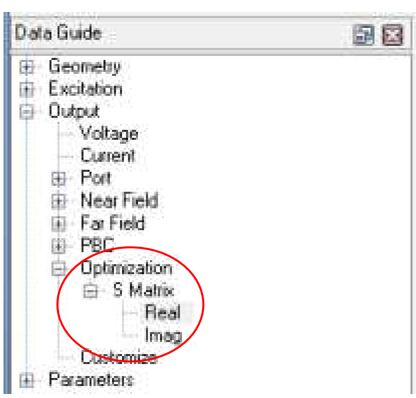


The folders "1" and "2" in the **Port->excitation_1** folder include the outputs (S_{11} and S_{12}) at the port "excitation_1" when the ports "excitation_1" and "excitation_2" are excited respectively.

The folders "1" and "2" in the **Port->excitation_2** folder include the outputs (S_{21} and S_{22}) at the port "excitation_2" when the ports "excitation_1" and "excitation_2" are excited respectively.



The real and imaginary parts of the S-parameter matrix are stored in the **Optimization->S Matrix** folder, which can be exported to a text file in the TouchStone format by selecting both of them and select the **Export** option in the **File** menu.



5

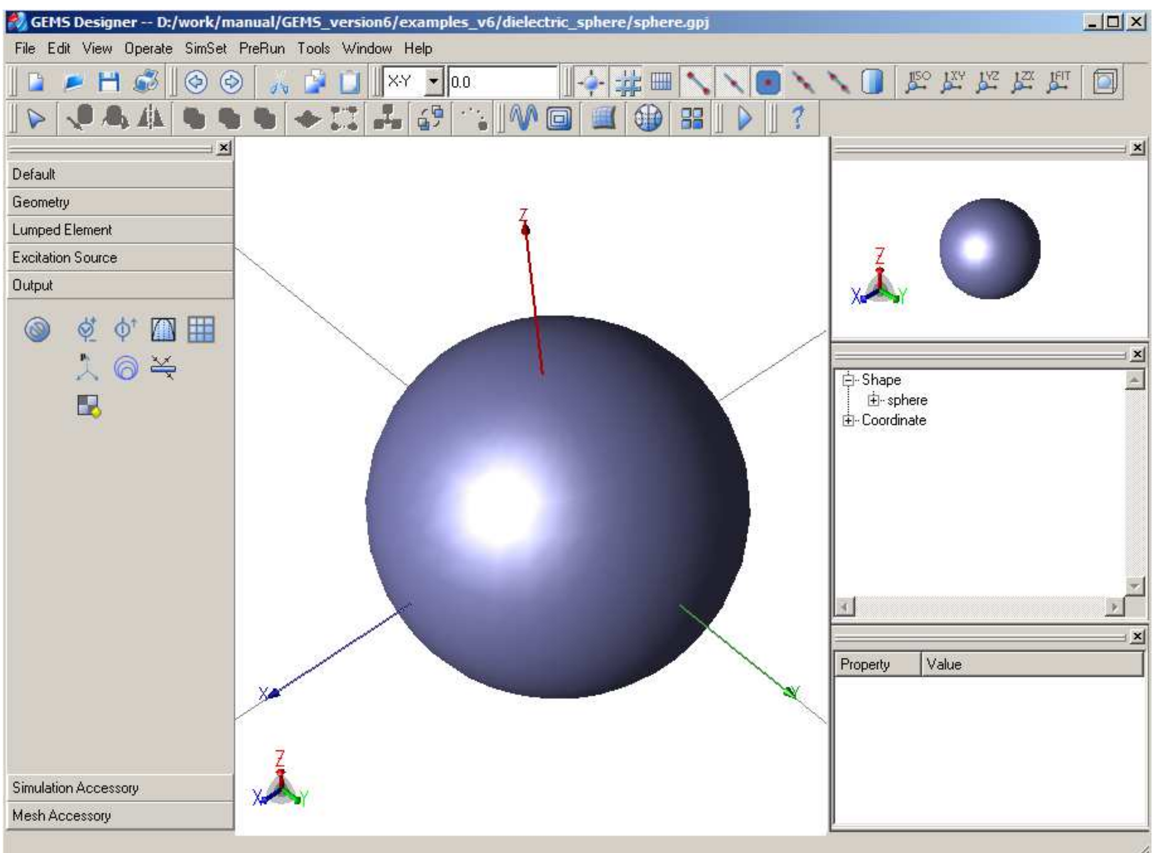
Example 5. RCS

Description: A dielectric sphere is used to demonstrate the RCS prediction.

Keywords: Bistatic RCS, dielectric sphere, and plane wave source, open boundary (add space).

5.1 Dielectric Sphere Configuration

A dielectric sphere (radius = 15mm and dielectric constant = 4) is illumined by a plane wave and the output parameter is the RCS (Radar Cross Section).



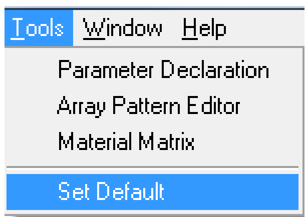
5.2 Create Dielectric Sphere Model

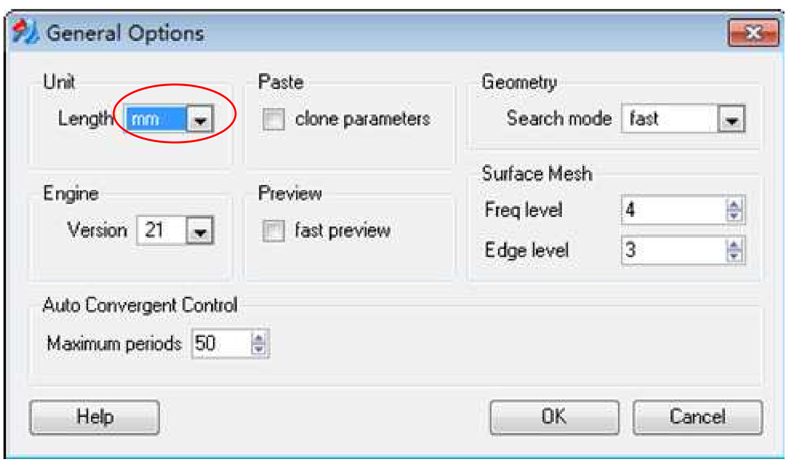
Follow the steps to generate the dielectric sphere model:

(1) Open the GEMS designer

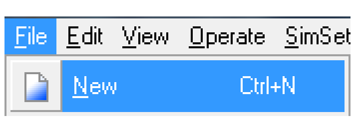
(2) Specify the project unit, which cannot be changed during the project modeling though you can input a variable in any units, however, the default unit can be only specified once at the beginning.

Select the **Tools->Set Default** option, and then select the "mm" in the **Unit->Length** box as the project unit, click on the **OK** button to close the window.



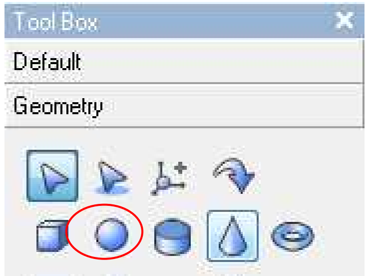


(3) Click on the **New** button in the toolbar or select the **New** option in the **File** menu.



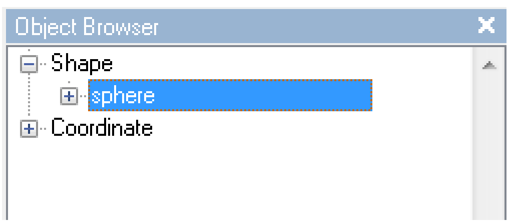
(4) Use the default drawing plane (X-Y) and height (0.0mm)

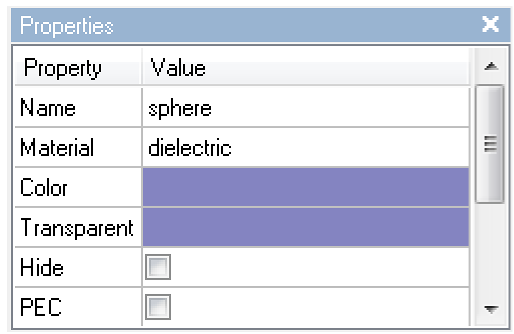
Click on the **Sphere** icon in the **Tool Box->Geometry** box.



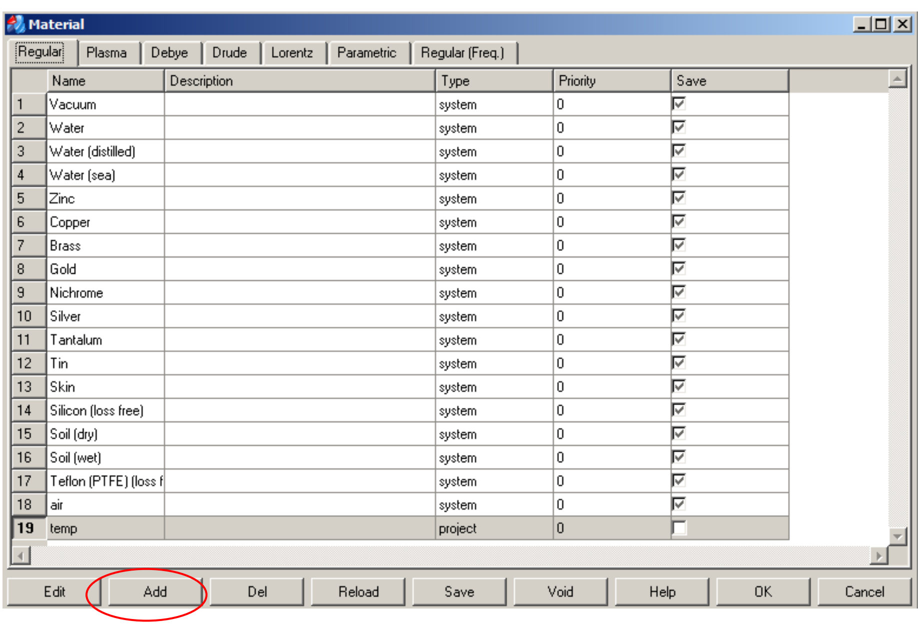
(5) Change its name and assign its material

Select the sphere option in the **Object Browser** box and change its name to "sphere" in the **Properties** box.

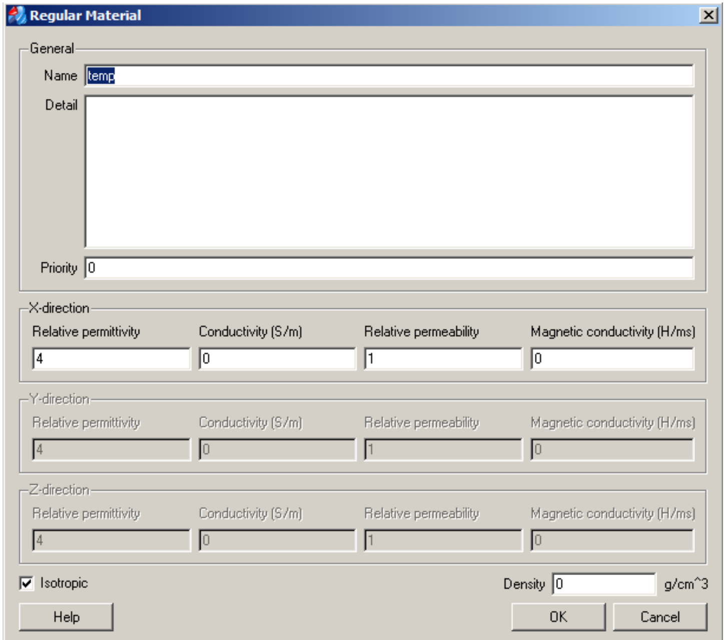




Click on the **Material** box in the **Properties** box and then click on the **Add** button to add a new material.

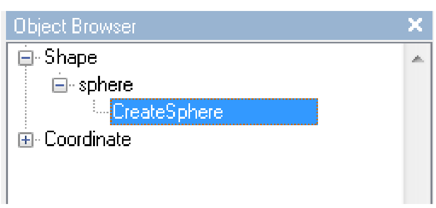


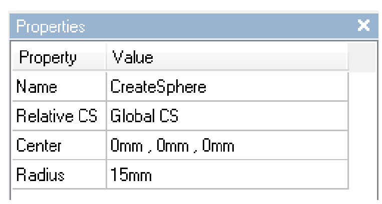
Specify its dielectric constant.



(6) Change its dimensions

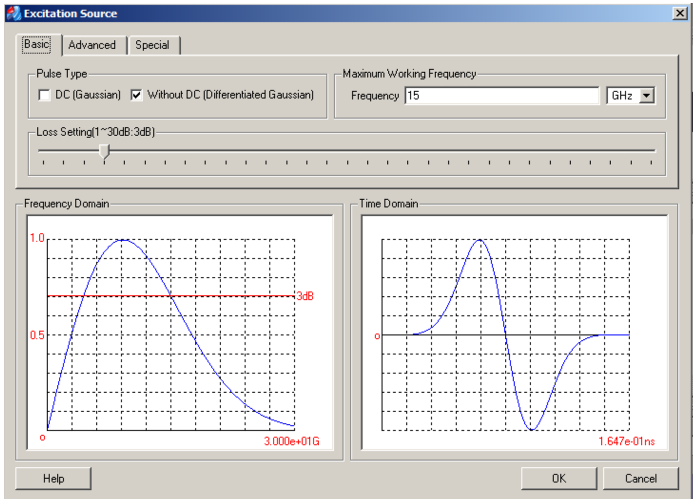
Select the “CreateSphere” option under the **sphere** entrance in the **Object Browser** box and change its dimensions in the **Properties** box.





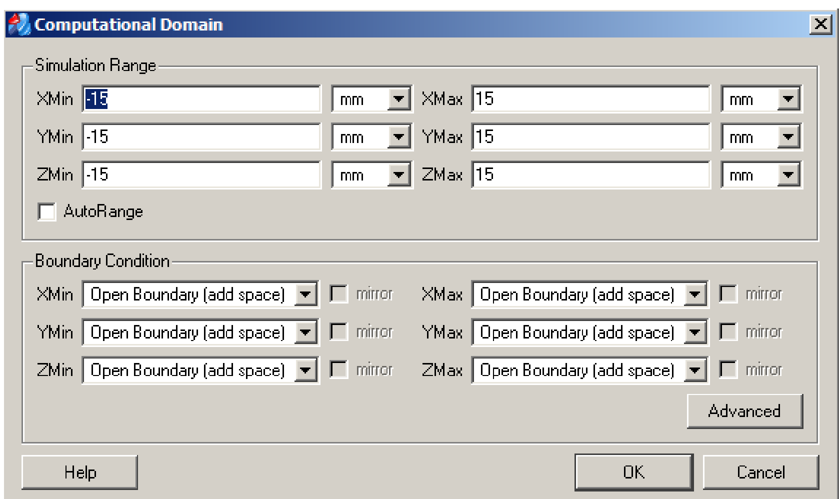
5.3 Select Excitation Pulse

Click on the **Excitation source** button in the toolbar. Since the maximum frequency of interest is 10GHz, so we type “10” in the **Frequency** box (unit is GHz). Click on the **OK** button to confirm the excitation pulse setting.



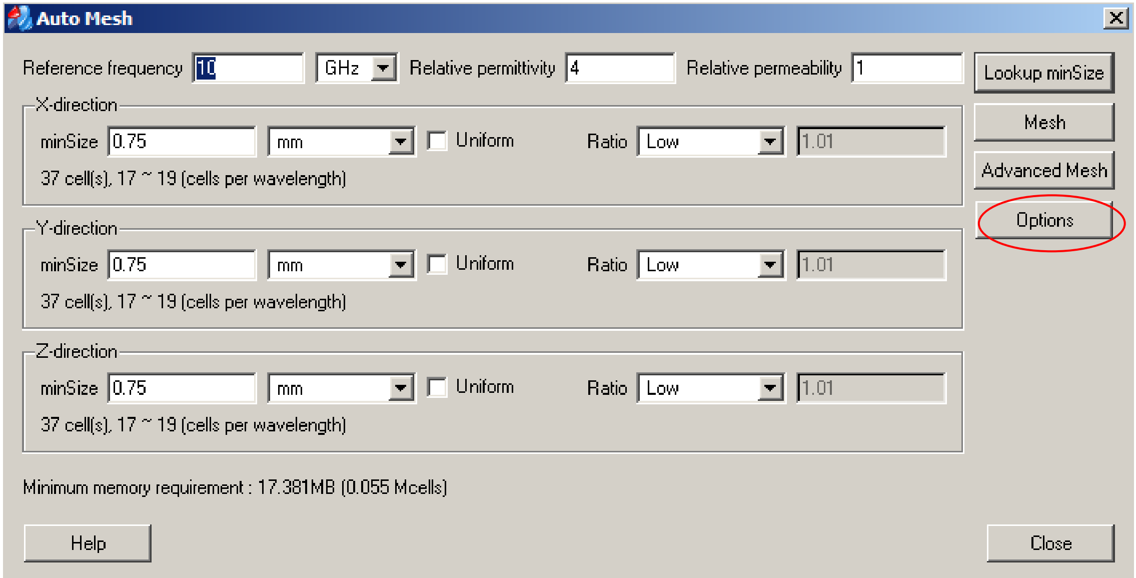
5.4 Set Domain and Boundary Condition

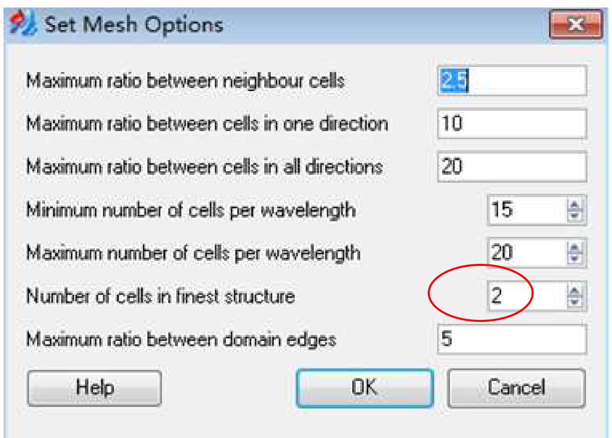
This is a scattering problem, and the "Open Boundary (add space)" is used to truncate the domain.



5.5 Design Mesh Distribution

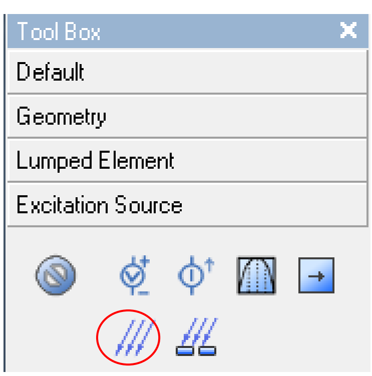
There is no fine structure inside the domain; hence, we need to adjust the number to "20" in the **maximum number of cells per wavelength** box. Click on the **Lookup minSize** button to get the default cell size, and then click on the **Mesh** button to generate the mesh distribution.





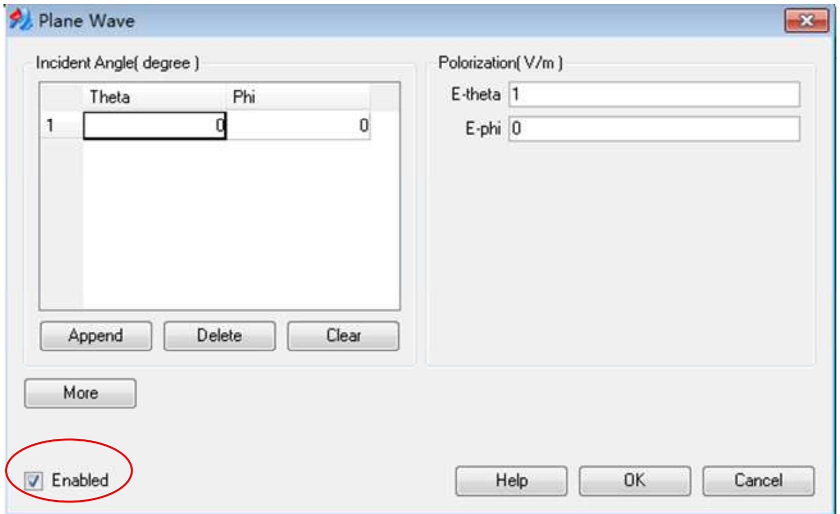
5.6 Set Plane Wave Source

Both the plane wave source and far field pattern settings require the mesh distribution; hence, we need to design the mesh before the plane wave source and far field pattern settings. Click on the **Plane wave** icon in the **Tool Box->Excitation Source** box.



Check the **Enabled** box to set the plane wave parameters.

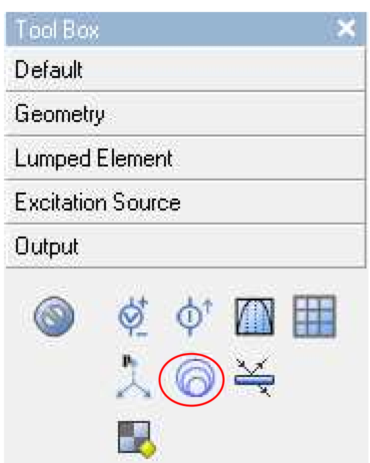
If the plane wave comes from the +z-direction with the *Ex* polarization, the settings for the plane wave are shown in the **Plane Wave** window below.



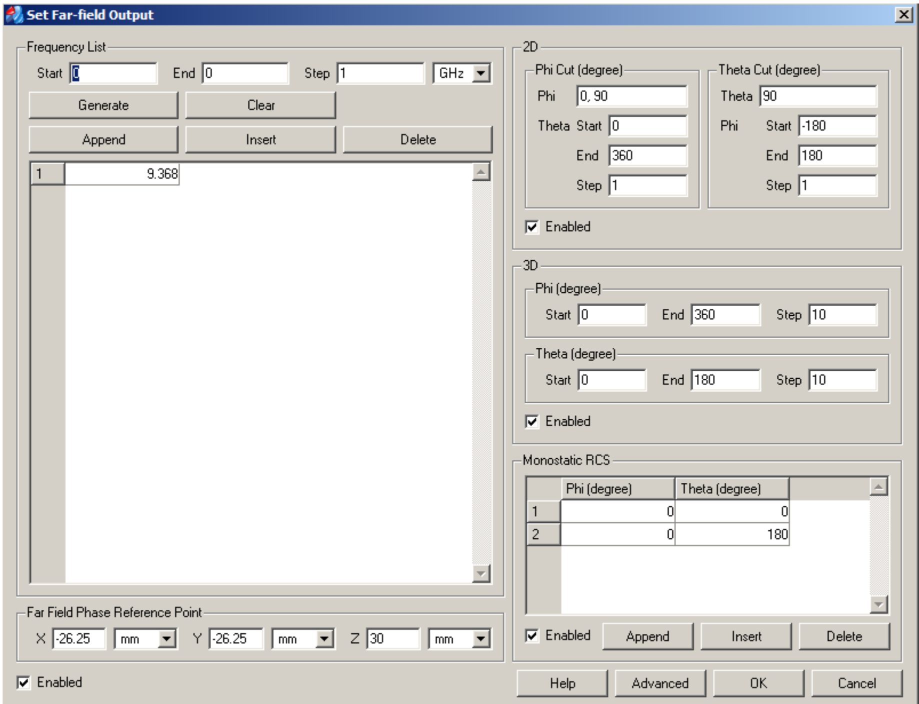
GEMS allows you to scan the incident angle by inputting multiple incident angles in the Incident Angle list. GEMS does not allow you to add the local source and plane wave source in one project.

5.7 Set Far Field Outputs

Click on the **Far-field** icon in the **Tool Box->Output** box.



Check the **Enabled** box to set the far field parameters. You can insert a frequency “9.368” in the frequency list for the 2D and 3D RCS patterns.

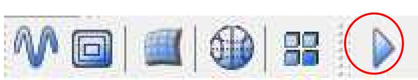


5.8 Save Project

Click on the **Save** button in the toolbar to save the project.

5.9 Generate Simulation File

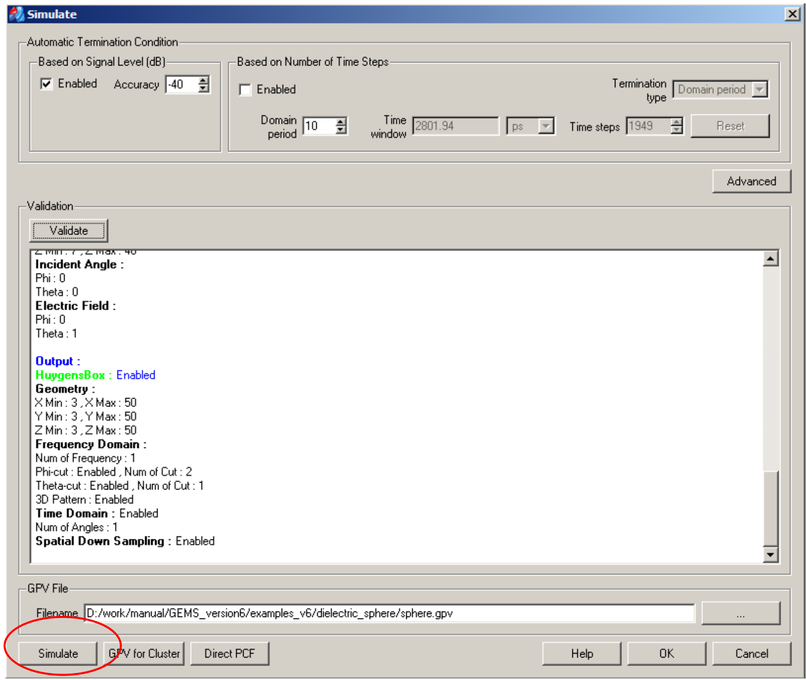
Click on the **PreCalculate** button in the toolbar to generate the simulation file (*.gpv).



Sometimes, "-30dB" convergence criterion may be not sufficient to get the accurate results, we need to use -40dB as the convergence criterion.

*If you do not need to apply the window function on the time signature, you can uncheck the option in the **Advanced Options** window.*

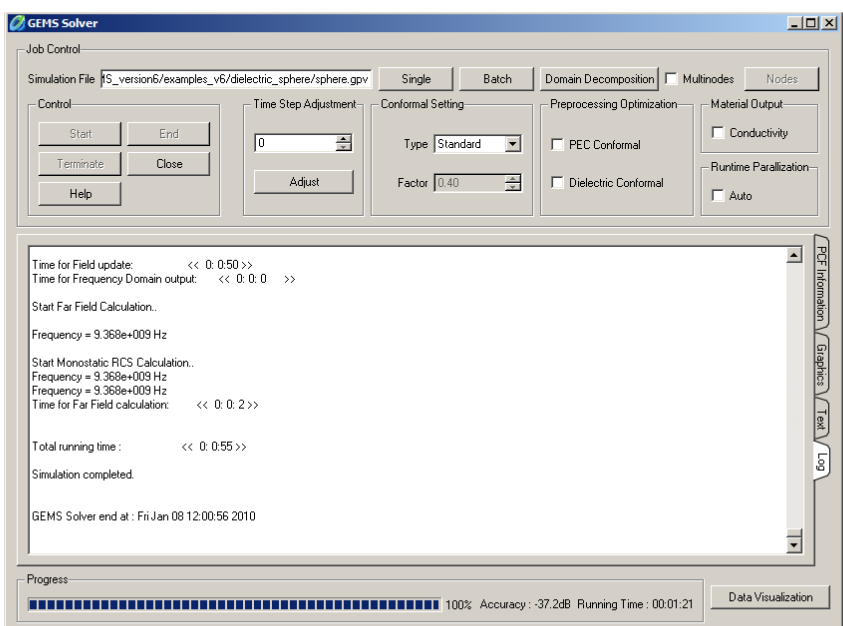
Click on the **Validate** button to validate the project settings. If there is any message in the red color, you need to check the project setting and make sure it is correct.



Click on the **Simulate** button to open the *GEMS Solver* window to start the simulation.

5.10 Simulate Project

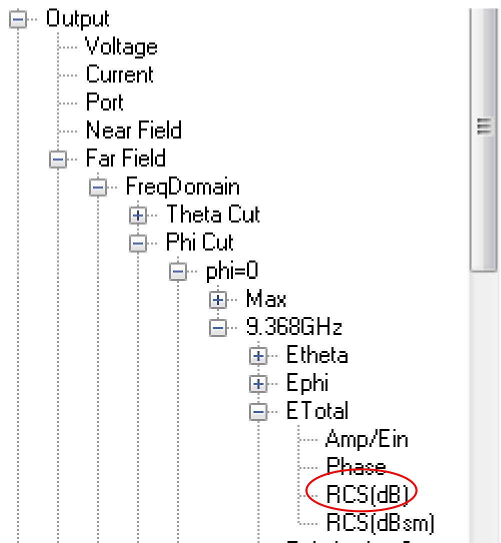
Click on the ***Simulate*** *button in the Simulate window. For a large dielectric/PEC object, you can check the* ***Preprocessing Optimization-> Dielectric/PEC conformal*** *box to speed up the preprocessing if the preprocessing is slow; however, the memory usage is more than the usual.*



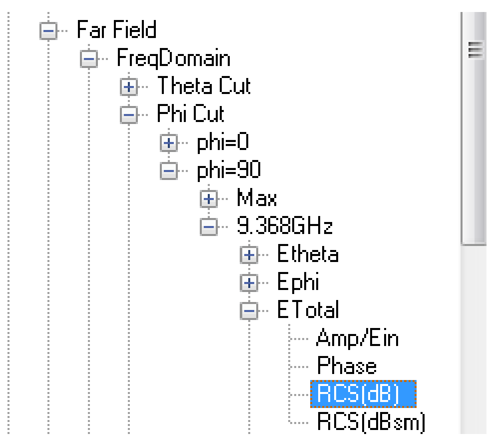
5.11 Result Visualization

Click on the **Data Visualization** button in the *GEMS Solver* window.

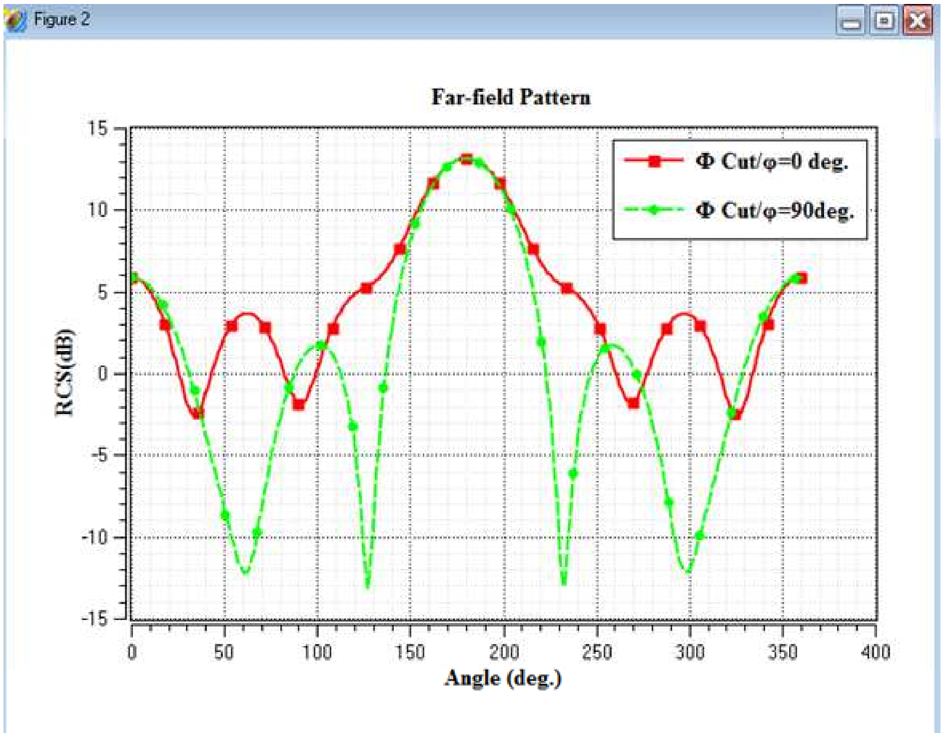
Double-click on the **Output->Far Field->FreqDomain->Phi=0->Etotal->RCS(dB)** option.



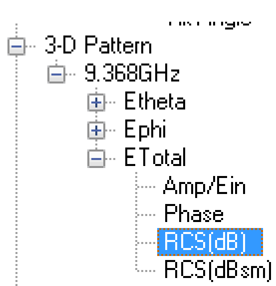
Select the **Output->Far Field->FreqDomain->Phi=90->Etotal->RCS(dB)** option, and then click on the **Add to current window** button in the toolbar.

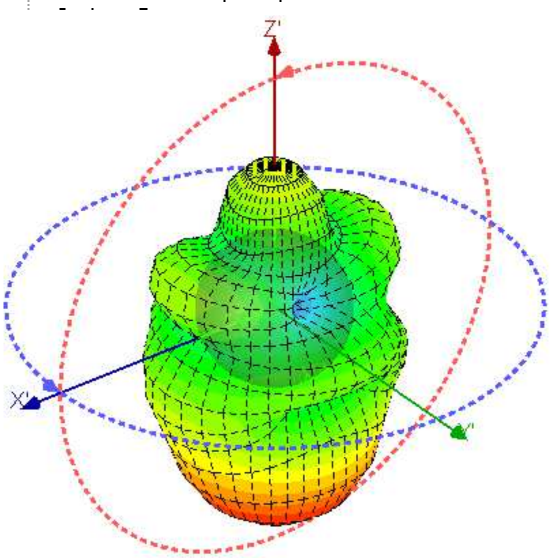


Two RCS patterns at phi=0 and 90 deg. will be plotted in the same figure.



Click on the 3D RCS pattern option in the result tree to plot the 3D RCS pattern.





6

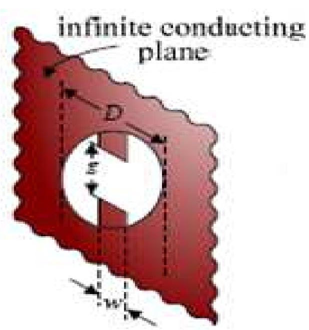
Example 6. Coupling

Description: A small hole on an infinitely large PEC plate is illumined by a plane wave. The output parameter is the transmitted power.

Keywords: Transmitted power, plane wave source, slot on an infinitely large PEC screen.

6.1 Problem Configuration

The configuration of the slot is shown in the figure below:



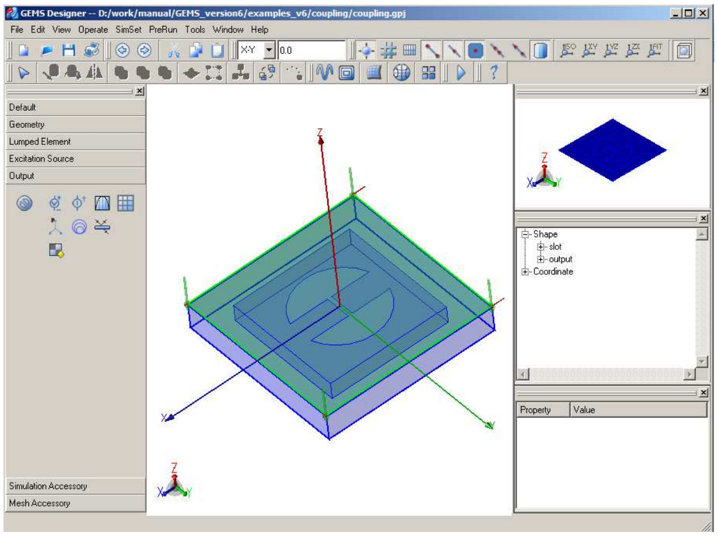
- D = 10 mm
- w = 3mm
- g = 0.487 mm

GEMS allows you to add a plane wave excitation for the infinitely large structure. In this case, the plane wave surface is degenerated to a single surface, for example, along the –z-axis direction propagation, and the plane wave surface is located above the structure.

The PEC sheet may have a finite thickness or infinitely thin depending on the importance of the thickness.

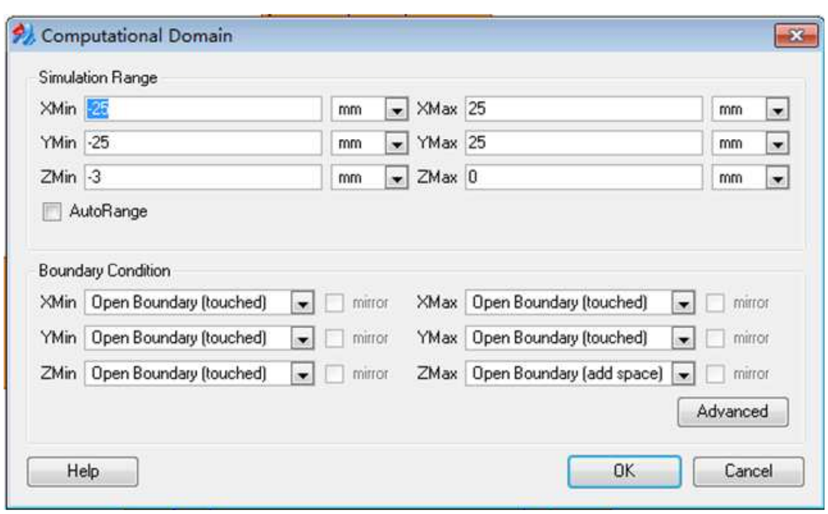
Since the PEC sheet is infinitely large in the horizontal directions, its four sides should be truncated by using the "Open Boundary (touched)", which means that the PEC will be extended to infinity by the absorbing boundary. The top boundary of the domain will be truncated by the "Open Boundary (add space)". The bottom boundary of the domain will be truncated by the "Open Boundary (touched)".

The transmitted power is calculated in an opened box without the top surface underneath the PEC sheet. The transmitted power equals to the real part of integral of Poynting vector on the five surfaces.

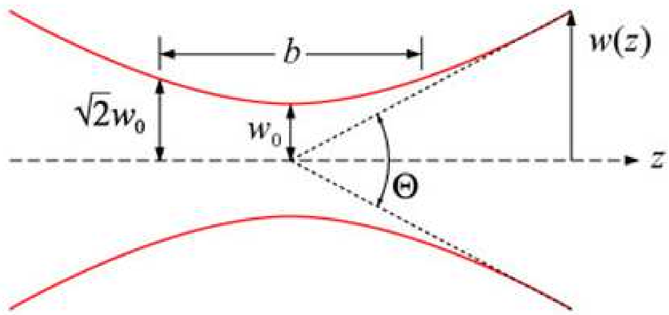


GEMS provides two types of plane wave excitations.

(1) The first one is based on the scattering/total field formulation. This option is designed for the Gaussian beam excitation. It can also be used to the case that requires a single plane wave surface excitation, in which the field behind the plane wave surface is the scattered field, and the field in front of the plane wave surface is total field. The scattering/total field excitation strictly requires that the boundary conditions are Open Boundary (touch) in the five directions, and Open Boundary (Add space) in the top boundary.

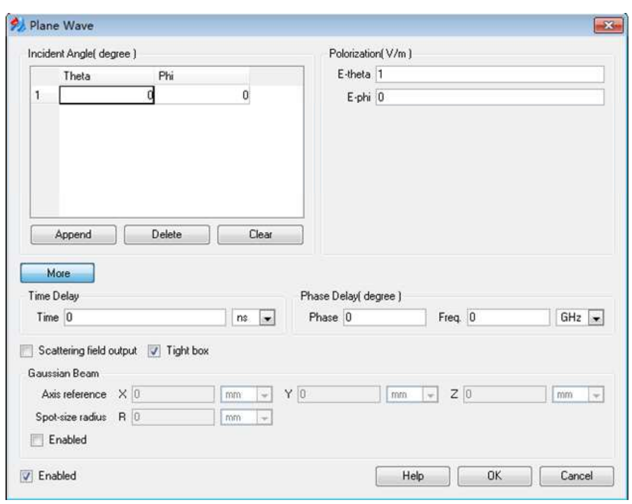


In the Gaussian beam excitation, the plane wave box degenerates to be a sheet, whose cross section size is same as the computational domain.



For example, the Gaussian beam coming from the +Z maximum direction, the boundary should be: “Open Boundary (Touched)” in the -x, +x, +y, -y and -z-directions, and “Open Boundary (Add space)” in the +z-direction. Click on the “plane wave” icon in the “Tool Box -> Excitation” box to open the plane wave excitation window. Click on the **More** button in the **Plane Wave** window, and the Gaussian Beam box will appear.

Enabled: check it to enable the Gaussian beam excitation.
X: *x* reference of the Gaussian beam center.
Y: *y* reference of the Gaussian beam center.
Z: *z* reference of the Gaussian beam center.
The spot size radius: spot radius of Gaussian beam cross the spot center.

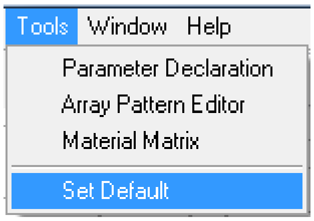


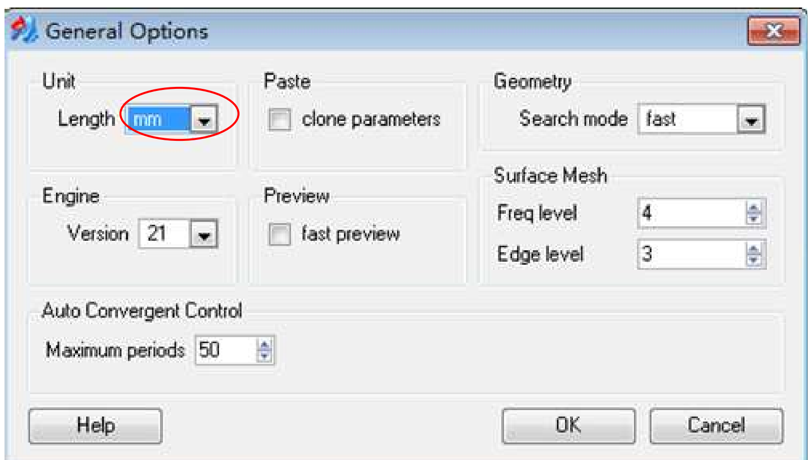
(2) The second one is based on the scattering field formulation. This option is designed for the general scattering problems to calculate the RCS, in which the far field is the scattered field, and the near field is the total field in default setting. You can output the near field as the scattered field by clicking the **More** button in the **Plane Wave** window and then checking the **Scattering field output** box.

6.2 Create Project Model

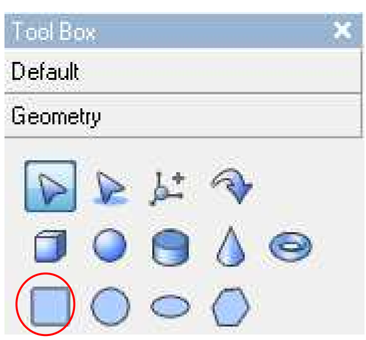
Follow the steps below to create the project model:

(1) Select the project unit



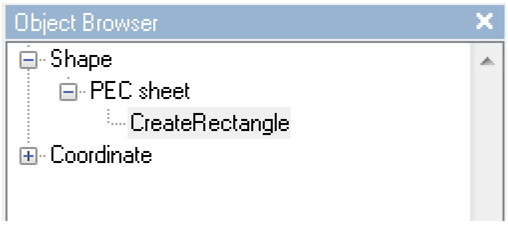


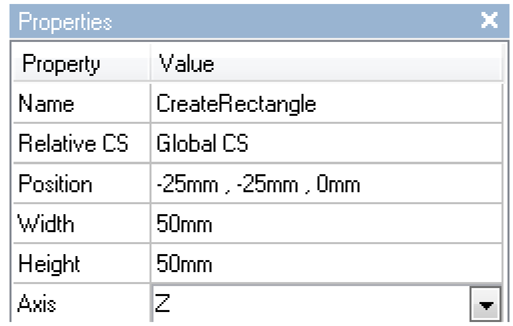
(2) Click on the **Rectangle** icon in the **Tool Box->Geometry** box



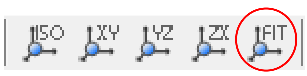
Select the rectangle option in the **Object Browser** box, and change its name to "PEC sheet" in the **Properties** box.

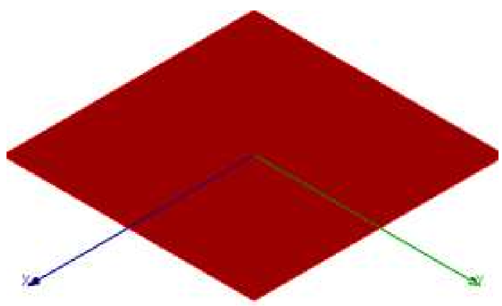
Select the "CreateRectangle" option under the "PEC sheet" entrance in the **Object Browser** box, and change its dimensions. The coordinate of the start point is (-25mm, -25mm, 0mm), and both the width and length are 50mm.



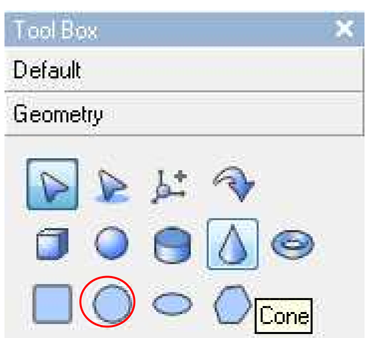


Click on the **FIT** button in the toolbar to fit the rectangle inside the figure region, and change its color to red.





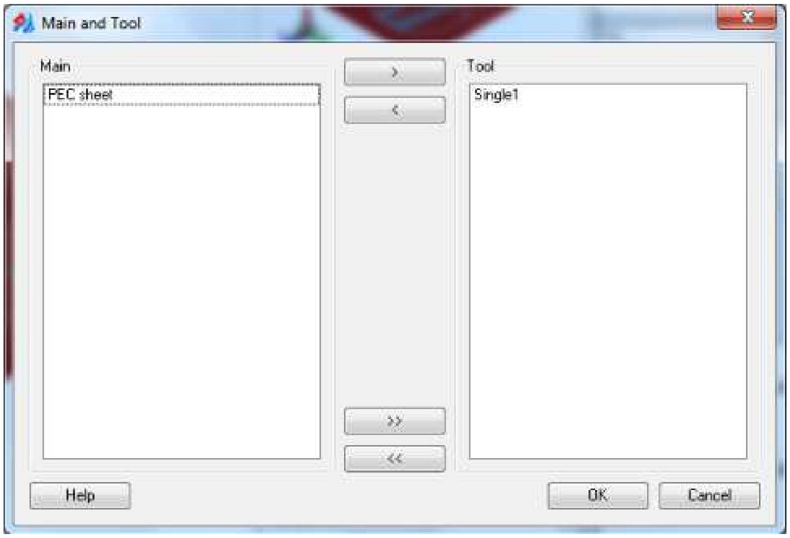
(3) Click on the **Circle** icon in the **Tool Box->Geometry** box. Draw a circle in the figure region.



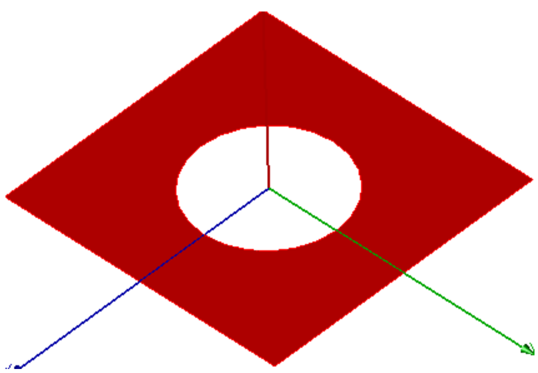
Select the "CreateCircle" in this option in the **Object Browser** box, and change its coordinate to (0mm, 0mm, 0mm) and the radius to 5mm, respectively.

(4) Select the "PEC sheet" and the circle option in the **Object Browser** box, and then click on the **Subtract** button in the toolbar.

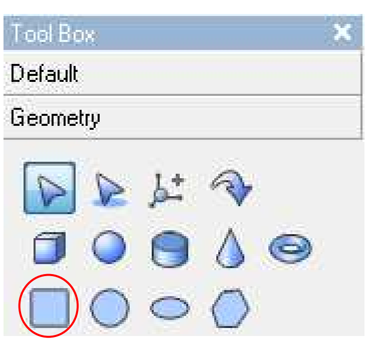
The “PEC sheet” and circle should be in **Main**, and **Tool** boxes, respectively.



Click on the **OK** button to confirm the operation.



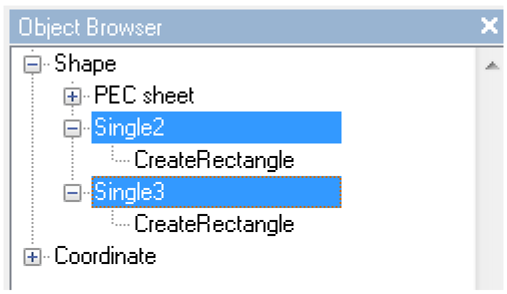
(5) Click on the **Rectangle** icon in the **Tool Box->Geometry** box



Select the “CreateRectangle” under this option in the **Object Browser** box, and change its coordinate to (-5mm, -1.5mm, 0mm), and the width and length to 3mm and 10mm, respectively.

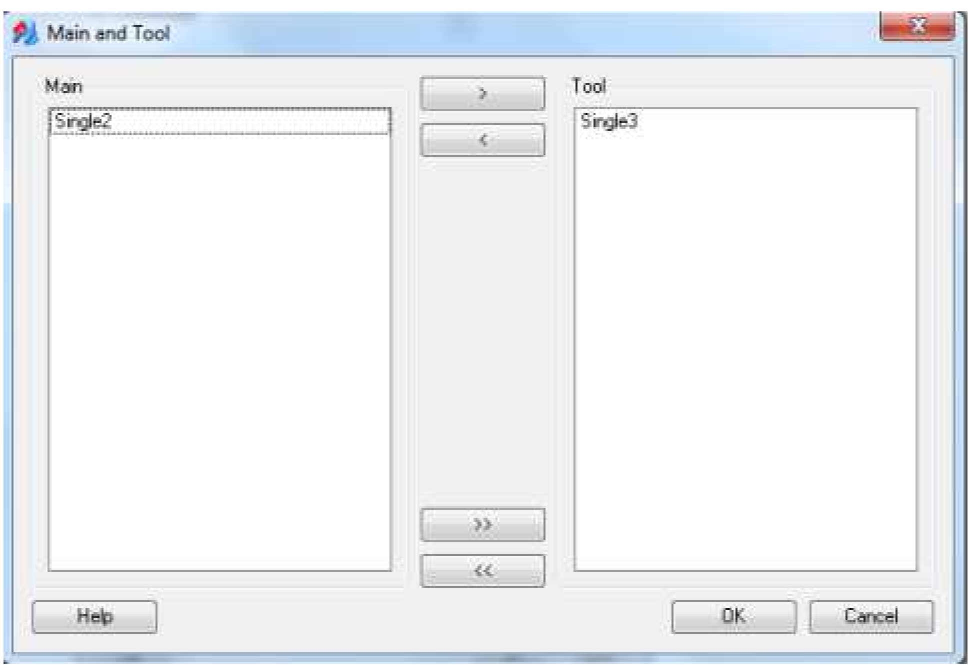
Draw another rectangle and change its coordinate to (-0.2435mm, -1.5mm, 0mm), and the width and length to 3mm and 0.487mm, respectively.

Select the two rectangle options “Single2” and “Single3” in the **Object Browser** box.

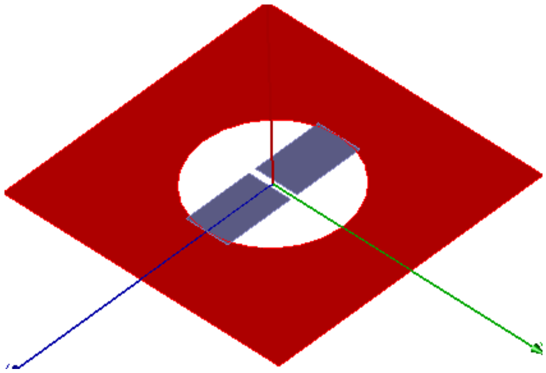


Click on the **Subtract** button in the toolbar.

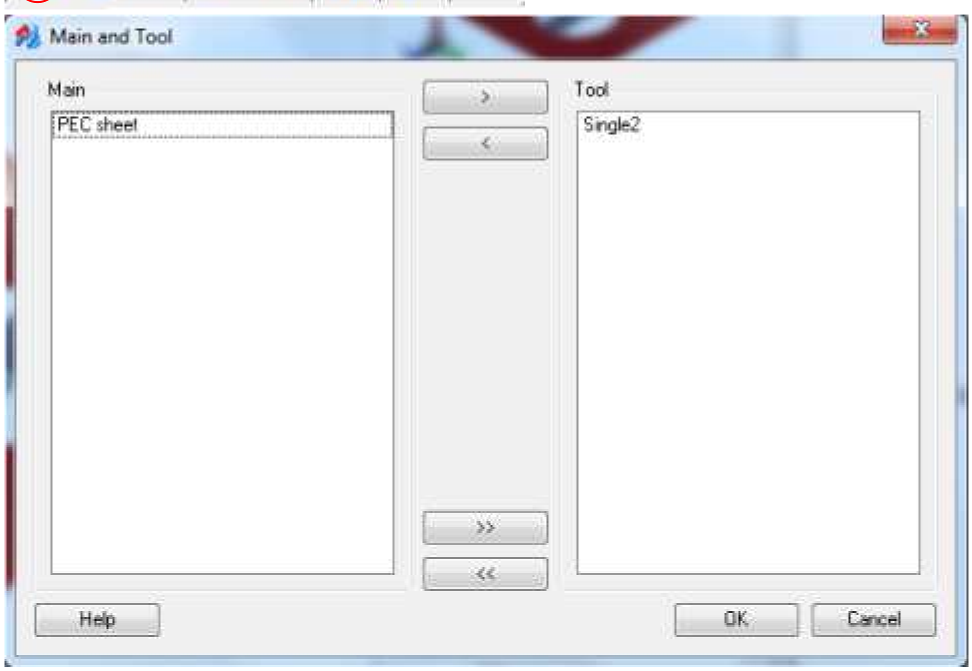
The “Single2” and “Single3” should be in **Main**, and **Tool** boxes, respectively.



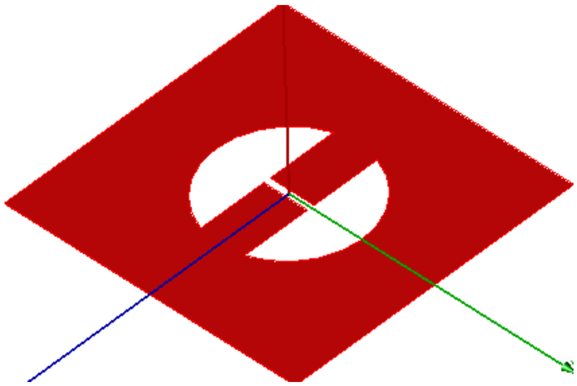
Click on the **OK** button to confirm the operation.



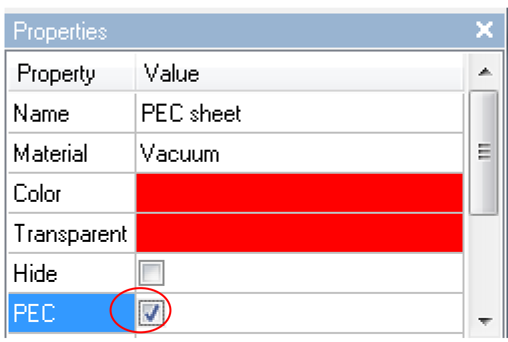
(6) Select the "PEC sheet" and "single2" options in the **Object Browser** box, and then click on the **Unite** button in the toolbar.



The "PEC" and "Single2" should be in **Main**, and **Tool** boxes, respectively. Click on the **OK** button to confirm the operation.

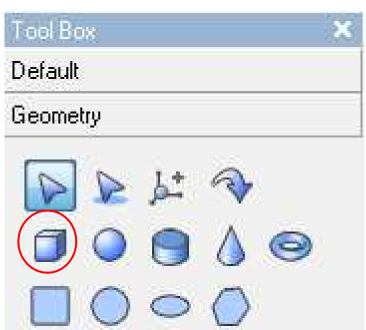


(7) Select the "PEC sheet" in the **Object Browser** box, and then check the "PEC" box in the **Properties** box to assign the "PEC sheet" to be the PEC material.



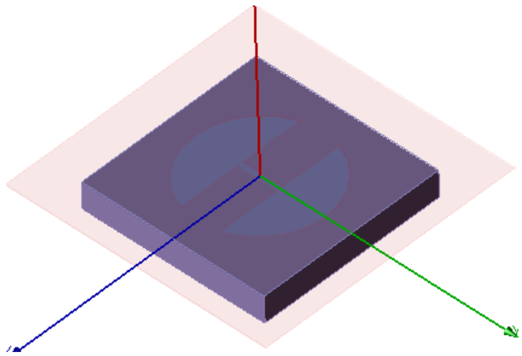
6.3 Output Setting

Click on the **Cuboid** icon in the **Tool Box->Geometry** box

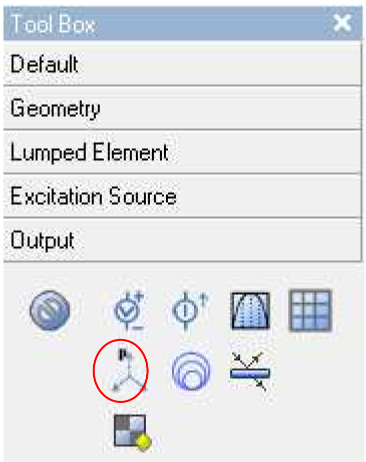


Draw a box and its corner is at (-20mm, -20mm, 0) and its width and length are 40mm, respectively.

Select this option in the **Object Browser** box, and change its name to "output" in the **Properties** box.



Select the "output" option in the **Object Browser** box, and click on the *Poynting vector* icon in the **Tool Box->Output** box.

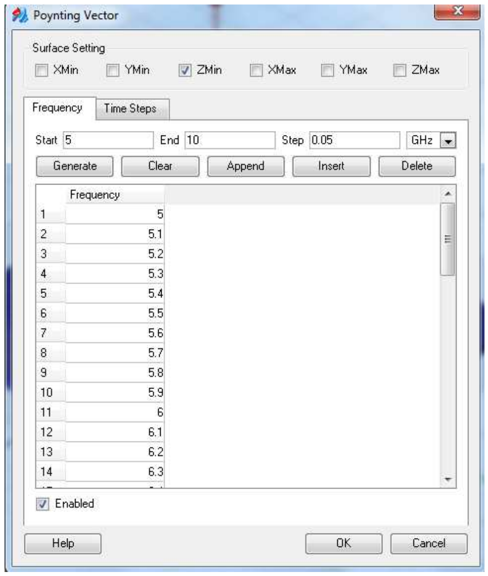


The six surfaces in the **Surface Setting** box correspond to the six surfaces of the drawn box. GEMS will calculate the integral of Poynting vector on the selected surfaces and then calculate the power summation through the selected surfaces.

In this example, we need to calculate the power through the slot. We draw a box and use the integral of Poynying vector on the five surfaces except the top surface (Zmax), which equals the power through the slot.

Check the **Enabled** box and uncheck the **ZMax** box to remove the power contribution from the "Zmax" surface.

You need to specify a frequency list at which the transmitted power will be generated. If the frequency is not in the frequency list, you can manually add it in the frequency list. Click on the **OK** button to confirm the parameter setting.

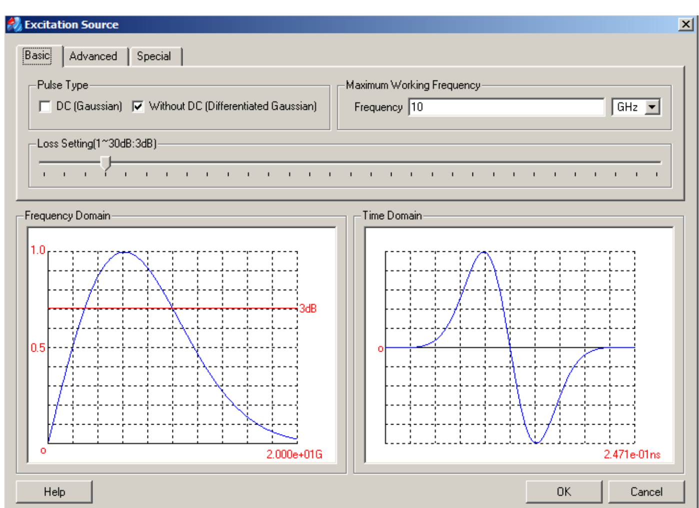


6.4 Excitation Pulse Setting

Click on the **Set excitation source** button in the toolbar.

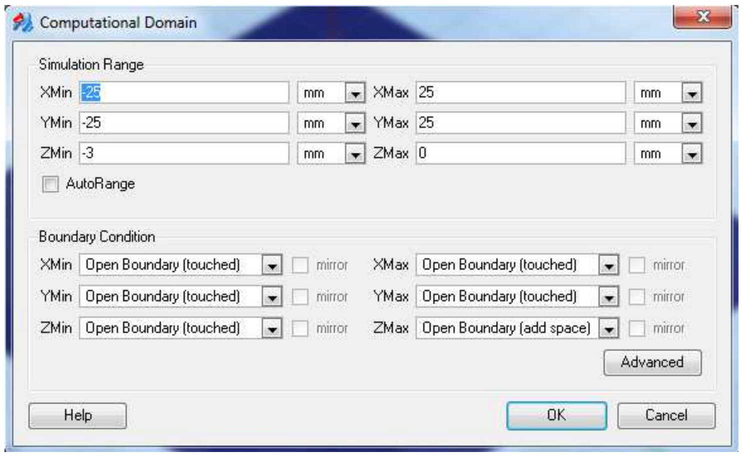
The maximum frequency we are interested in here is 10GHz, so that the **maximum frequency** in the **Excitation Source** window should be 10GHz.
You cannot use the Pure Gaussian as the excitation pulse for the plane wave source; otherwise, the simulation result will be incorrect.

Click on the **OK** button to confirm the excitation pulse setting.

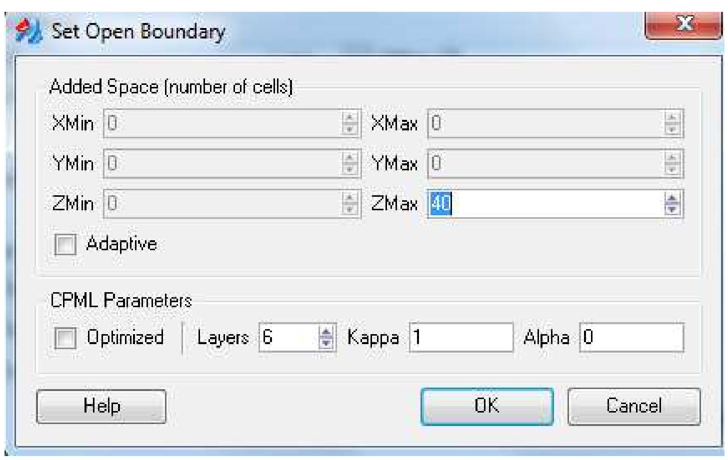


6.5 Set Domain and Boundary Condition

Click on the **Set boundary condition** button in the toolbar.



Click on the “Advanced” button and type “40” in the **ZMax** box. This number “40” generates a large white space between the object and domain boundary to form a regular domain shape.



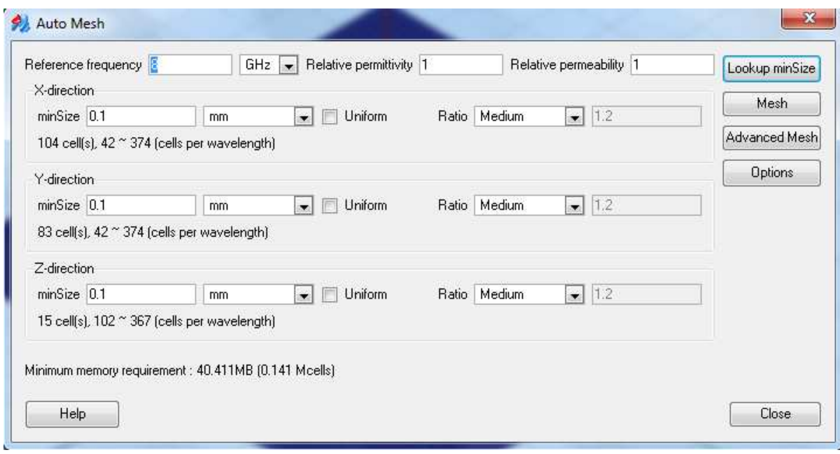
Since the PEC sheet is infinitely large in the horizontal directions, there is no gap between the PEC sheet and domain boundary in the horizontal direction.

*To remove the contribution from the bottom of the plane wave surface, the boundary should be selected to be “**Open Boundary (touched)**”. However, the boundary in the Z maximum direction should be “**Open Boundary (add space)**”.*

The power output box cannot touch the domain boundary. If we check the **AutoRange** box, the power output box will touch the domain boundary in the Z minimum direction. We need to uncheck the **AutoRange** box and add a small distance between the power output box and the domain boundary in the Z minimum direction; for example, change the “-2” to “-3” in the **ZMin** box, namely, add 1mm space.

6.6 Design Mesh Distribution

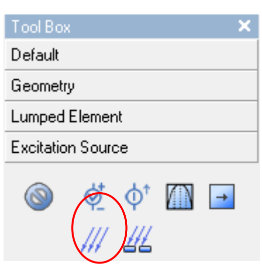
Click on the **Auto mesh** button in the toolbar.

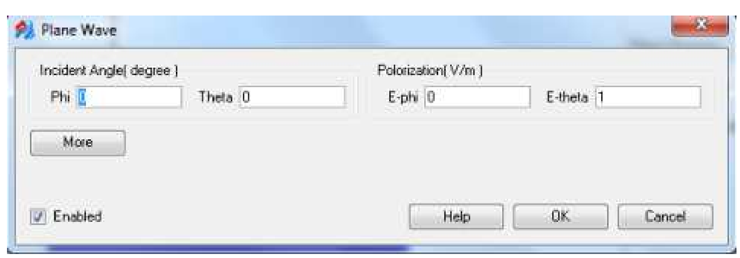


The size of the minimum gap is 0.487mm, and, generally speaking, we can select the minimum cell size to be 0.24mm. However, for this problem, we need to use 0.1mm to get the good results.

6.7 Add Plane Wave Source

Click on the **Plane Source** icon in the **Tool Box->Geometry** box.

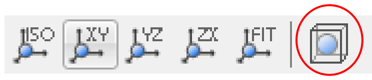


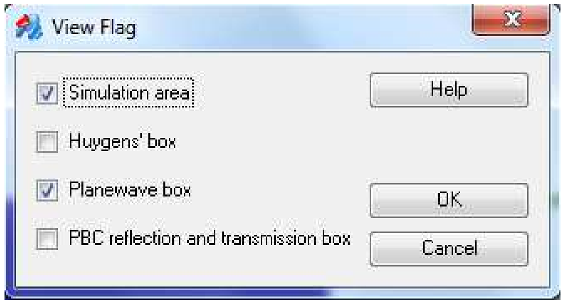


The plane wave illumines from the +z direction if phi=0 degree and theta =0 degree, and the polarization is *Ex* for *Etheta*=1 and *Ephi*=0.

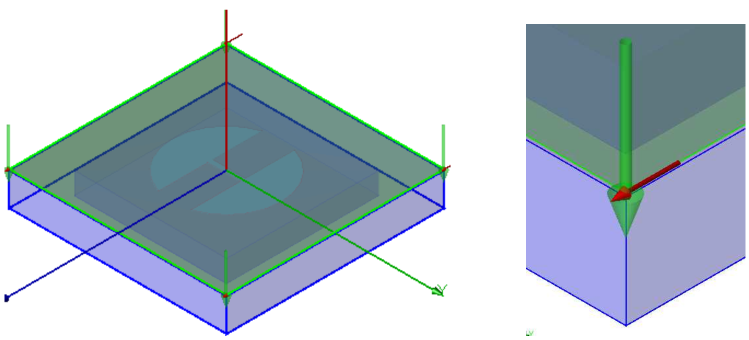
Click on the **OK** button to confirm the setting.

Click on the **View defined box** button in the toolbar.





Check the **Simulation area** and **Plane box** boxes.



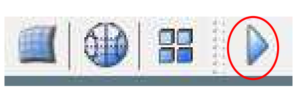
The green and red arrows indicate the directions of plane wave propagation and polarization.

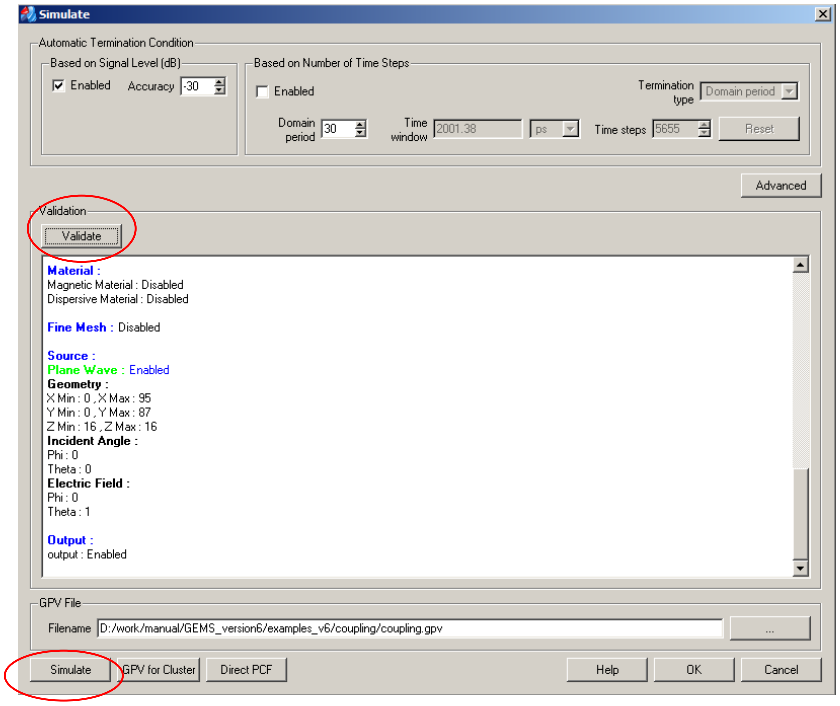
6.8 Save Project

Click on the **Save** button in the toolbar to save the project.

6.9 Generate Simulation File

Click on the **PreCalculate** button in the toolbar.

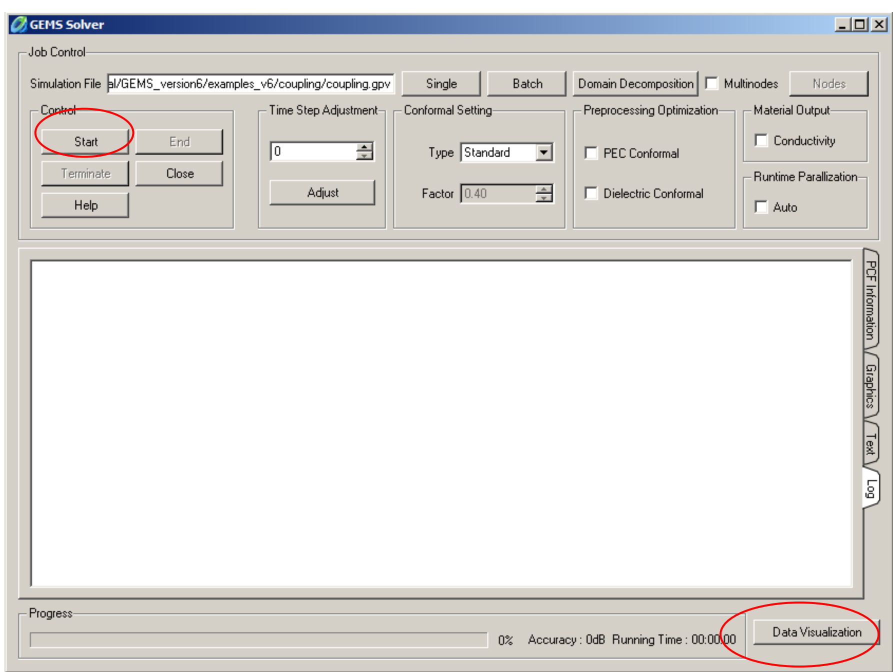




In the default status, the **Apply window function** box is checked, and the Hamming window function will be added in the time signature in the GEMS simulation. In this project, we do not need the window function, so the **Apply window function** box is unchecked.

6.10 Simulate Project

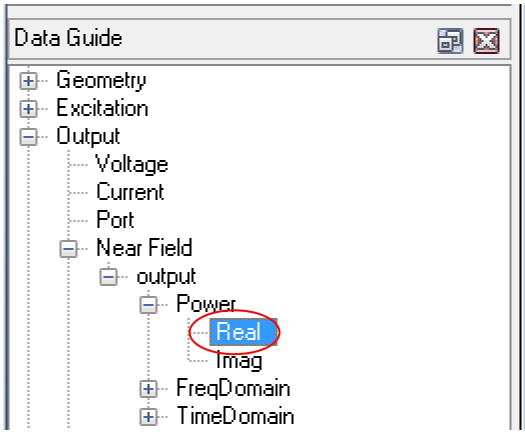
Click on the **Start** button to start the simulation.

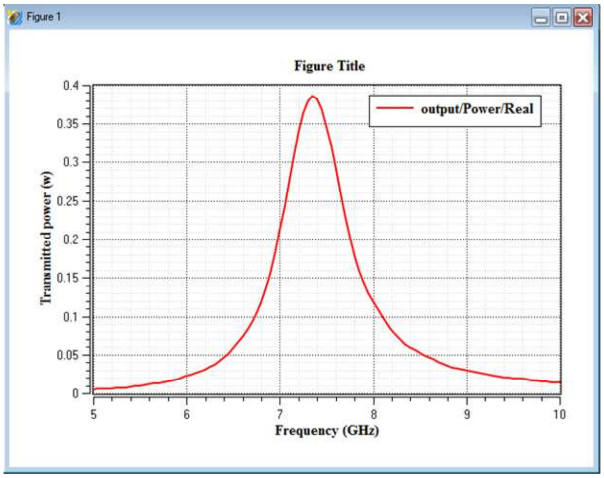


Click on the **Data Visualization** button to start the result visualization.

6.11 Result Visualization

Double-click on the **Output->Near Field->Power->Real** option to plot the transmitted power through the hole. This transmitted power is normalized to the power density of the incident lane wave.

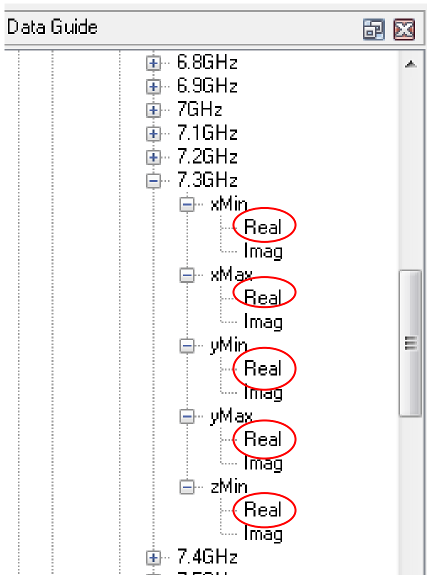




The transmitted power was normalized to incident power density in the version before 7.0. It is normalized to the incident electric field in the latter version.

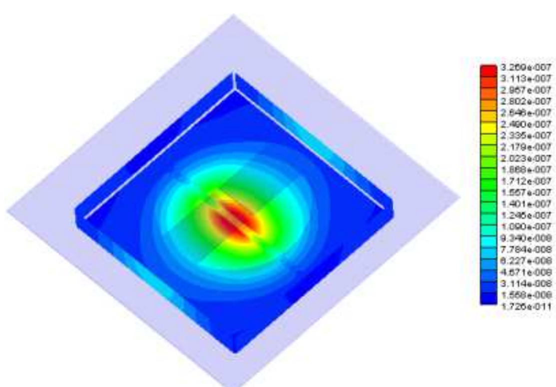
Click on the **View model** button and then on the **Transparent mode** button in the toolbar.

Select the **Output->Near Field->FreqDomain->7.3GHz->Zmin->Real** option.

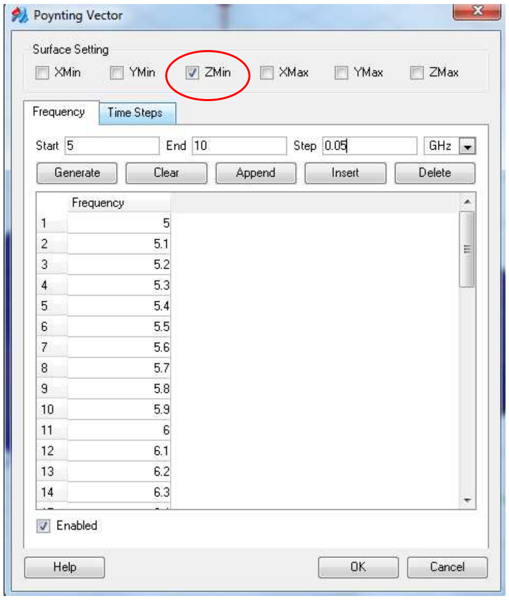


Click on the **Add to current window** button in the toolbar.

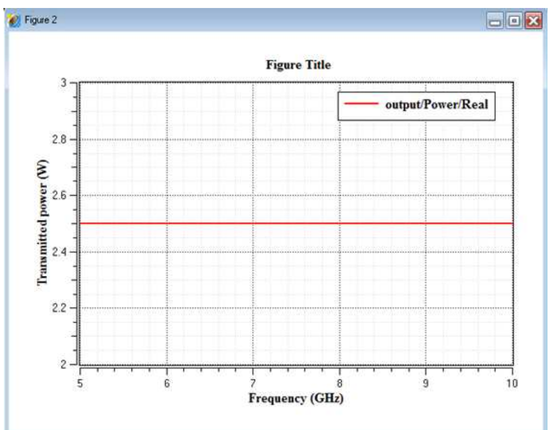
Click on the **Play** button in the toolbar.



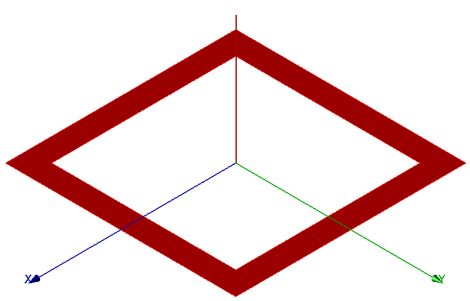
Select the **Output->Near Field->FreqDomain->7.3GHz->Xmin->Real** option, and then click on the **Add to current window** button in the toolbar. Follow the same procedure to add the five Poynting vector distribution on the five surfaces.
To verify the result, we remove the slot and PEC screen, let the power measurement box touch the domain boundary in the horizontal direction and check the **Zmin** box only.



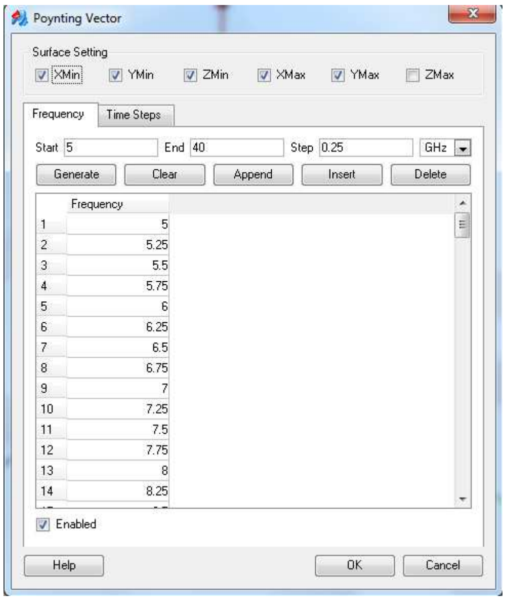
The cross section of the domain is 50mm x 50mm, so the transmitted power is 0.0025 Watts, as expected.

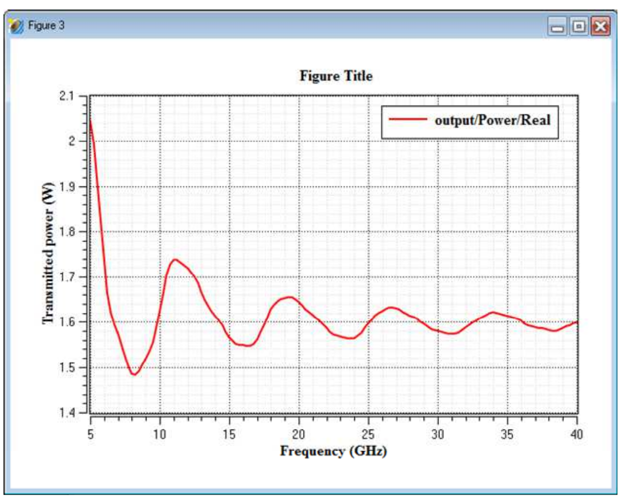


Next, we replace the slot as a simple rectangle slot with a cross section of 40mmx40mm . The transmitted power should be 0.0016Watts at the high frequency.

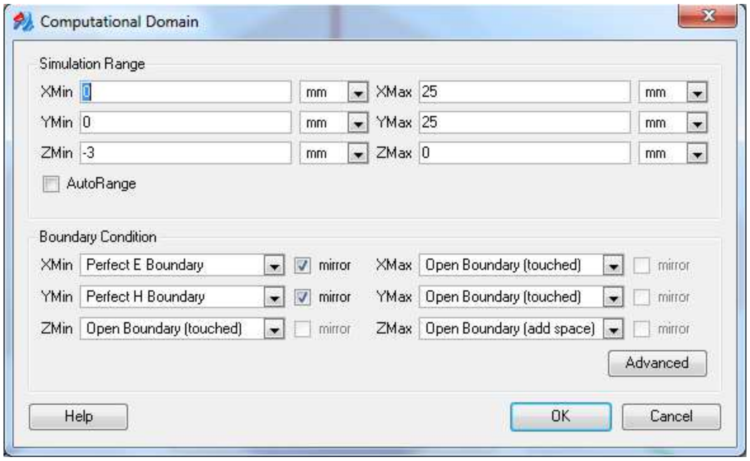


We will calculate the transmitted power in the frequency range between 5GHz to 40GHz. So the maximum frequency in the excitation pulse is changed to 40GHz. The transmitted power output box is changed to x: -22.5mm to 22.5mm; y: -22.5mm to 22.5mm; and z:-1mm to 0mm, and use the five walls to measure the transmitted power.

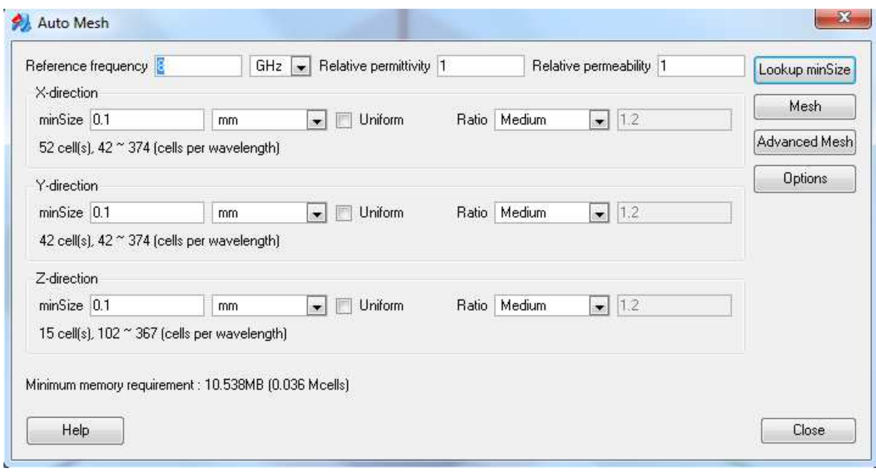




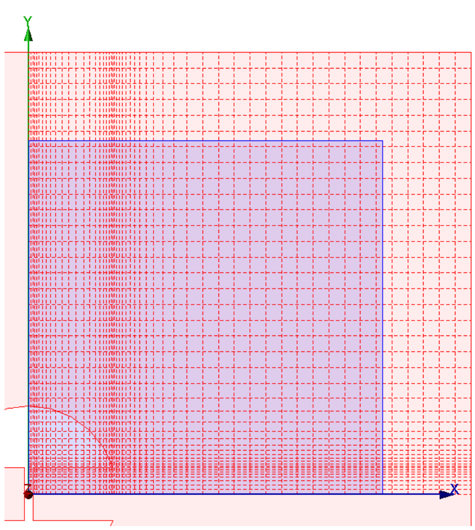
Finally, we use the symmetric property of the problem to reduce the simulation domain to one quarter of the original structure. In the original project, change its domain size to one quarter and the boundary to the "Perfect E Boundary" and "Perfect H boundary" in the X- and Y-directions, respectively. In order to get the transmitted power of the complete structure, we need to check the "mirror" options corresponding to the "Perfect E Boundary" and "Perfect H boundary" boundaries.



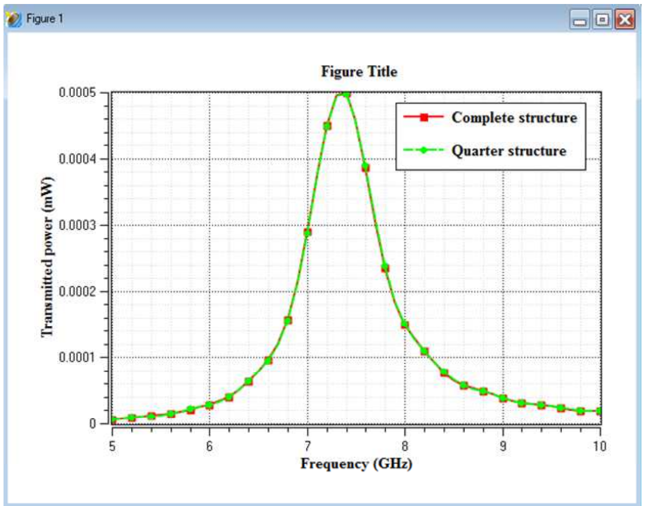
Generate the mesh distribution.



View the mesh distribution.



The results are same obtained by simulating one quarter structure using the symmetric property and directly simulating the original problem.



7

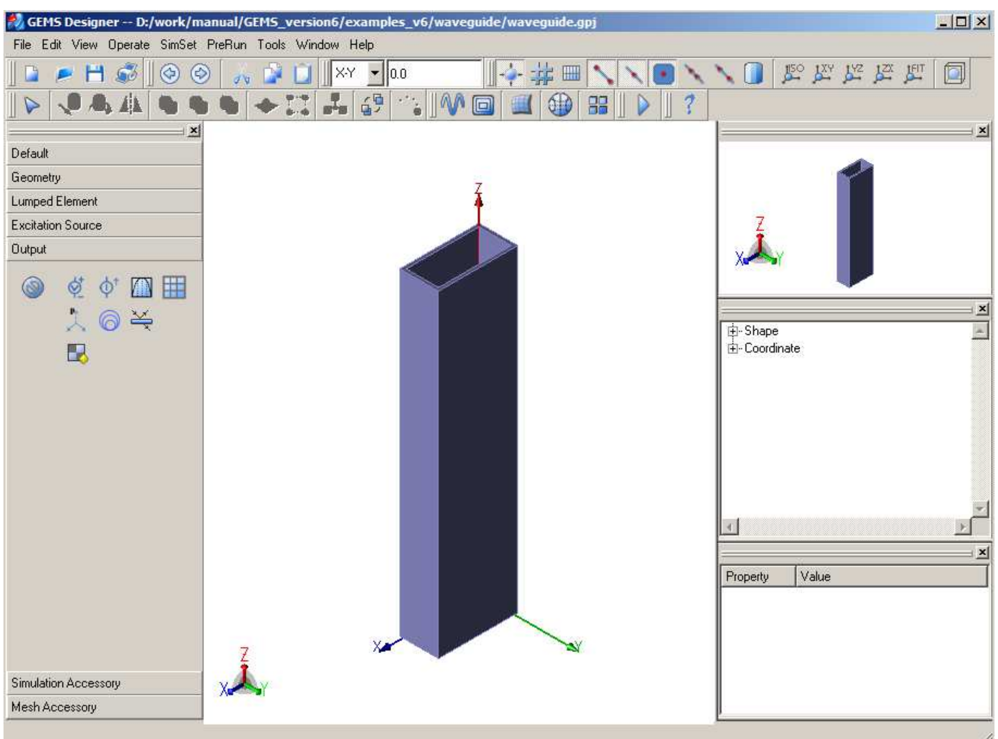
Example 7. Rectangular Waveguide

Description: A X90 waveguide with a rectangular cross section is excited by using a TE_{11} mode. The output parameters include S-parameters, propagation constant, attenuation factor and mode impedance.

Keywords: Rectangular waveguide, mode impedance, S-parameters, mode excitation and output, propagation constant, and attenuation factor.

7.1 Problem Configuration

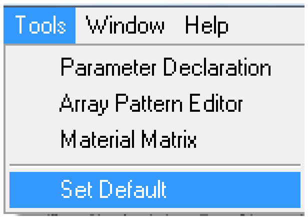
The inner dimensions of waveguide are 22.86mm and 10.16mm, which is excited by a TE_{10} mode. Its thickness does not affect the simulation results because the wall is PEC material.

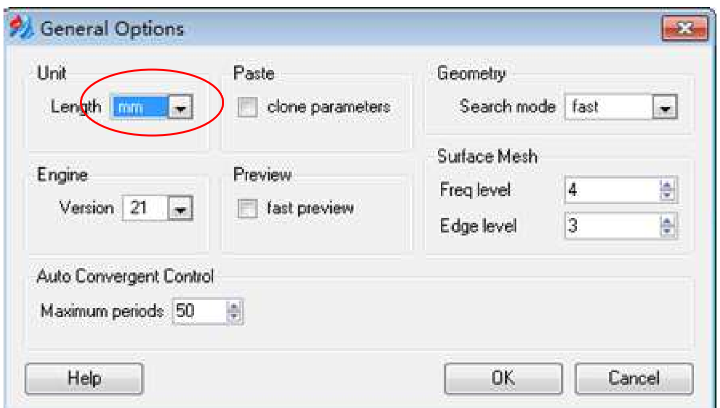


7.2 Waveguide Modeling

Follow the steps below to create the waveguide model:

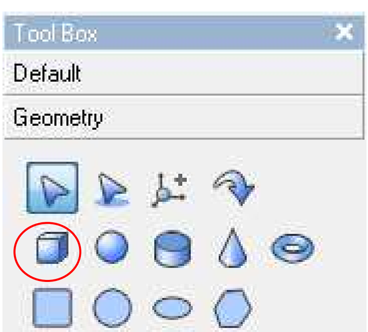
(1) Open GEMS Designer
(2) Select the **Set Default** option in the **Tools** menu





(3) Click on the **New** button in the toolbar to create a new project

(4) Select the **Cuboid** icon in the **Tool Box->Geometry** box.



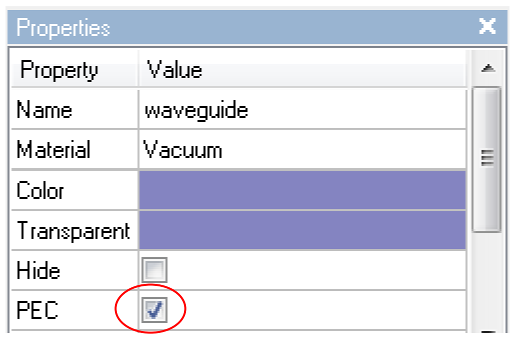
There are two ways to create the waveguide geometry:

(i) Draw a larger box and a smaller box.

- Draw a larger box in the figure region.

  Select the **Cuboid** option in the **Object Browser** box and change its name to "waveguide" in the **Properties** box.

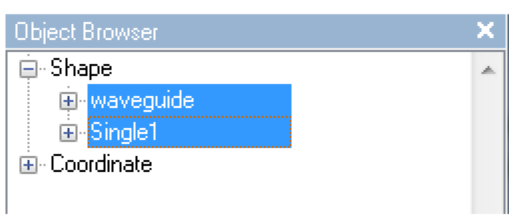
  Check the **PEC** box in the **Properties** box.



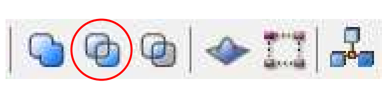
  Select the "CreateCuboid" option under the "waveguide" entrance and change its dimensions to: start point (-1, -1, 0), width=24.86mm, height=12.16mm, and length=100mm.

- Draw a smaller box: start point (0, 0, 0), width=22.86mm, height=10.16mm, and length=100mm.

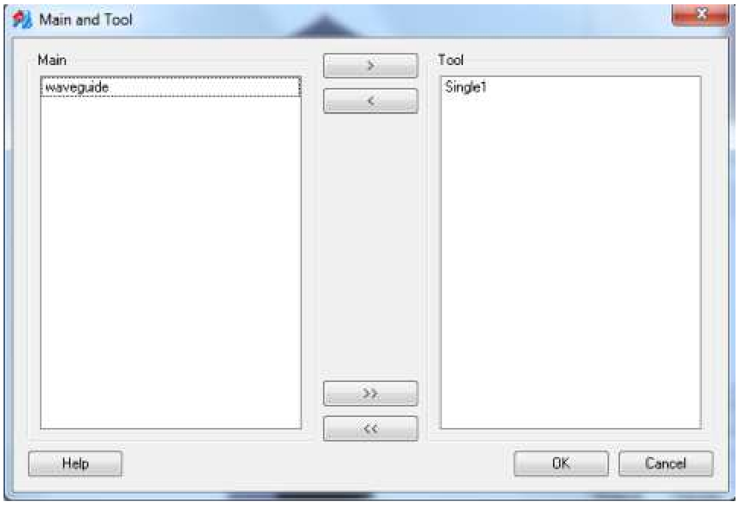
- Select the "waveguide" option in the **Object Browser** box, and press and hold the "Ctrl" key and use the mouse to select the smaller box.

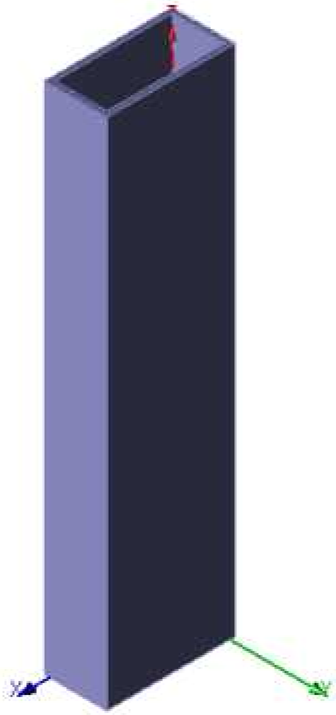


- Click on the **Subtract** button in the toolbar.



- The “waveguide” should be in the **Main** box. Click on the **OK** button to confirm the setting.

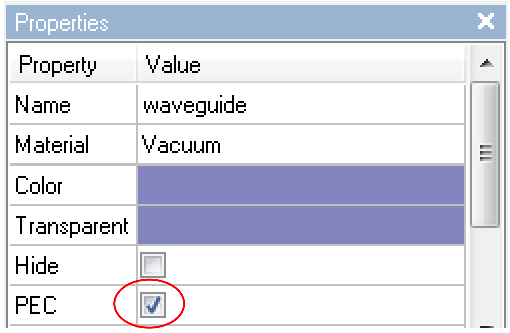




(ii) Draw one box

- Draw a box in the figure region.
  Select the **Cuboid** option in the **Object Browser** box and change its name to “waveguide” in the **Properties** box.

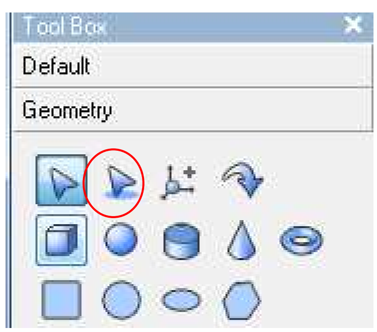
Check the **PEC** box in the **Properties** box.

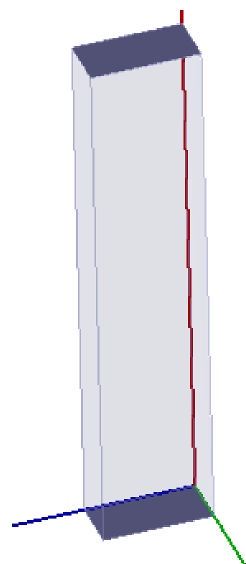


Select the “CreateCuboid” option in the “waveguide” folder in the **Object Browser** box and change its dimensions to: start point (0, 0, 0), width=22.86mm, height=10.16mm, and length=100mm.

- Generate a waveguide model

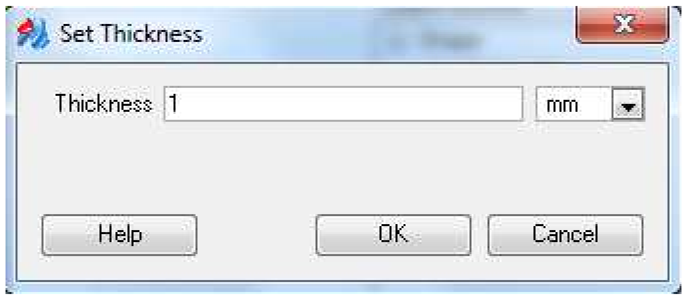
Select the “waveguide” entrance in the **Object Browser** box. Click on the **Select face** icon in the **Tool Box->Geometry** box. Use the mouse to select two surfaces.

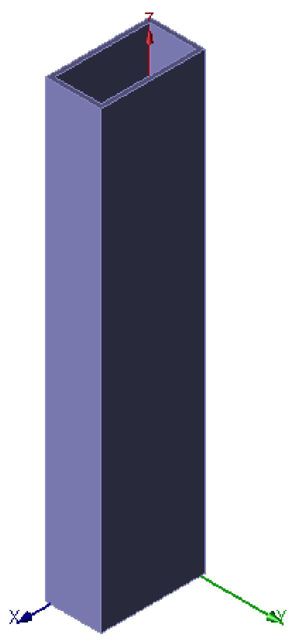




Select the “waveguide” option in the **Object Browser** box and click on the “Shelling” option in the **Operate** menu.

Type “1” in the box, and click on the **OK** button.



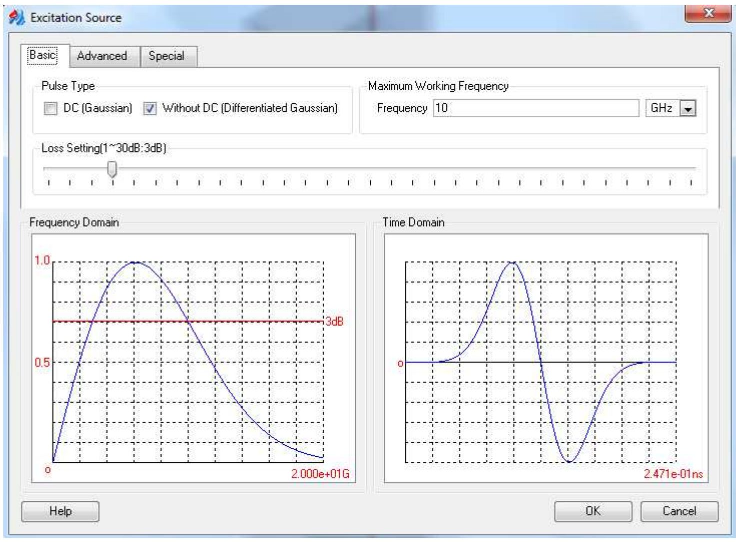


7.3 Excitation Pulse Setting

Click on the **Set excitation source** button in the toolbar.

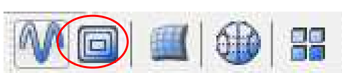
The maximum frequency we are interested in here is 10GHz, so that the maximum frequency in the **Excitation Source** window should be 10GHz.

Click on the **OK** button to confirm the excitation pulse setting.

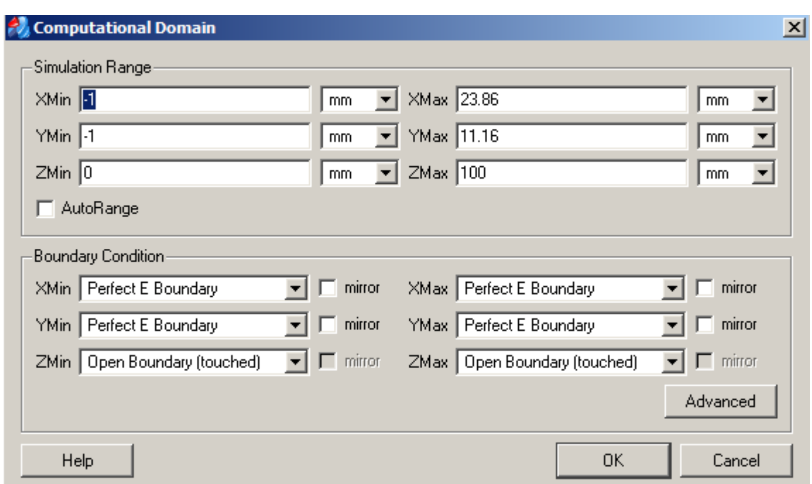


7.4 Set Domain and Boundary Condition

Click on the **Set boundary condition** button in the toolbar.

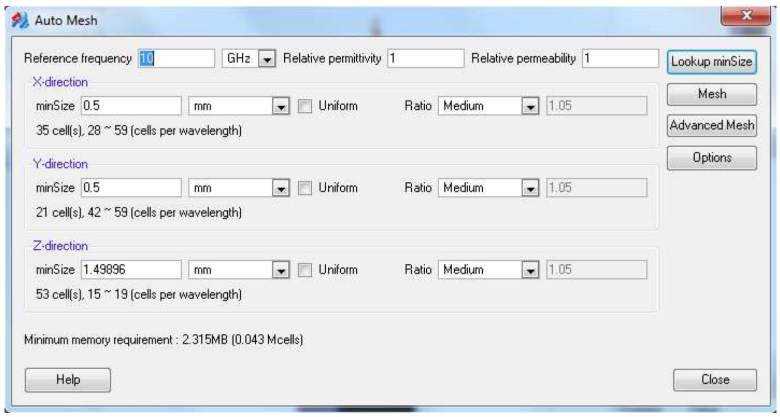


The boundary in the x- and y-directions is the **Perfect E Boundary**; however, the boundary in the z-direction is the **Open Space (touched)** boundary to extend the waveguide in the propagation direction to infinity.



7.5 Design Mesh Distribution

Click on the **Auto mesh** button in the toolbar.

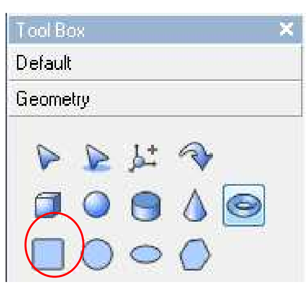


Since there is no fine structure in the waveguide, the minimum cell size is selected to be 15 cells per wavelength. The **Ratio** can be select to be “Medium”.

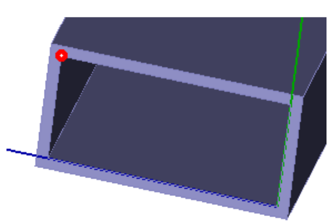
7.6 Set Excitation Port

Follow the steps below to draw an excitation mode port:

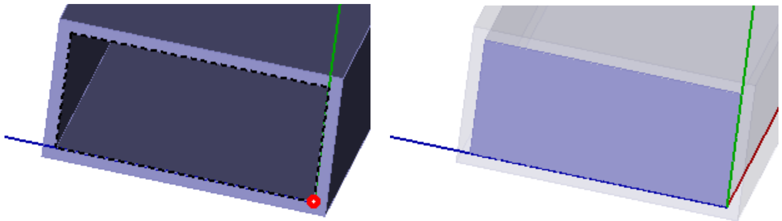
(i) Select the **Rectangle** icon in the **Tool Box->Geometry** box.



(ii) Use the mouse icon to snap to the inner corner of the waveguide.



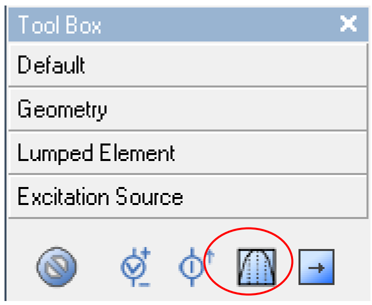
(iii) Draw a rectangle with the size of the inner cross section of waveguide.



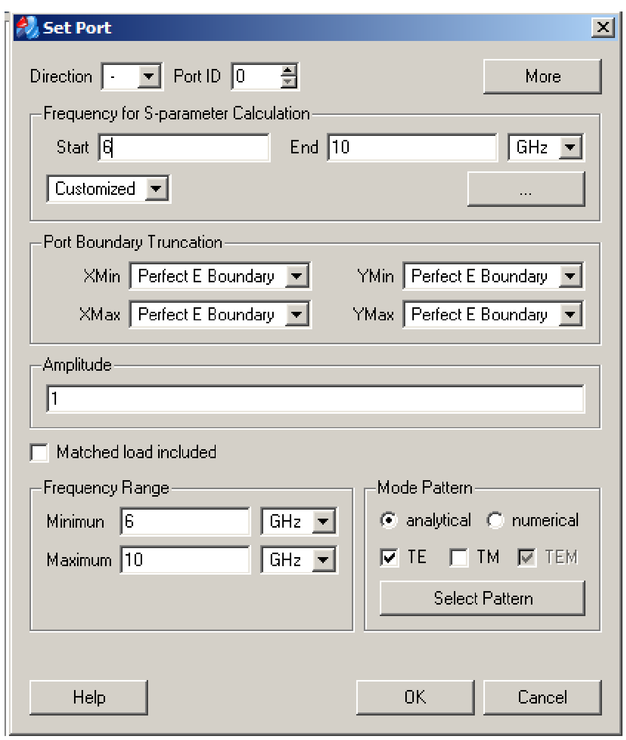
(iv) Select the rectangle option in the **Object Browser** box and change its name to “excitation” in the **Properties** box.

(v) Select the **Port** icon in the **Tool Box->Excitation Source** box.

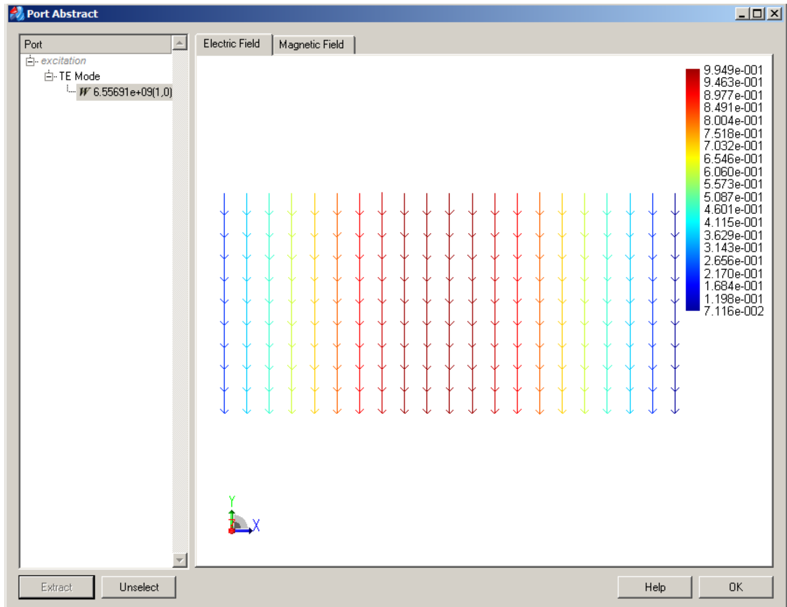
.



Specify the frequency range for the S-parameter outputs in the **Frequency for S-parameter Calculation** box. Select the boundary condition for the port truncation. Uncheck the **Matched load included** box.

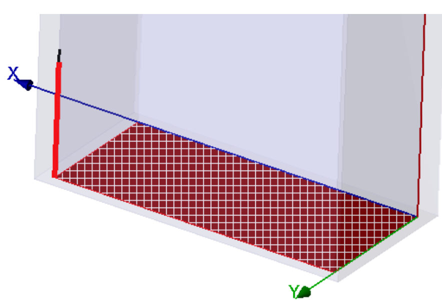


Specify the frequency range in which you like to extract the modes. For the rectangular waveguide, check the **analytic** radio button in the **Mode Pattern** box. Click on the **Select Pattern** button to view the excitation mode pattern.



The mode with "**W**" is the excitation mode. You can click on the **Unselect** button to deselect the selected mode.

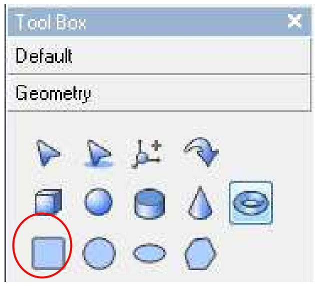
Ensure the port direction pointing to the power propagation direction, which is controlled by the "-" or "+" in the **Direction** box.



7.7 Set Output Port

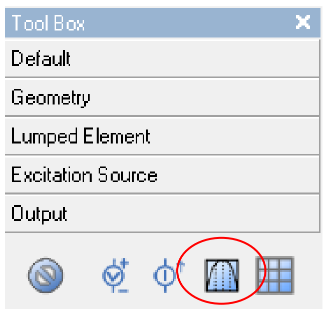
Follow the steps below to draw an output mode port:

(i) Select the **Rectangle** icon in the **Tool Box->Geometry** box. Draw a rectangle at the top of waveguide.

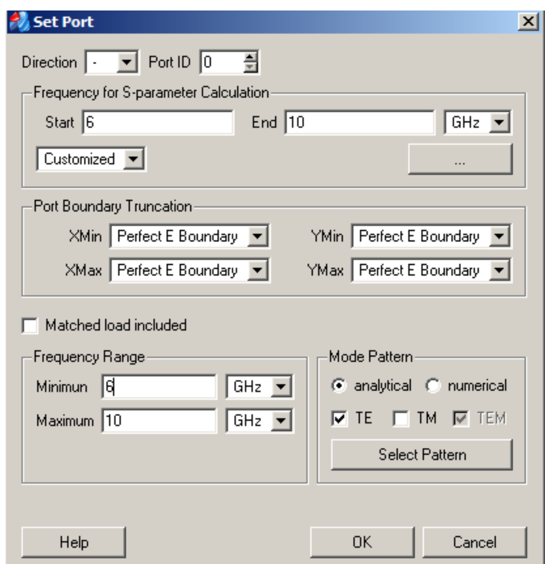


(ii) Select this option in the **Object Browser** box and change its name to "output" in the **Properties** box. Select the **Port** icon in the **Tool Box->Output** box.

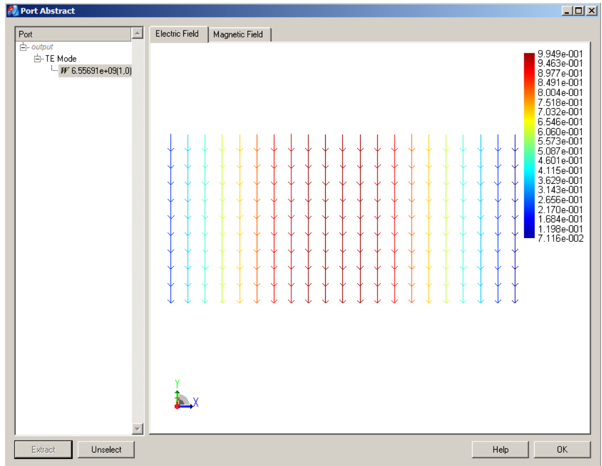
.



Specify the frequency range for the S-parameter output in the **Frequency for S-parameter Calculation** box. Select the boundary condition for the port truncation.

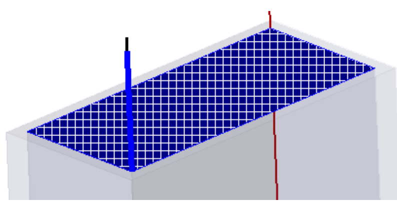


Specify the frequency range in which you like to extract the modes. For the rectangular waveguide, check the **analytic** radio button in the **Mode Pattern** box. Click on the **Select Pattern** button to view the excitation mode pattern.



The mode with "**W**" is the output mode. You can click on the **Unselect** button to deselect the selected mode.

Ensure the port direction pointing to the power propagation direction, which is controlled by the "-" or "+" in the **Direction** box.

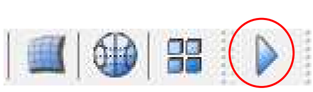


7.8 Save Project

Click on the **Save** button in the toolbar to save the project.

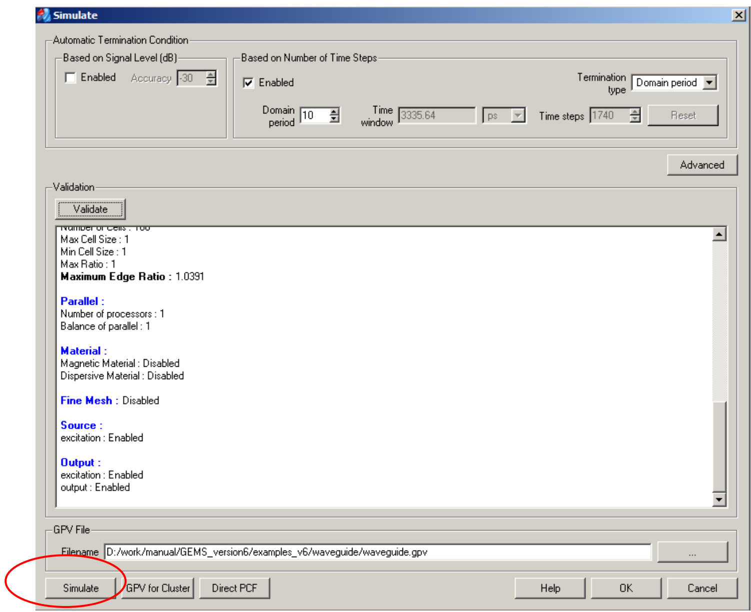
7.9 Generate Simulation File

Click on the **PreCalculate** button in the toolbar to save the project.



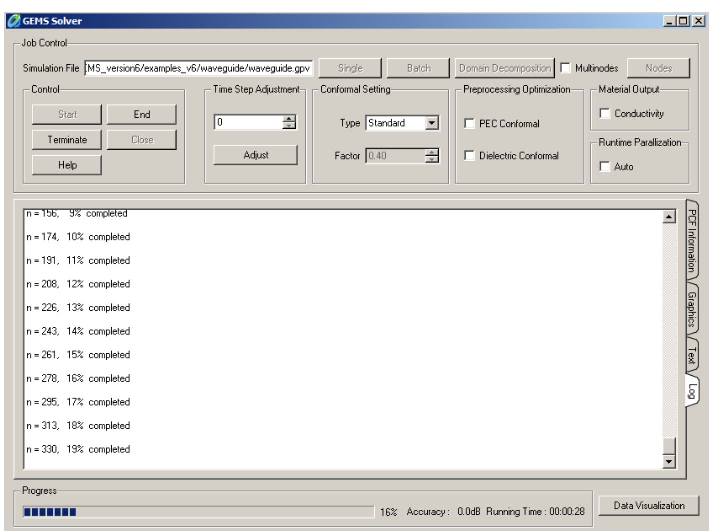
Support you know how many domain periods you need to run for this problem, you can set the simulation termination using the domain period. For example, in the example, you need to run 10 domain periods.

You can decide if you need apply the Hamming window function in the time domain signature. Click on the **Simulate** button to open the *GEMS Solver* window and start the simulation.



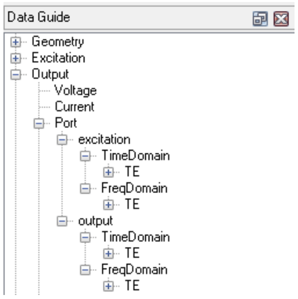
7.10 Simulate Project

Click on the **Simulate** button in the *Simulate* window. Click on the **Data Visualization** button to open the *Display* window after the simulation is completed.



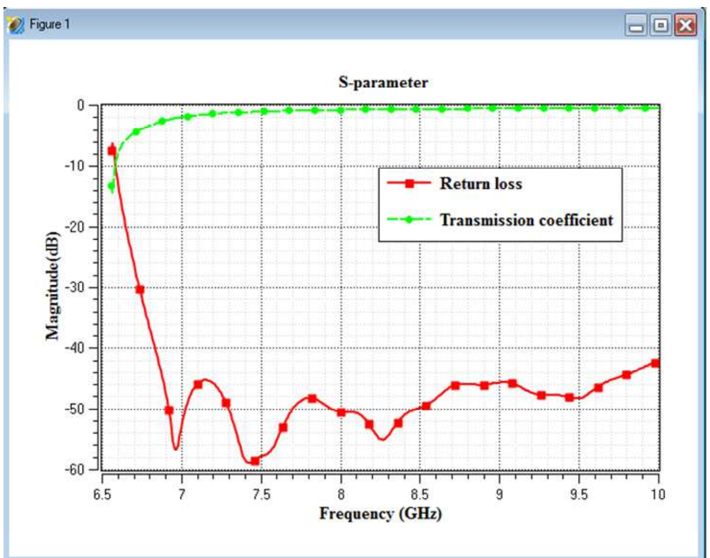
7.11 Result Visualization

The port outputs are listed in the **Output->Port** folder.

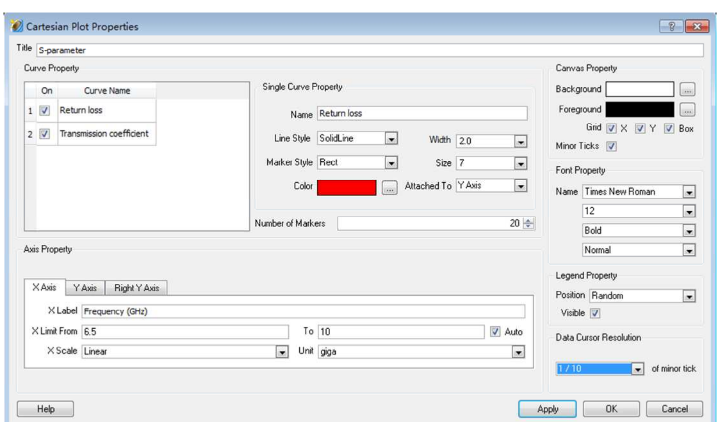


Double-click on the **Output -> Port -> excitation - >FreqDomain -> TE -> (1,0) -> S parameter -> Magnitude(dB)** option.

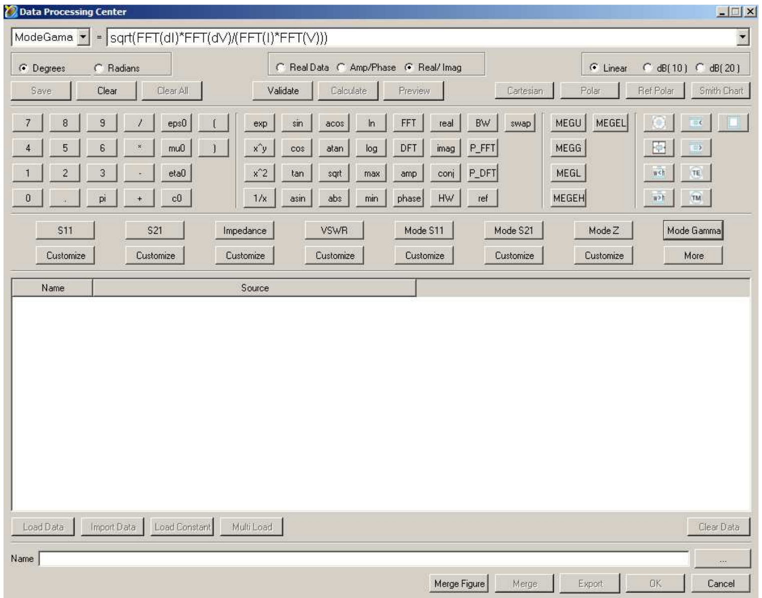
Select the **Output -> Port -> output -> FreqDomain -> TE -> (1,0) -> S parameter -> Magnitude(dB)** option, and click on the **Add to current window** option in the toolbar.



Double-click inside the figure to adjust the display options.

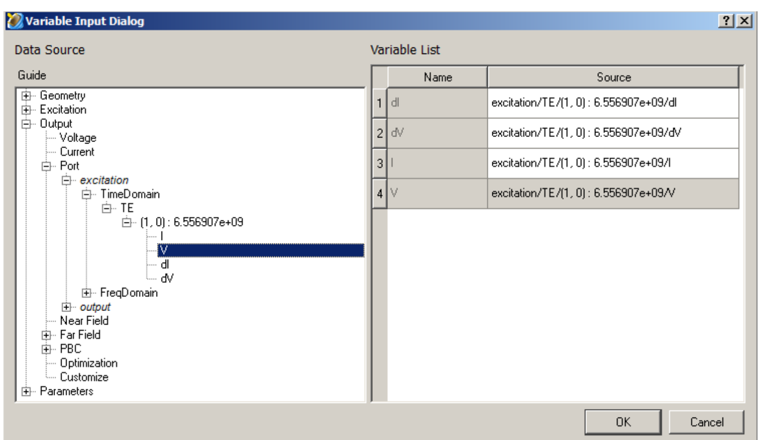


Click on the **Data Process** button the toolbar.

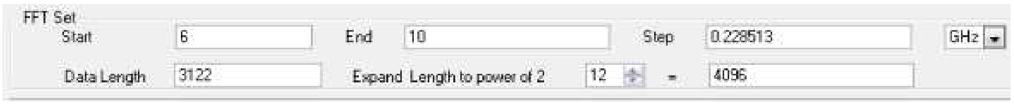


Click on the **Mode Gamma** button and then on the **Validate** button, the variables in the formula will be listed in the variable list.

Click on the **MultiLoad** button to assign the GEMS simulation results to the variables.

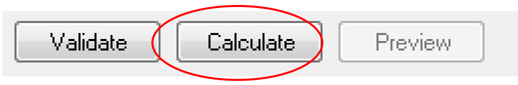


Click on the variable name in the **Name** list, and select an option in the result tree and double-click on it to assign it to the selected variable. Click on the **OK** button to return to the main window. Specify the frequency range.

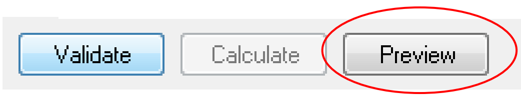


You can pad more zeros by increasing the number in the “Expand Length to power of 2” box.

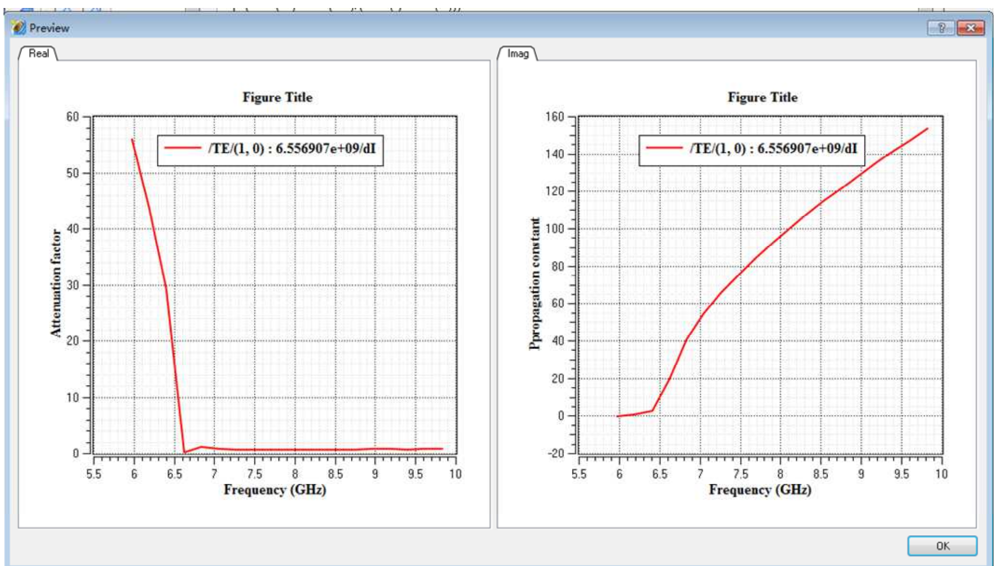
Click on the **Calculate** button to process the formula.



Click on the **Preview** button to view the results.

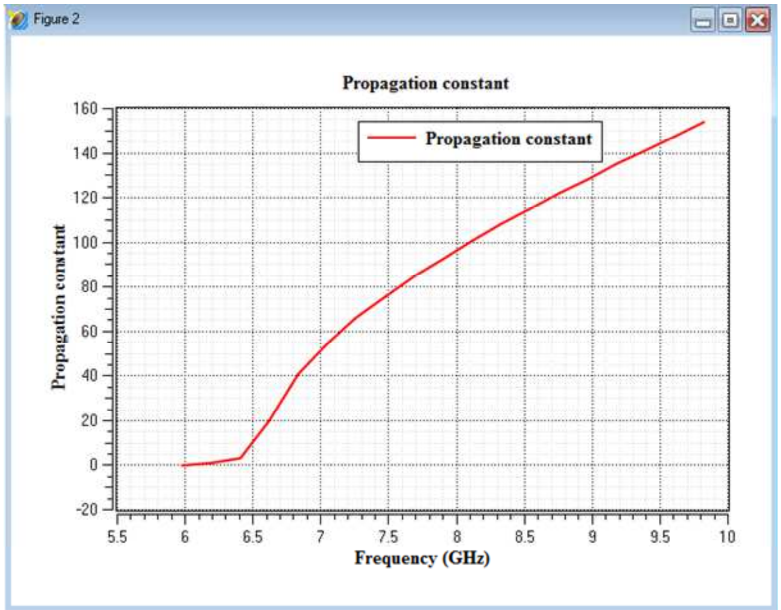


The attenuation factor and propagation constant are shown in the figure below:

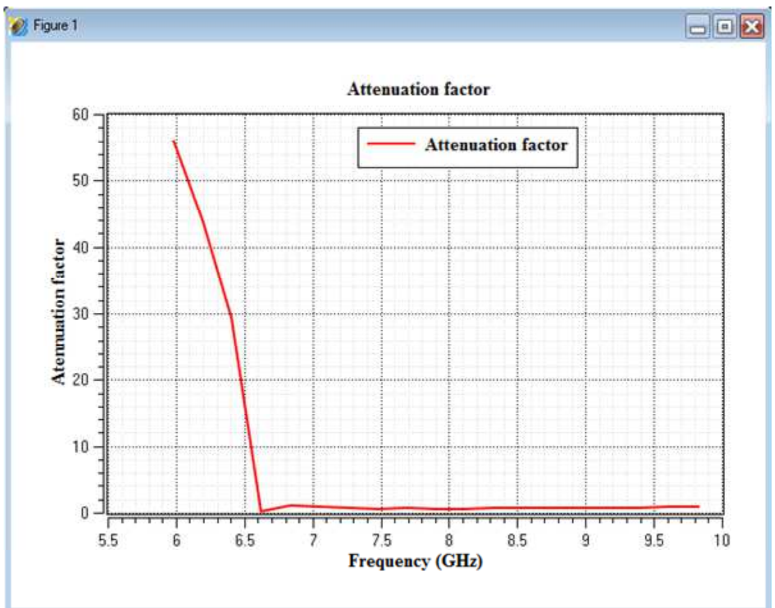


Click on the **Cartesian** button to plot the result in the **Display** window. Click on the margin of the figure to add the labels and title.

Propagation constant



Attenuation factor



8

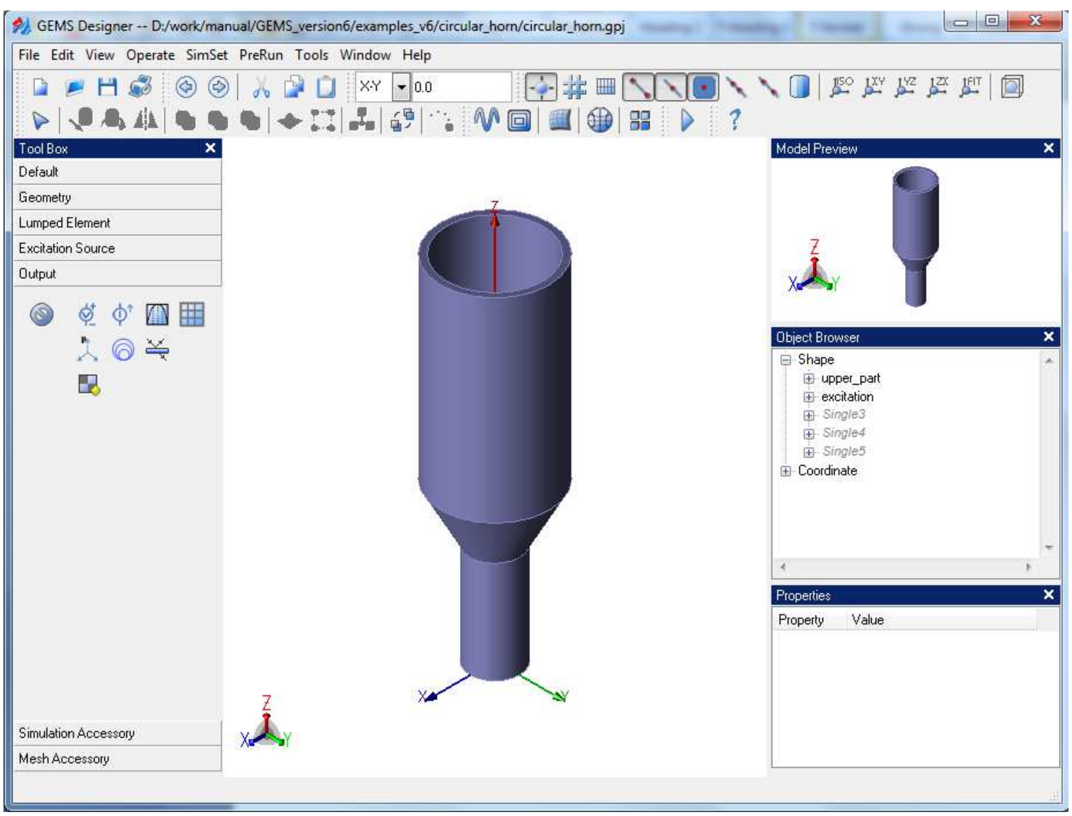
Example 8. Dual Mode Horn Antenna

Description: A circular horn antenna with three segments is excited by using the TE_{11} mode. The feed guide is infinitely long in one direction. The output parameters include return loss and far field pattern.

Keywords: Horn antenna, far field pattern, mode excitation, return loss, and match load.

8.1 Problem Configuration

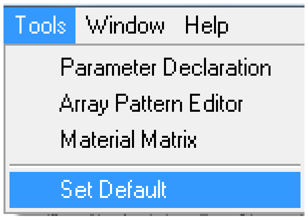
The horn antenna includes three parts: (i) the feed waveguide (infinitely long, radius=8.875mm); (ii) the waveguide transit (length=22mm); and (iii) circular guide (length=92.25mm and radius=23.25mm).

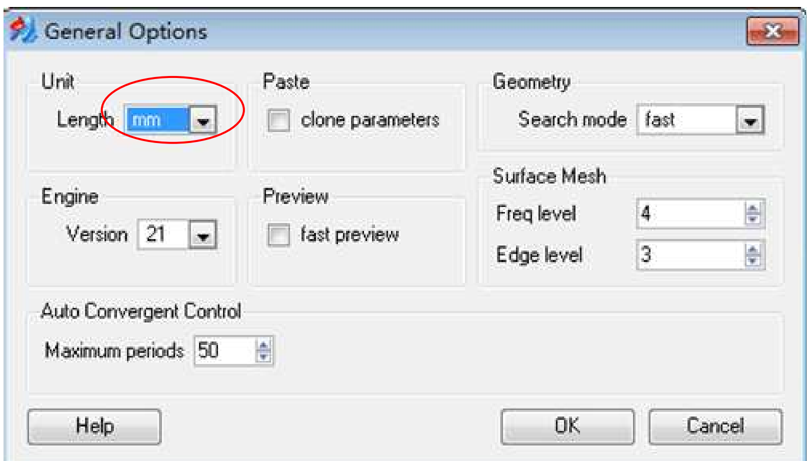


8.2 Create Project Model

Follow the steps below to create the horn antenna model:

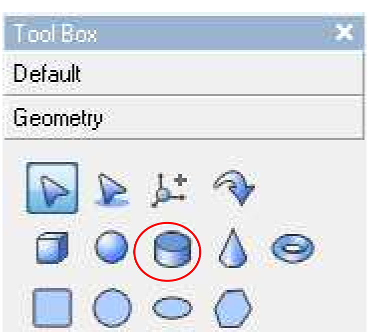
(1) Open GEMS Designer
(2) Select the **Set Default** option in the **Tools** menu

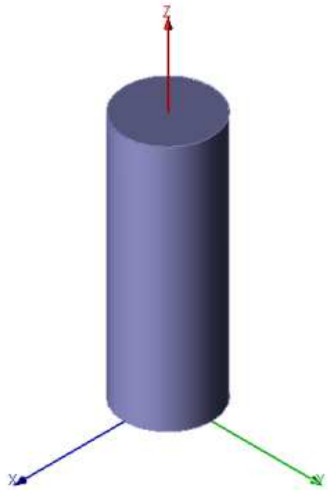




(3) Click on the **New** button in the toolbar to create a new project

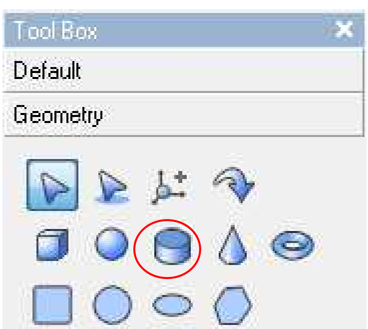
(4) Select the **Cylinder** icon in the **Tool Box->Geometry** box





Select the cylinder option in the **Object Browser** box, and change its name to “lower_part”. Select the “CreateCylinder” option in the cylinder folder in the **Object Browser** box, and change its center to (0, 0, 0), length to 50mm, and radius to 8.875mm.

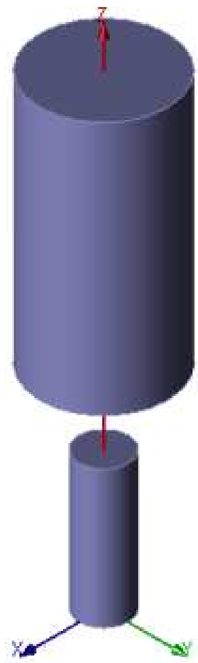
(5) Select the **Cylinder** icon in the **Tool Box->Geometry** box

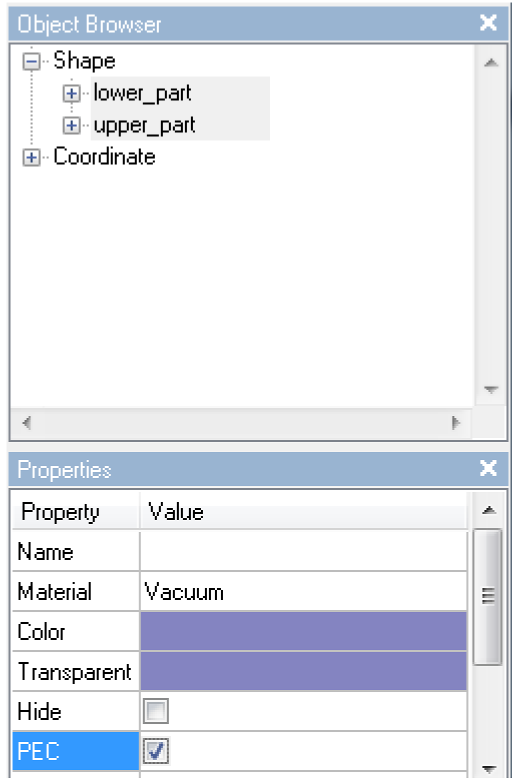


Set the drawing plane to z=77mm.

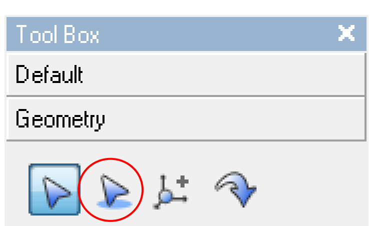
Draw a cylinder in the figure region. Select the cylinder option in the **Object Browser** box, and change its name to “upper_part”. Select the “CreateCylinder” option in the cylinder folder in the **Object Browser** box, and change its center to (0, 0, 77mm), length to 92.25mm, and radius to 23.25mm.

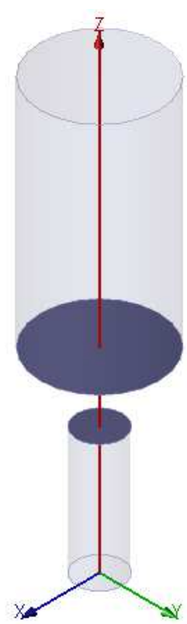
You will see the two cylinders in the figure region. Select the “upper_part” and “lower_part” in the **Object Browser** box and check the **PEC** box in the **Properties** box.



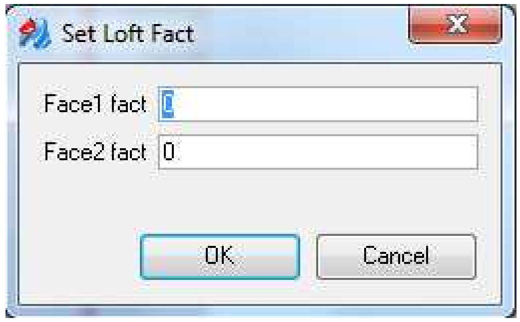


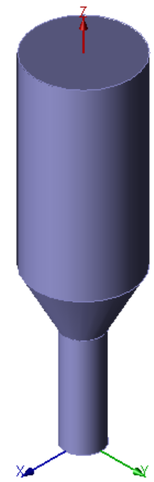
(6) Click on the **Select face** in the **Tool Box->Geometry** box and select two inner surfaces.



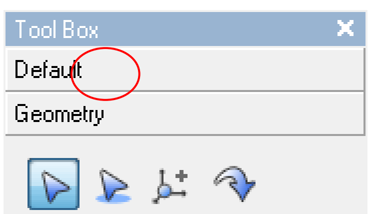


(7) Click on the **Loft** button in the **Operate** menu

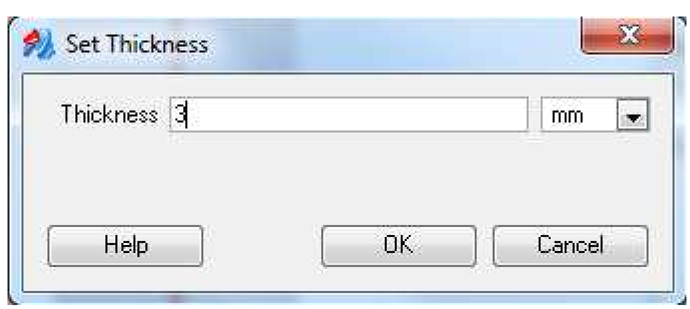


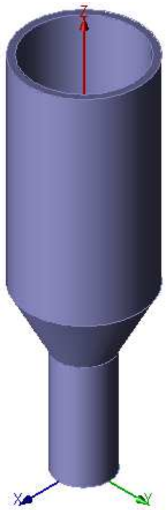


(8) Click on the **Select face** in the **Tool Box->Geometry** box and select two outer surfaces



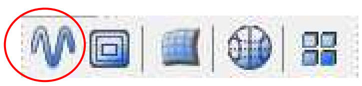
(9) Click on the **Shelling** button in the **Operate** menu. Type "3" in the **Thickness** box, and click on the **OK** button.



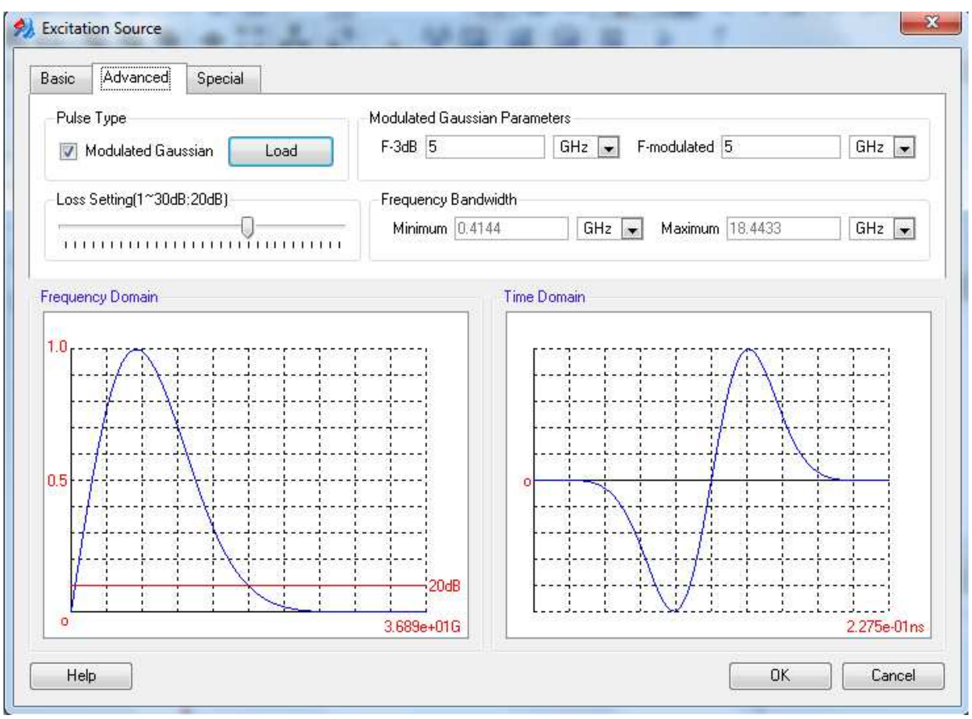


8.3 Set Excitation Pulse

Click on the **Set excitation pulse** in the toolbar.

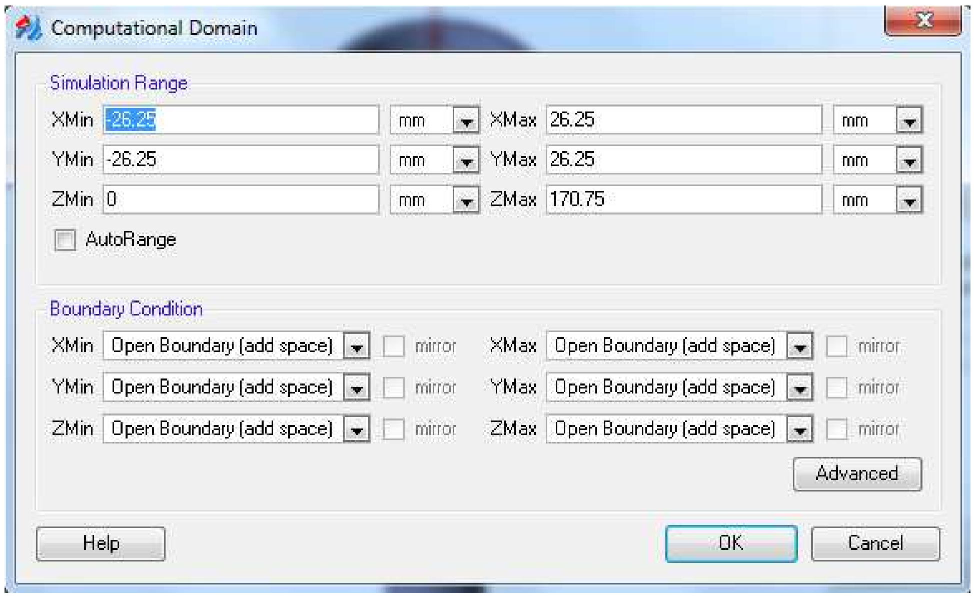


Type the number “15” in the **Maximum Working Frequency->Frequency** box.



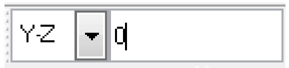
8.4 Set Domain and Boundary Condition

Click on the **Set boundary condition** in the toolbar.

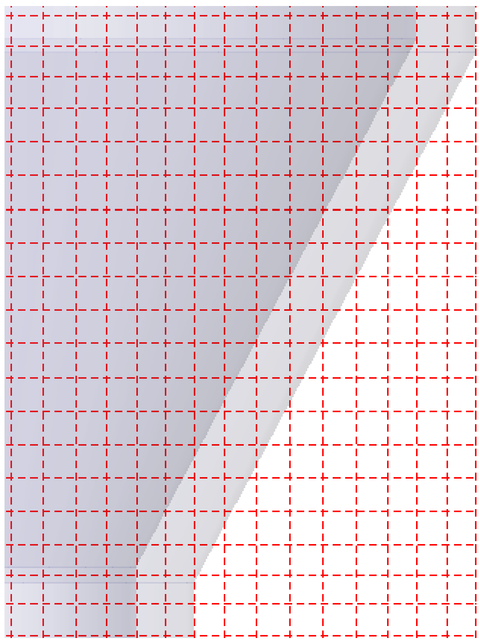


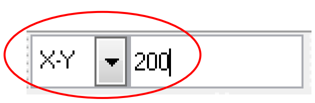
8.5 Design Mesh Distribution

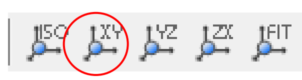
Click on the **Auto Mesh** in the toolbar.

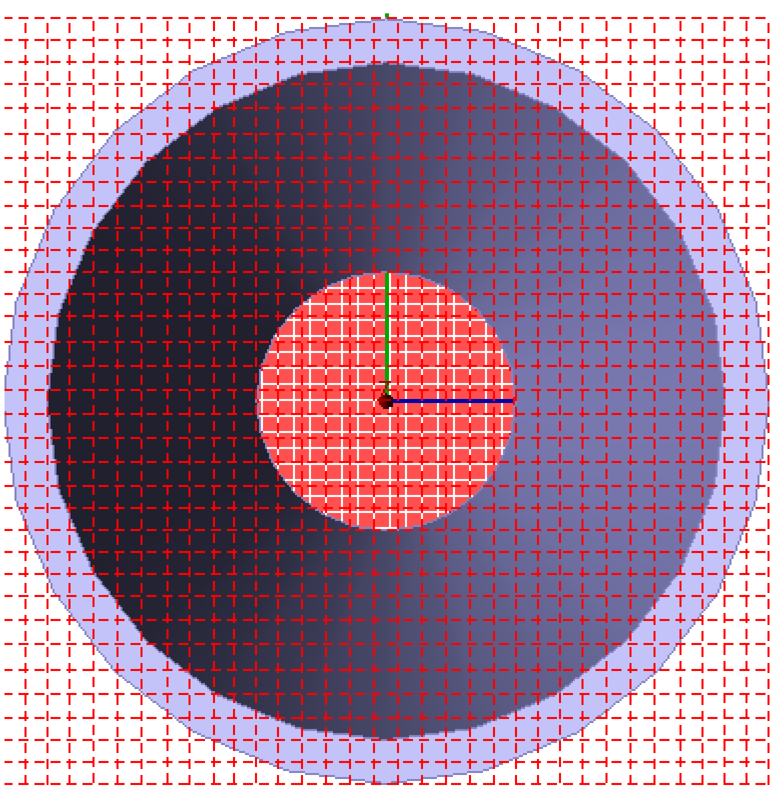

Y-Z
0


ISO
XY
YZ
ZX
FIT



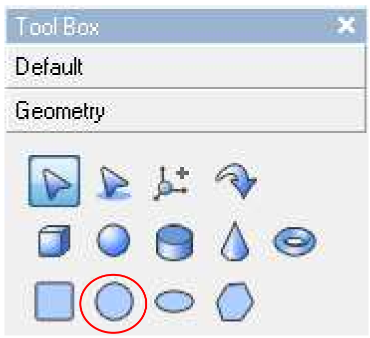

X-Y
200

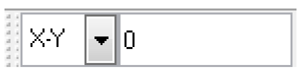

ISO
XY
YZ
ZX
FIT



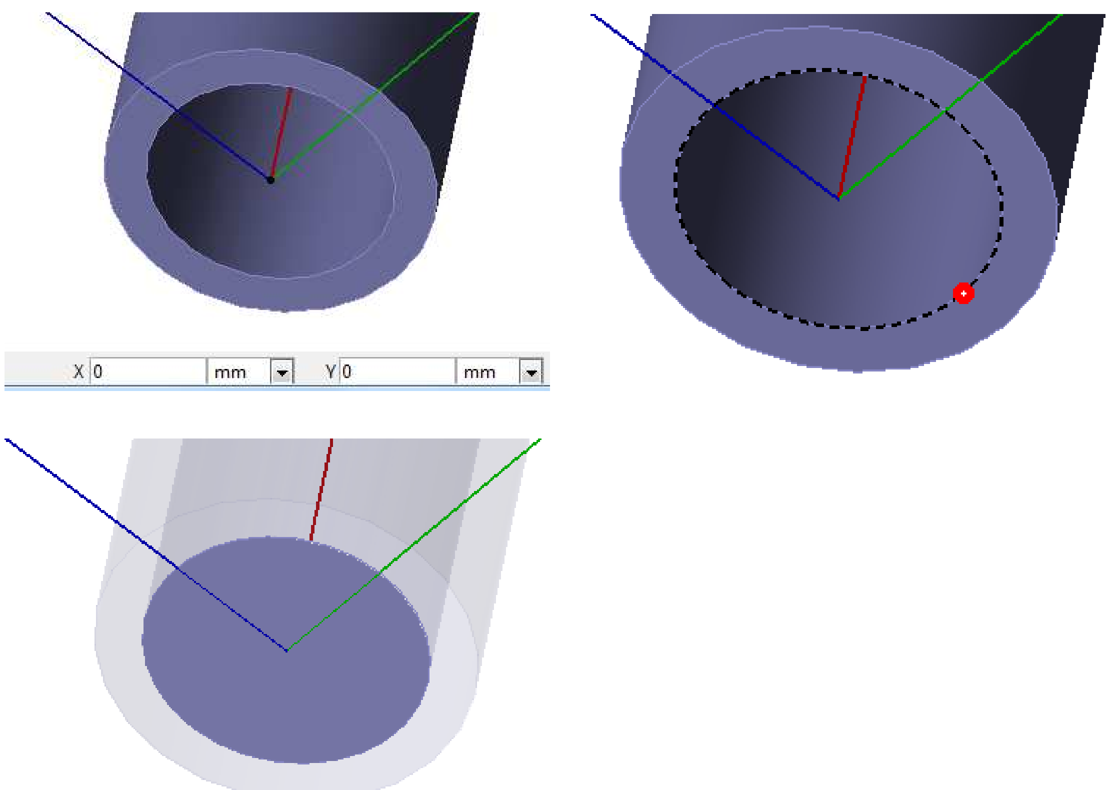
8.6 Add Excitation Source

Select the **Circle** icon in the **Tool Box->Geometry** box. Set the drawing plane to “X-Y” and the height to “0”.

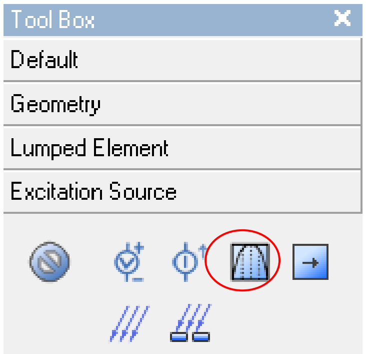




Select the origin point (x=0 and y=0) and then draw a circle (radius=8.875mm).

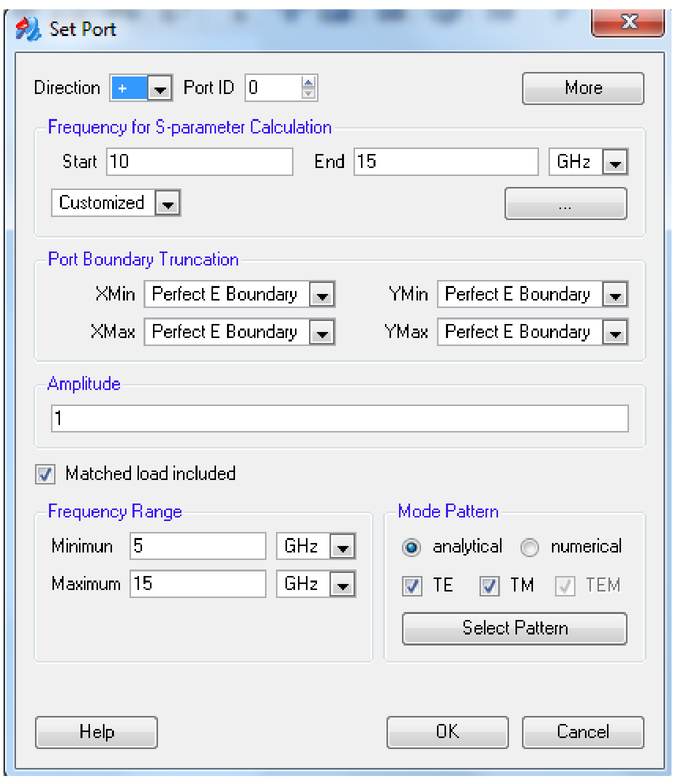


Select the excitation option in the **Object Browser** box and change its name to “excitation” in the **Properties** box. Click on the **Port** icon in the **Tool Box->Excitation Source** box.

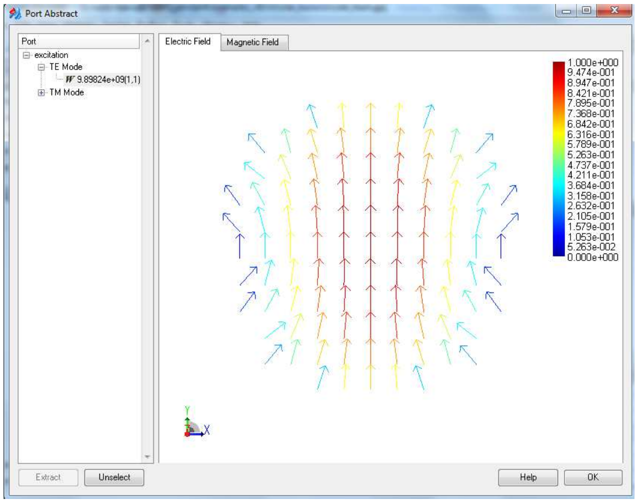


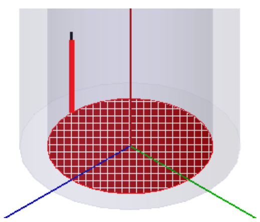
Select the "-" and "+" in the **Direction** box to ensure the direction of excitation port pointing to the power propagation direction.

Select the frequency band of interest from 8GHz to 15GHz. Check the **Matched load included** box since the feed guide cannot be truncated by the absorbing boundary of the domain and we need a closed Huygens' surface to calculate the 3D far field patterns.



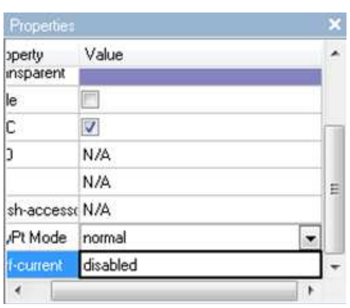
The TE_{11} mode of the circular waveguide is used to excite the horn antenna. Select the **analytic** radio button and the **TE** mode. Only when the mode of a guide cannot be extracted by using the analytic method, we will select the **numerical** mode. Click on the **Pattern** button to view the mode pattern.



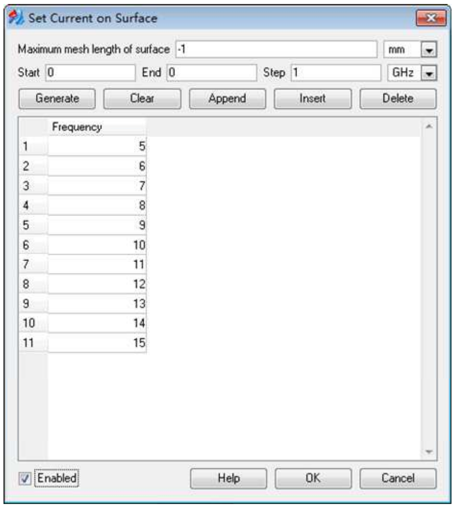


8.7 Set Surface Current Output

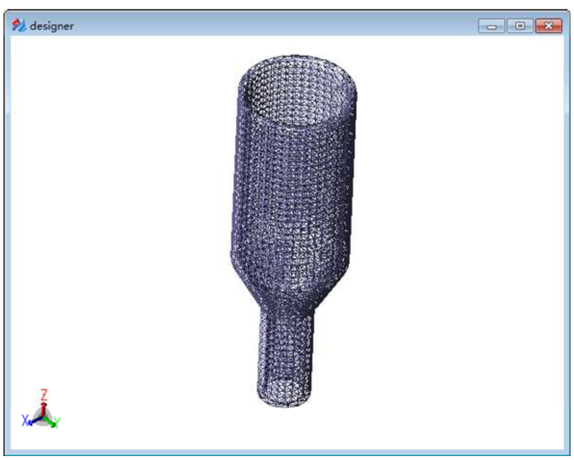
Select the horn (select the tree pieces if they are separated) structure in the figure region or horn option in the **Object Browser** box, click on “Disabled” in the “Sur_current” option in the **Properties** box.

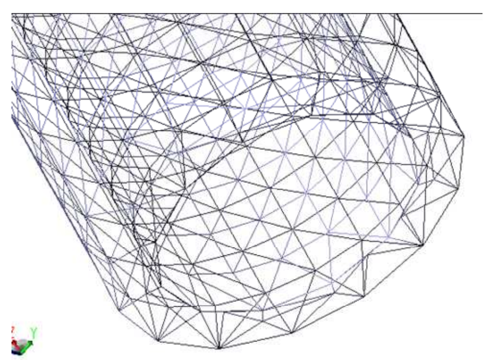


Check the “Enabled” box and then specify the frequency information in the popup window. The output options in the frequency domain such as surface current, far field pattern, field and current distributions on a specified plane and field distribution on a line share the same frequency list. If you change the frequency list for one output option, the frequency list will change for all the output options.



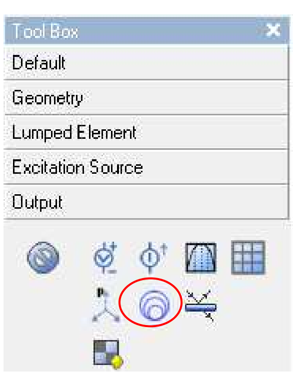
Select the horn structure in the figure region, and select the "Tools->Surface Mesh" option. The surface mesh will appear in the popup window. GEMS will generate the surface current distribution on the surface mesh.



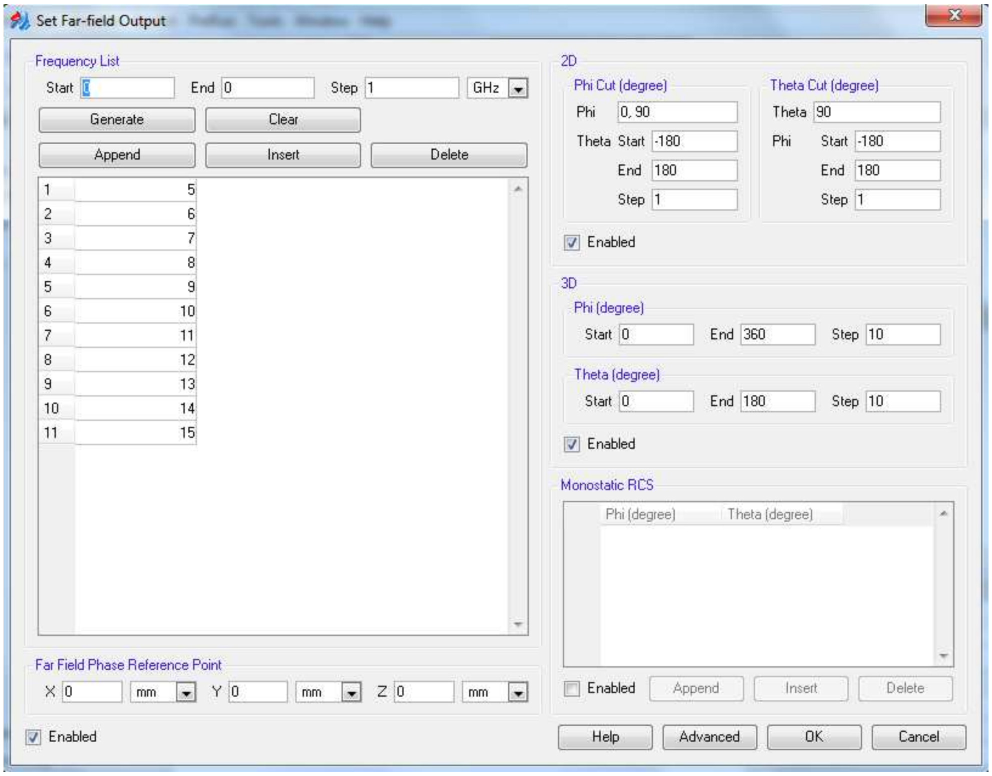


8.8 Set Far Field Output

Click on the **Far-field** icon in the **Tool Box->Output** box.

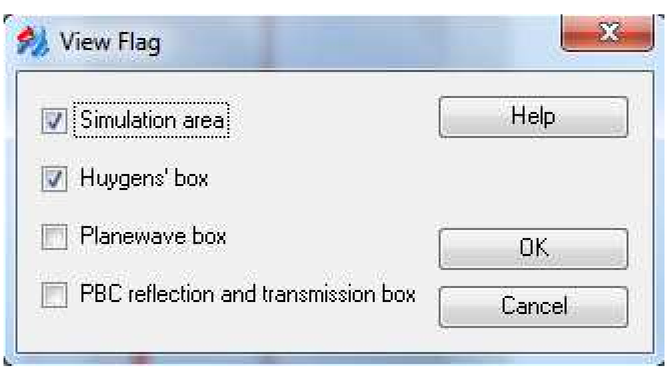


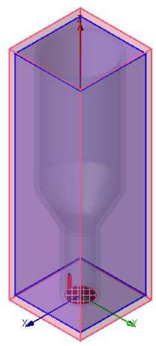
Check the **Enabled** box to set the far field parameters. Specify the frequency list for the far field pattern output. Check the **3D->Enabled** box to generate the 3D far field pattern.



To view the relative position of the simulation area and Huygens' surface, click on the **View defined box** in the toolbar.

Check the **Simulation area** and **Huygens' box** boxes. Click on the **OK** button to confirm the selection.



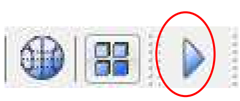


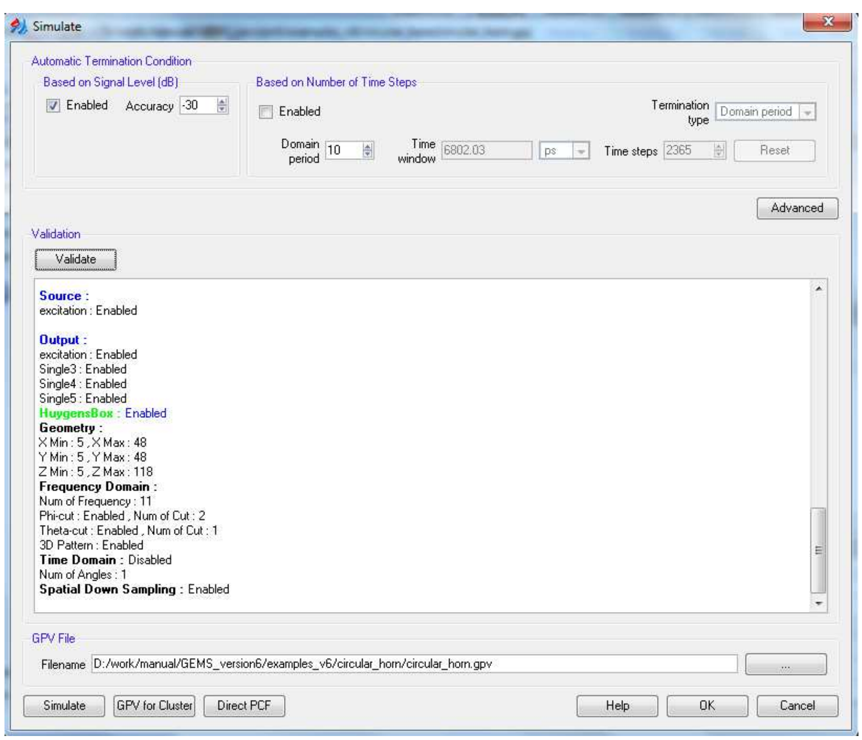
8.9 Save Project

Click on the **Save** button in the toolbar to save the project.

8.10 Generate Project File

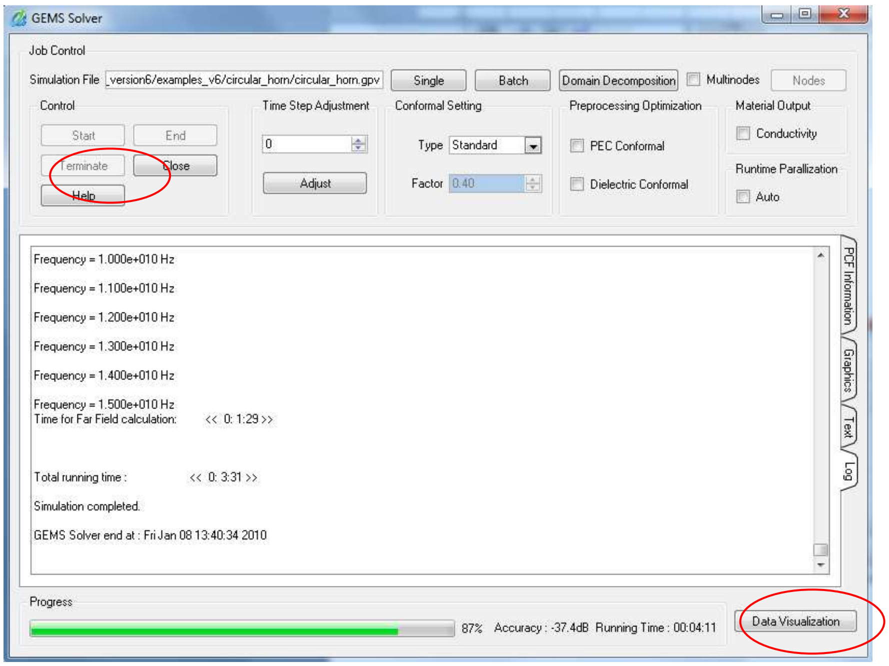
Click on the **PreCalculate** button in the toolbar. Click on the **Validate** and then on the **Simulate** button.





8.11 Simulate Project

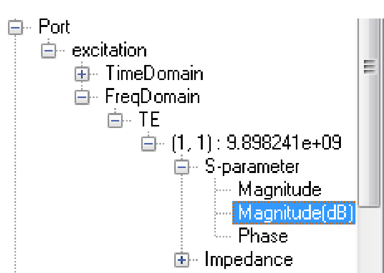
Click on the **Simulate** button in the *Simulate* window. Click on the **Start** button to start the simulation.

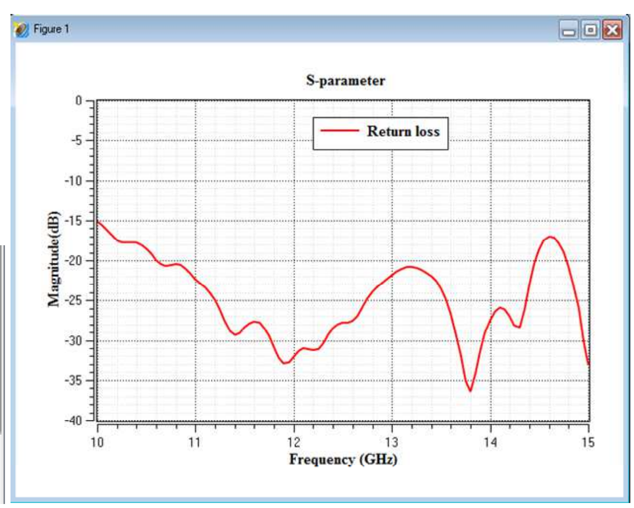


8.12 Result Visualization

Click on the **Data Visualization** button in the *GEMS Solver* window.

Click on the **Magnitude(dB)** of S-parameter option in the result tree to view it in the Cartesian system.

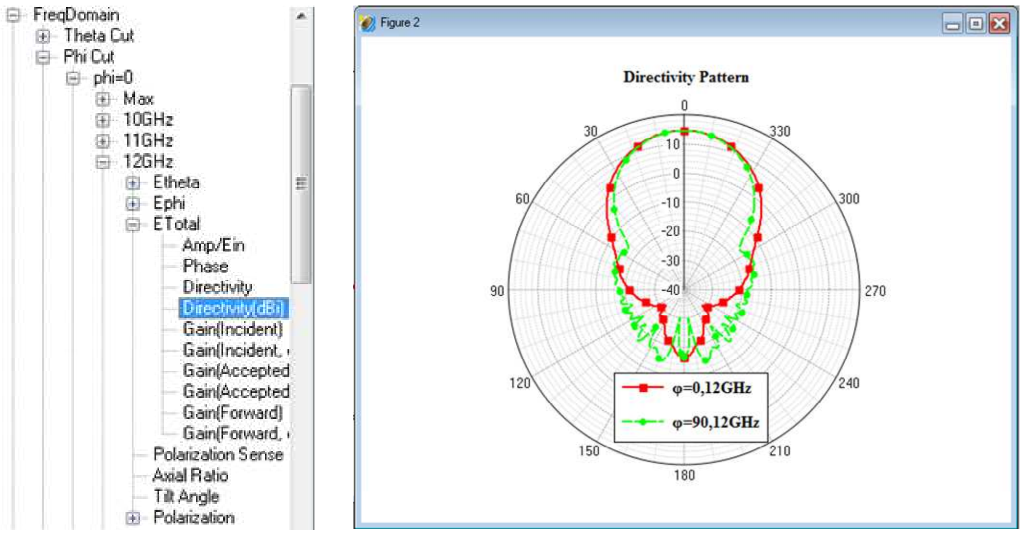




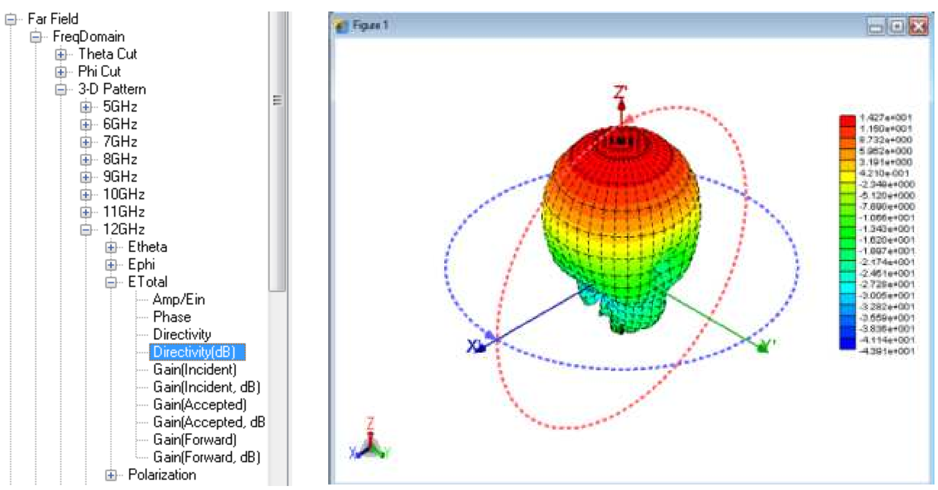
Follow the steps below to plot tow directivity pattern curves in the polar coordinate system:

(i) Select the “Directivity(dBi)” option in the “phi=0” subfolder and click the “View in polar” button in the toolbar.

(ii) Select the “Directivity(dBi)” option in the “phi=90” subfolder and click the “Add to current window” button in the toolbar to plot two curves in the same figure.

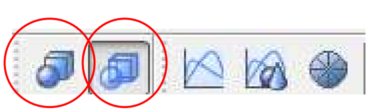


Double-click on the 3D directivity pattern option to view it.

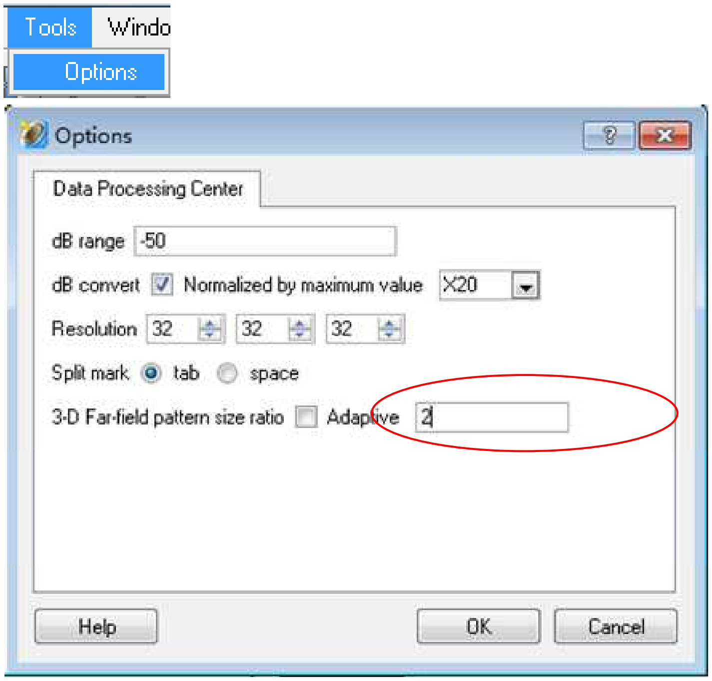


You can plot it with the project model following the steps below:

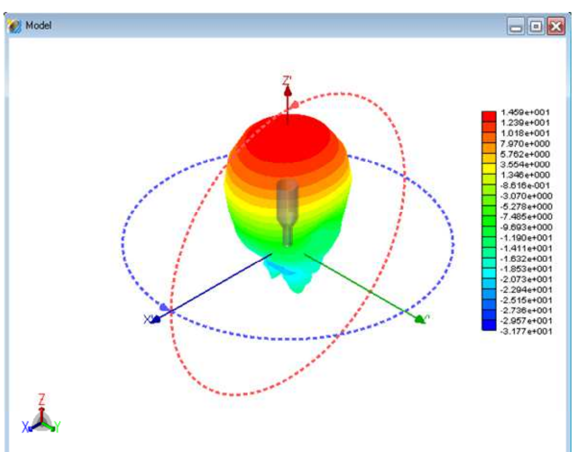
(i) Click on the “View model” button and “Transparent mode” button in the toolbar



(ii) Select the display ratio between far field and model to decide the relative size of the plot.

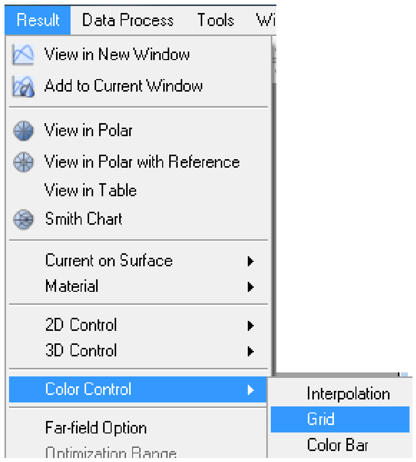


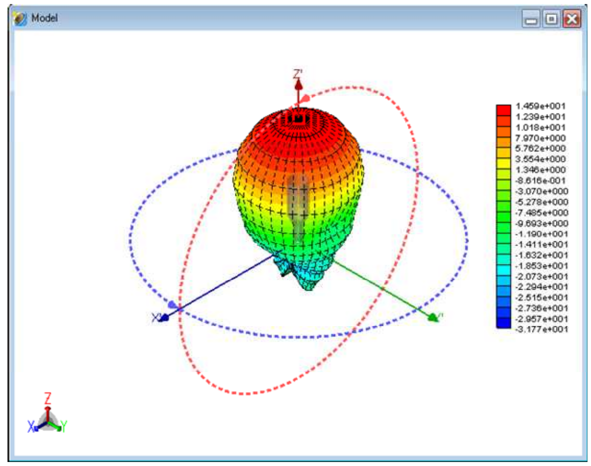
(iii) Click on the **ADD to current window** button in the toolbar, and then click the “Play” button in the toolbar to show the plot.



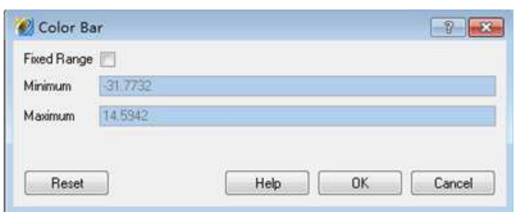
Three options in the **Result->Color Control** option control the display styles: (a) Interpolation: graduate color variation; (b) Grid: with grids; (c) Color Bar: specify the display range.

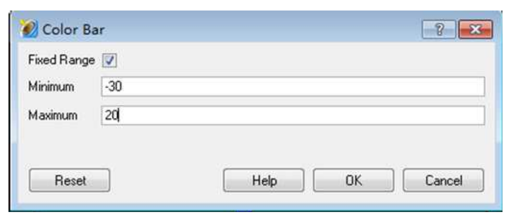
Select the **Color Bar** in the **Color Control** option in the **Result** menu. In the default case, the GEMS picks up the default range of the values in the data file.

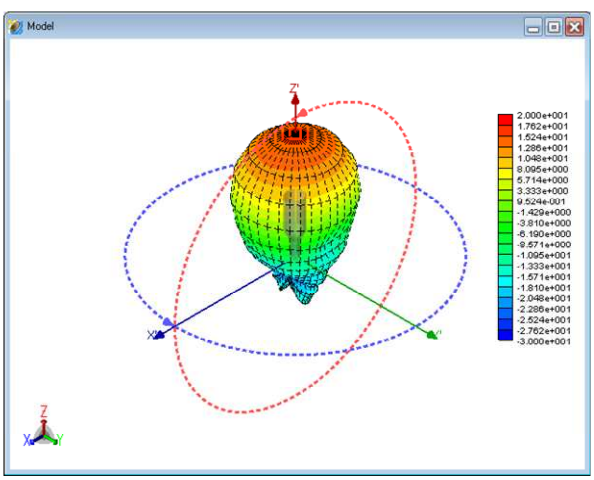




You can change the display range to the desired values. Check the “Fixed Range” box and adjust the minimum and maximum numbers that you like to display, and then click the “OK” button.





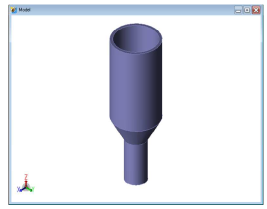


8.13 View Surface Current Distribution

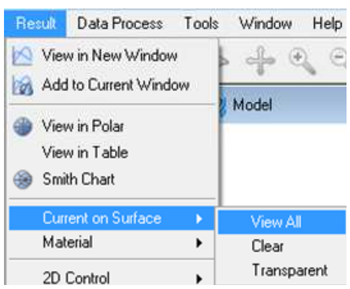
GEMS provides two ways to view the surface current distribution on the conductor surface:

(1) View the surface current distribution on the entire conductor structure you selected during the project design.

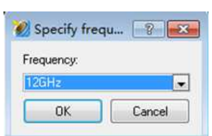
Step 1: Click the “View Model” button in the toolbar. Select the “Geometry” folder in the result tree, click the right mouse button and select the “Hide” option to hide the geometry.



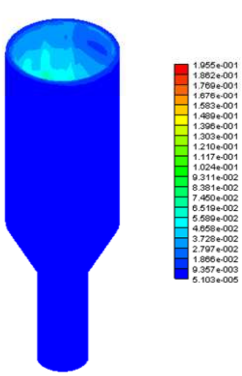
Step 2: Select the “View All” option in the “Result” menu.



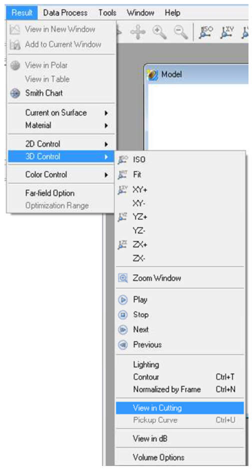
Step 3: Select a frequency in the frequency list.

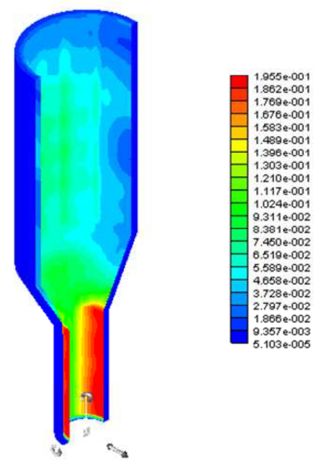


Step 4: Select a frequency in the frequency list. Click on the “Play” button in the toolbar.

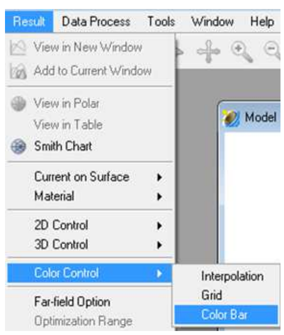


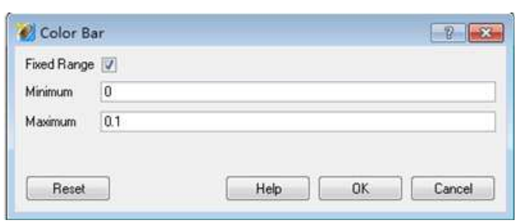
Step 5: Select a View in cutting" option in the "Result->3D Control" menu.

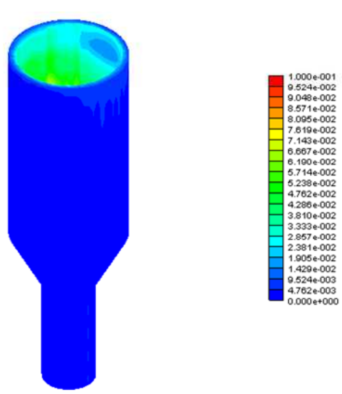


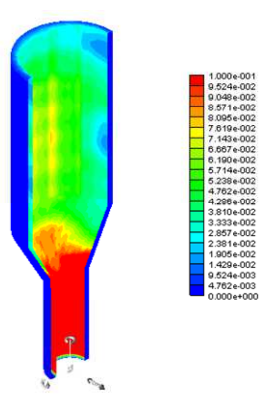


Step 6: Adjust the display range.





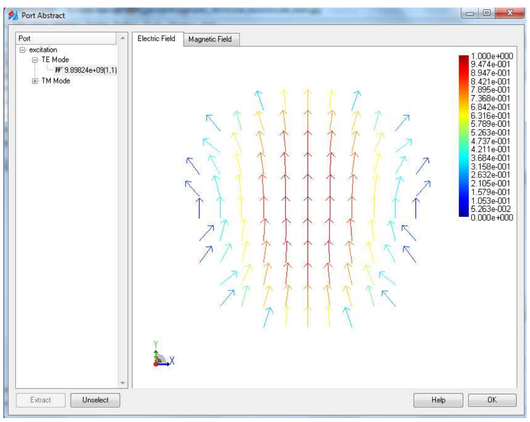




(2) View the surface current distribution on the individual conductor structure you selected during the project design. However, in this example, we only have a conductor that was united together in the simulation. If they are three pieces in the project, you can double-click on the frequency in the corresponding object in the "Output->Nearfield" folder in the result tree.

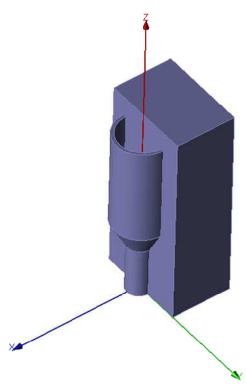
8.14 Symmetric Structure Simulation

It is evident that circular horn is rotationally symmetric. If the excitation is symmetric about one axis, we can reduce the simulation domain to be the half of the original problem size. The excitation pattern is symmetric about both the x- and y-axes. In this part, we consider the symmetric in the x-axis. Since the electric field is parallel to the x-minimum wall, we need to use the PMC (Perfect H boundary) at the x-minimum wall to ensure that the electric field in the imaging region is symmetric with that in the simulated region.

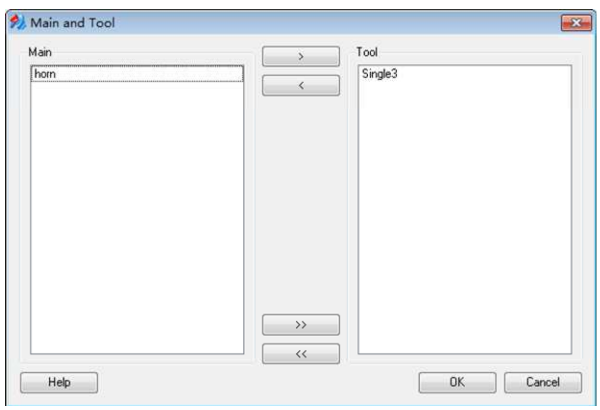


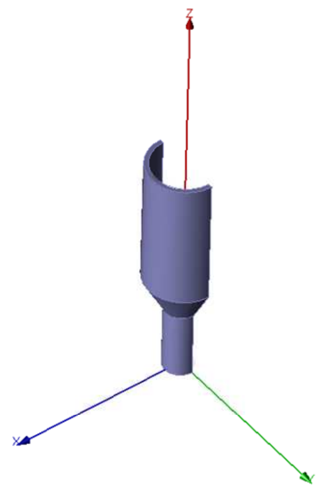
Based on the project you have for the complete geometry simulation, you can modify the project simulation options following the steps below:

(1) Model modifications
   (i) Draw a cuboid object inside the domain that can be used to erase the half of the horn structure.



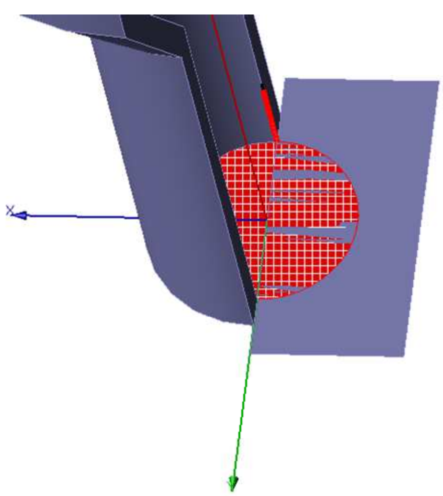
(ii) Use the horn to subtract the cuboid structure, and the leftover is the half horn that we need to simulate.

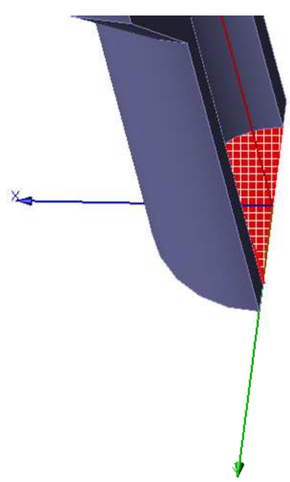




(2) Excitation port modifications

Follow the similar steps above to draw a rectangle that can be used to erase the half of the excitation circle. Use the excitation circle to subtract the rectangle and the leftover is the half excitation circle. However, after the mesh generation we need to extract the mode pattern again for the new feed structure.

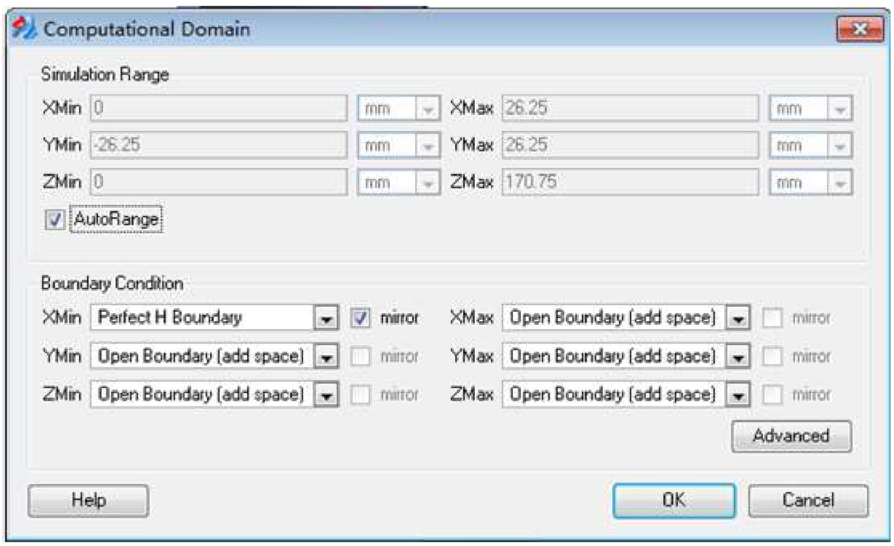




(3) Boundary and domain modifications

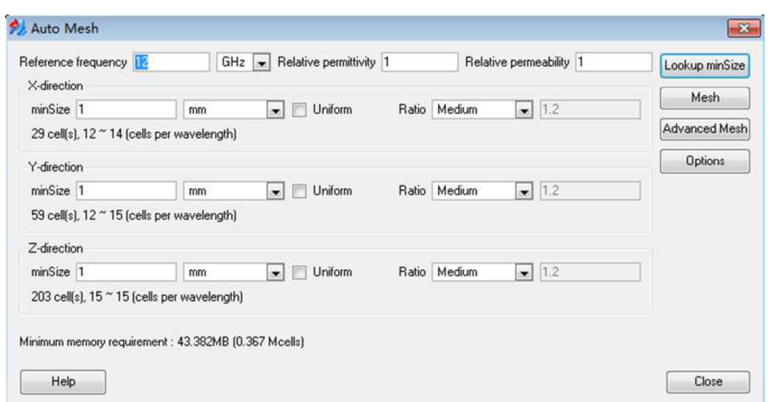
If the “AutoRange” box is checked, the domain size will be adjusted automatically. Otherwise, you may need to adjust the domain size manually.

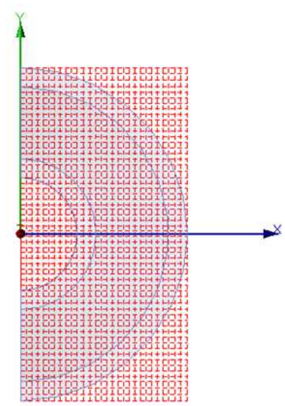
The boundary at the x-minimum wall should be changed to be “Perfect H Boundary” and the “mirror” box should be checked to generate the correct near and far fields.



(4) Redesign the mesh distribution

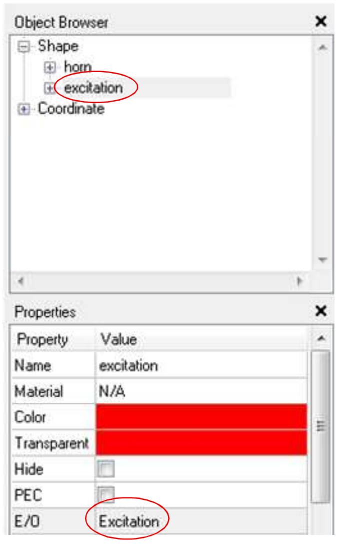
Use the same cell size and the number of cells in the x-direction will be reduced to be the half of the original number.

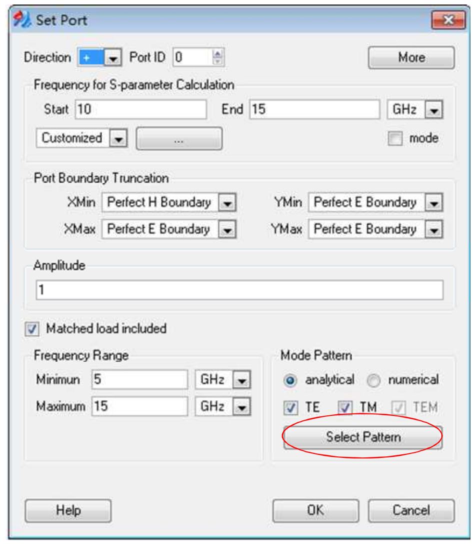




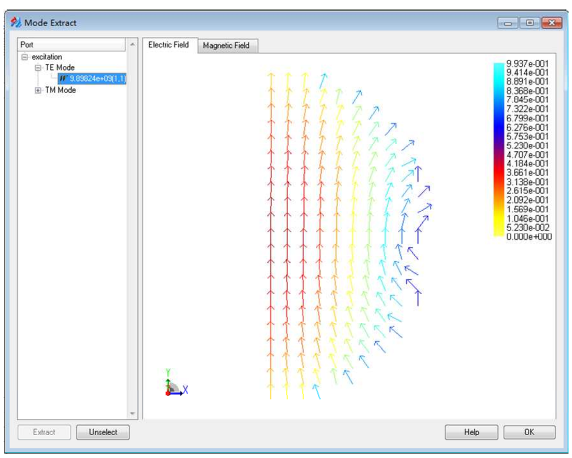
(5) Mode pattern extraction

Select the “excitation” option in the **Object Browser** box, and then click the “Excitation” option in the **Properties** box. Click the “Select pattern” button in the popup window.





Extract the modes supported by the port structure, and select the excitation mode in the mode list.

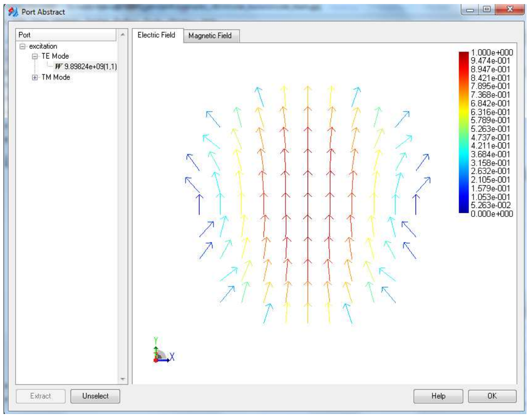


Follow the regular steps to simulate the problem, and you will get the similar results with the original problem.

8.15 Symmetric Structure Simulation

Since the circular horn is rotationally symmetric and the excitation is symmetric about both the x- and y-axes, we can reduce the simulation domain to be the quarter of the original problem size. In this part, we consider the symmetric in the x- and y-axes. Since the electric field is parallel to the x-minimum wall, we need to use the PMC (Perfect H boundary) at the x-minimum wall to ensure that the electric field in the imaging region is symmetric with that in the simulated region. Since the electric field is perpendicular to the y-minimum wall, we need to use the PEC (Perfect

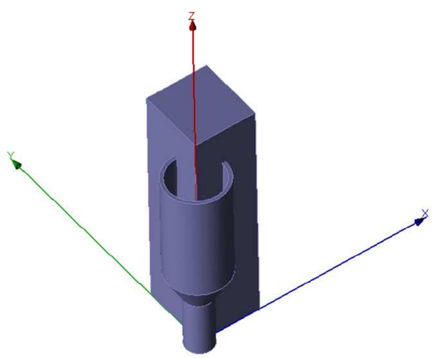
E boundary) at the y-minimum wall to ensure that the electric field in the imaging region is symmetric with that in the simulated region.



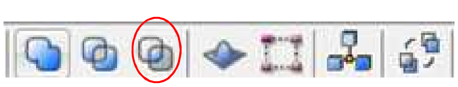
Based on the project you have for the complete geometry simulation, you can modify the project simulation options following the steps below:

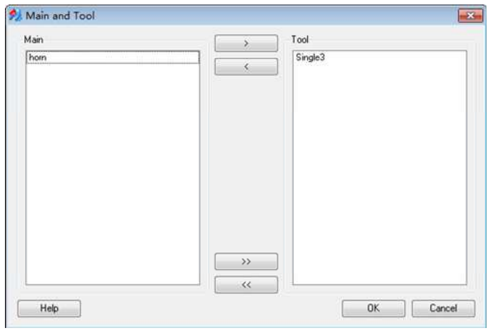
(1) Model modifications

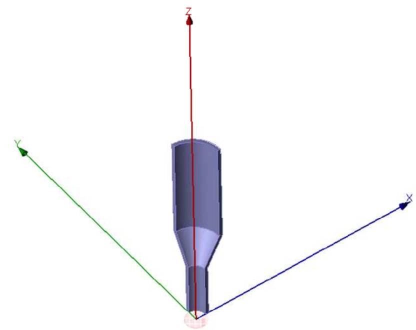
(i) Draw a cuboid object inside the domain that can be used to keep the quarter of the horn structure.



(ii) Use the “Intersect” option between the horn and the cuboid structure, and the common part is the quarter horn that we need to simulate.

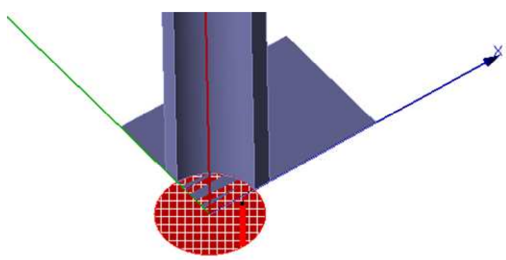


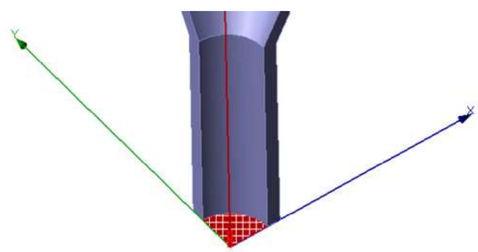




(2) Excitation port modifications

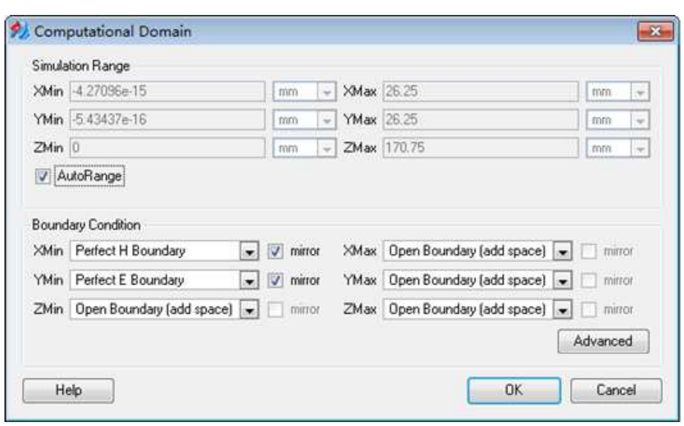
Follow the similar steps above to draw a rectangle that can be used to keep the quarter of the excitation circle. Use the excitation circle to intersect with the rectangle and the leftover is the quarter excitation circle. However, after the mesh generation we need to extract the mode pattern again for the new feed structure.





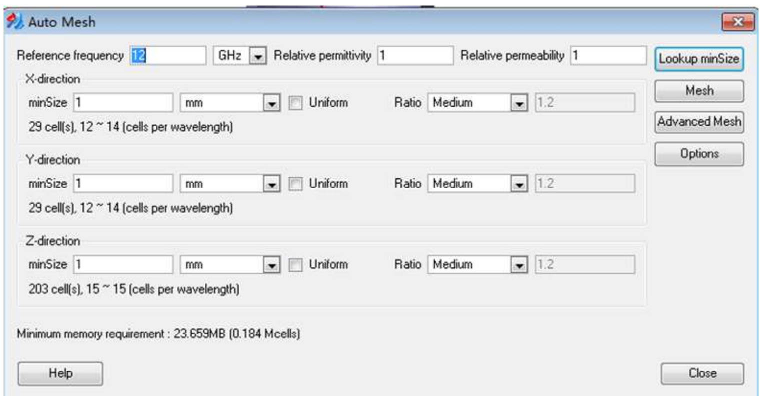
(3) Boundary and domain modifications

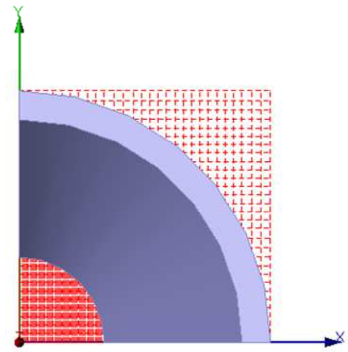
If the “AutoRange” box is checked, the domain size will be adjusted automatically. Otherwise, you may need to adjust the domain size manually. The boundary at the x-minimum wall should be changed to be “Perfect H Boundary” and the “mirror” box should be checked to generate the correct near and far fields. The boundary at the y-minimum wall should be changed to be “Perfect E Boundary” and the “mirror” box should be checked to generate the correct near and far fields.



(4) Redesign the mesh distribution

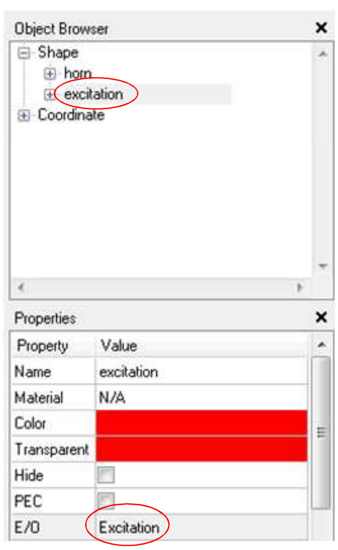
Use the same cell size and the number of cells in the x-direction will be reduced to be the half of the original number.

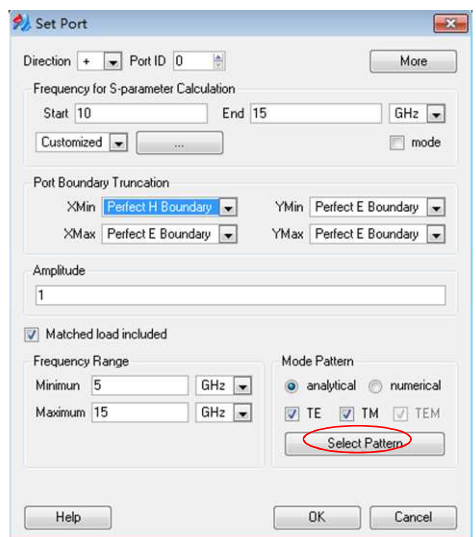




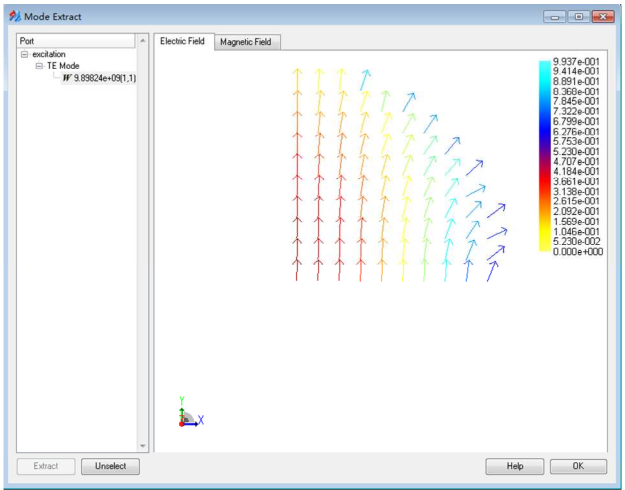
(5) Mode pattern extraction

Select the “excitation” option in the **Object Browser** box, and then click the “Excitation” option in the **Properties** box. Click the “Select pattern” button in the popup window.





Extract the modes supported by the port structure, and select the excitation mode in the mode list.



Follow the regular steps to simulate the problem, and you will get the similar results with the original problem.

9

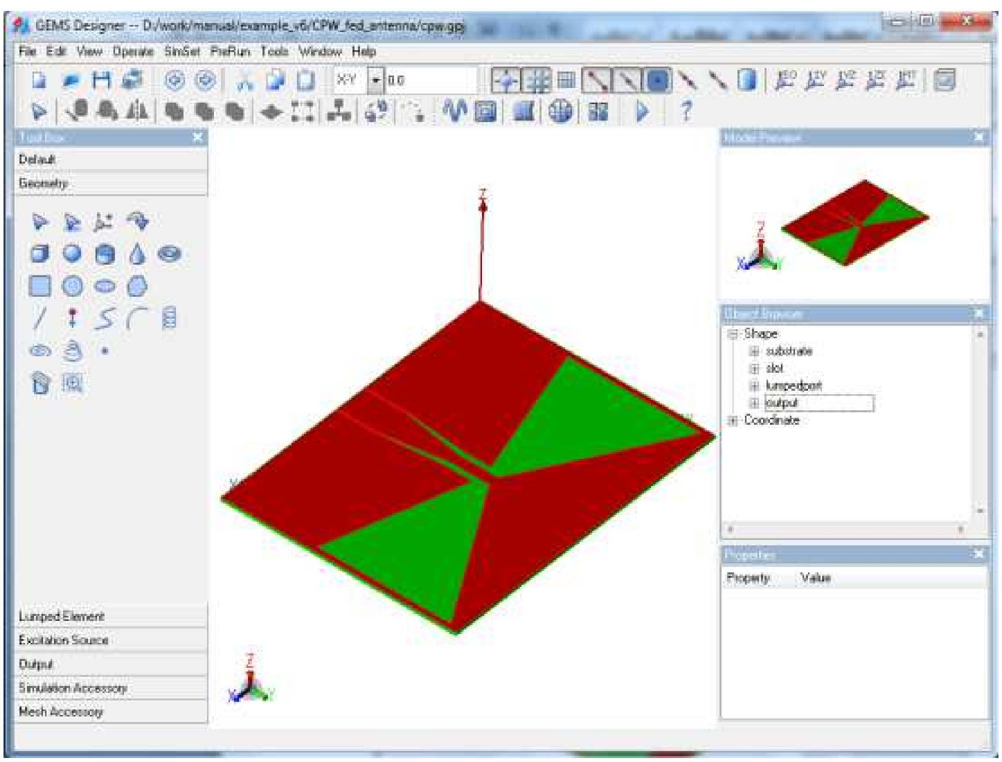
Example 9. Coplanar Feed Structure

Description: A slot antenna fed by a coplanar waveguide structure (CPW) is excited by using a lumped port. Using the symmetric property the problem size is reduced to half. The output parameters include return loss and far field pattern.

Keywords: CPW, DXF, loss tangent, return loss, far field pattern, radiation efficiency, symmetric structure, and lumped port.

9.1 Problem Configuration

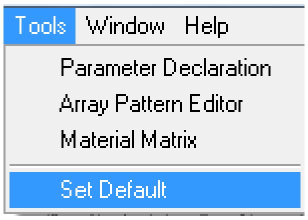
A slot antenna is fed by the CPW structure. Both the substrate (length=47mm, width=36mm and thickness=0.508mm; dielectric constant=3.38 and loss tangent=0.0027) and slot antenna structure are finite in the horizontal directions.

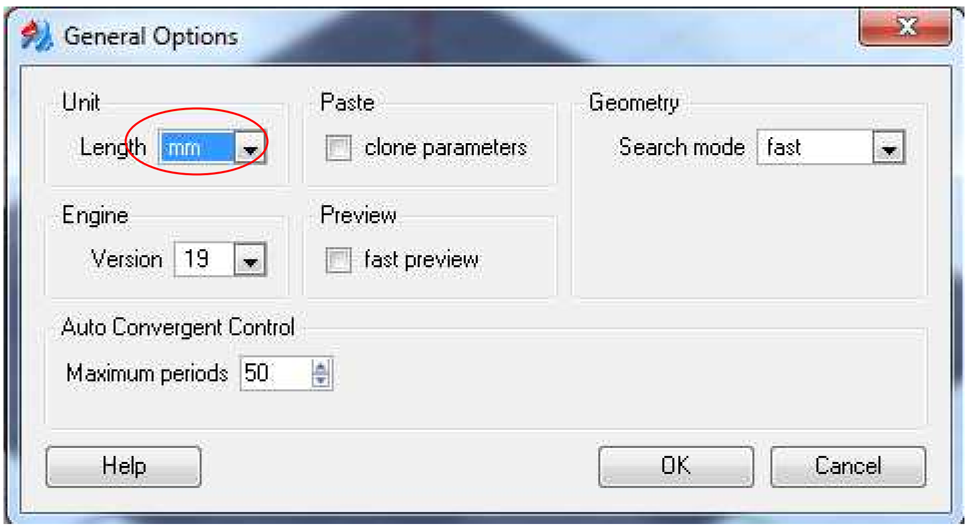


9.2 Create Project Model

Follow the steps below to create the slot antenna model:

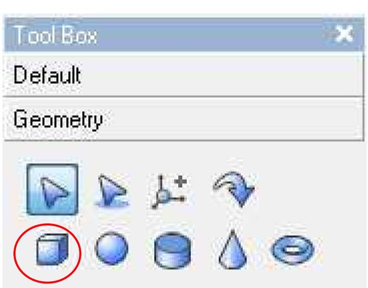
(1) Open GEMS Designer
(2) Select the **Set Default** option in the **Tools** menu





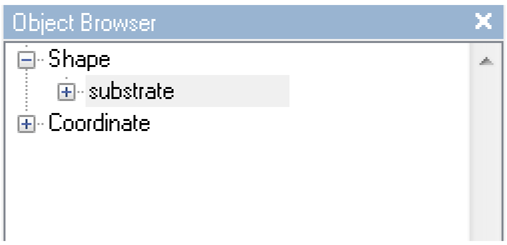
(3) Click on the **New** button in the toolbar to create a new project

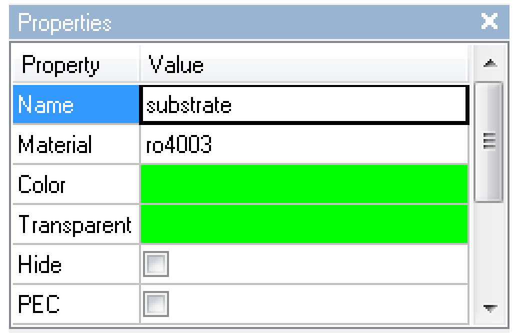
(4) Select the **Cuboid** icon in the **Tool Box->Geometry** box

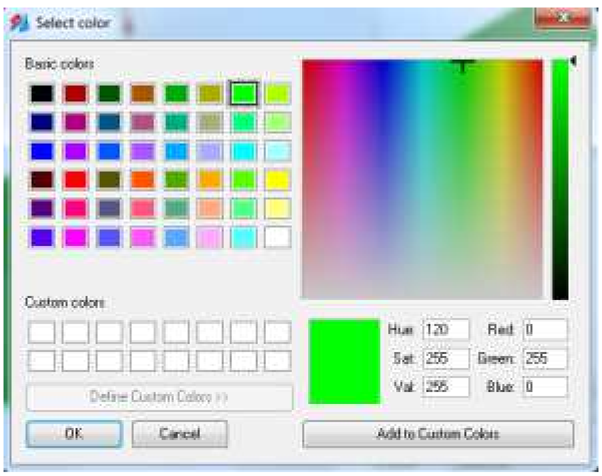


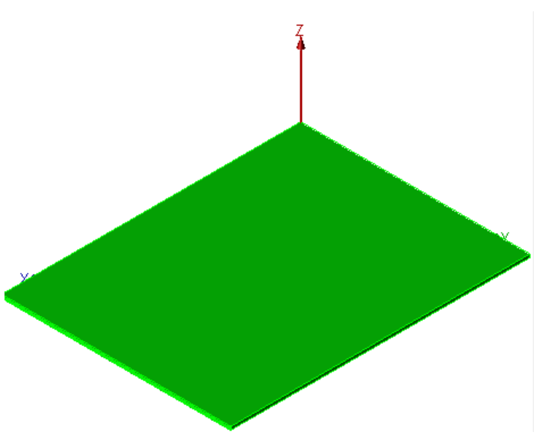
Select the object option in the **Object Browser** box and change its name to “substrate” in the **Properties** box. Change its color to green using the color panel opened by clicking on the **Color** box in the **Properties** box.

Select the “CreateRectangle” option under the **substrate** entrance in the **Object Browser** box, and change its width = 36mm, length=47mm, and thickness=0.508mm.

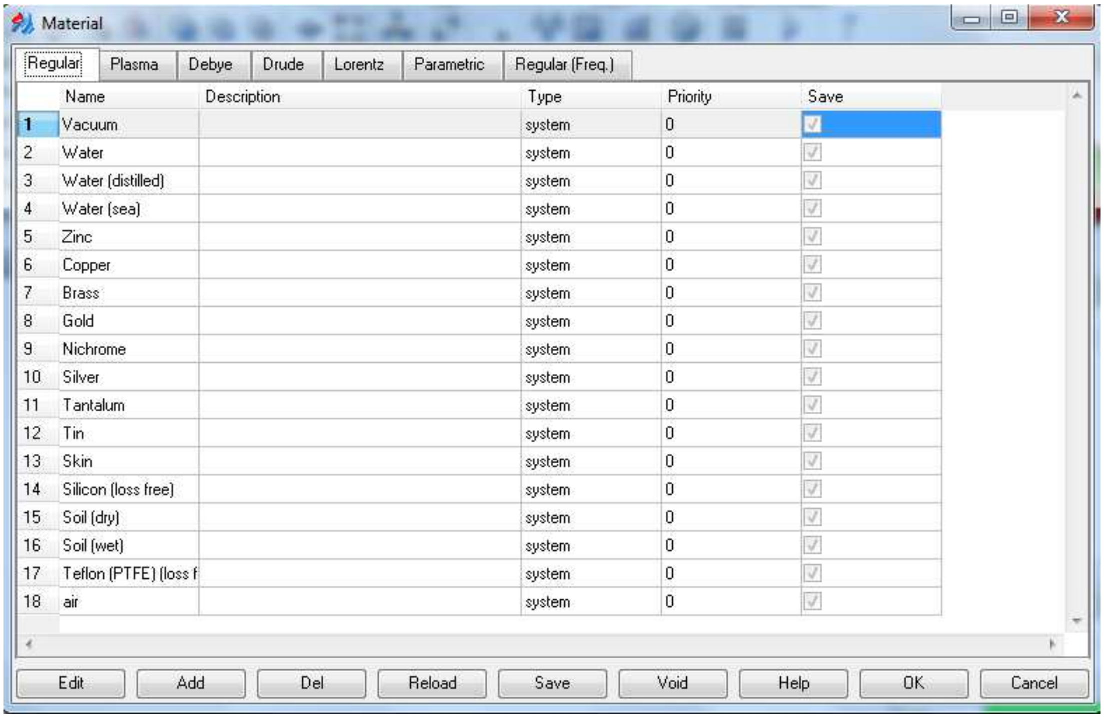


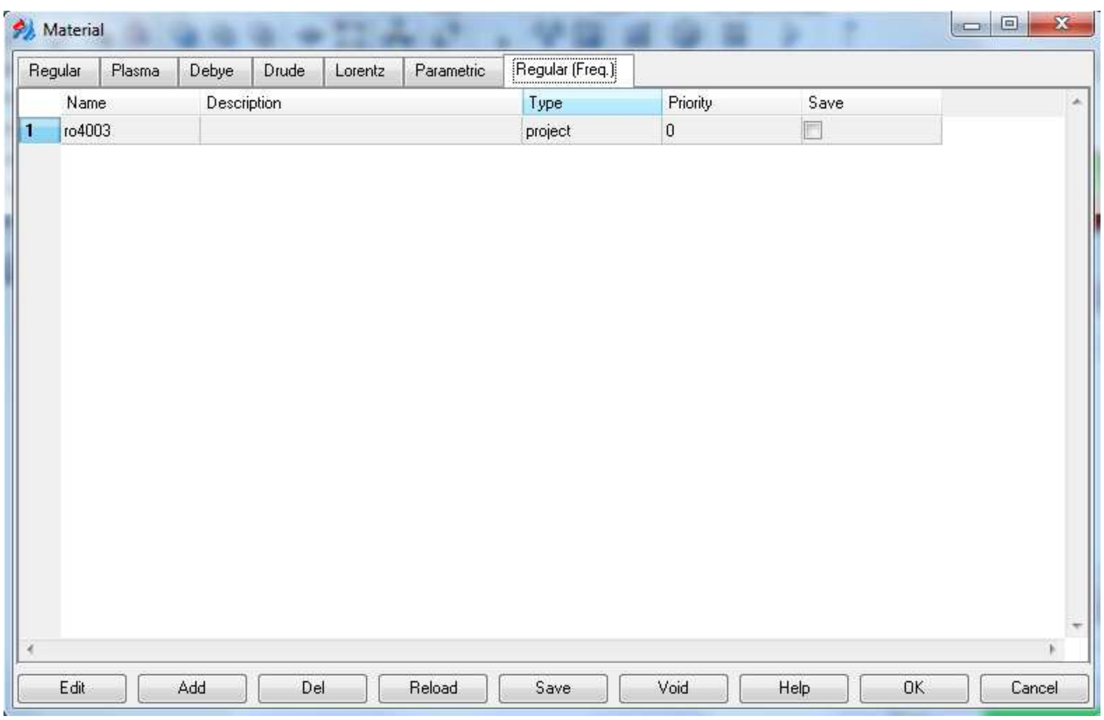




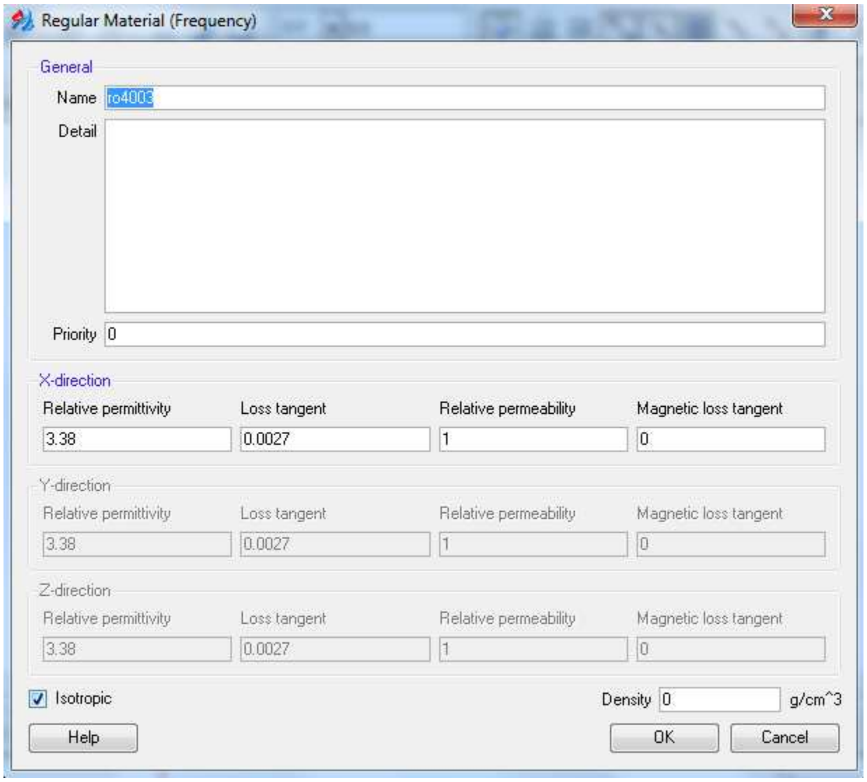


Select the **substrate** option in the **Object Browser** box, and click on the **Material** box in the **Properties** box. Click on the **Regular (Freq.)** (loss tangent) label and then on the **Add** button in the next window.



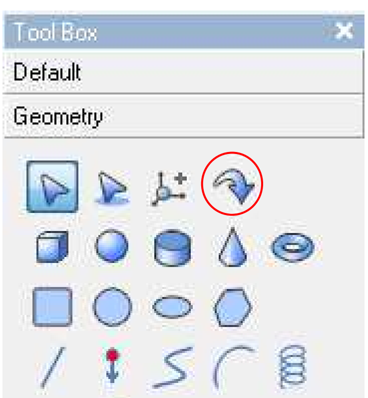


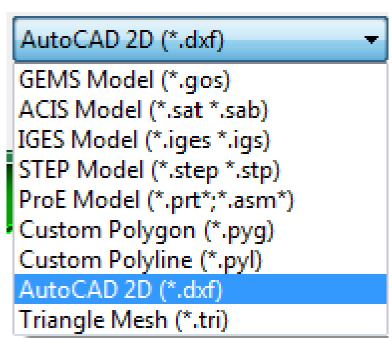
Specify the name and input the material parameters. Click on the **OK** button to confirm the parameter setting.



The slot antenna is in the DXF format (DXF 2004). To import the DXF file into the interface, click on the **Import model** icon in the **Tool Box->Geometry** box.

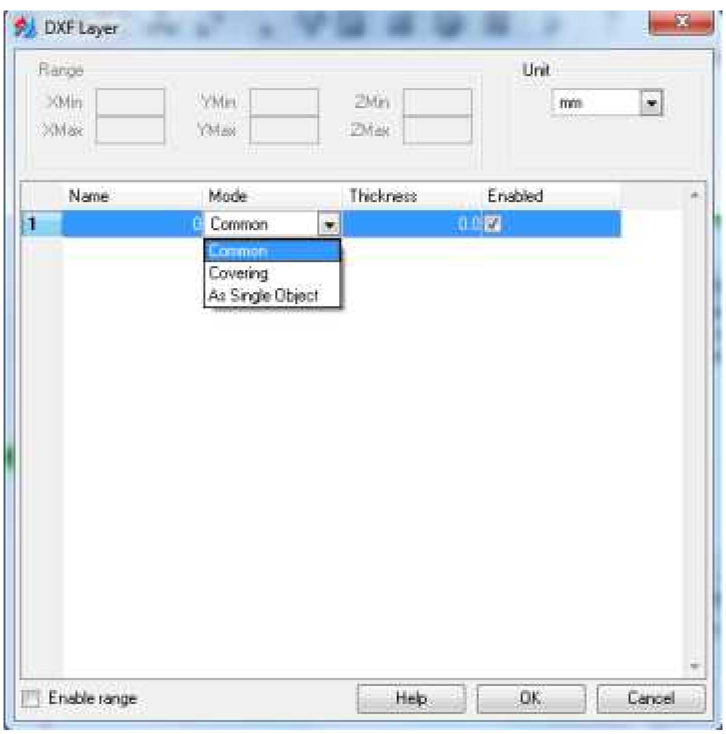
Select the **AutoCAD 2D (*.dxf)** option in the file style list.



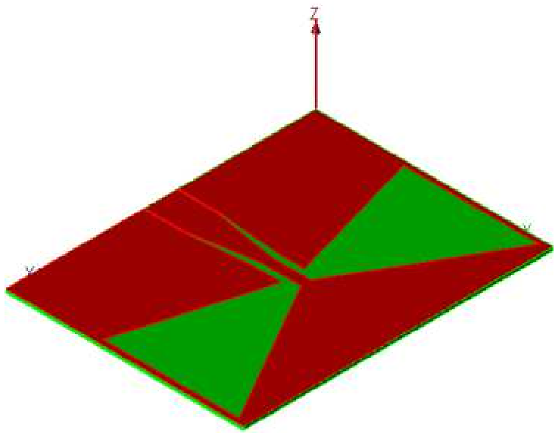


Search for the “slot.dxf” file, and double-click on it or click on the **Open** button.

*There are 3 options in the **Mode** box, namely, **Common**, **Covering**, and **As Single Object**. The **Common** option indicates that GEMS will load the original design regardless of the drawing types in the **AutoCAD**. The **Covering** option will fill-in the polygonal lines as a solid polygon (plate). The **As Single Object** option will treat the separated objects as a single one.*

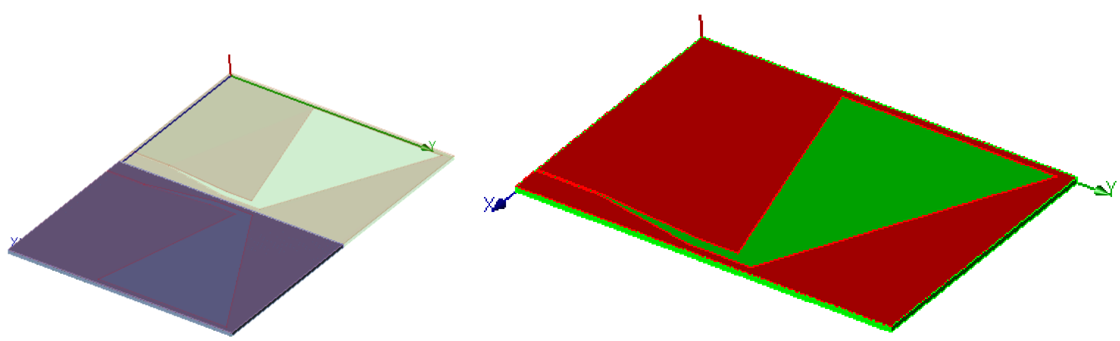


Select the **Common** option in the **Mode** column.



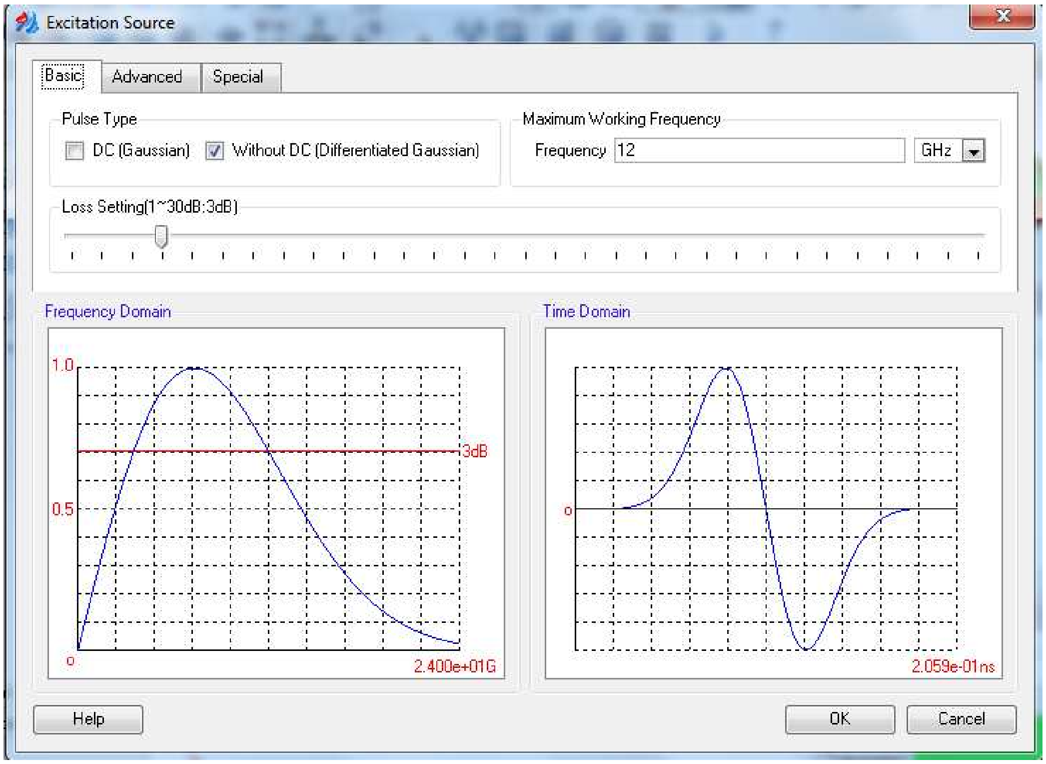
Select the **slot** entrance in the **Object Browser** box, and check the **PEC** box in the **Properties** box.

Since the structure is symmetric, we only need to simulate half geometry. Draw a box with the half size of the substrate, and use the slot and substrate to subtract the box.



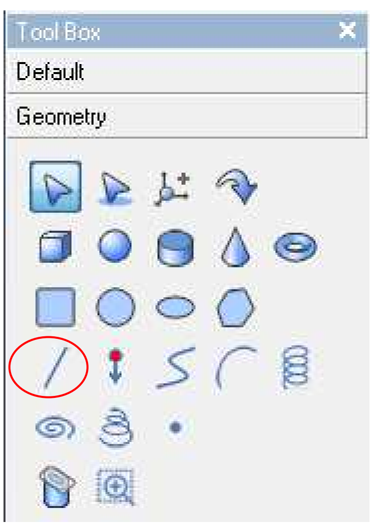
9.3 Set Excitation Pulse

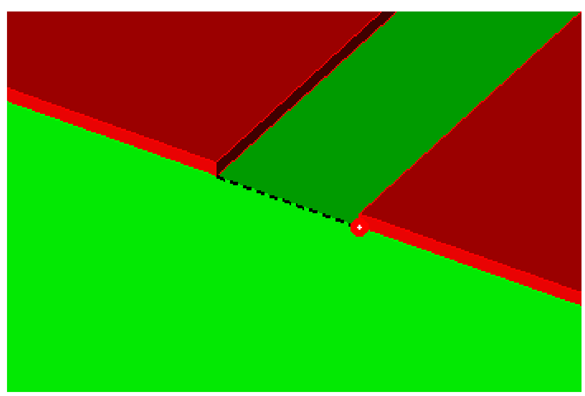
Click on the **Set excitation source** button in the toolbar.



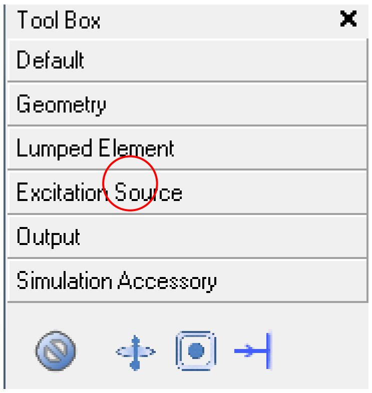
9.4 Set Excitation Source

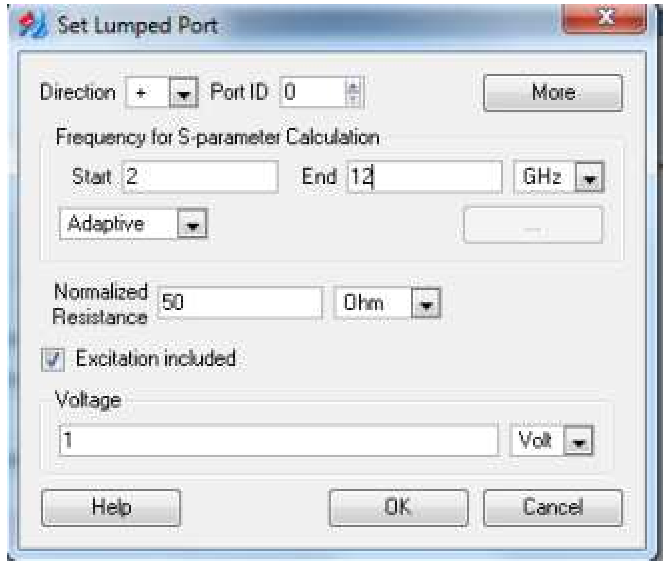
Click on the **Line** icon in the **Tool Box->Geometry** box. Draw a line inside the feed gap.

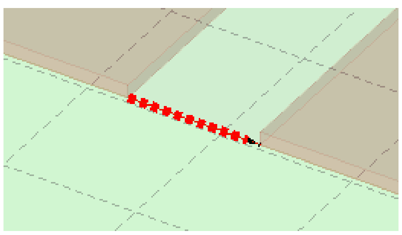




Select the line option in the **Object Browser** box and change its name to “excitation” in the **Properties** box. Click on the **Lumped port** icon in the **Tool Box->Simulation Accessory** box.

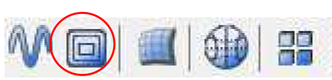


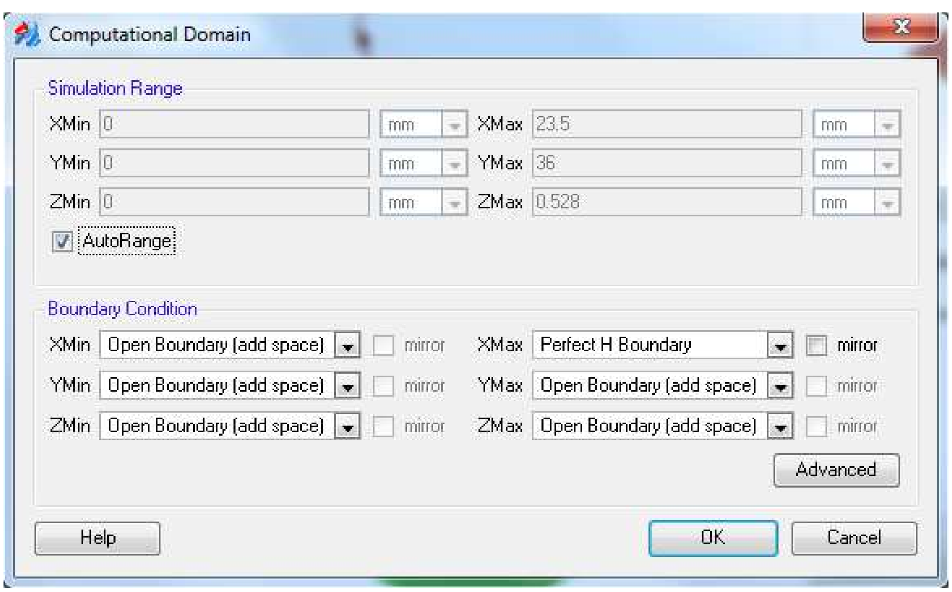




9.5 Set Domain and Boundary Condition

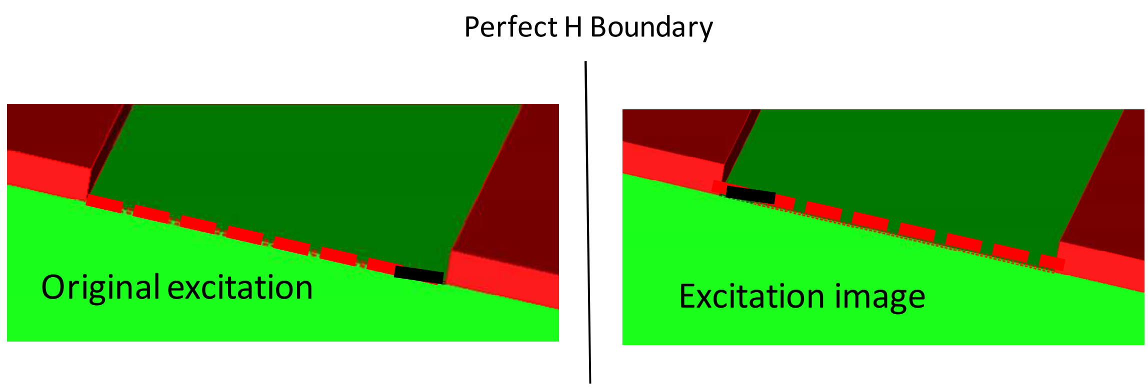
Click on the **Set boundary condition** button in the toolbar.





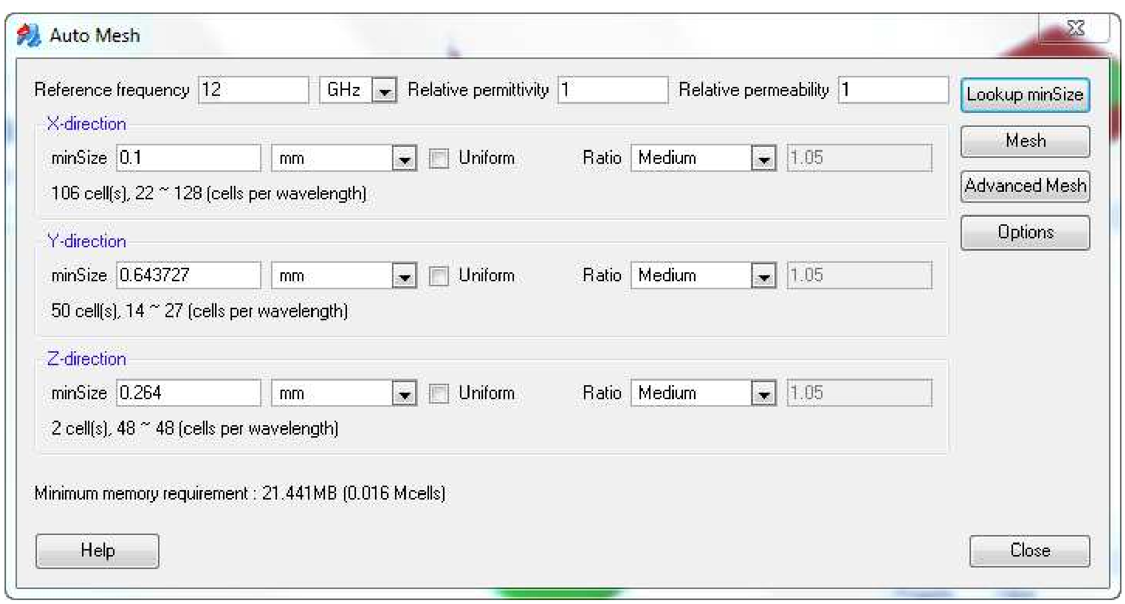
Considering the symmetric property, the boundary at the **XMax** direction should be the **Perfect H Boundary**. All other directions are truncated by using the **Open Boundary (add space)** boundary.

The excitation and its image should be in the following format.



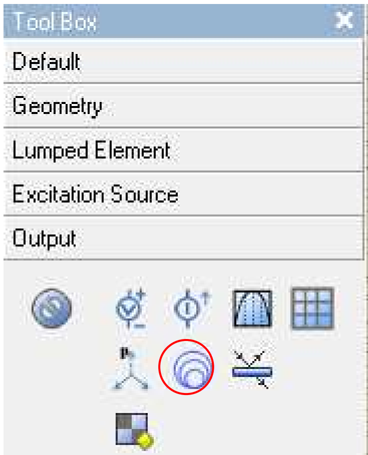
9.6 Design Mesh Distribution

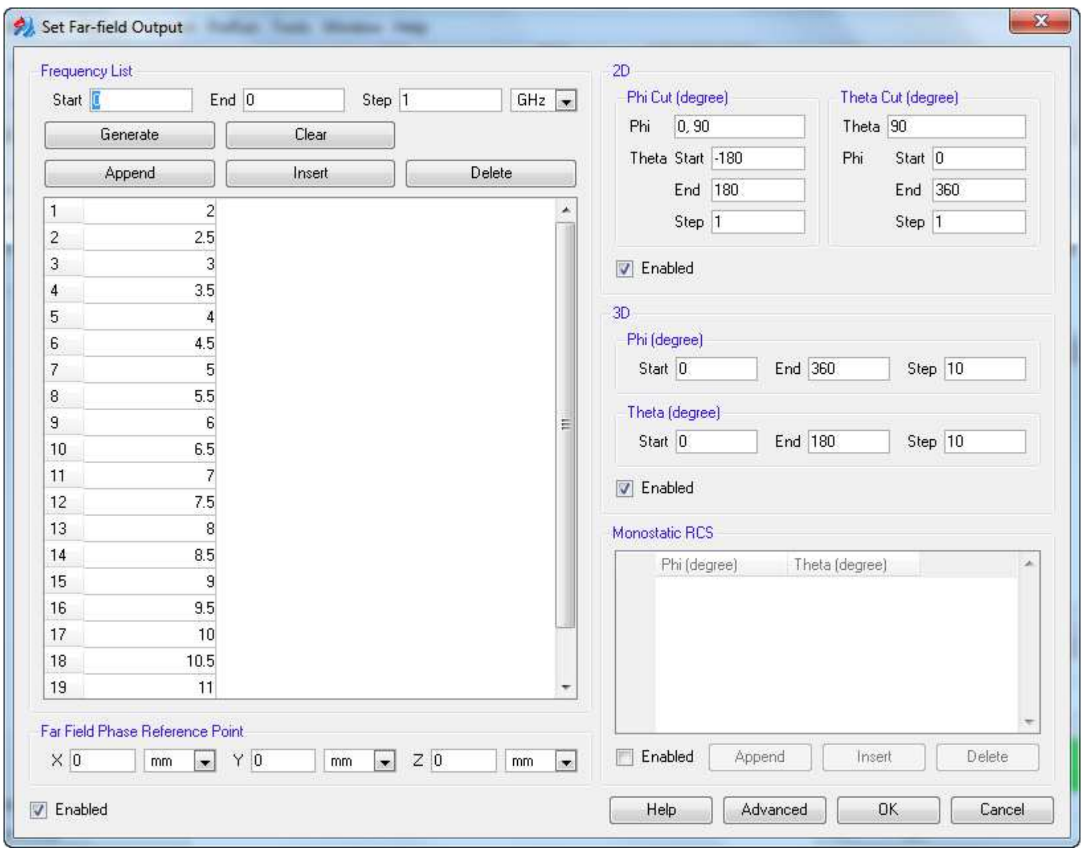
Click on the **Auto Mesh** button in the toolbar.



9.7 Set Far Field Output

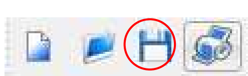
Click on the **Far-field** icon in the **Tool Box->Output** box.





9.8 Save Project

Click on the **Save** button in the toolbar.

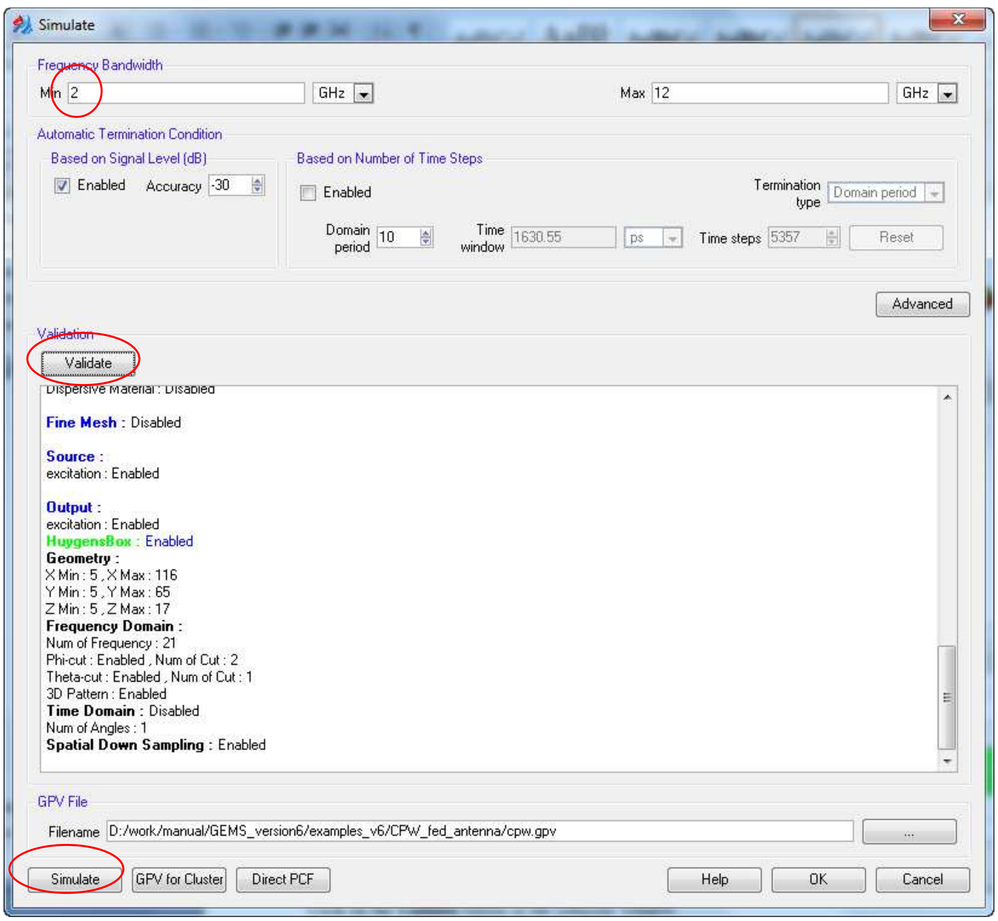


9.9 Generate Simulation File

Click on the **PreCalculate** button in the toolbar. Specify the frequency range of interest in the **Frequency Band** boxes.

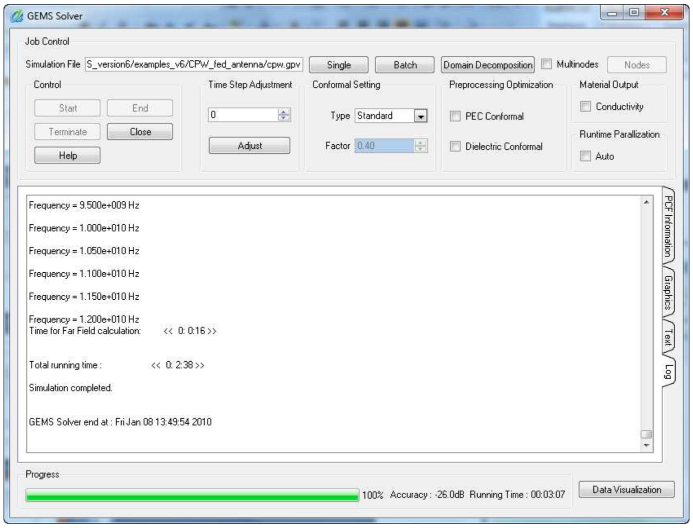
Click on the **Validate** button to validate the project settings.

Click on the **Simulate** button to open the *GEMS Solver* window.



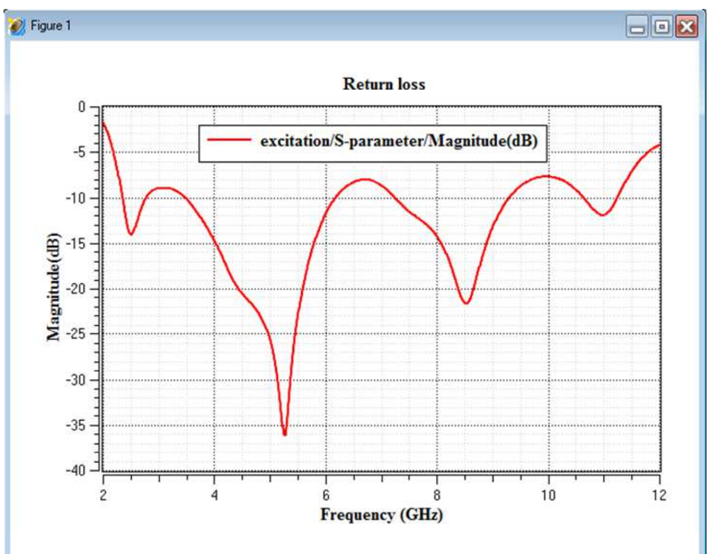
9.10 Simulate Project

Click on the **Validate** button in the *Simulate* window. Click on the **Start** button to start the simulation

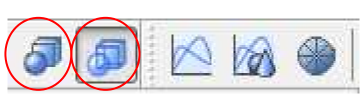


9.11 Result Visualization

Click on the **Data Visualization** button in the *GEMS Solver* window. Double-click on the **Port->excitation->FreqDomain->S parameter->Magnitude(dB)** option.

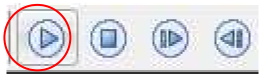


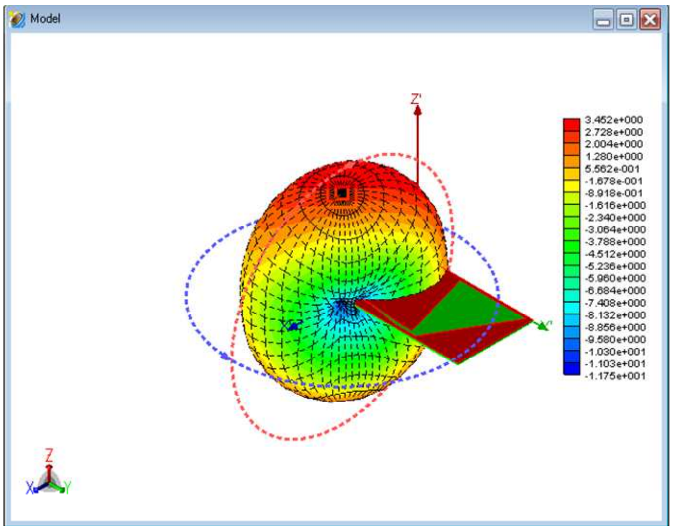
Click on the **View model** button in the toolbar, and then on the **Transparent mode** button.



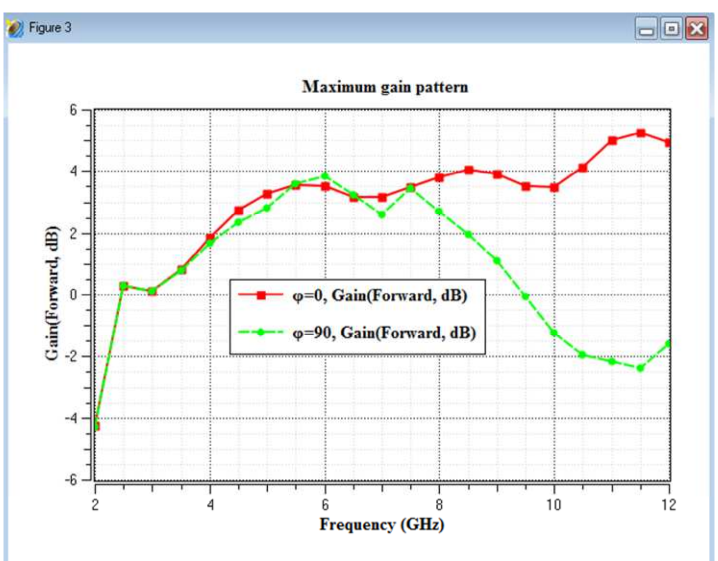
Select one 3D pattern option and then click on the **Add to current window** button.

Click on the **Play** button to display the 3D gain pattern with the project model.

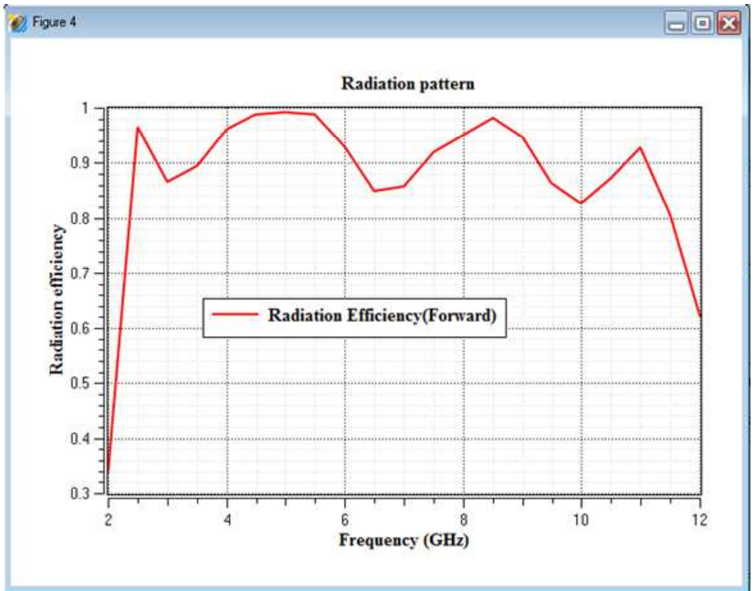




Double-click on the **Ouput->Farfield->FreqDomain->Phi Cut->Phi=0->Max->ETotal->Gain(Forward, dB)** option in the result tree. And then select the **Ouput->Farfield->FreqDomain->Phi Cut->Phi=90->Max->ETotal->Gain(Forward, dB)** option, and then click on the **Add to current window** in the toolbar.



Double-click on the **Ouput->Farfield->FreqDomain->Radiation Efficiency(Forward,dB)** option in the result tree.



10

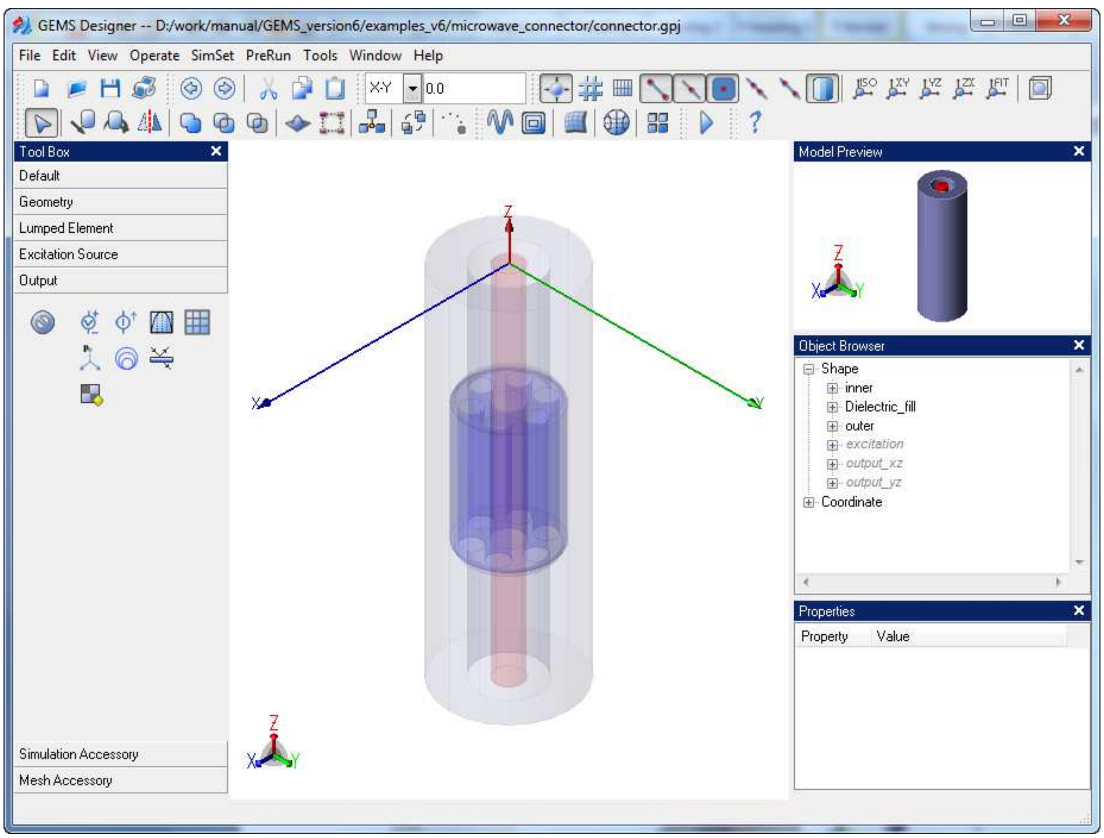
Example 10. Microwave Connector

Description: A simple microwave connector is excited by the TEM mode. The output parameters include TDR (Time Domain Reflectometry) and time domain impedance.

Keywords: Microwave connector, TDR, time domain impedance, and rise time.

10.1 Problem Configuration

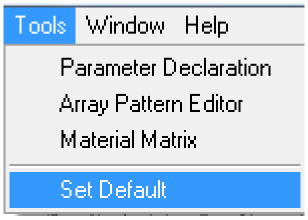
A simple microwave connector fed by a uniform coaxial waveguide is used to demonstrate the TDR calculation.

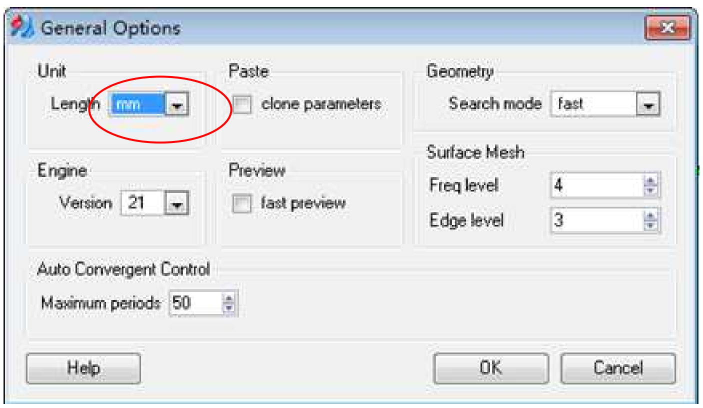


10.2 Create Project Model

Follow the steps below to create the connector model:

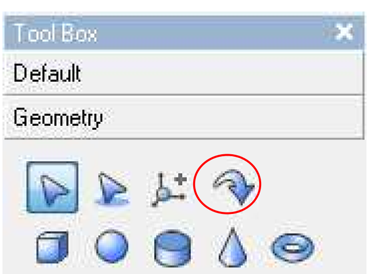
(1) Open GEMS Designer
(2) Select the **Set Default** option in the **Tools** menu



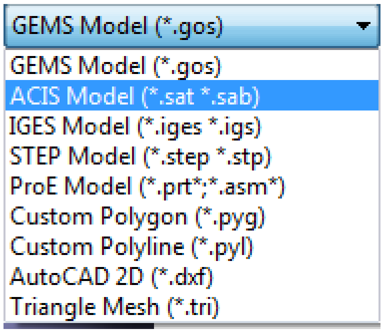


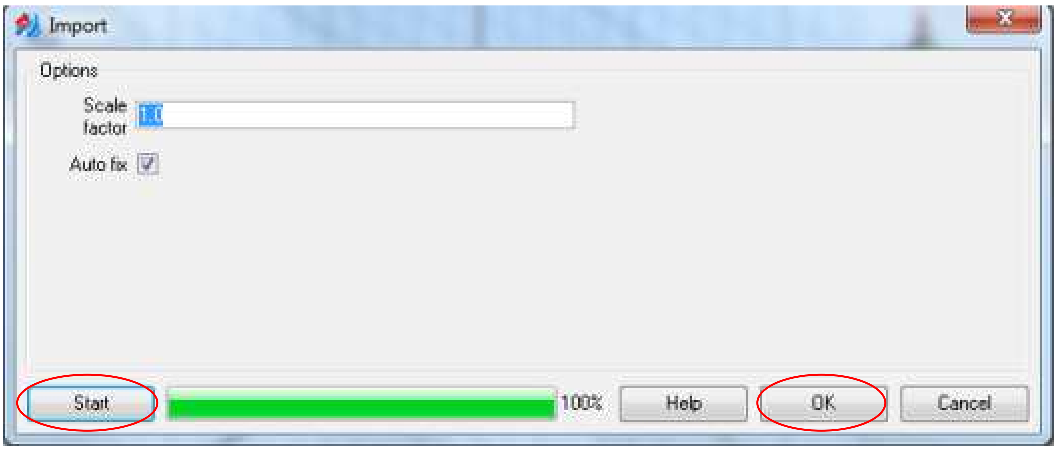
(3) Click on the **New** button in the toolbar to create a new project

(4) Click on the **Import Model** icon in the **Tool Box->Geometry** box.

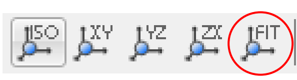


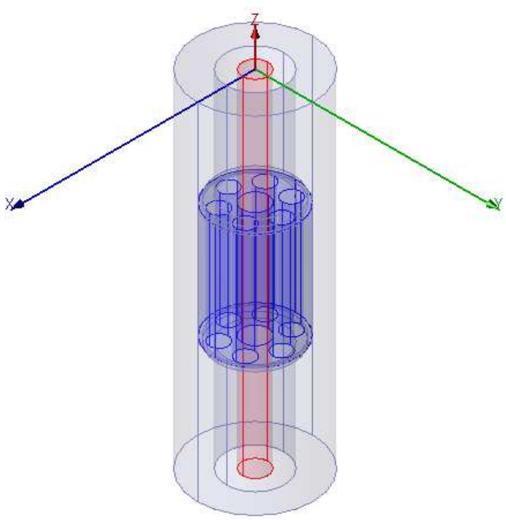
Select the “ACIS Model (*.sat, *.sab)” format.



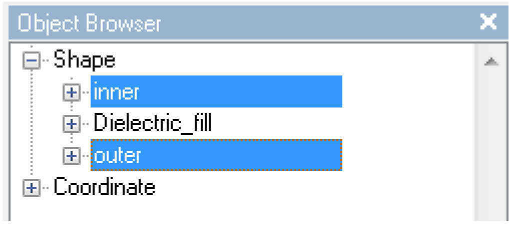


Click on the **FIT** button in the toolbar.

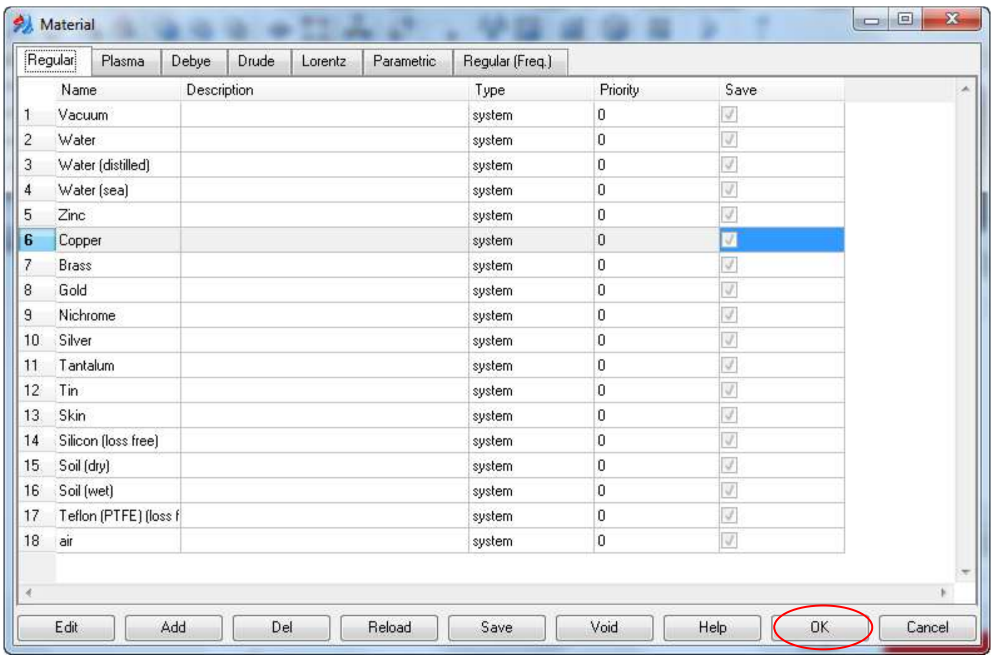




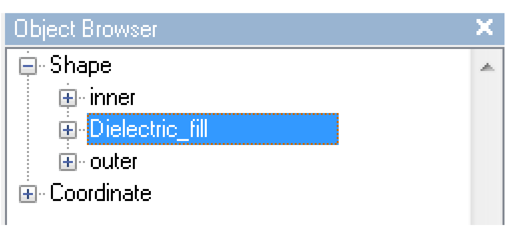
There are three parts with two types of materials in the model. Select the **outer** and **inner** conductors in the **Object Browser** box and assign them to the copper material in the **Properties** box.



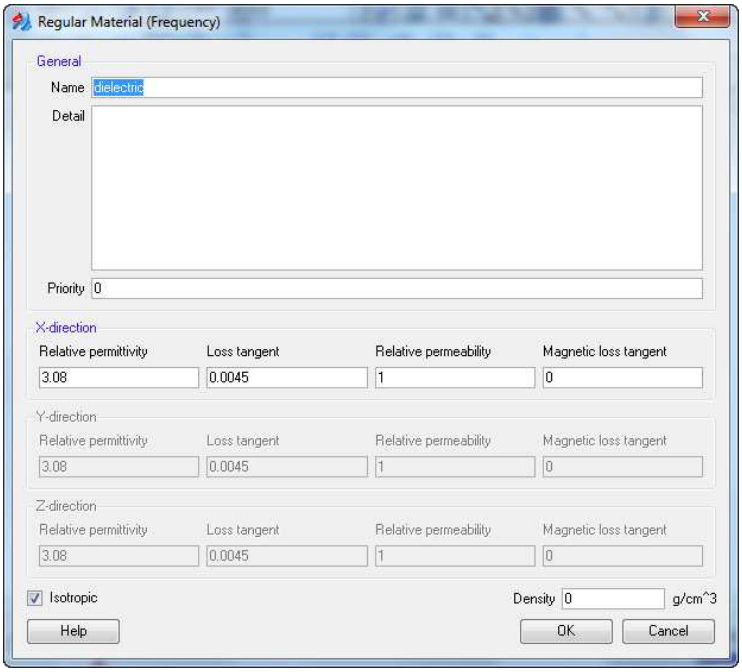
Select the **Copper** in the library and click on the **OK** button to confirm the selection.



Select the **Dielectric_fill** option in the **Object Browser** box and assign them to the dielectric material (dielectric constant=3.08, and loss tangent = 0.0045) in the **Properties** box.

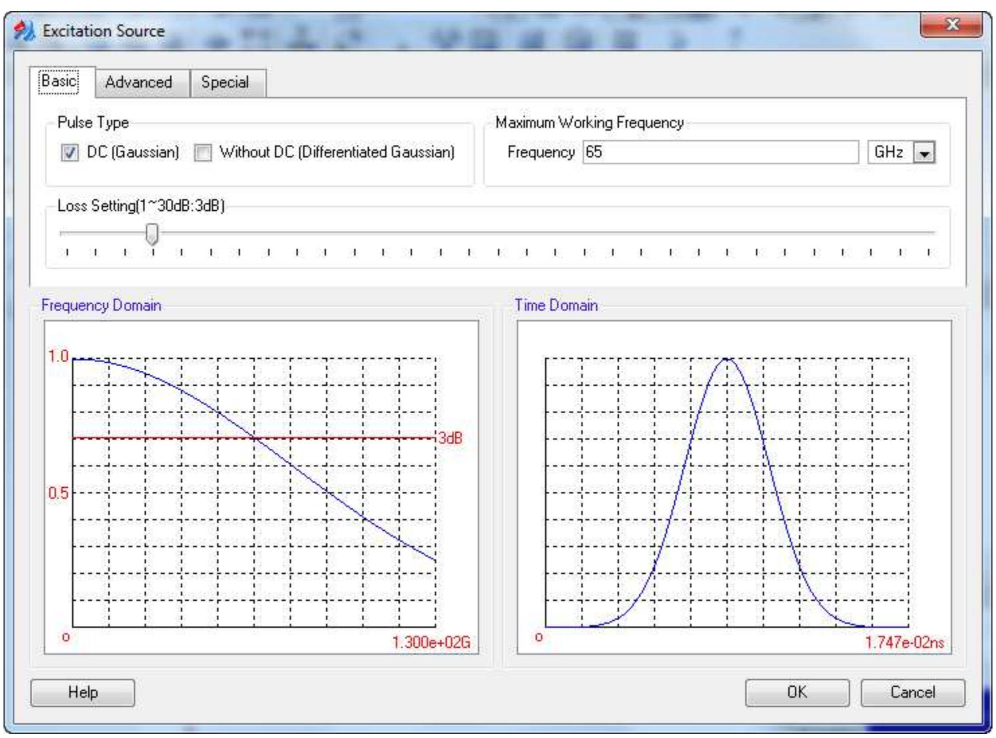


Type the “dielectric_fill” in the **Name** box, and the dielectric constant and loss tangent in the parameter boxes.



10.3 Set Excitation Pulse

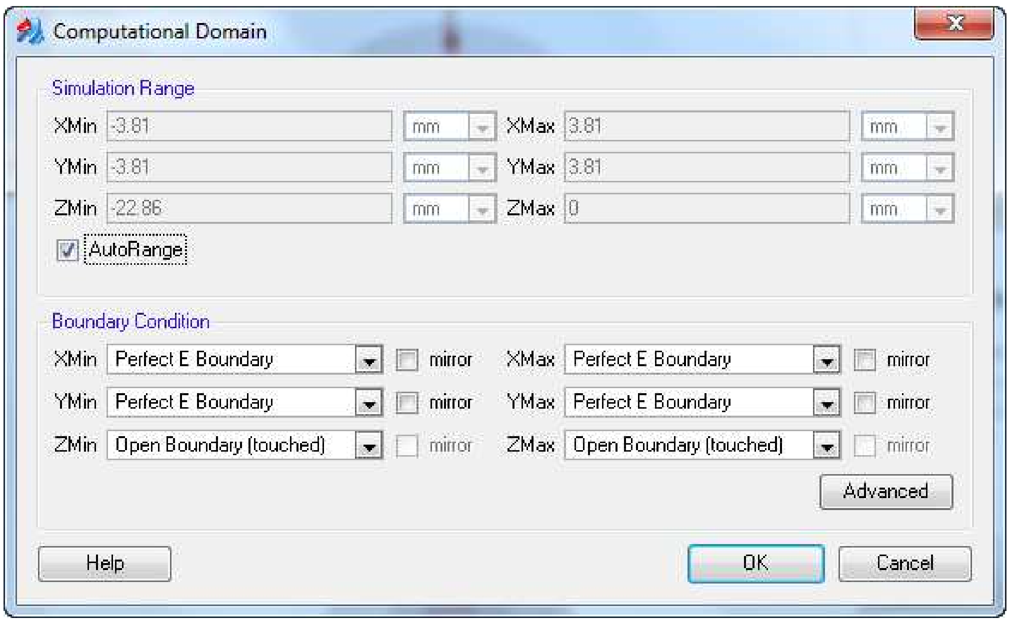
Click on the **Set excitation source** button in the toolbar.



The excitation pulse for the TDR simulation must be the pure Gaussian pulse. The rise time of the Gaussian pulse is 0.3394/frequency. The pulse width should be narrow enough to describe the fine structure of geometry.

10.4 Set Domain and Boundary Condition

Click on the **Set boundary condition** button in the toolbar.

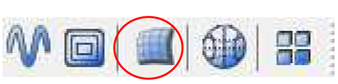


The power is limited inside the outer conductor and no power is outside the conductor. We use the “Perfect E boundary” to truncate the domain in the horizontal directions. Two ends are truncated by the absorbing boundary (Open Boundary (touched)) and extended to infinity.

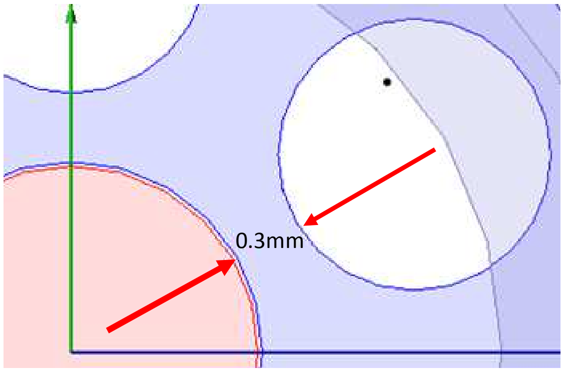
Click on the **OK** button to confirm the domain and boundary settings.

10.5 Design Mesh Distribution

Click on the **Design mesh** button in the toolbar.

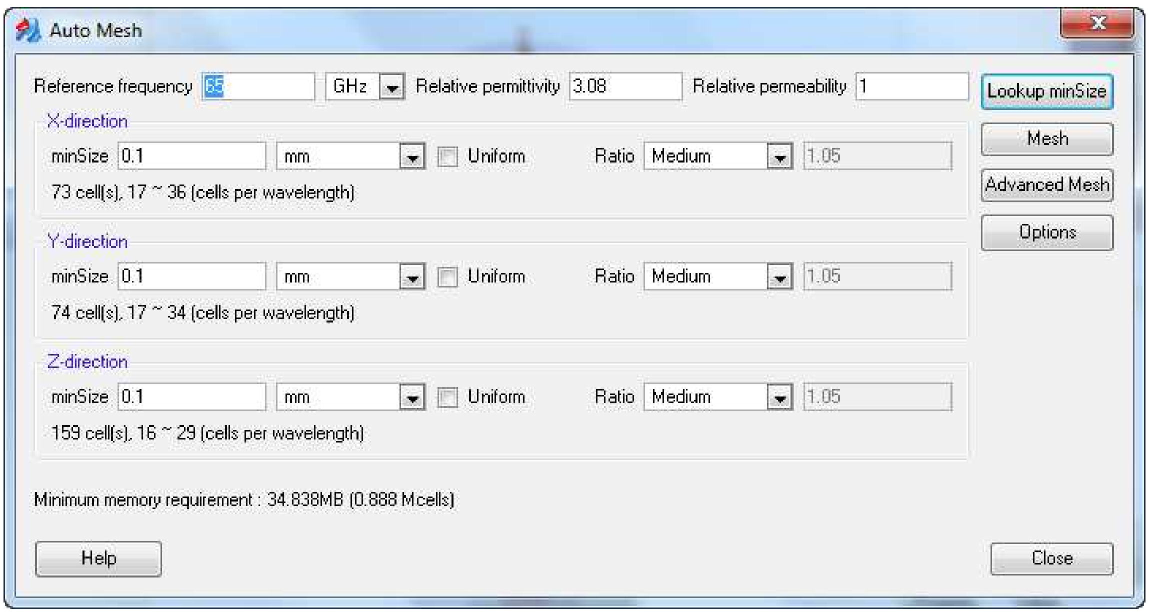


The fine structure of interest in this example is the gap shown in the figure below:



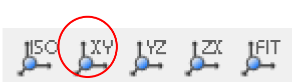
The cell size is taken to be 0.1mm for the safety. Set the ratio to be “Low” and click on the **Mesh** button to generate the proper mesh distribution.

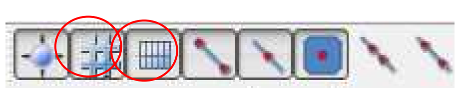
Click on the **OK** button to confirm the mesh design.

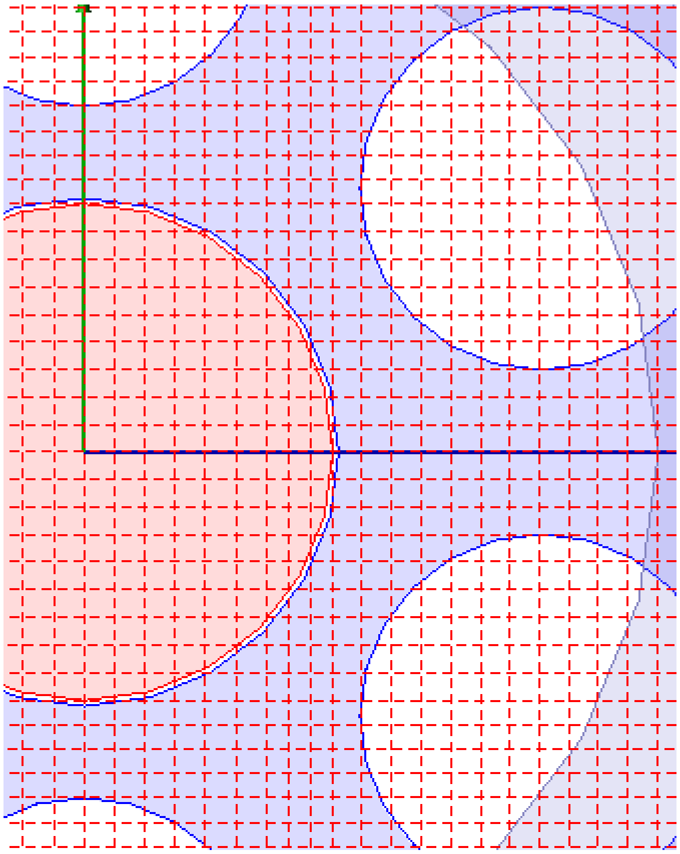


Set the drawing plane to be the “X-Y” plane’

Click on the “X-Y” button in the toolbar and then push down “View grid’ and “Mesh mode” buttons.

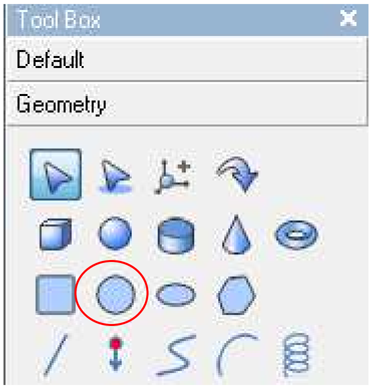


,


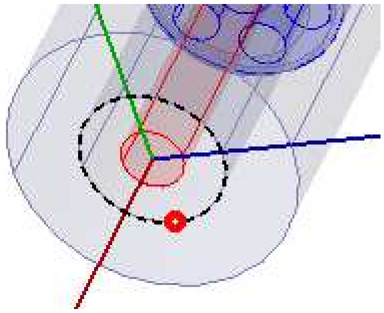
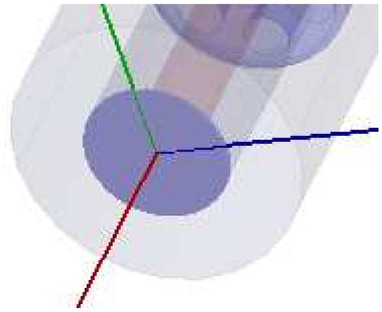


10.6 Set Excitation Source

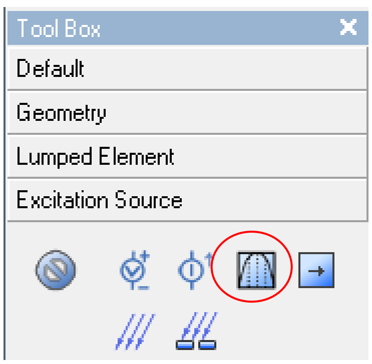
Click on the **Circle** icon in the **Tool Box->Geometry** box.



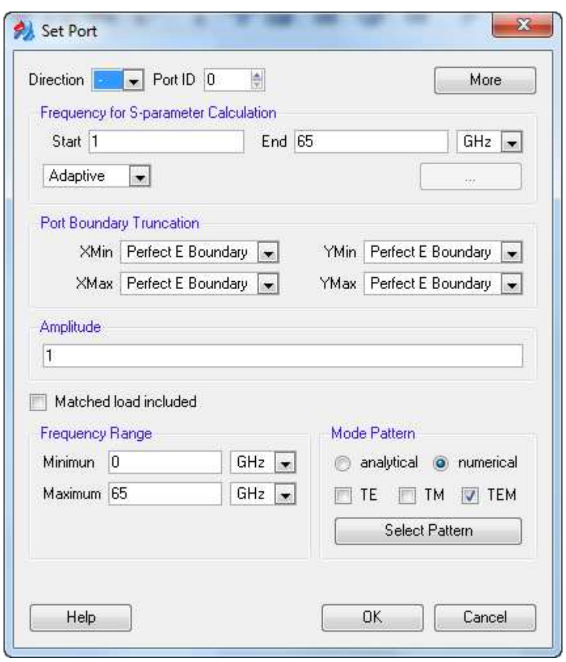
Move the mouse icon to the bottom center of the inner conductor and press the left mouse button, Move the mouse icon to the inner radius of the outer conductor and press the left mouse button.

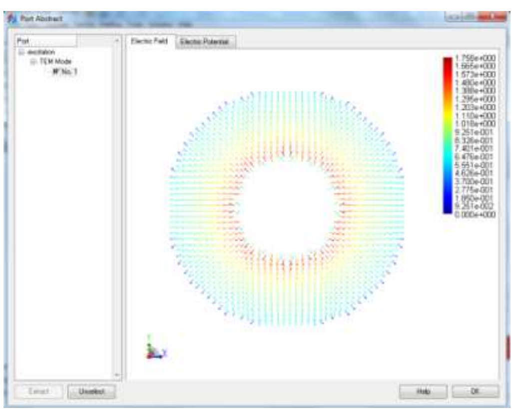
Select the circle in the **Object Browser** box and change its name to “excitation” in the **Properties** box. Click on the **Port** icon in the **Tool Box->Excitation Source** box.



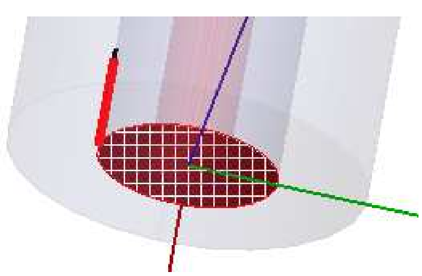
Click on the **numerical** radio button (not rectangular or circular waveguide), and check the **TEM** box. Push the **More** button down to expand the **Set Port** window, and check the TDR box to output the TDR during the simulation. Click on the **OK** button to close this window.



Select the “excitation” option in the **Object Browser** box and click on the “excitation” option in the **E/O** box. Click on the **Select Pattern** button to extract the mode pattern. Select the “excitation” option in the **Port** tree. And then click on the **Extract** button. View the mode pattern by clicking on the mode option in the **TEM Mode** entrance. If the excitation mode is not marked with “**W**”, select it and click on the **Select** button.



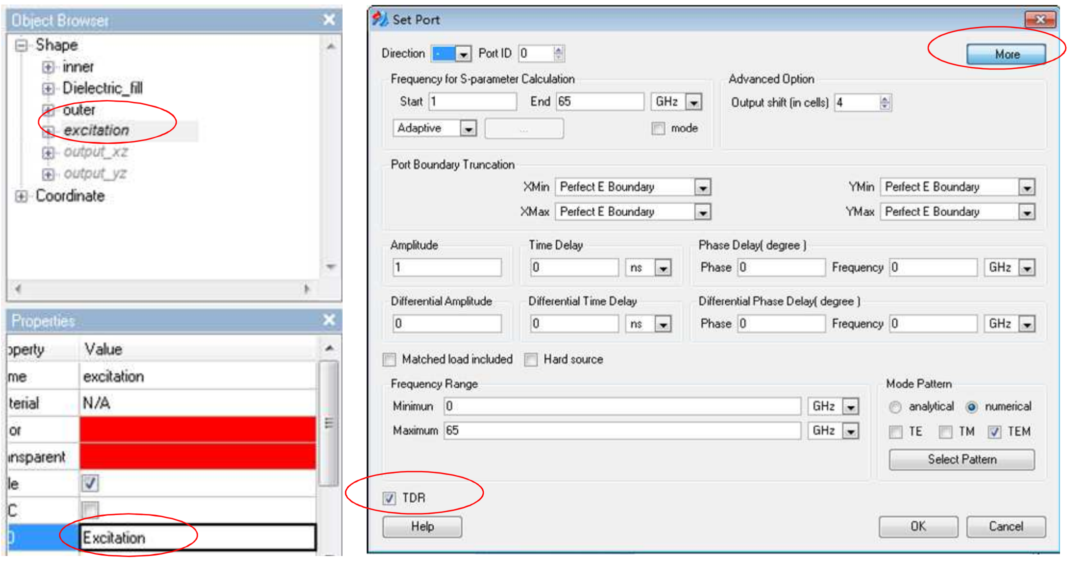
The arrow of the port should point to the power propagation direction.



10.7 Set Outputs

Select the **excitation** option in the **Object Browser** box, and then click the Excitation option in the **Properties** box.

Click the **More** button in the **Set Port** window, and check the **TDR** box. Check the “mode” box, the TDR calculation is based on the mode pattern, otherwise, based on the port voltage measured between the inner and outer conductors.



10.8 Save Project

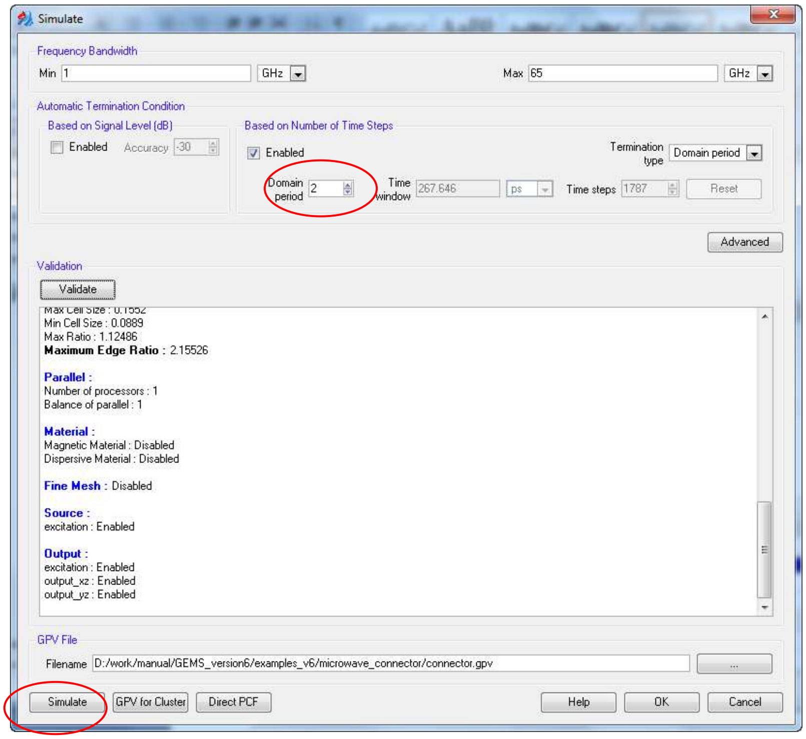
Click on the **Save** button in the toolbar.

10.9 Generate Simulation File

Click on the **PreCalculate** button in the toolbar. Specify the frequency range of interest in the **Frequency Band** boxes.

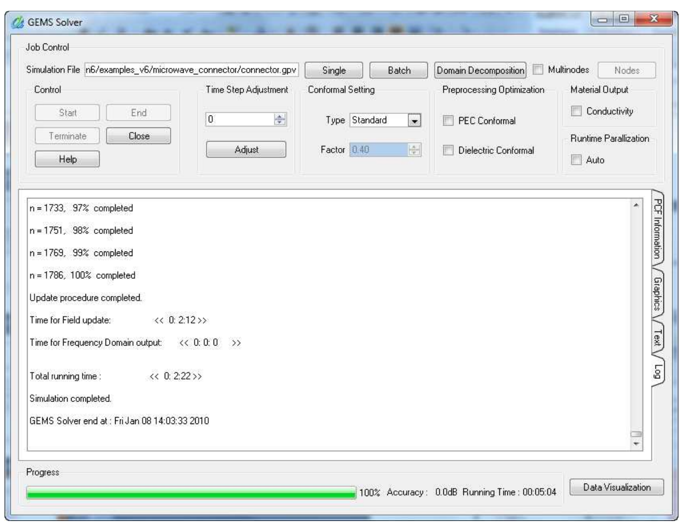
Click on the **Validate** button to validate the project settings.

Click on the **Simulate** button to open the *GEMS Solver* window. Specify the frequency range of interest in the **Frequency Band** boxes.



10.10 Simulate Project

Click on the **Start** button in the *GEMS Solver* window.

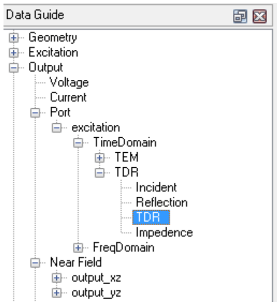


Click on the **Data Visualization** button to open the result visualization window after the simulation is completed.

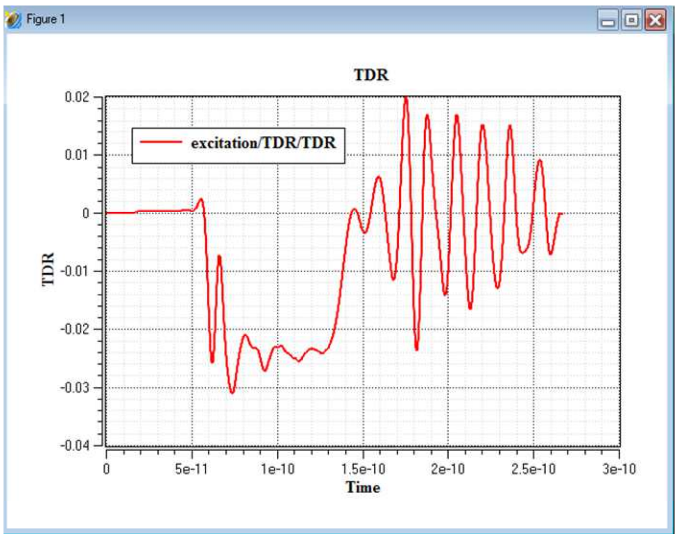
10.11 Result Visualization

Click on the **Data Visualization** button in the *GEMS Solver* window.

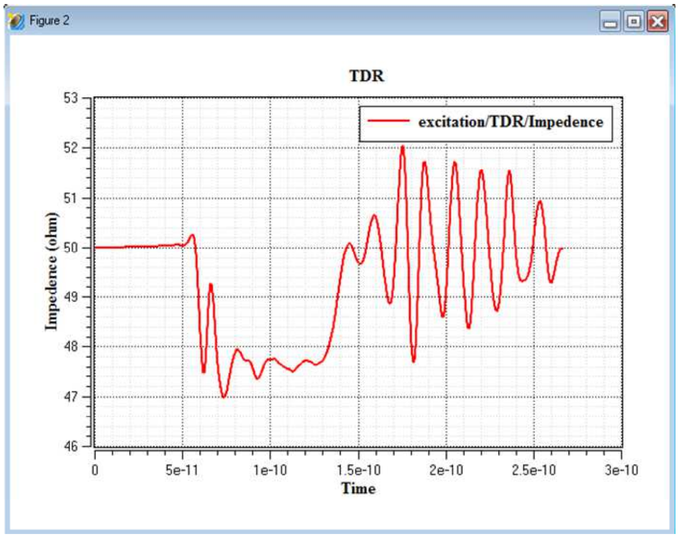
The simulation results are listed in the result tree.



Double-click on the **TDR** option.



Double-click on the **Impedance** option.



11

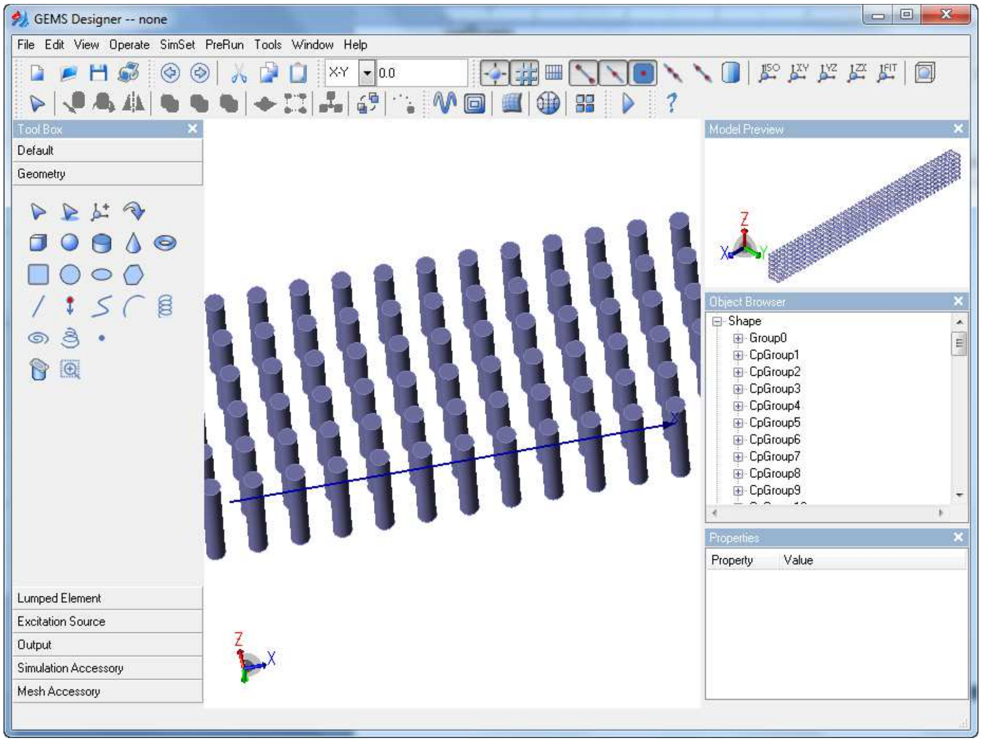
Example 11. EBG Structure

Description: An EBG (Electronic Band Gap) structure is periodic in two directions and illumined by a plane wave. Output parameters include the reflection and transmission coefficients.

Keywords: PBC boundary, EBG structure, reflection and transmission coefficients, and array generation.

11.1 Problem Configuration

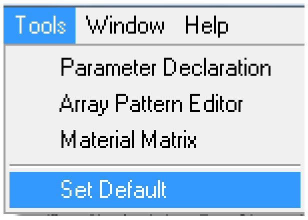
The EBG structure includes a group of dielectric cylinders (6-layers of cylinders, radius=2mm and the distance between the adjacent cylinders = 5mm). We need to simulate one element (6 cylinders) and use the PBC boundary to truncate the domain.

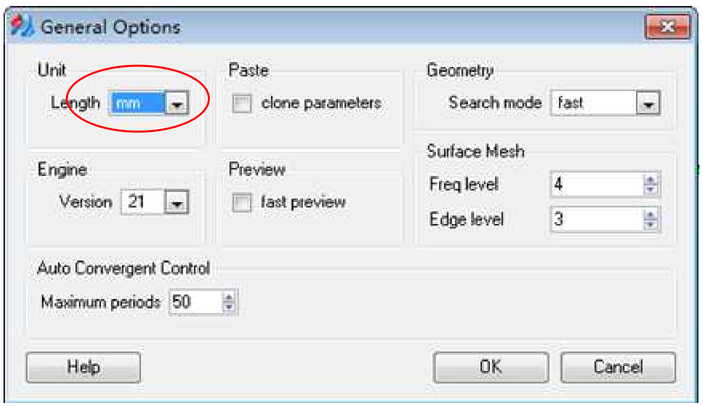


11.2 Create EBG Model

Follow the steps below to create the EBG structure:

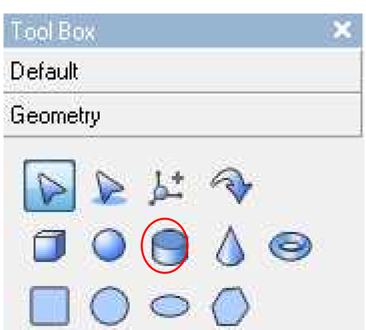
(1) Open GEMS Designer
(2) Select the **Set Default** option in the **Tools** menu





(3) Click on the **New** button in the toolbar to create a new project

(4) Click on the **Circle** icon in the **Tool Box->Geometry** box.



Select the cylinder option in the **Object Browser** box and change its name to "cylinder" in the **Properties** box.

Select the "CreateCylinder" option under the "cylinder" entrance in the **Object Browser** box, and change its coordinates to: center (4.5mm, 0, 4.5mm), radius=2mm, and height=2mm.

The length of cylinder is infinitely long; however, we only need to simulate a finite structure.

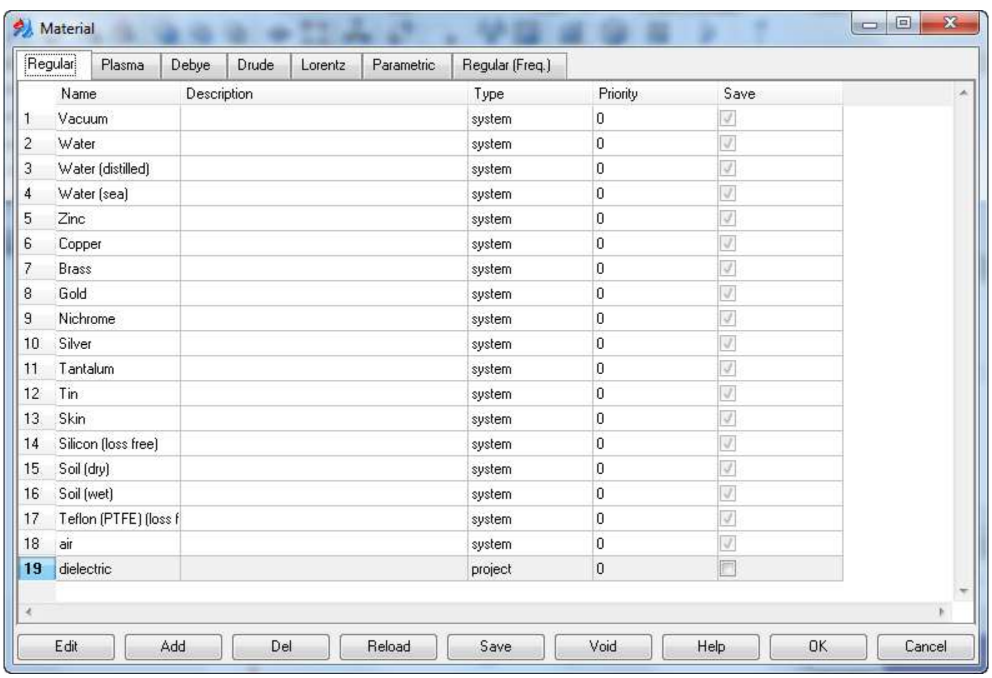
The simulation range is one element rather than complete structure.

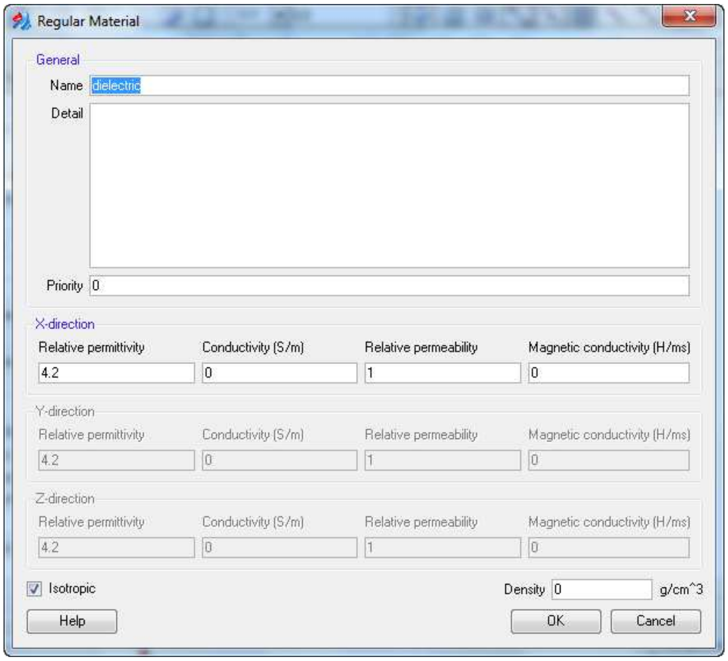
Select the **cylinder** option in the **Object Browser** box and click on the **Material** box in the **Properties** box. Click on the **Add** button to add a new material type in the material library.

*If checking the box in the **Save** column, this material will be available for the current user, otherwise, it only survives in the current project.*

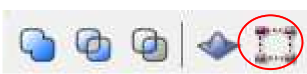
Since the material is lossless, it can be added in the **Regular** category. For the isotropic material, we only need to specify the parameters in one direction and check the **Isotropic** box.

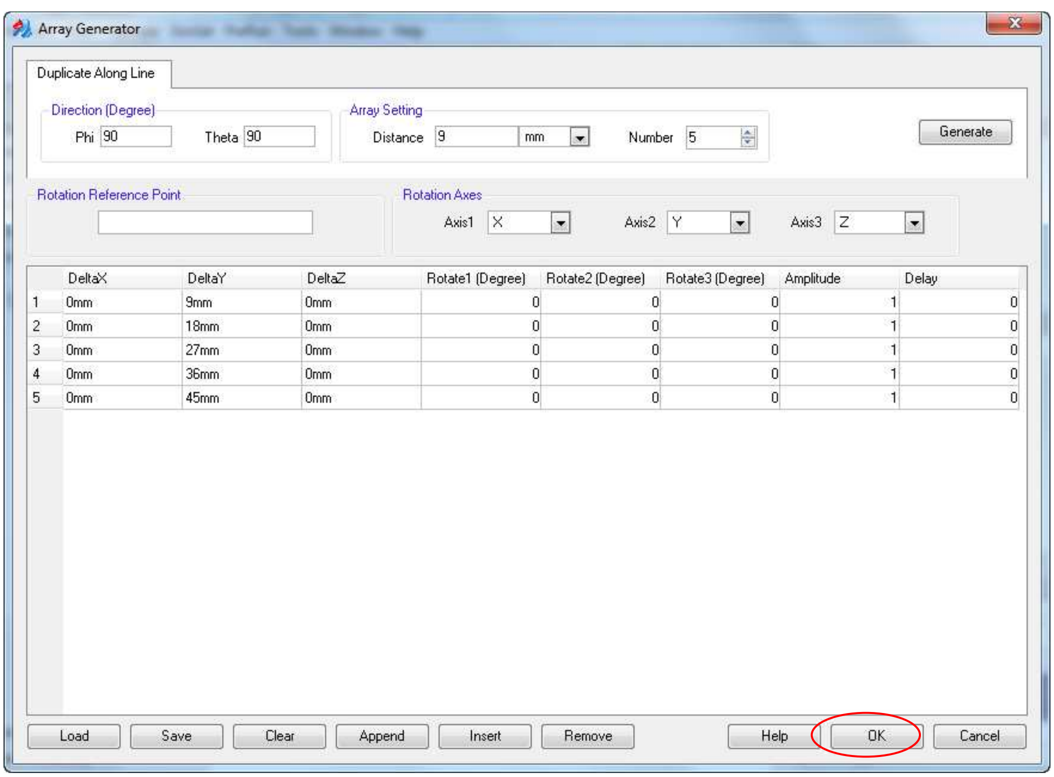
Specify the material name "dielectric" in the **Name** box.

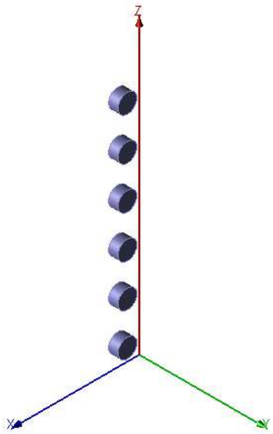




(5) Select the **cylinder** in the **Tool Box->Geometry** box and click on the **Hard clone** button in the toolbar.

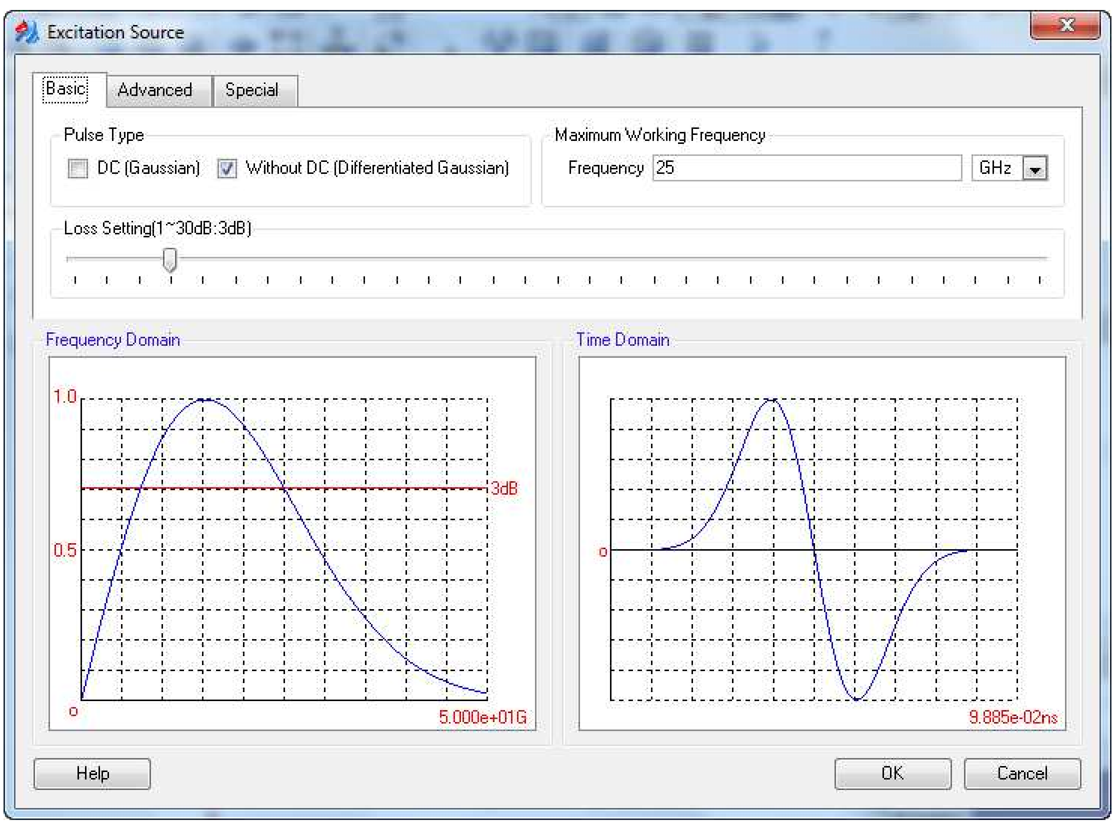






11.3 Set Excitation Pulse

Click on the **Set excitation source** button. Click on the **OK** to confirm the setting.



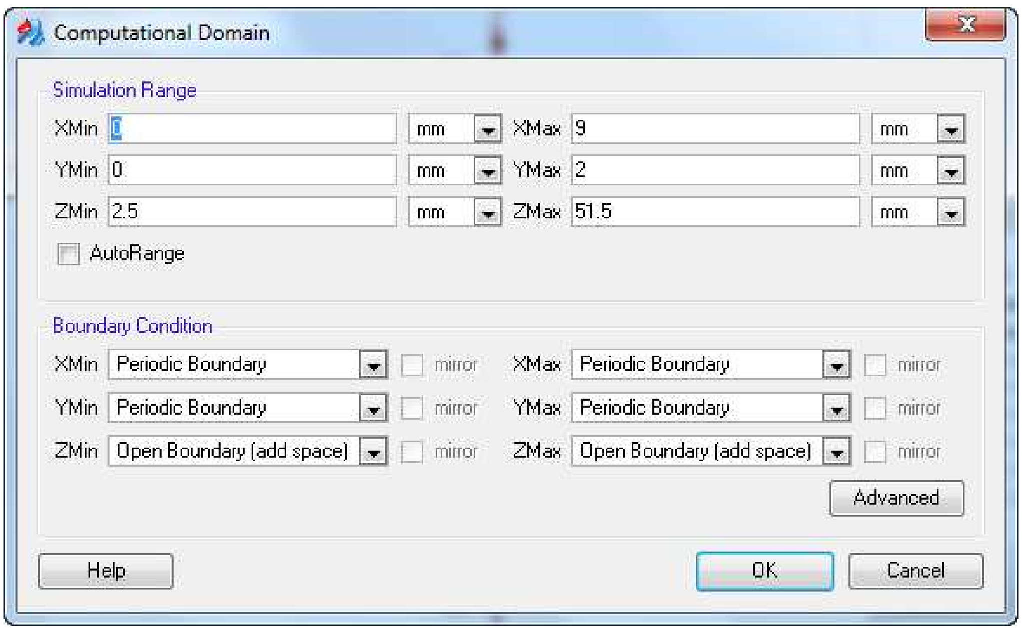
11.4 Set Domain and Boundary Condition

Click on the **Set boundary condition** button in the toolbar.

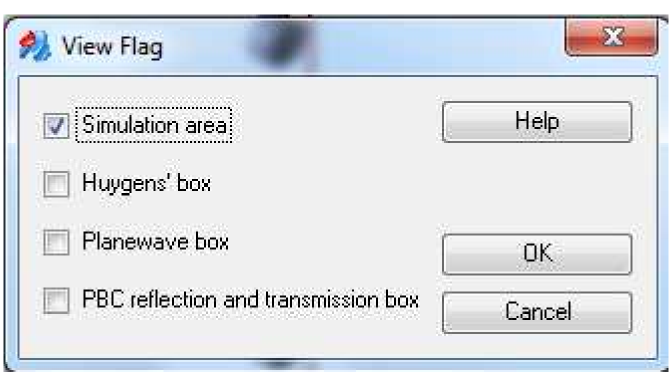
Uncheck the **AutoRange** box and set the domain size (0, 0, 0) and (9mm, 2mm, 54mm) equal to one element dimension.

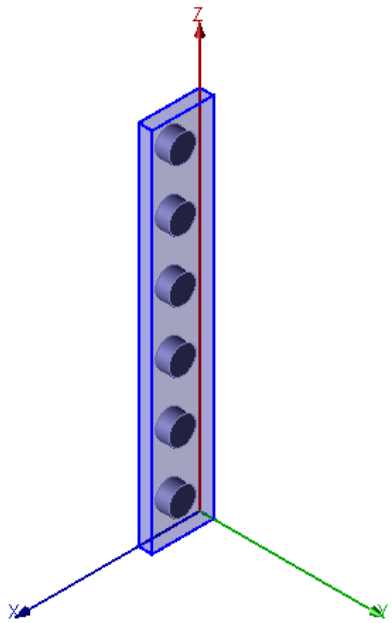
PBC boundary is in the X-Y directions and the **Open Boundary (add space)** is in the z-direction.

GEMS only allows you to use the PBC boundary in the X-Y directions.



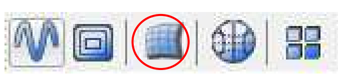
Click on the **View defined box** button in the toolbar, and check the **Simulation area** box.

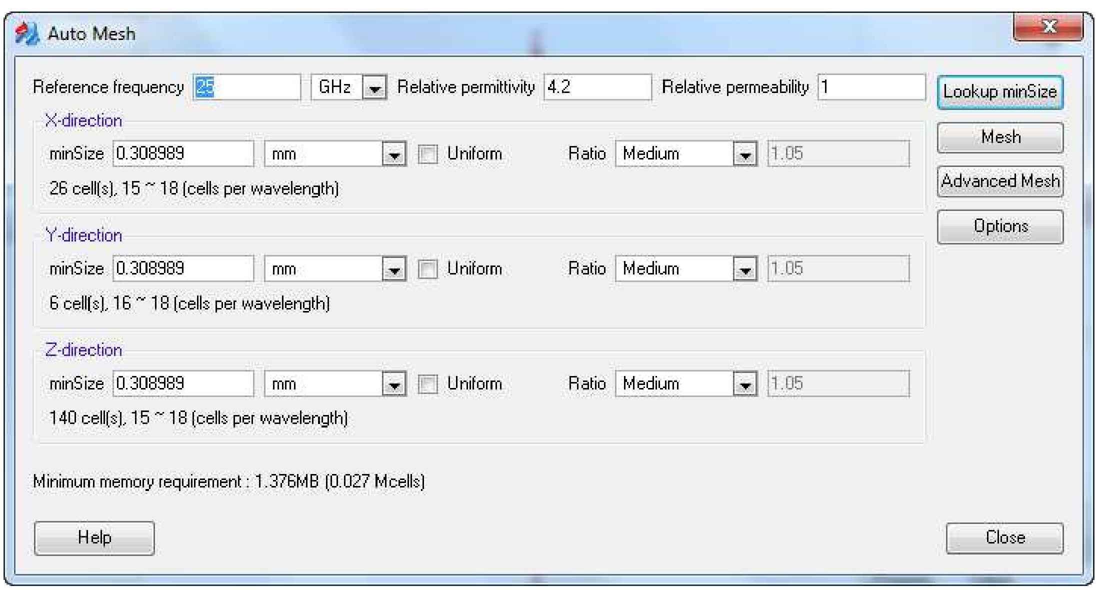




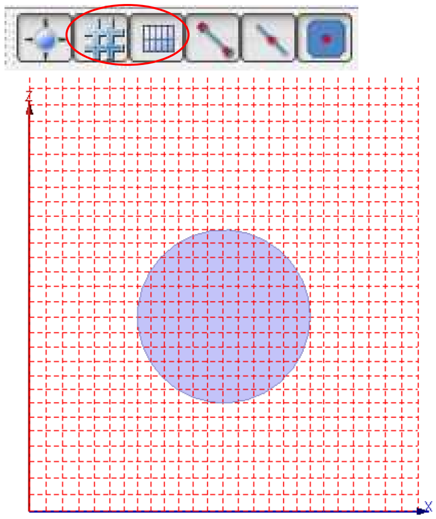
11.5 Design Mesh Distribution

Click on the **Auto Mesh** button in the toolbar.





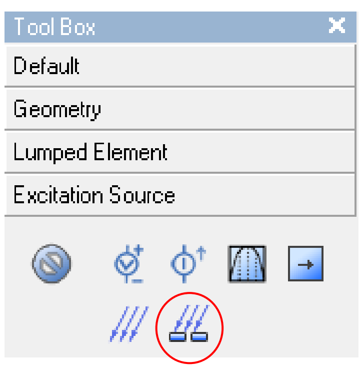
Push the **View grid** and **Mesh mode** button down in the toolbar.



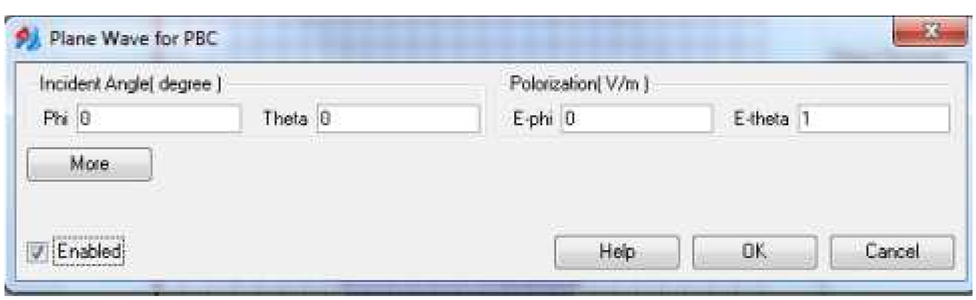
To view the mesh distribution, ensure the drawing plane and view plane to be same.

11.6 Add PBC Excitation

Click on the **PBC plane wave** icon in the **Tool Box->Excitation Source** box.

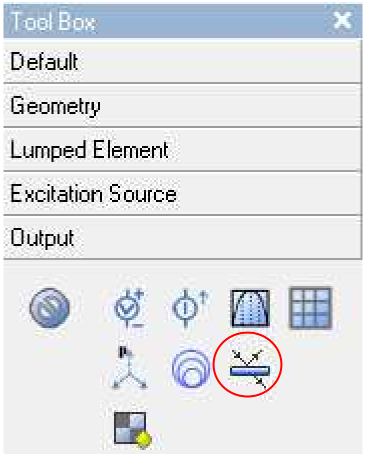


The plane wave illumines from the +z-direction with the *Ex* polarization. Check the **Enabled** box and click on the **OK** button to confirm the settings.

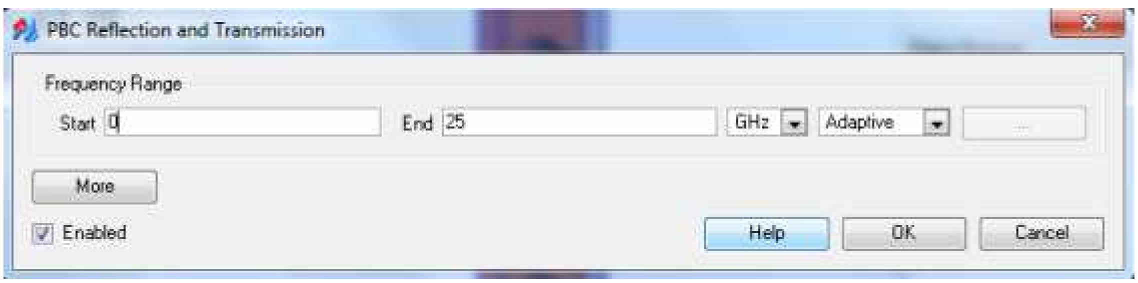


11.7 Add EBG Output

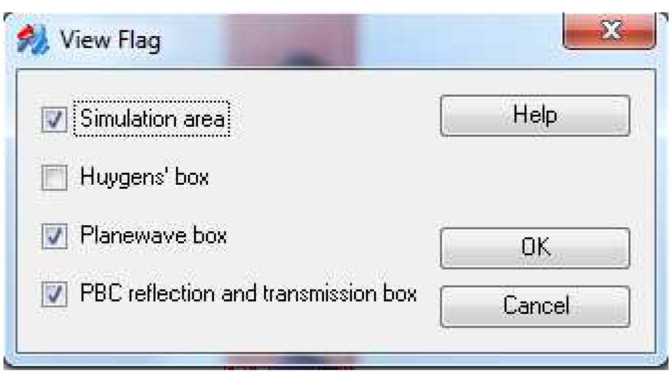
Click on the **PBC reflection&transmission** icon in the **Tool Box->Output** box.

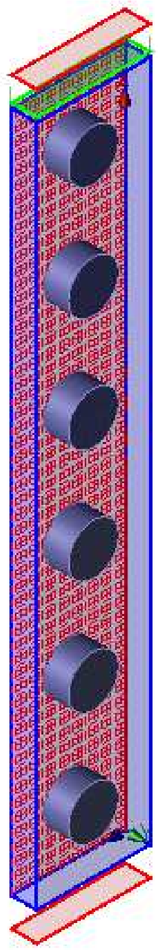


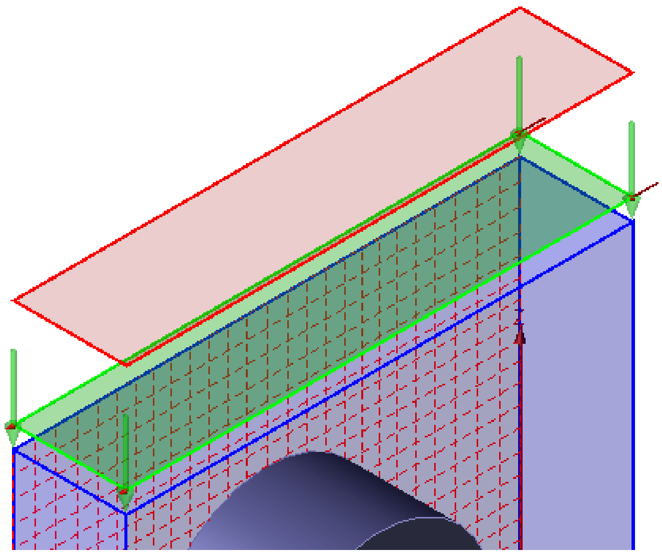
Check the **Enabled** box and specify the frequency band of interest. Click on the **OK** button to confirm the settings.



Click on the **View defined box** button in the toolbar, and check the **Simulation area**, **Planewave box**, and **PBC reflection and transmission** boxes.







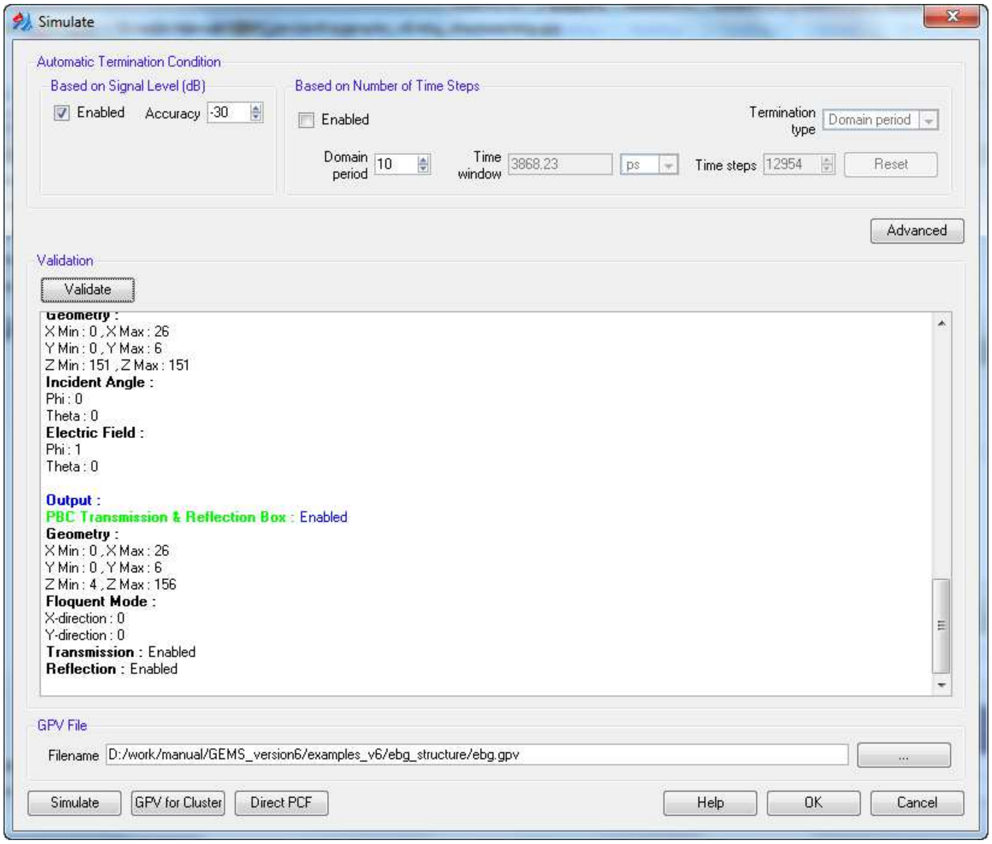
11.8 Save Project

Click on the **Save** button in the toolbar.

11.9 Generate Simulation File

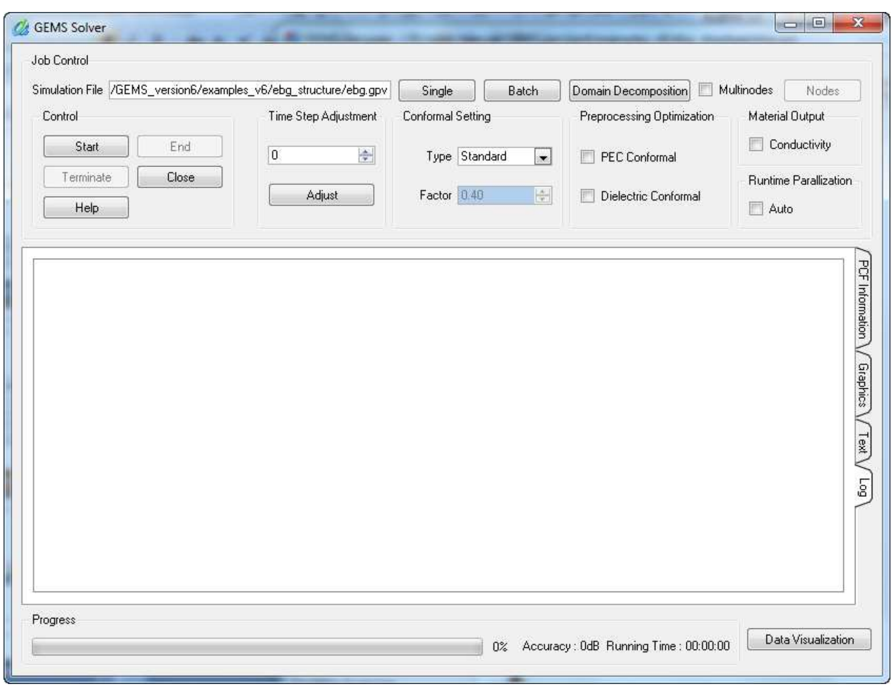
Click on the **PreCalculate** button in the toolbar.

Click on the **Validate** button to validate the project settings and click on the **Simulate** button to open the *GEMS Solver* window.



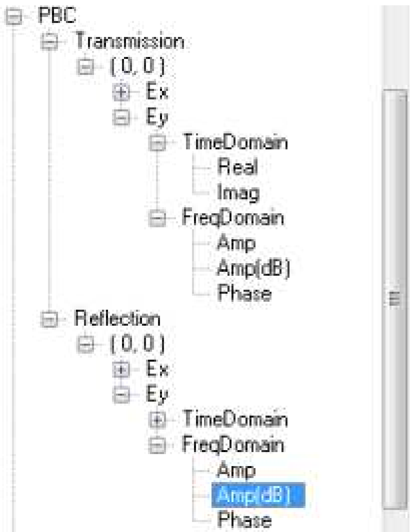
11.10 Simulate Project

Click on the **Simulate** button in the **Simulate** window.

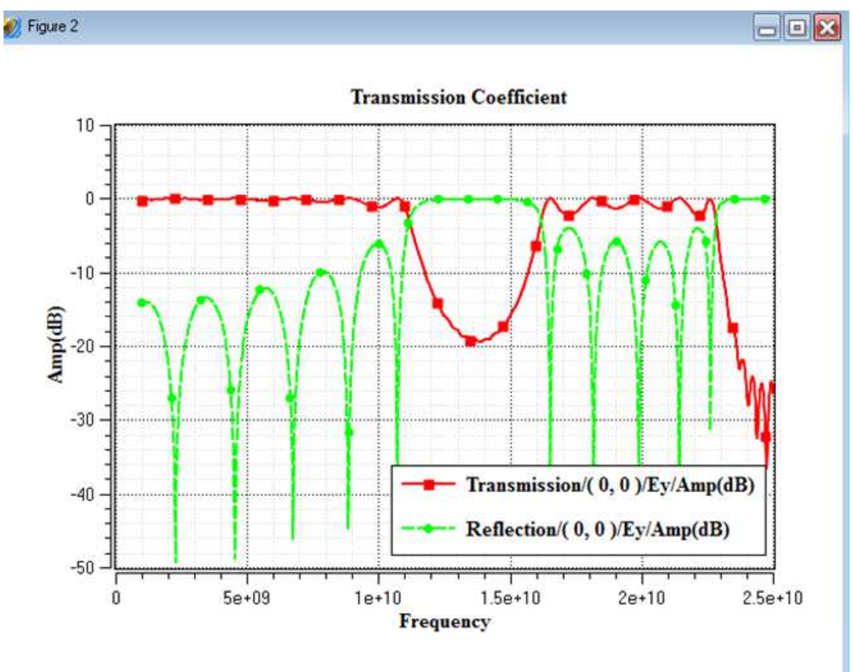


11.11 Result Visualization

Click on the **Data Visualization** button in the *GEMS Solver* window. The EBG results include the reflection and transmission coefficients.



Double-click on the **AMP(dB)** option in the *Ey* entrance of **transmission** folder. Select the **AMP(dB)** option in the *Ey* entrance of **reflection** folder and click on the **Add to current window** button in the toolbar.



12

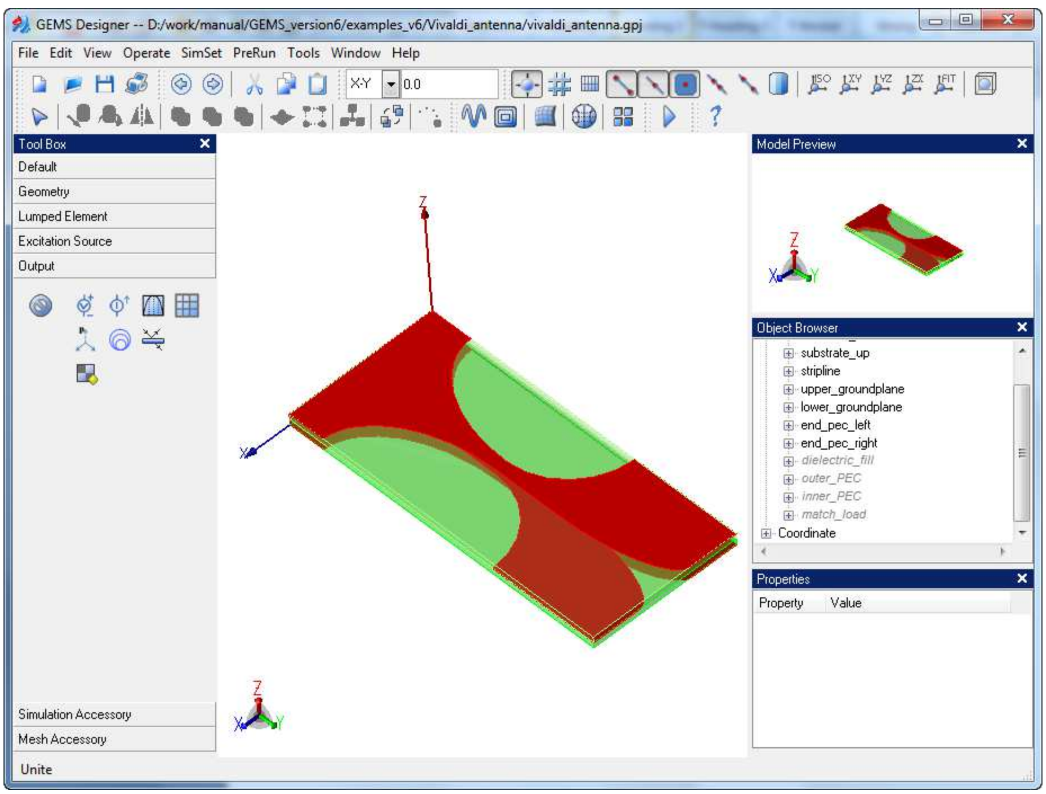
Example 12. Vivaldi Antenna

Description: A Vivaldi antenna includes three pieces of PEC plates, and two layers of dielectric slabs. The output parameters are return loss and far field pattern.

Keywords: Vivaldi antenna, return loss, and far field pattern.

12.1 Problem Configuration

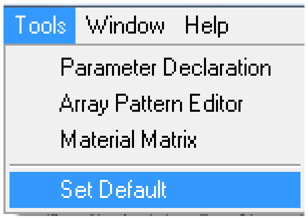
A Vivaldi antenna includes three layers of PEC (two of them are used as ground), and two layers of dielectrics. The feed is a coaxial cable, and we need to add an extended cable for excitation.

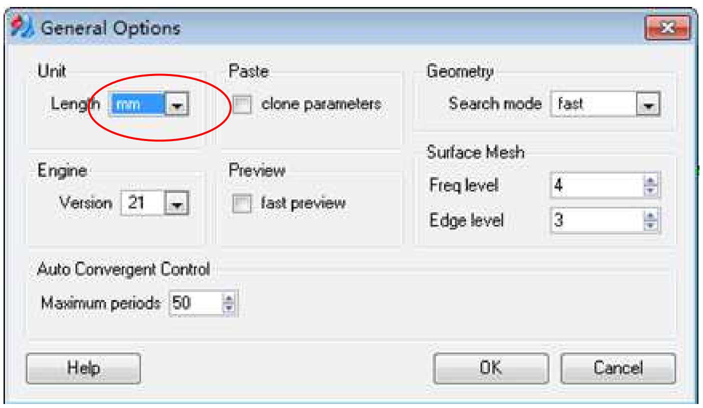


12.2 Create Vivaldi Model

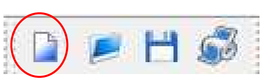
Follow the steps below to create the Vivaldi structure:

(1) Open GEMS Designer
(2) Select the **Set Default** option in the **Tools** menu

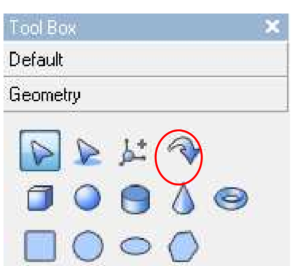




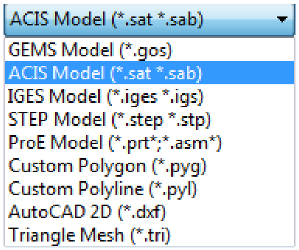
(3) Click on the **New** button in the toolbar to create a new project

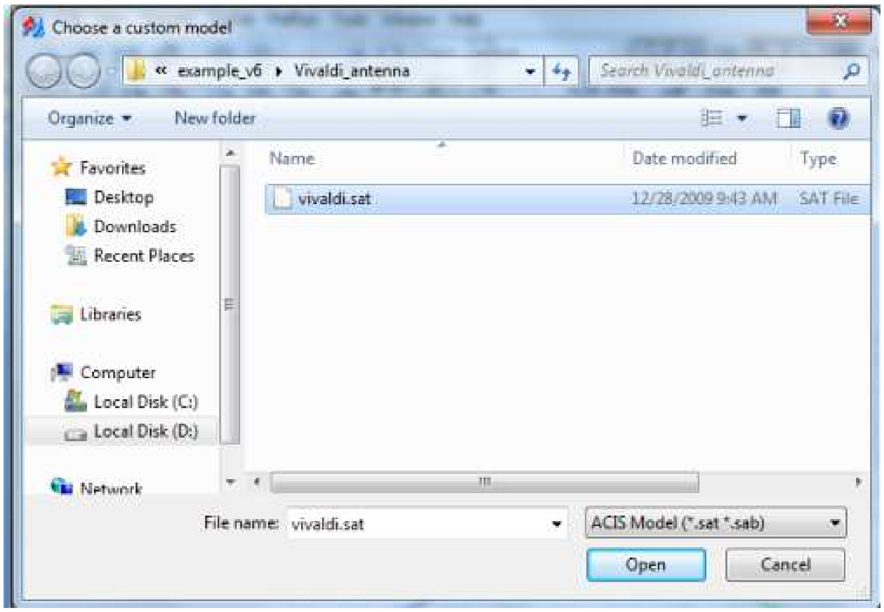


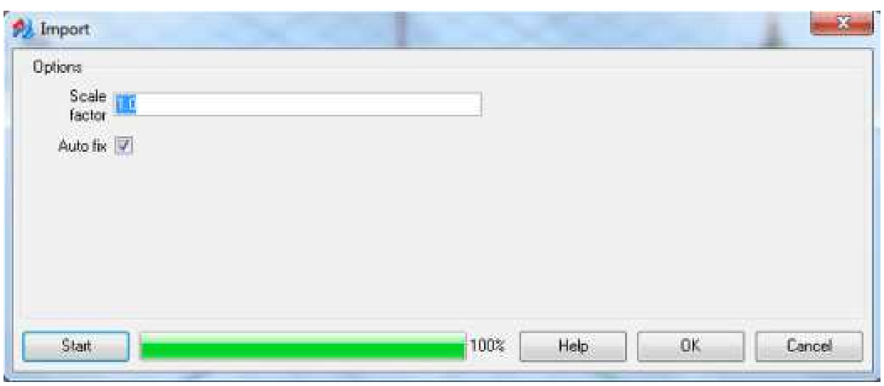
(4) Click on the **Import model** icon in the **Tool Box->Geometry** box.



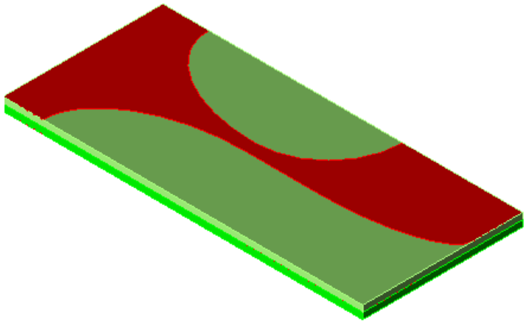
Select the “ACIS Model(*.sat, *.sab)” format in the file format list. Search for and select the “Vivaldi.sat” file and click on the **Open** button. Click on the **Start** button to start the processing, and then click on the **OK** button after the processing is completed.



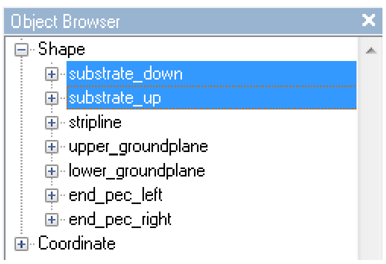


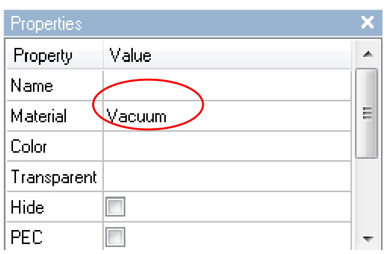


Click on the **FIT** button to fit the model in the figure region, and click on the **Select** button to the view mode.

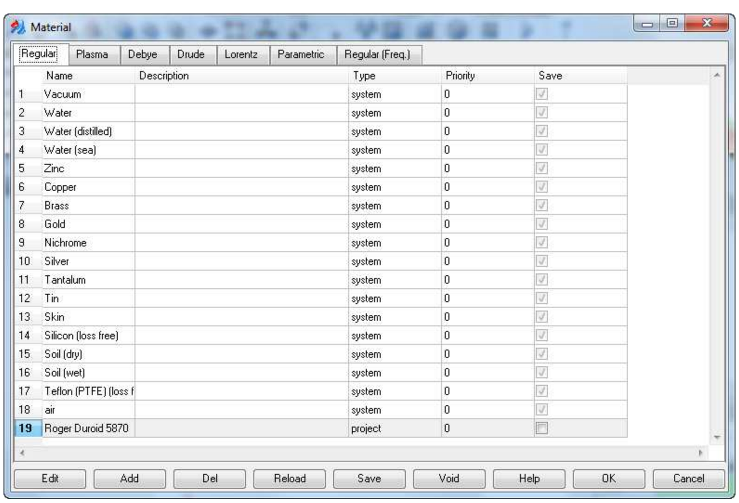


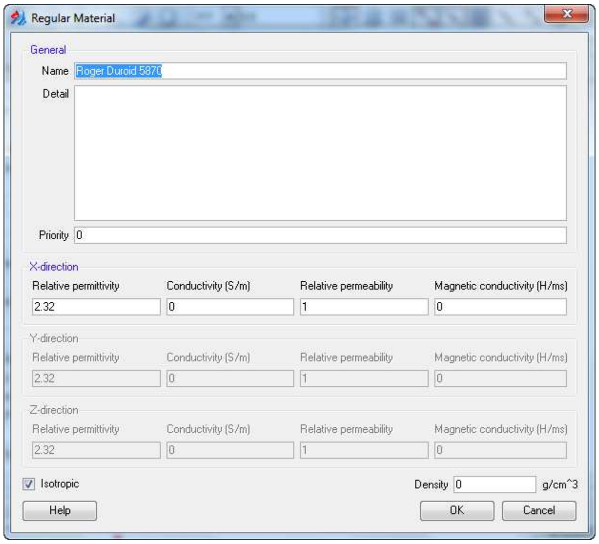
Select the “Substrate_down” and “substrate_up” options in the **Object Browser** box and then click on the **Material** box in the **Properties** box.



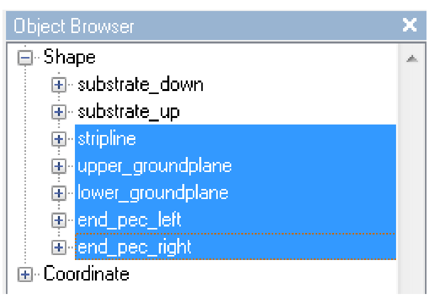


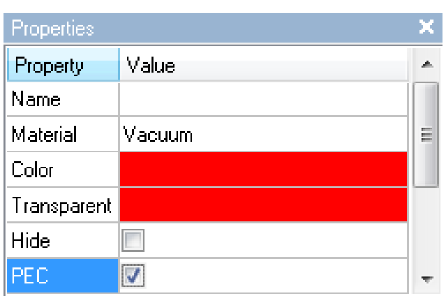
Click on the **Add** button and add the parameter for the “Roger Duroid 5870” material in the *Regular Material* window.





Select the “Stripline”, “upper_groundplane”, “lower_groundplane”, “end_pec_left”, and “end_pec_right” options in the **Object Browser** box and then check the PEC box in the **Properties** box.



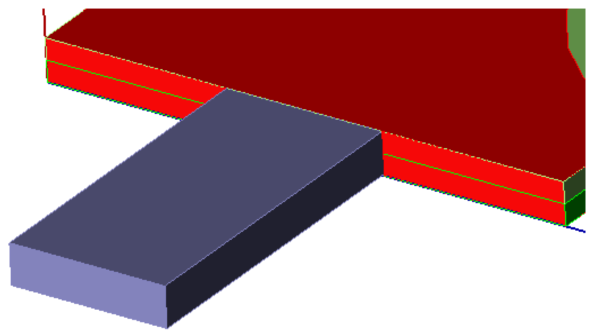


12.3 Add Feed Port Structure

Follow the steps below to create the dielectric slab model:

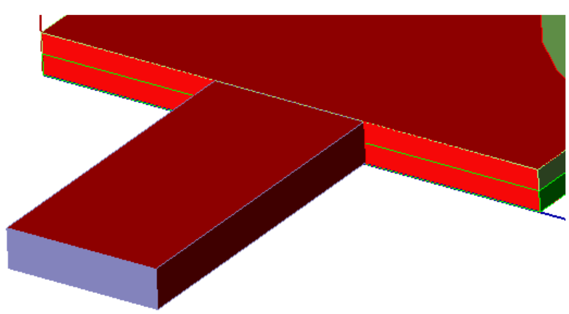
(1) Change the drawing plane to the “X-Z” plane.
(2) Click on the **Cuboid** icon in the **Tool Box->Geometry** box.

(3) Move the mouse icon to select one corner of the feed structure and press the left mouse button, and move the mouse icon to snap to another corner and press the left mouse button. Move the mouse icon along the –Y-direction to get a box.
(4) Select the box in the **Object Browser** box and change its name to “dielectric_fill” in the **Properties** box. Click on the **Material** box and assign the “Roger Duroid 5870” to it.
(5) Select the “CreateCuboid” option under the “dielectric_fill” entrance and change its length of the dielectric to “30mm” in the **Properties** box.



Follow the steps below to create the PEC guide structure:

(1) Click on the **Select face** icon in the **Tool Box->Geometry** box
(2) Use the mouse icon to select the four surfaces in the y-, and z-directions
(3) Press the “Ctrl + c” and then “Ctrl + v”, and there are four options, face_0, face_1,face_2, and face_3, in the **Object Browser** box.
(4) Select the “face_0”, “face_1”, “face_2”, and “face_3”, in the **Object Browser** box, and check the **PEC** box in the **Properties** box. Click on the **Unite** button in the toolbar. Change its name to “outer_PEC” in the **Properties** box.



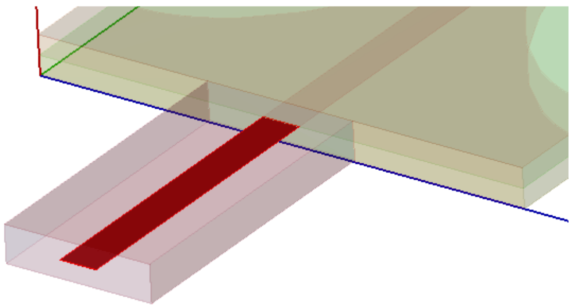
Follow the steps below to create the inner PEC structure:

(1) Select “dielectric_fill” and “outer_PEC” options in the **Object Browser** box.
(2) Check the **Hide** box in the **Properties** box.
(3) Click on the **Rectangle** icon in the **Tool Box->Geometry** box
(4) Change the drawing plane to the “X-Y” plane.

(5) Move the mouse icon to the corner of stripline, and draw a rectangle.

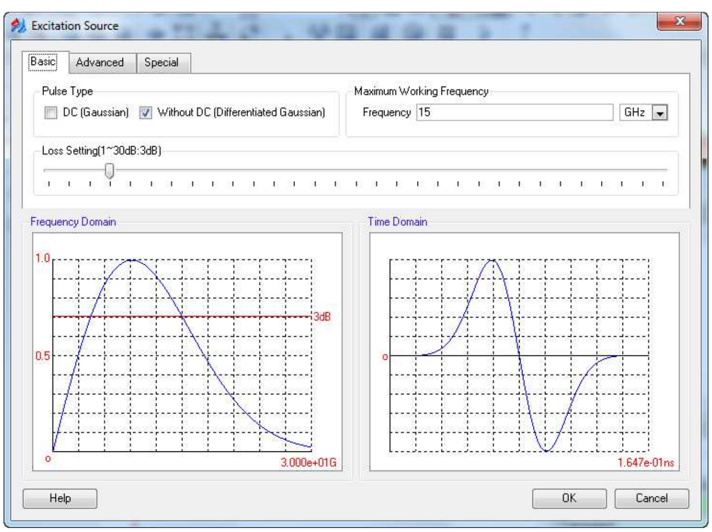
(6) Select the rectangle option in the **Object Browser** box, change its name to “inner_PEC” in the **Properties** box. And check the **PEC** box.

(7) Select the “CreateRectangle” option under the “inner_PEC” entrance in the **Object Browser** box, change its dimensions to the same width of the stripline and the same lengthy of “outer_PEC” in the **Properties** box.



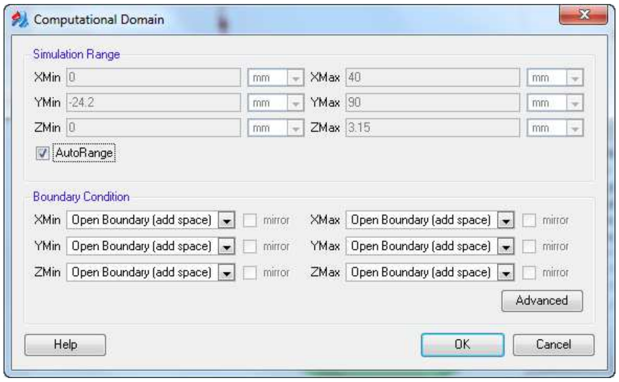
12.4 Set Excitation Pulse

Click on the **Set excitation source** button. Click on the **OK** to confirm the setting.



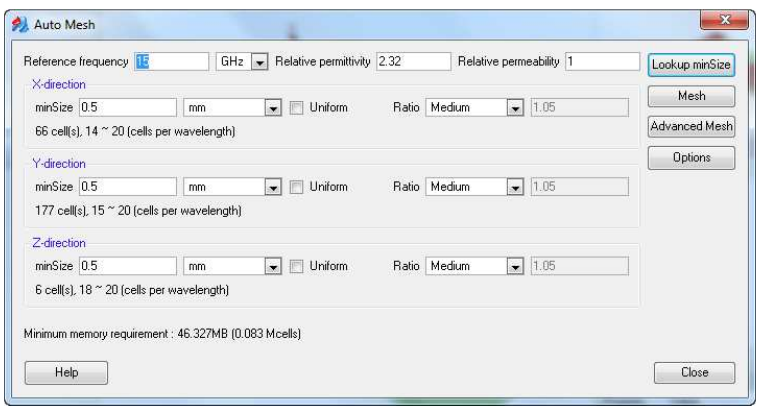
12.5 Set Domain and Boundary Condition

Click on the **Set boundary condition** button in the toolbar.

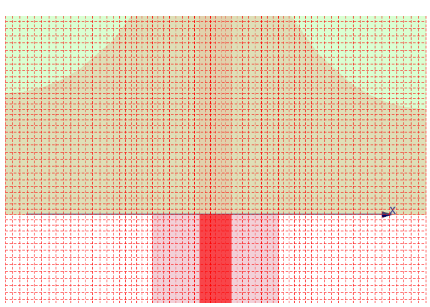


12.6 Design Mesh Distribution

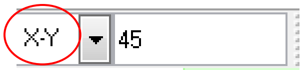
Click on the **Auto Mesh** button in the toolbar.



Push the **View grid** and **Mesh mode** button down in the toolbar.



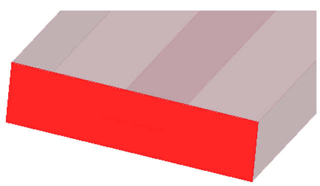
To view the mesh distribution, ensure the drawing plane and view plane to be same.



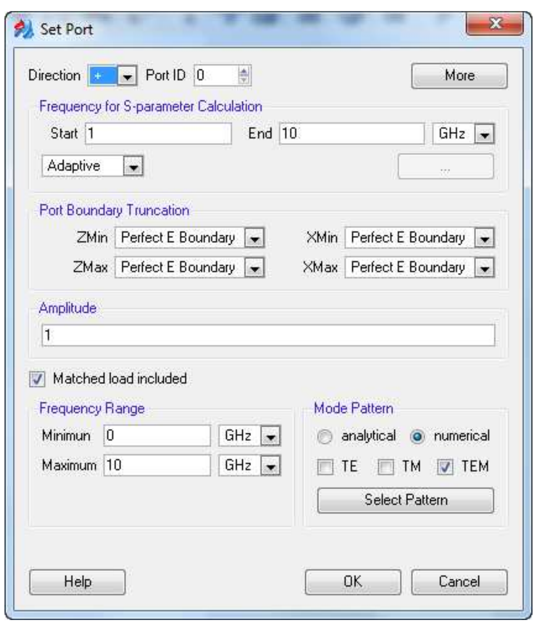
12.7 Add Excitation

Follow the steps below to create the excitation port:

(1) Click on the **Rectangle** icon in the **Tool Box->Geometry** box
(2) Change the drawing plane to the "X-Z" plane
(3) Use the mouse icon to select the corner of the feed structure and press the left mouse button.
(4) Move the mouse icon to another corner of the feed structure and press the left mouse button.
(5) Select the rectangle option in the **Object Browser** box, and change its name to "match_load".



(6) Select the **match_load** option in the **Object Browser** box, and click on the **Port** icon in the **Tool Box->Excitation Source** box.



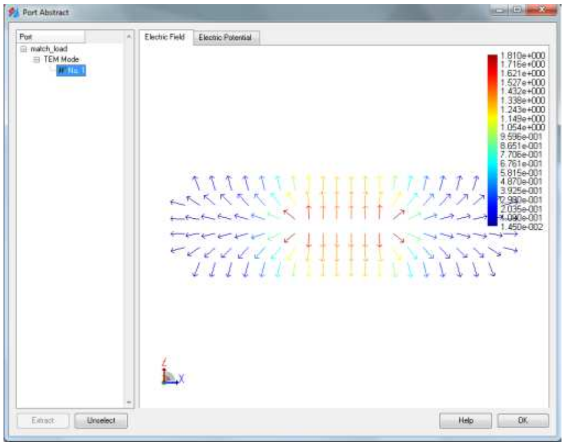
Adjust "-" or "+" in the **Direction** box to ensure the direction of port to the same as the power propagation direction.

Specify the frequency band of interest.

Check the **Matched load included** box for an open port.

Check the **numerical** radio button, and the **TEM** box in the **Mode Pattern** box.

Click on the **Pattern** button to extract the mode pattern supported by the feed structure.

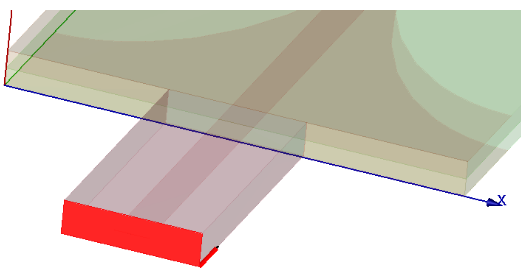


Select the “match_load” option in the **Port** tree. And click on the **Extract** button to extract the modes.

The excitation mode is marked with a bold character “**W**”.

Click on the “Electric Field” label to view the field distribution for the selected excitation mode.

Click on the **OK** button to return the **Set Port** window. Click on the **OK** button in the **Set Port** window to confirm the port settings.



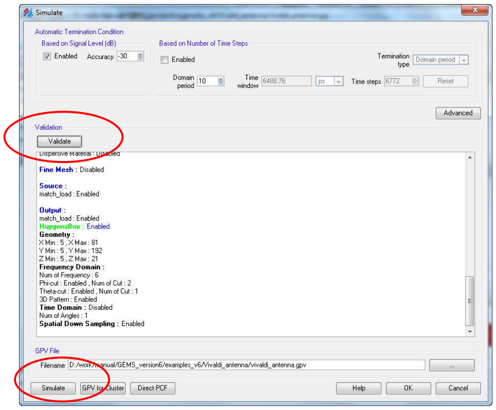
*It is worthwhile to mention that you may need to set the mode port setting first and close the **Set Port** window, and then open the **Set Port** window and extract the mode pattern.*

12.8 Save Project

Click on the **Save** button in the toolbar to save the project.

12.9 Generate Simulation File

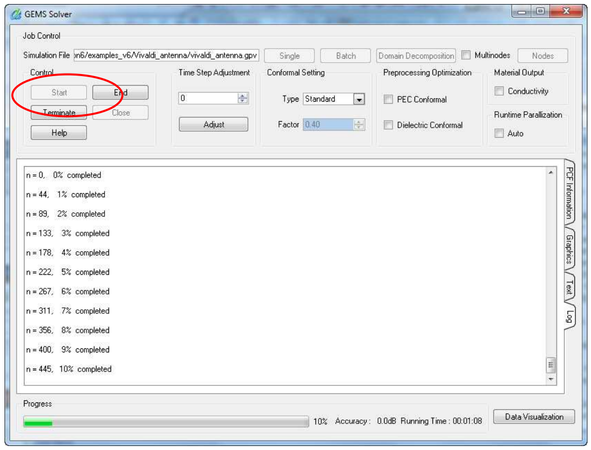
Click on the **PreCalculate** button in the toolbar.



Click on the **Validate** button and then on the **Simulate** button.

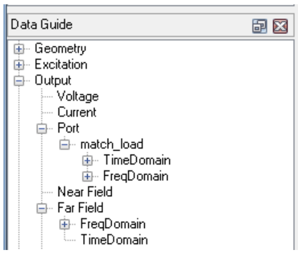
12.9 Simulate Project

Click on the **Start** button to start the simulation.

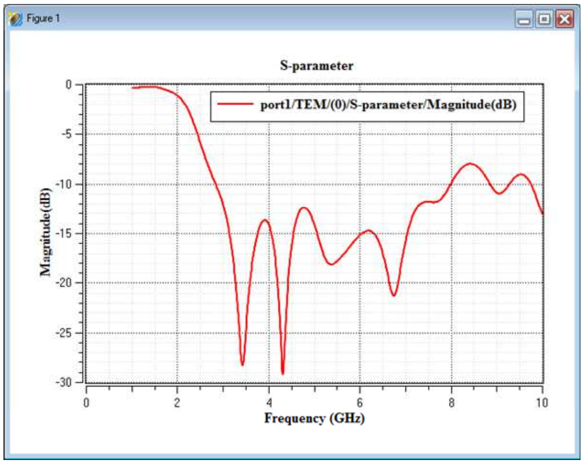


12.10 Result Visualization

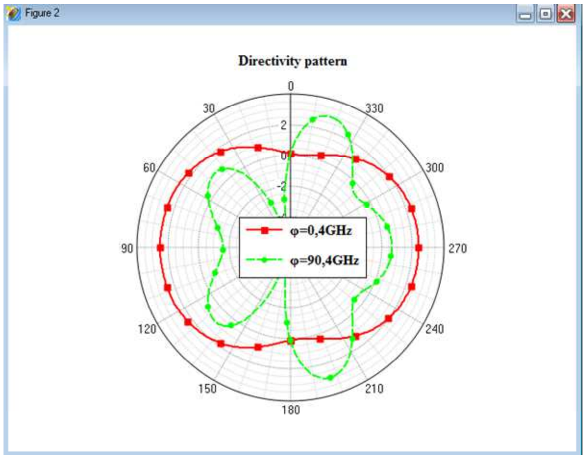
Click on the **Data Visualization** button in the *GEMS Solver* window. The outputs include the port information and far field patterns.



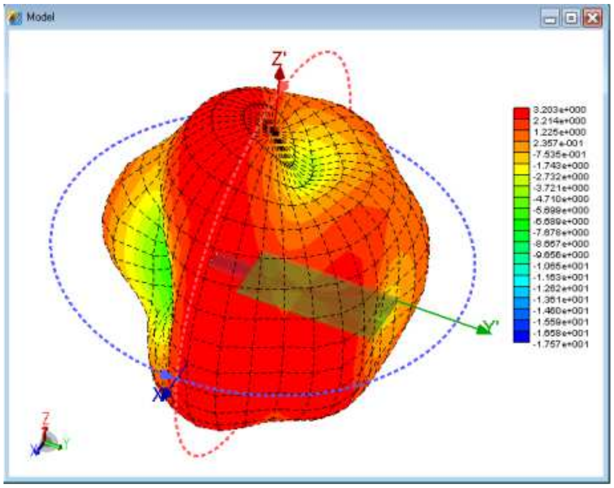
Click on the **Output->Port->match_load->FreqDomain->TEM->(0)->S parameter->Magnitude(dB)** option in the result tree.



The far field pattern at 4GHz in the H and E planes.



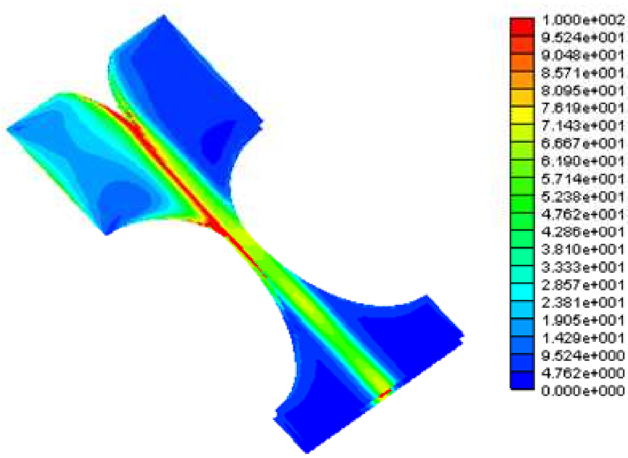
The 3-D far field pattern at 4GHz.



Click the **View model** button in the toolbar, and then select the **Geometry** folder in the result tree.

Select the "Result->View All" option, and select the "4GHz" frequency in the frequency list. Click the Play button in the toolbar to show the surface current distribution.

Select the "Result->Color Control->Color bar" option, and check the **Fixed Range** box and adjust the minimum and maximum display values.



13

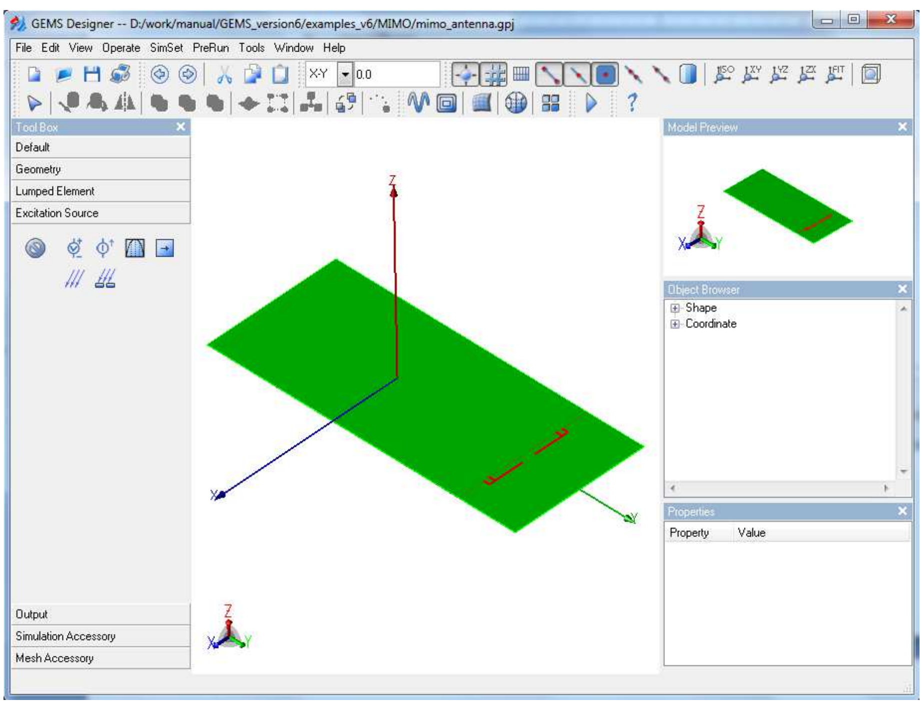
Example 13. MIMO Antenna

Description: A MIMO antenna includes two "F" shaped elements. The output parameters are the MEG (Mean Gain), and ECC (Envelope Correlation Coefficient), and far field pattern.

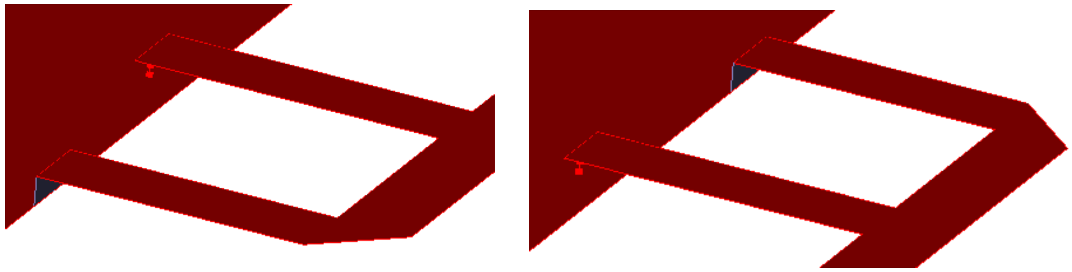
Keywords: MIMO, MEG, ECC, S-parameter matrix, and far field pattern.

13.1 Problem Configuration

The width of the F shape is 0.5mm. The width, length, and thickness of the substrate are 46.6mm, 108.7mm, and 0.2mm, respectively; and its dielectric constant and loss tangent are 3.38 and 0.0027, respectively. The dimensions of the PEC ground are 46.6mm x 88.7mm.



Two ports are shorted by two vias and another two ports are excited independently.

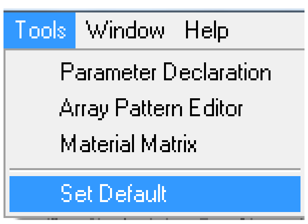


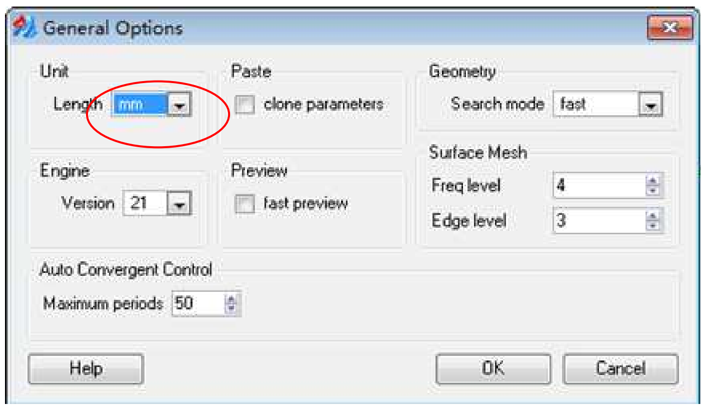
13.2 Create MIMO Antenna Model

Follow the steps below to create the MIMO antenna structure:

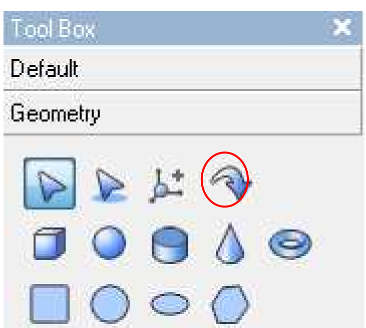
(1) Open GEMS Designer

(2) Select the **Set Default** option in the **Tools** menu

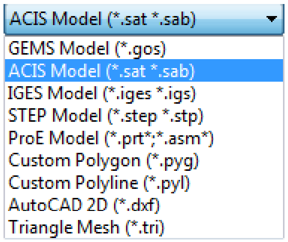


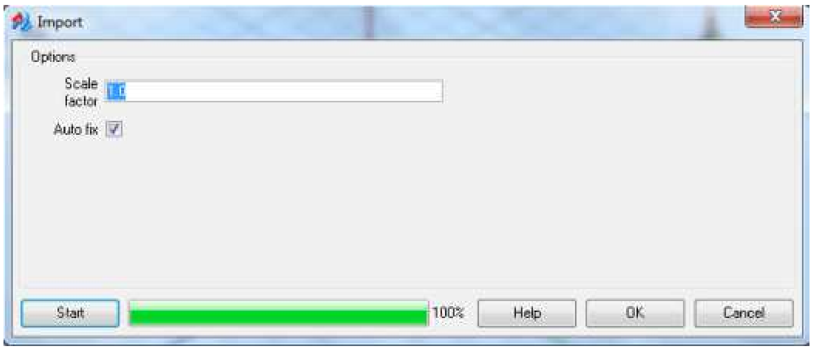


(3) Click on the **Import model** icon in the **Tool Box->Geometry** box.

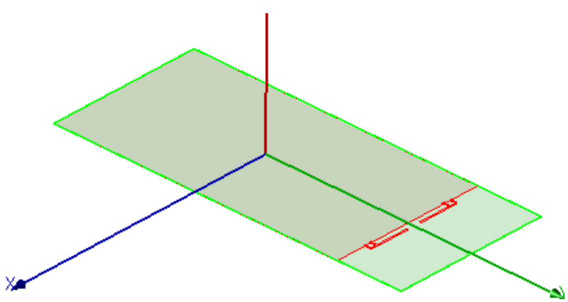


Select the "ACIS Model(*.sat, *.sab)" format in the file format list. Search for and select the "MIMO.sat" file and click on the **Open** button. Click on the **Start** button to start the processing, and then click on the **OK** button after the processing is completed.

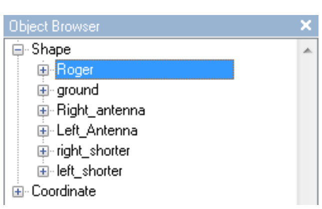




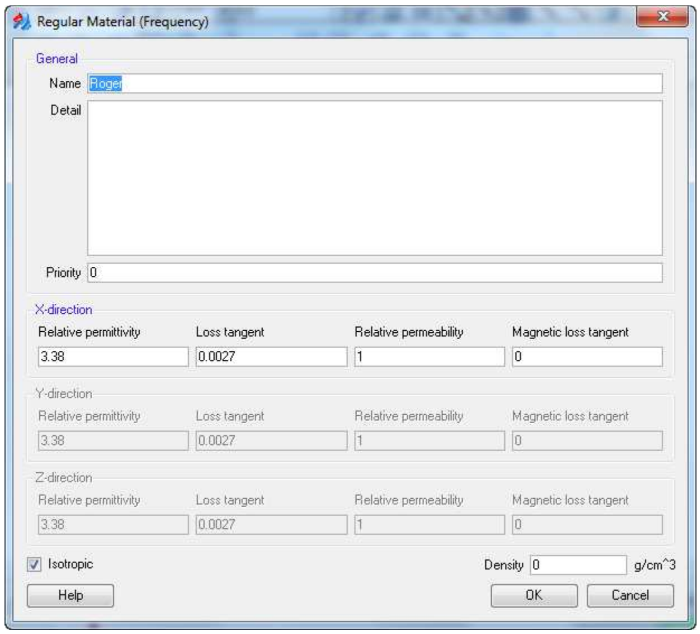
Click on the **FIT** button to fit the model in the figure region, and click on the **Select** button to the view mode.



Select the “Roger” option in the **Object Browser** box and then click on the **Material** box in the **Properties** box.



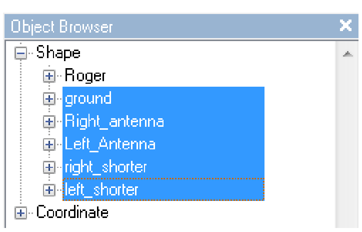
Click on the “Regular (Freq.)” label to input the dielectric parameters.

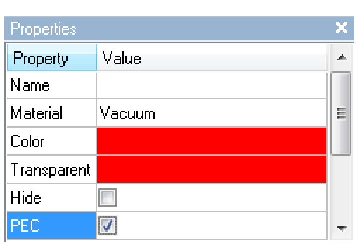


Click on the **OK** button to confirm the settings and return to the **Material** window.

Click on the **OK** button in the **Material** window to finish the material assignment.

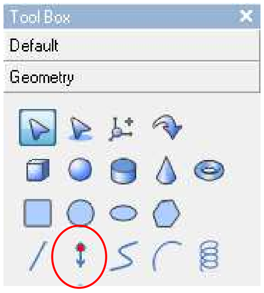
Select the “ground”, “Right_antenna”, “Left_Antenna”, “right_shorter”, and “left_shorter” options in the **Object Browser** box and then check the **PEC** box in the **Properties** box.





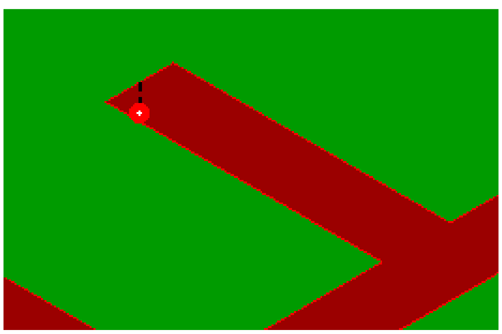
13.3 Add Excitation Ports

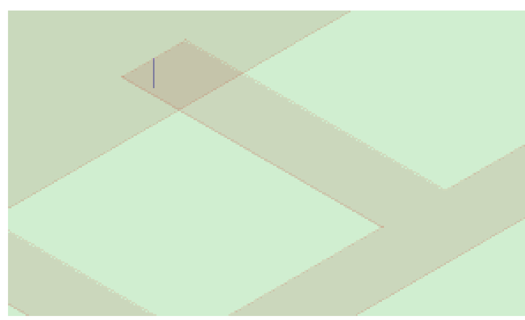
Click on the **Plumb line** icon in the **Tool Box->Geometry** box.



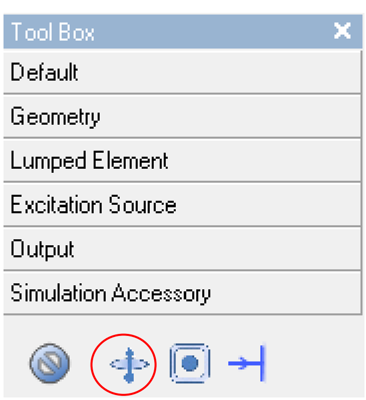
Select the drawing plane to the “X-Y” plane.

Move the mouse icon to the center of the feed structure, and move the mouse icon long the -z-direction and stops at the ground.

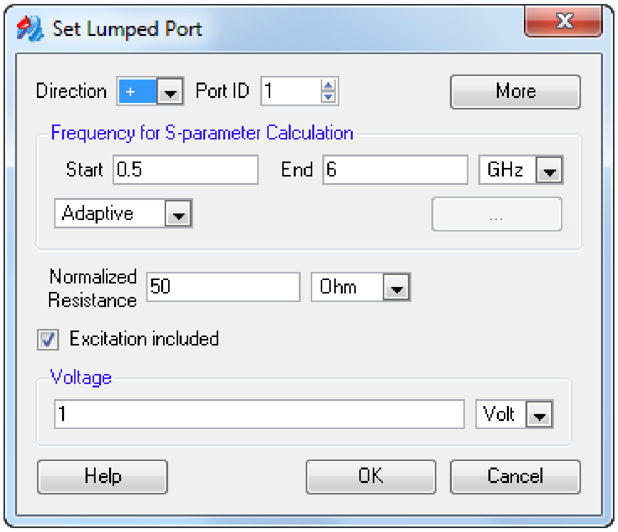


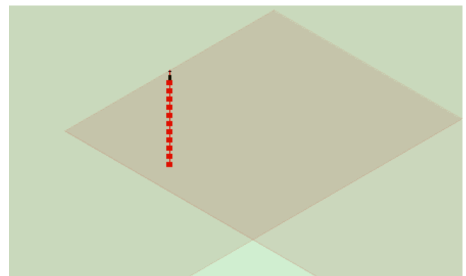


Select the **Plumb line** in the **Object Browser** box and change its name to “port1” in the **Properties** box. Click on the **Lumped port** icon in the **Tool Box->Simulation Accessory** box.

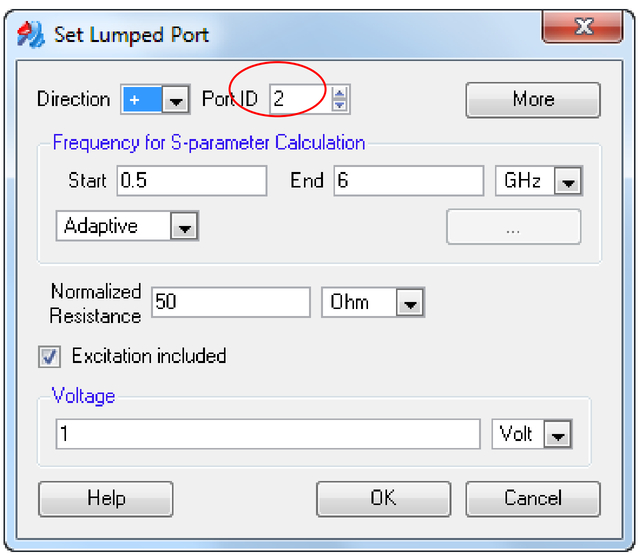


Change the Port ID to “1” and specify the frequency band of interest (4.5 ~ 6GHz).



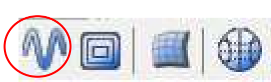


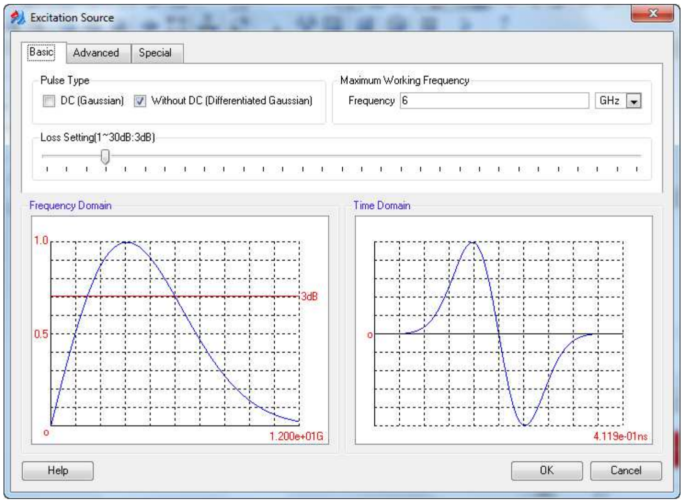
Follow the same procedure to add the second port with the Port ID “2”.



13.4 Set Excitation Pulse

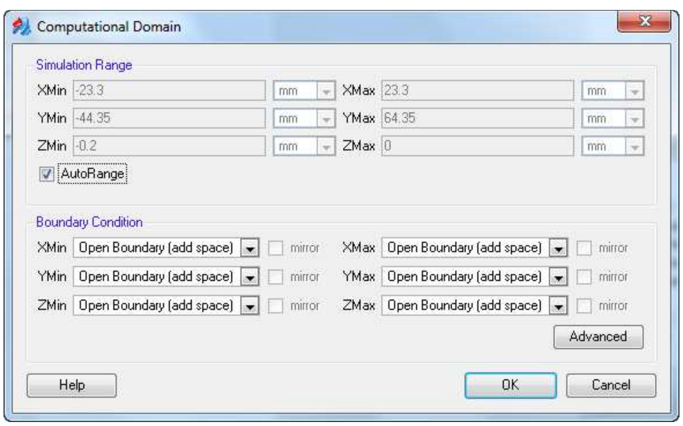
Click on the **Set excitation source** icon in the toolbar.





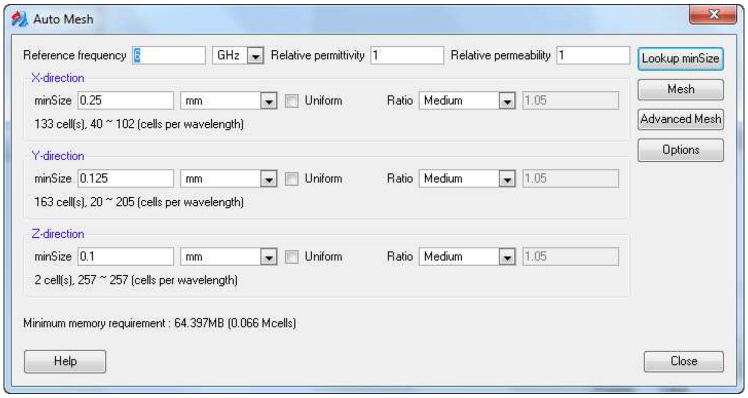
13.5 Set Domain and Boundary Condition

Click on the **Set boundary condition** icon in the toolbar.

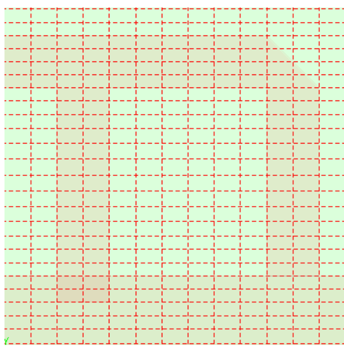


13.5 Design Mesh Distribution

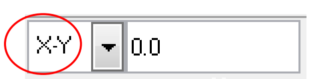
Click on the **Auto Mesh** icon in the toolbar.

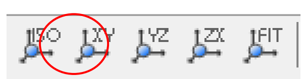


Push the **View grid** and **Mesh mode** button down in the toolbar.



To view the mesh distribution, ensure the drawing plane and view plane to be same.





13.6 Save Project

Click on the **Save** icon in the toolbar.

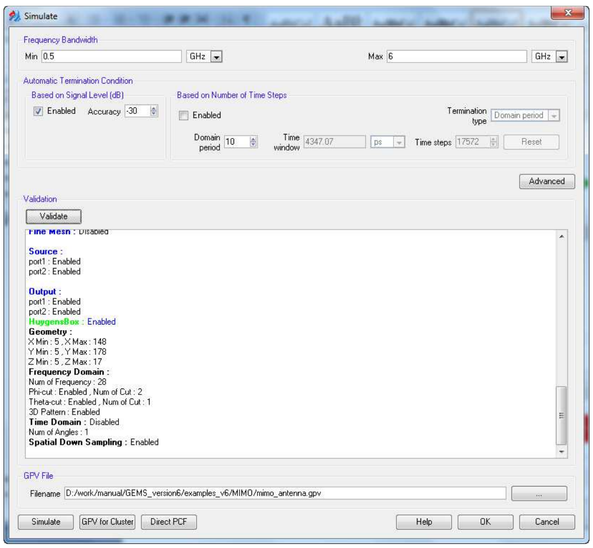
Create a folder and specify a name for this project.

13.7 Generate Simulation File

Click on the **PreCalculate** icon in the toolbar.

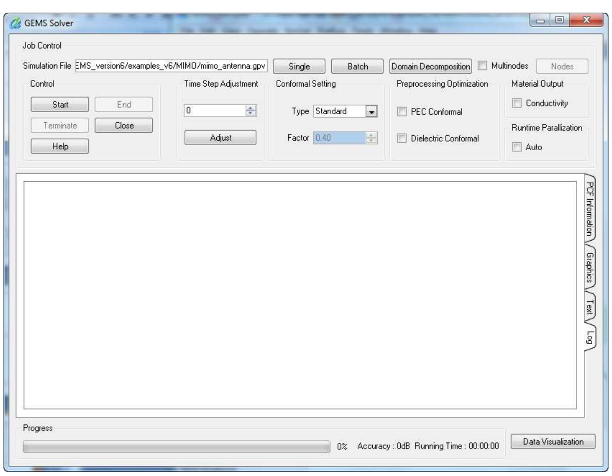
Specify the frequency range of interest in the **Frequency Band** box.

Click on the **Validate** button and **Simulate** button to open the *GEMS Solver* window.



13.8 Simulate Project

Click on the **Simulate** button in the *Simulate* window. Click on the **Start** button to start the simulation.



13.9 Data Visualization

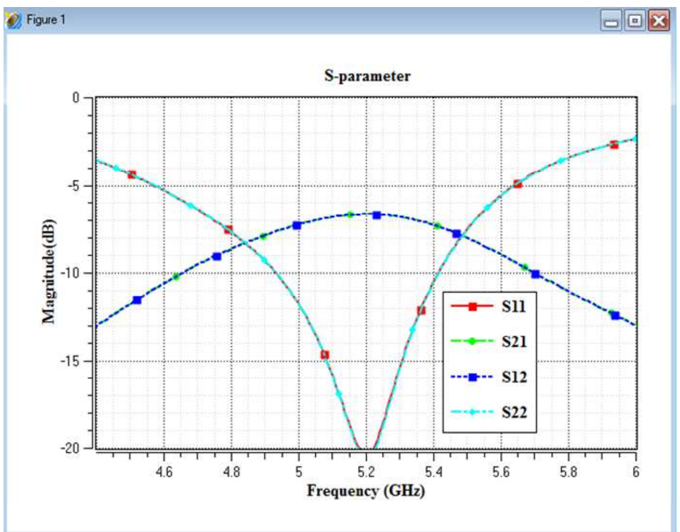
Click on the **Data Visualization** button in the *GEMS Solver* window.

Double-click on the **Output->Port->Port1->FreqDomain->1->S parameter->Magnitude(dB)** option in the result tree.

Select the **Output->Port->Port1->FreqDomain->2->S parameter->Magnitude(dB)** option in the result tree, and click on the **Add to current window** button in the toolbar.

Select the **Output->Port->Port2->FreqDomain->1->S parameter->Magnitude(dB)** option in the result tree, and click on the **Add to current window** button in the toolbar.

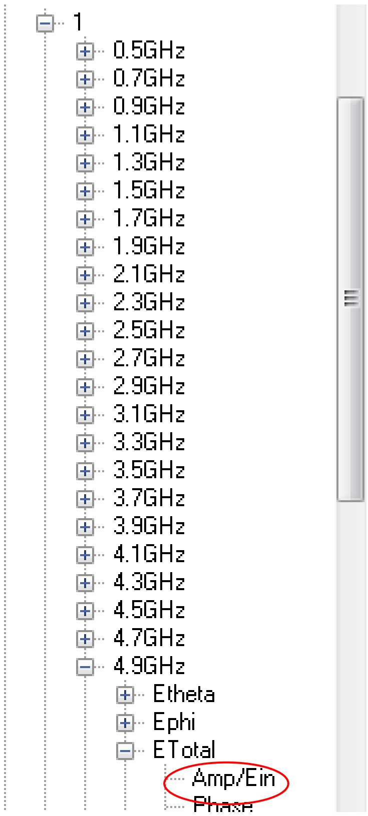
Select the **Output->Port->Port2->FreqDomain->2->S parameter->Magnitude(dB)** option in the result tree, and click on the **Add to current window** button in the toolbar.

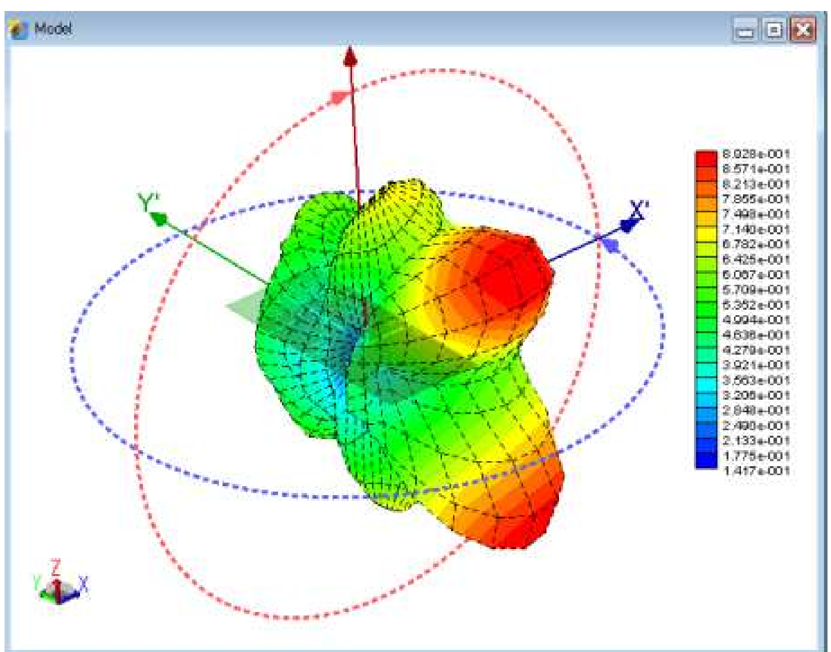


Double-click on the margin of figure and change the display range to “4.5 ~ 6.0GHz” in the x-direction and the display range to “0 ~ -30dB” in the y-direction.

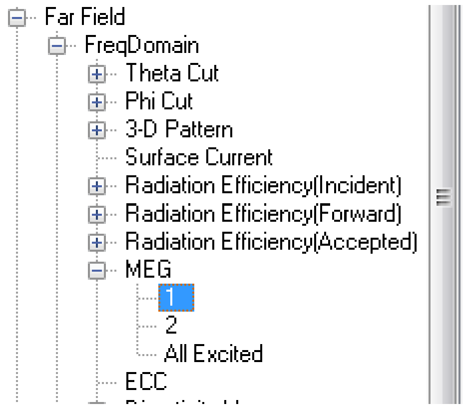
Double-click on the **Output->Port->Port1->FreqDomain->1->S parameter->Magnitude(dB)** option in the result tree.

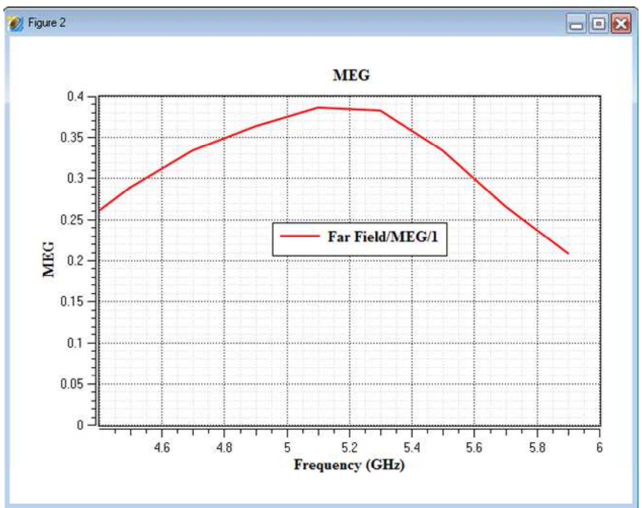
Click on the **View model** button in the toolbar, select the option of 3D far field pattern at 4.9GHz in the result tree, and then click on the **Add to current window** button in the toolbar, and then on the **Play** button in the toolbar..



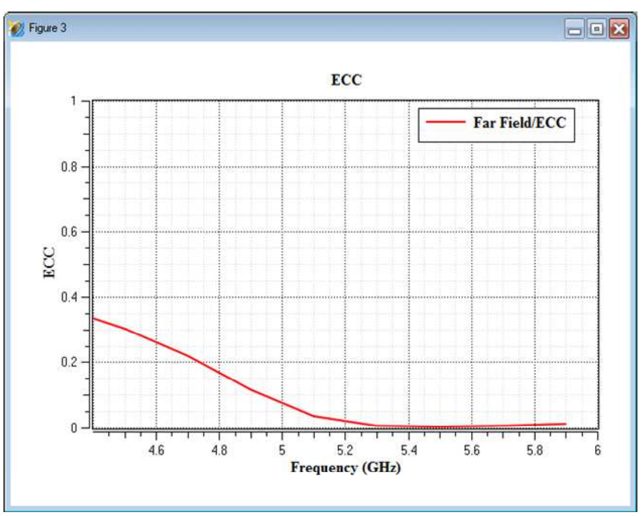


Double-click on the **MEG** option in the **Far field** folder in the result tree.





Double-click on the **ECC** option in the **Far field** folder in the result tree.



14

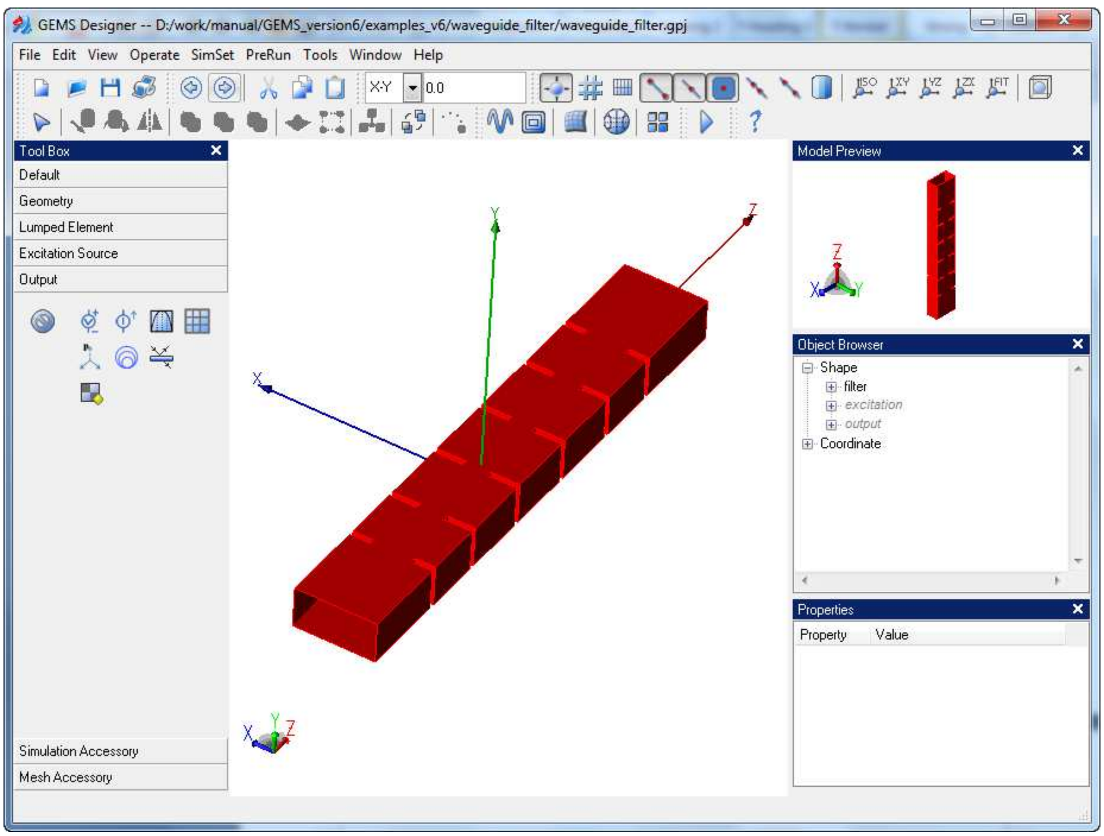
Example 14. Microwave Filter

Description: A microwave filter is consisted of five cavities and is excited by the TE_{10} mode. The output parameters are S-parameters.

Keywords: High-Q system, microwave filter, and S-parameters.

14.1 Problem Configuration

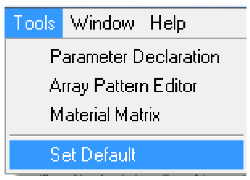
A waveguide WR75 (width=19.5mm, and height=9.525mm) includes five cavities with the different lengths. The cut-off frequencies of TE_{10} mode is 7.868GHz. The waveguide filter is excited by the TE_{10} mode and the outputs are measured at the two ports.

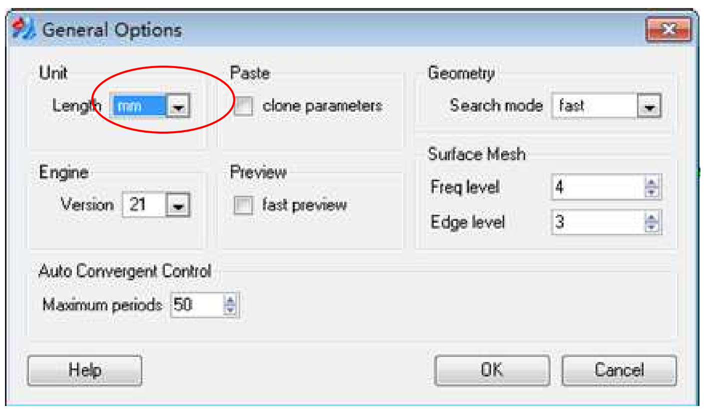


14.2 Create Filter Model

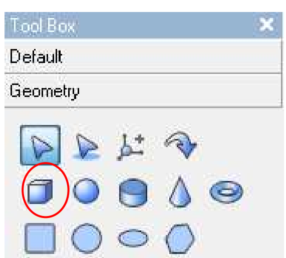
Follow the steps below to create the filter structure:

(1) Open GEMS Designer
(2) Select the **Set Default** option in the **Tools** menu





(3) Click on the **Cuboid** icon in the **Tool Box->Geometry** box.

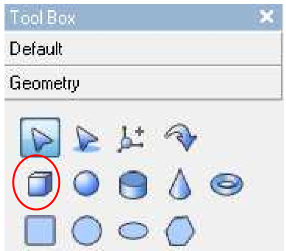


Draw a box with the dimensions 19.5mm x 9.525mm x 110.5434mm.

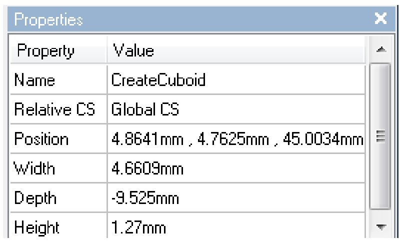
Select the box option in the **Object Browser** box and change its name to “filter”.

Select the **filter** option in the **Object Browser** box and check the **PEC** box in the **Properties** box.

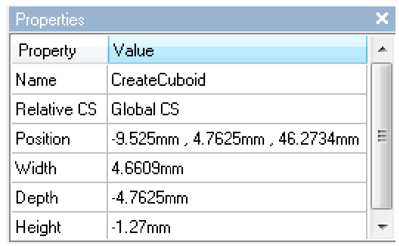
(4) Click on the **Cuboid** icon in the **Tool Box->Geometry** box.



Draw two boxes with the dimensions shown in the tables below:

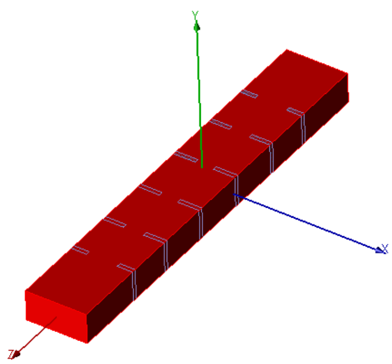


Property	Value
Name	CreateCuboid
Relative CS	Global CS
Position	4.8641mm , 4.7625mm , 45.0034mm
Width	4.6609mm
Depth	-9.525mm
Height	1.27mm

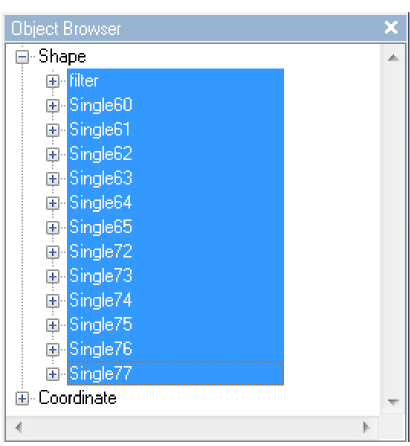


Property	Value
Name	CreateCuboid
Relative CS	Global CS
Position	-9.525mm , 4.7625mm , 46.2734mm
Width	4.6609mm
Depth	-4.7625mm
Height	-1.27mm

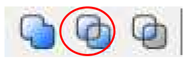
Draw 10 more boxes in the figure region.

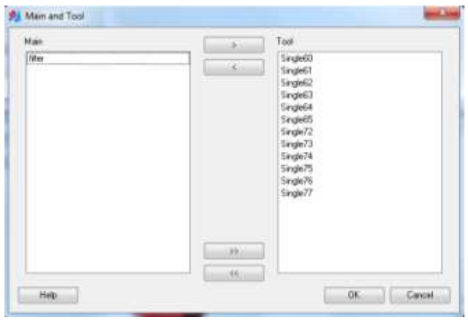


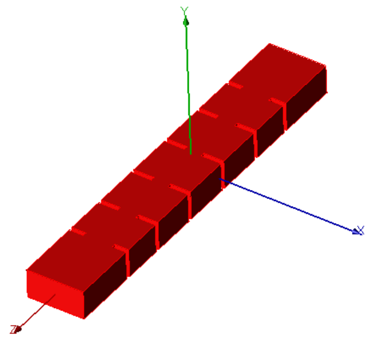
(5) Select the **filter** and 12 box options in the **Object Browser** box.



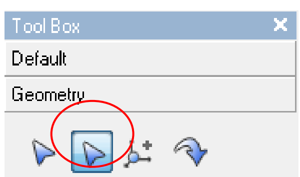
(6) Click on the **Subtract** button in the toolbar. Click on the **OK** button to confirm the operation.







(7) Select the **filter** option in the **Object Browser** box, and click on the **Select face** icon. Move the mouse icon to select one surface of the filter and then press the “Ctrl + a” keys.

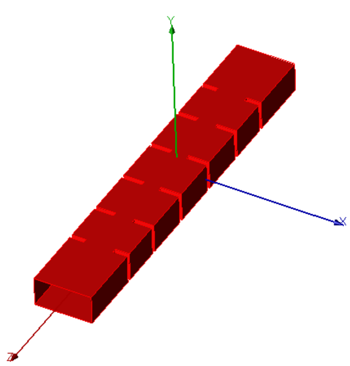


(8) Deselect the two end surfaces, and then press the “Ctrl + c” and then “Ctrl + v” keys.

(9) Select the **filter** option in the **Object Browser** box and delete it.

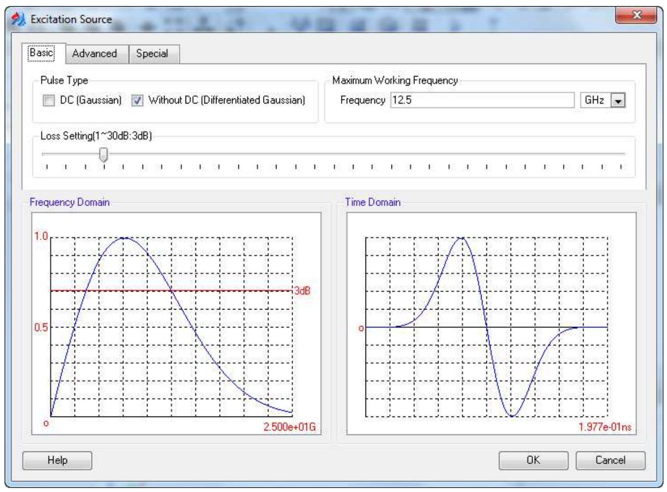
(10) Select all the options in the **Object Browser** box and then click on the **Unite** button in the toolbar.

Select the united object in the **Object Browser** box, change its name to “filter”. Check the **PEC** box, and change its color to red.



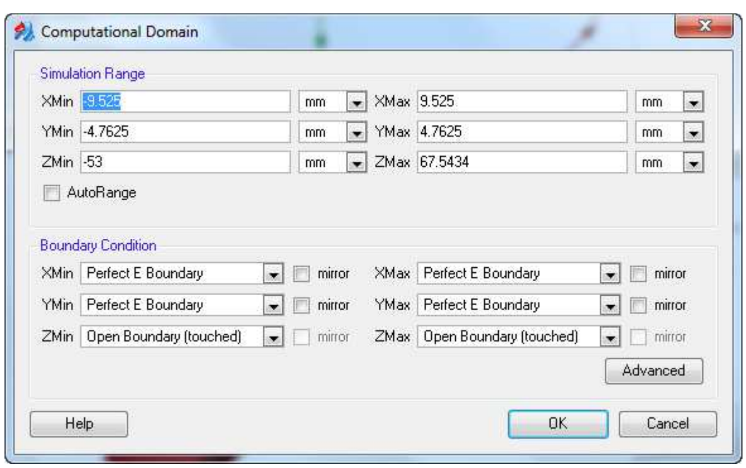
14.3 Set Excitation Pulse

Click on the **Set excitation source** button in the toolbar.



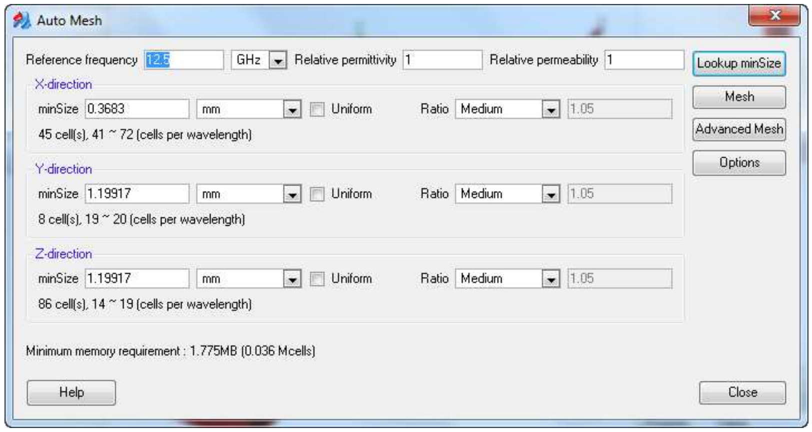
14.4 Set Domain and Boundary Condition

Click on the **Set boundary condition** button in the toolbar.



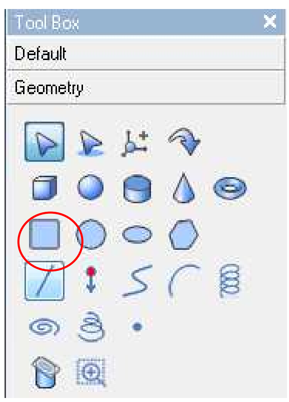
14.5 Design Mesh Distribution

Click on the **Auto mesh** button in the toolbar.

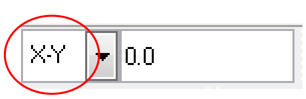


14.6 Create Excitation Port

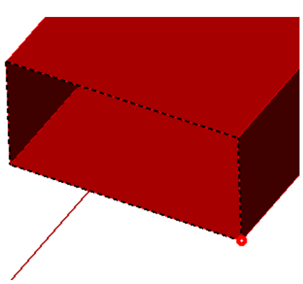
Click on the **Rectangle** icon in the **Tool Box->Geometry** box.

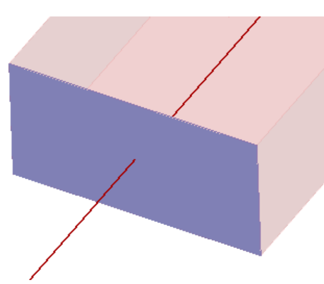


Select the drawing plane to the “X-Y” plane.



Use the mouse icon to snap to the corners of end waveguide.

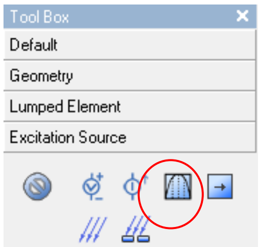


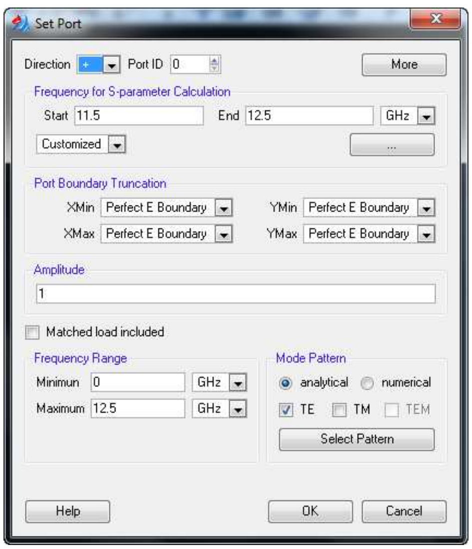


Select the rectangle option in the **Object Browser** box and change its name to "excitation" in the **Properties** box.

Select the "excitation" option in the **Object Browser** box and click on the **Port** icon in the **Tool Box->Excitation Source** box.

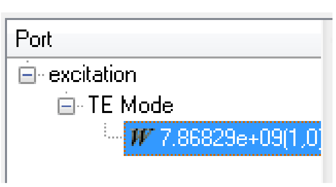
Specify the frequency range of interest, and adjust the port direction to the power propagation direction.



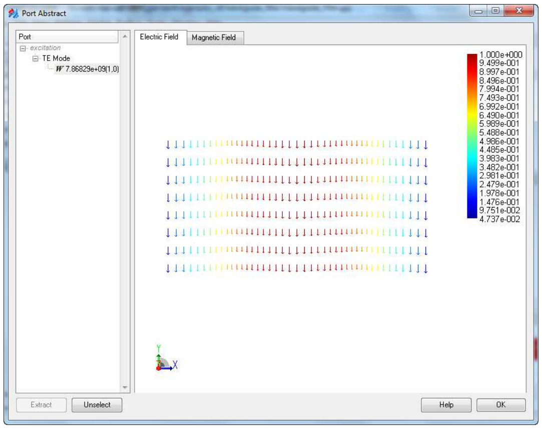


Click on the radio button "analytic" and check the **TE** mode. Click on the **Select Pattern** button.

Select the frequency under the **TE mode** entrance

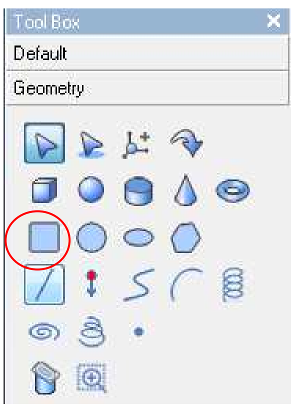


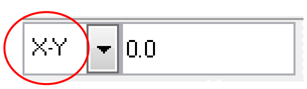
Click on the **Electric Field** label to view the mode pattern.



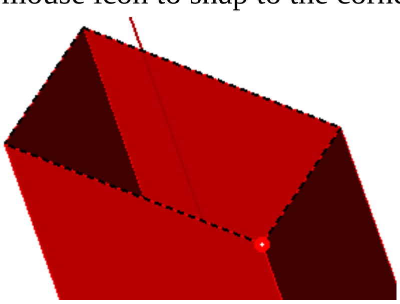
14.7 Create Output Port

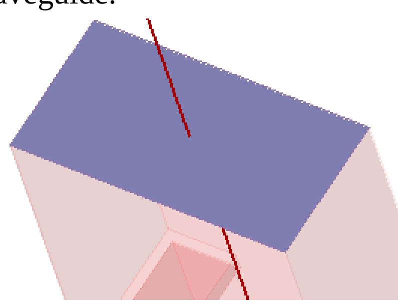
Click on the **Rectangle** icon in the **Tool Box->Geometry** box. Select the drawing plane to the "X-Y" plane.





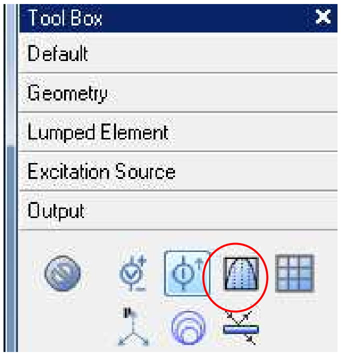
Use the mouse icon to snap to the corners of end waveguide.

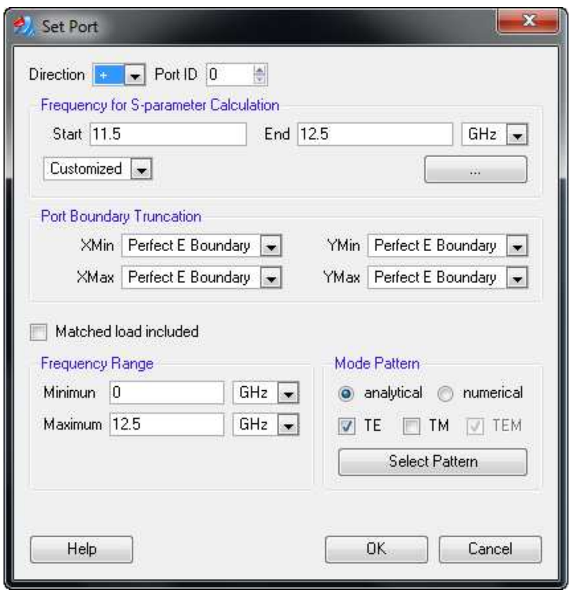




Select the rectangle option in the **Object Browser** box and change its name to "output" in the **Properties** box.

Select the “output” option in the **Object Browser** box and click on the **Port** icon in the **Tool Box->Output** box. Adjust the port direction to the power propagation direction.



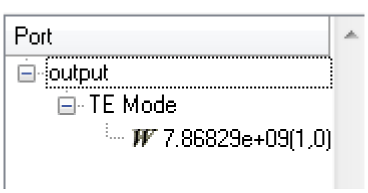


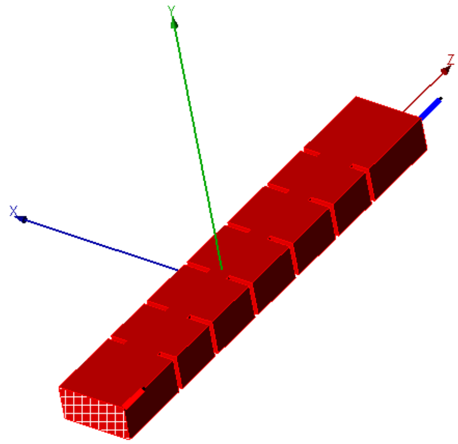
Click on the radio button “analytic” and check the **TE** mode.

Click on the **Select Pattern** button.

Select the **Output** option in the Port tree and then click on the **Extract** button.

Select the **Output->TE Mode->7.86829GHz(1,0)** option, and then click on the **Select** button.



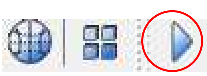


14.8 Save Project

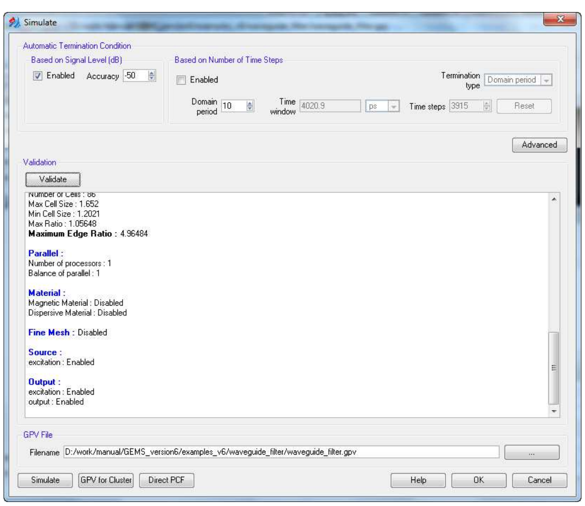
Click on the **Save** icon in the toolbar.

14.9 Generate Simulation File

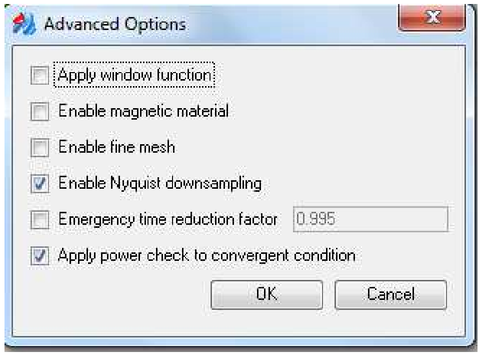
Click on the **PreCalculate** icon in the toolbar.



Change the convergence criterion to “-50dB” and then click on the **Advanced** button.



For the high-Q system, to check the power convergence, we need to check the **Apply power check to convergence condition** box.

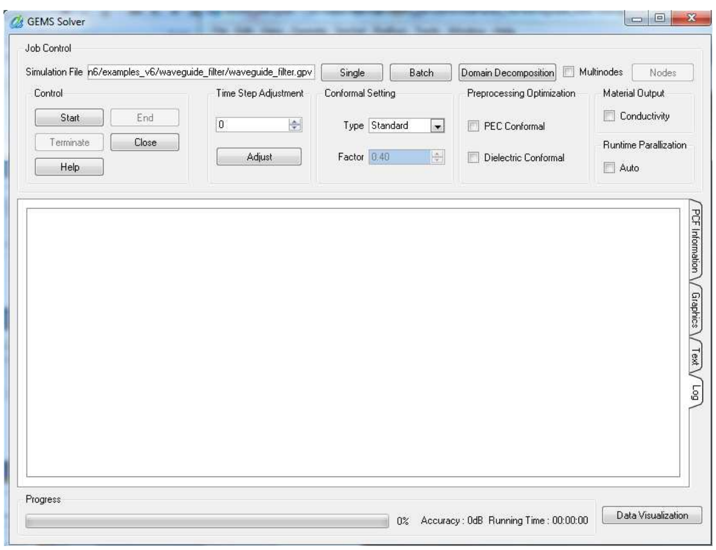


Click on the **OK** button to confirm the selection.

Click on the **Validate** button to validate the project settings, and then click on the **Simulate** button to open the *GEMS solver* window.

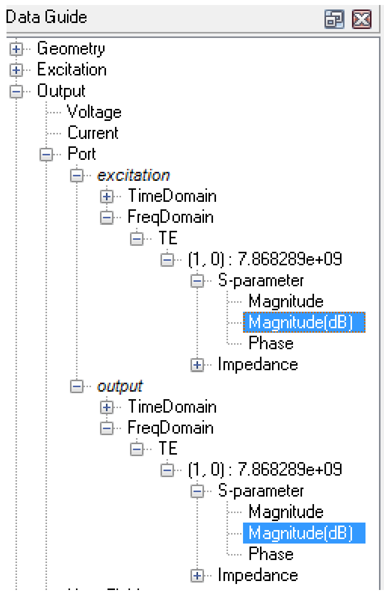
14.10 Simulate Project

Click on the **Start** button to start the simulation.

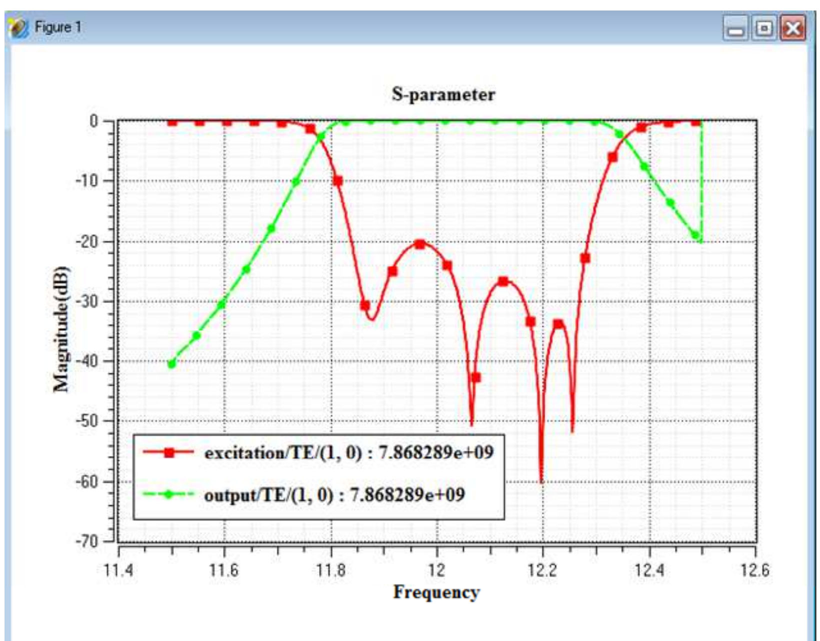


14.11 Result Visualization

Click on the **Data Visualization** button in the *GEMS Solver* window.



Click on the **Data Visualization** button in the *GEMS Solver* window. Double-click on the **Port->excitation->FreqDomain->TE->(1,0)->S-parameter->Magnitude(dB)** option. Select the **Port->output->FreqDomain->TE->(1,0)->S-parameter->Magnitude(dB)** option, and click on the **Add to current window** button in the toolbar.



15

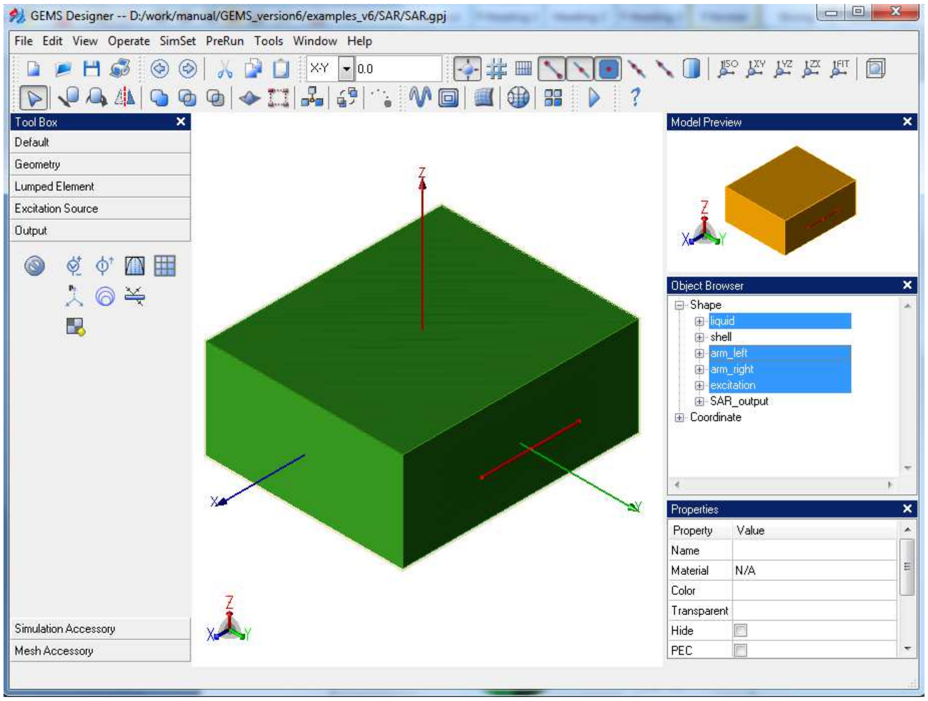
Example 15. SAR (Specific Absorption Rate)

Description: A SAR test case was taken from IEEE 1528 Standards to validate GESM software. A dipole antenna is used to excite the phantom. The output parameters are 1-g, 10-g and peak SAR, and 3D SAR distribution.

Keywords: 1-g, 10-g and peak SAR, IEEE 1528 Standards, dipole antenna, and far field pattern.

15.1 Problem Configuration

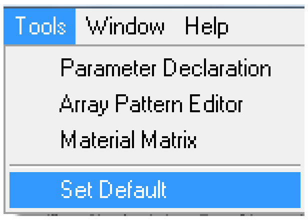
The phantom dimensions in this example for 900MHz are taken from the IEEE 1528 Standards. The length and radius of the dipole are 149mm and 1.8mm, respectively. The feed gap size is 1.8mm. The distance between the plastic shell and dipole (center) is 15mm. The dielectric constant, conductivity, and mass density of the human equivalent liquid (360x300x150mm) are 41.5, 0.97S/m, and 1000kg/m^3 respectively. The dielectric constant of the plastic shell is 3.7.

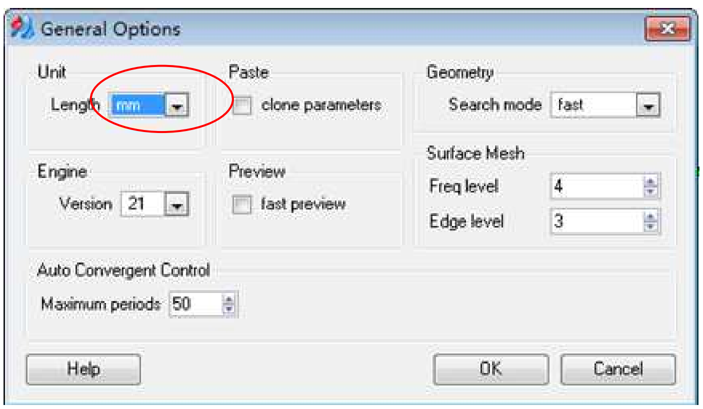


15.2 Create Project Model

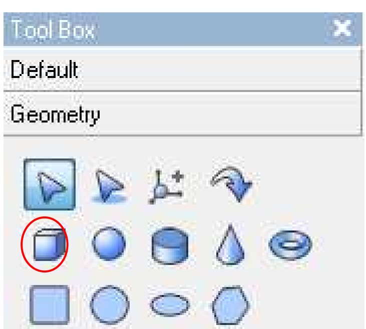
Follow the steps below to create the phantom model:

(1) Open GEMS Designer
(2) Select the **Set Default** option in the **Tools** menu





(3) Click on the **Cuboid** icon in the **Tool Box->Geometry** box.

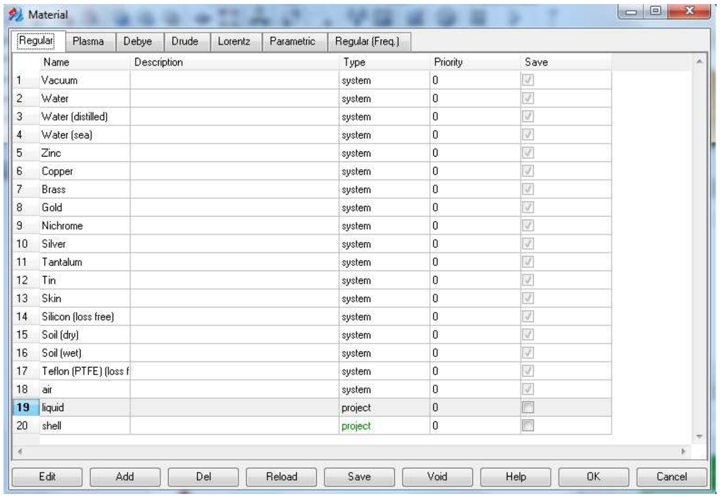


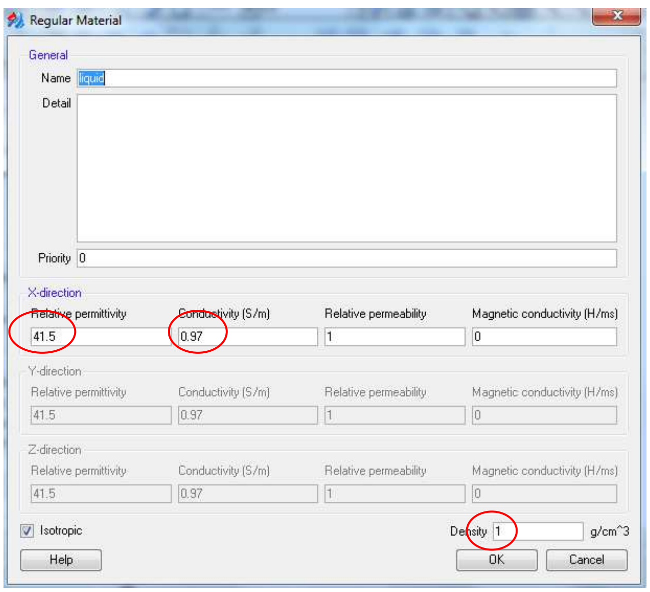
Draw a box with the dimensions 360mm x 300mm x 150mm.

Select the box option in the **Object Browser** box and change its name to “liquid”.

Select the **liquid** option in the **Object Browser** box and click on the **Material** box in the **Properties** box.

Click on the **Add** button to add a new material.

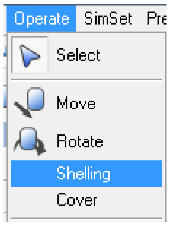




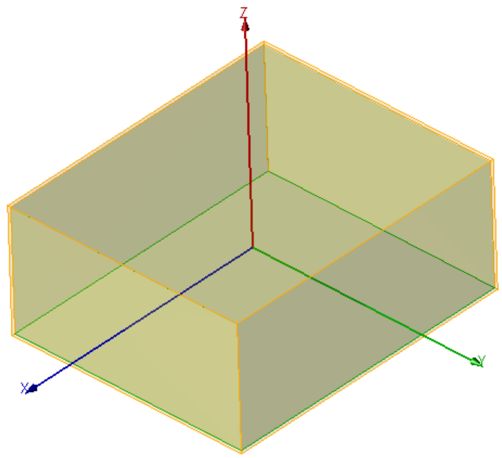
(4) Select the "liquid" option in the **Object Browser** box, and press the "Ctrl + c" and then "Ctrl + v" keys.

Change name of the pasted box to "shell" in the **Properties** box.

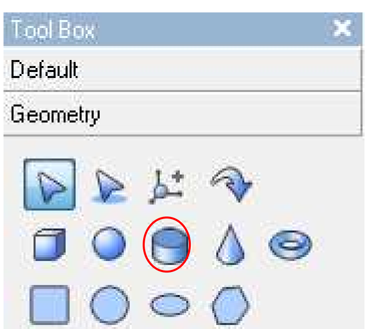
Select the "shell" option in the **Object Browser** box, and select the "Shelling" option in the **Operate** menu.



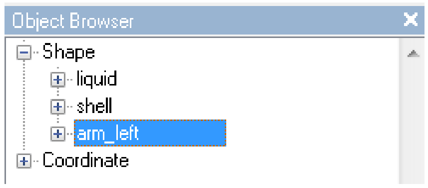
Select the “shell” option in the **Object Browser** box, and assign its dielectric constant to 3.7.



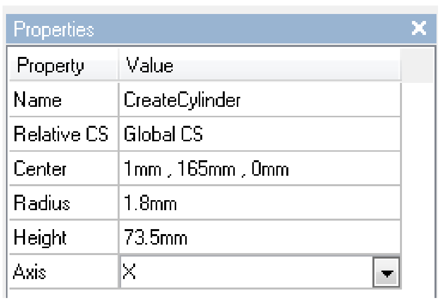
(5) Click on the **Cylinder** icon in the **Tool Box->Geometry** box.

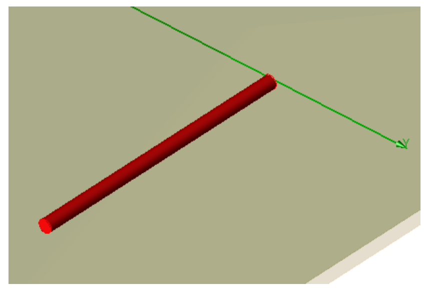


Select the cylinder option in the **Object Browser** box and change its name to “arm_left”, and check the **PEC** box in the **Properties** box.



Select the "CreateCylinder" option under the **arm_left** entrance in the **Object Browser** box and change its dimensions to (1, 165, 0), radius=1.8mm, and length=73.5mm in the **Properties** box.





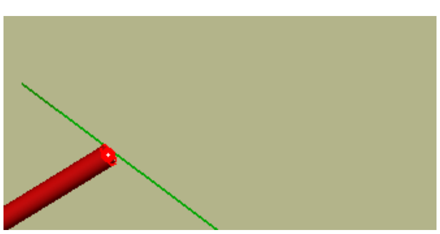
(6) Select the **arm_left** option in the **Object Browser** box.

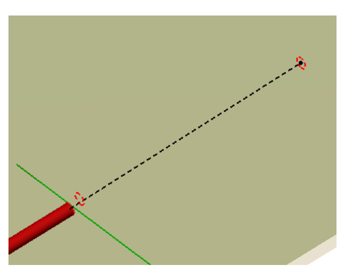
Press the "Ctrl + c" and then "Ctrl + v" keys.

Select the pasted cylinder option in the **Object Browser** box and change its name to "arm_right".

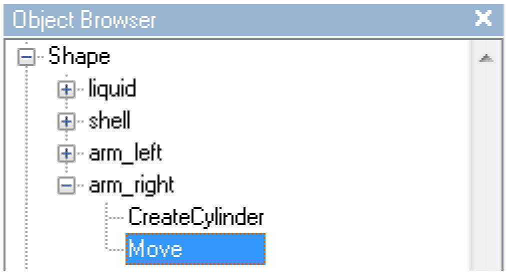
Select the **arm_right** option in the **Object Browser** box and change the drawing plane to "X-Y" plane, and click on the **Move** button in the toolbar.

Snap to a point on the "arm_left" object, and move the mouse icon along the –x-direction.



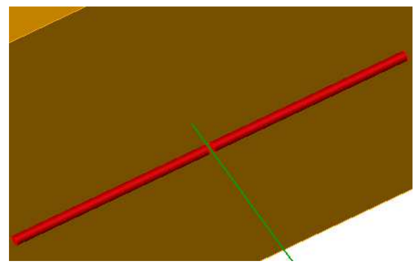


Select the **Move** option in the **arm_right** entrance.



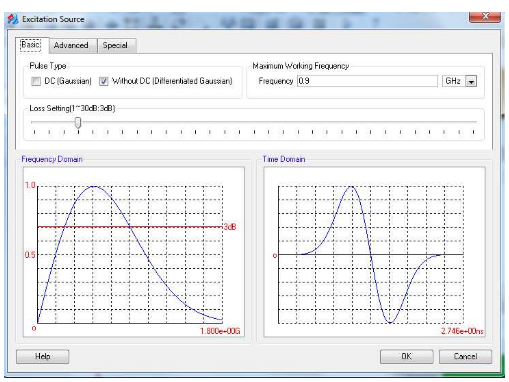
Change the moving parameters to (-75.5mm, 0, 0).

Property	Value
Name	Move
Relative CS	Global CS
Delta	-75.5mm , 0mm , 0mm



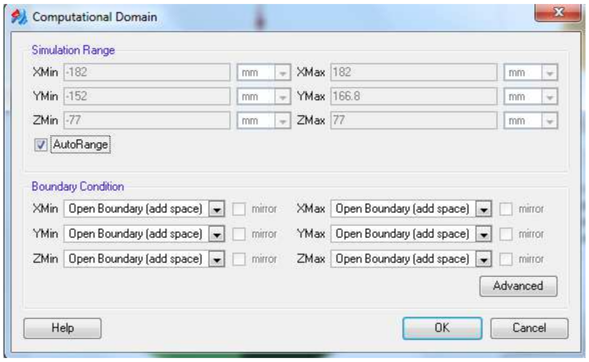
15.3 Set Excitation Pulse

Click on the **Set excitation source** button in the toolbar.



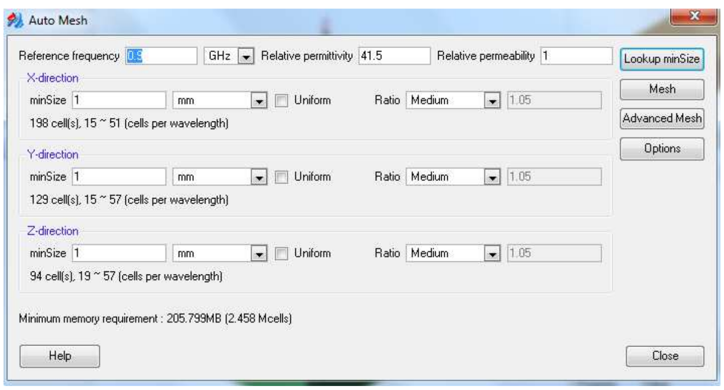
15.4 Set Domain and Boundary Condition

Click on the **Set boundary condition** button in the toolbar.



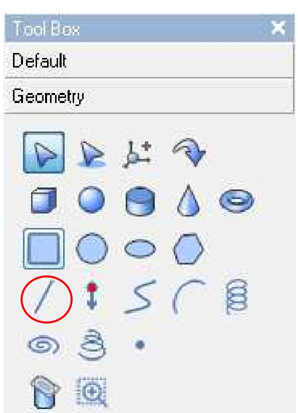
15.5 Design Mesh Distribution

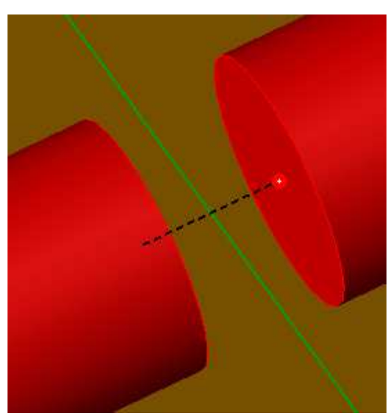
Click on the **Auto Mesh** button in the toolbar.



15.6 Add Excitation Source

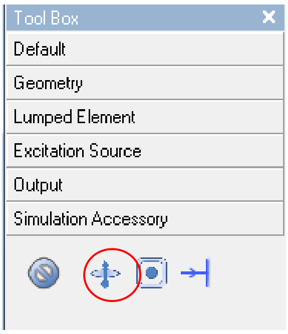
Click on the **Line** button in the **Tool Box->Geometry** box. Move and snap to the center of left arm and press the left mouse button, and move the mouse icon and snap to the center of the right arm, and press the left mouse button.

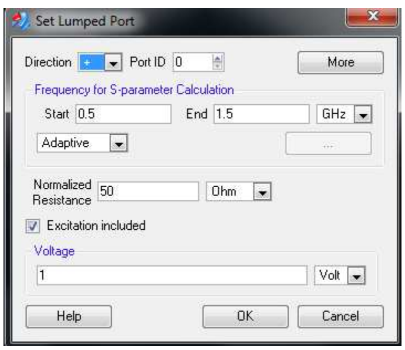


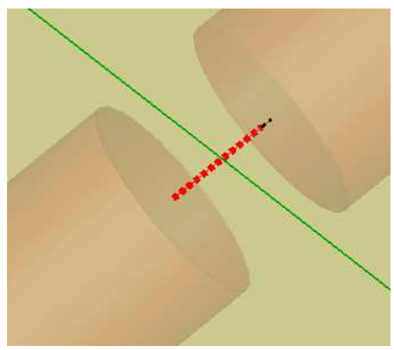


Select the line option in the **Object Browser** box, and change its name to "excitation" in the **Properties** box.

Select the "excitation" option in the **Object Browser** box, and click on the **Lumped port** icon in the **Tool Box->Accessory** box.





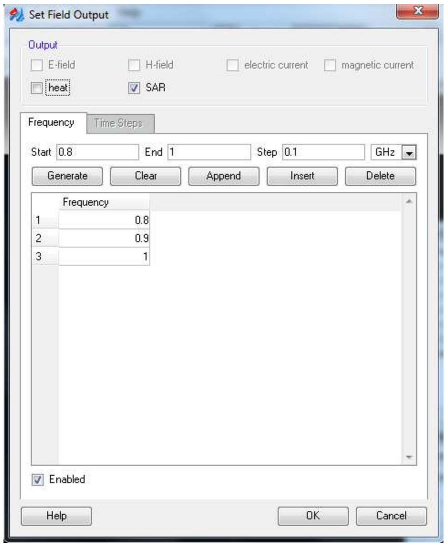


15.7 Add Outputs

Follow the steps below to add SAR output:

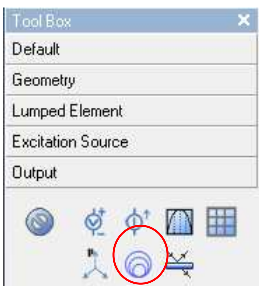
(1) Select the **liquid** option in the **Object Browser** box
(2) Press the "Ctrl + c" and then "Ctrl + v" keys
(3) Select the pasted object option in the **Object Browser** box and change the name of the pasted object to "SAR_output" in the **Properties** box.
(4) Select the "SAR_output" option in the **Object Browser** box and click on the **Field** icon in the **Tool Box->Output** box.

(5) Check the **Enabled** box and leave the **SAR** box checked. Click on the **Insert** button and type 0.9GHz in the **Frequency** list. Click on the **OK** button to confirm the setting.

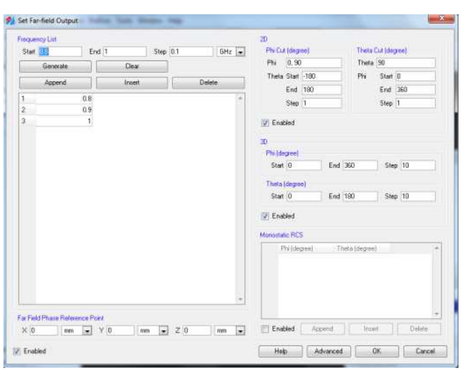


Follow the steps below to add far field output:

(1) Click on the **Far field** icon in the **Tool Box->Output** box.



(2) Check the **Enabled** box
(3) Type “0.8” in the **Start** box, “1.0” in the **End** box, and “0.1” in the **Step** box
(4) Click on the **Generate** button
(5) Check the **3D->Enabled** box to output the 3D far field pattern
(6) Click on the **OK** button to confirm the settings.



15.8 Save Project

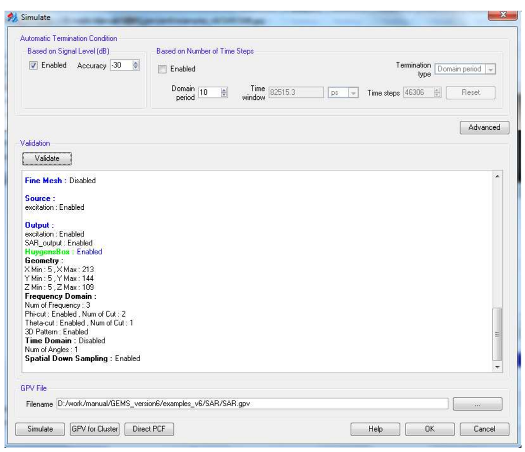
Click on the **Save** button in the toolbar to save the project.

15.9 Generate Simulation File

Click on the **PreCalculate** button in the toolbar.

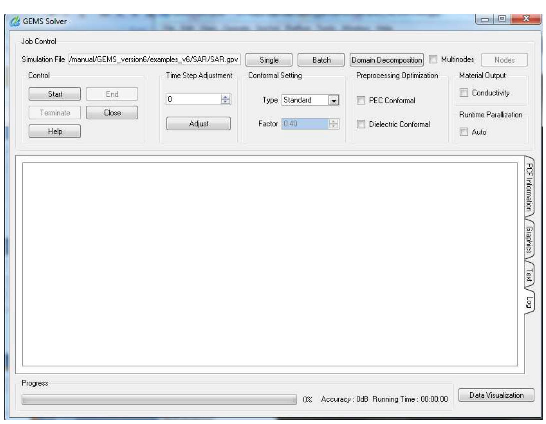
Click on the **Validate** button to validate the project settings.

Click on the **Simulate** button to open the **GEMS Solver** window.



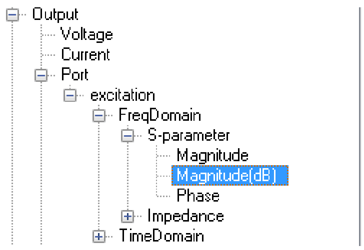
15.10 Simulate Project

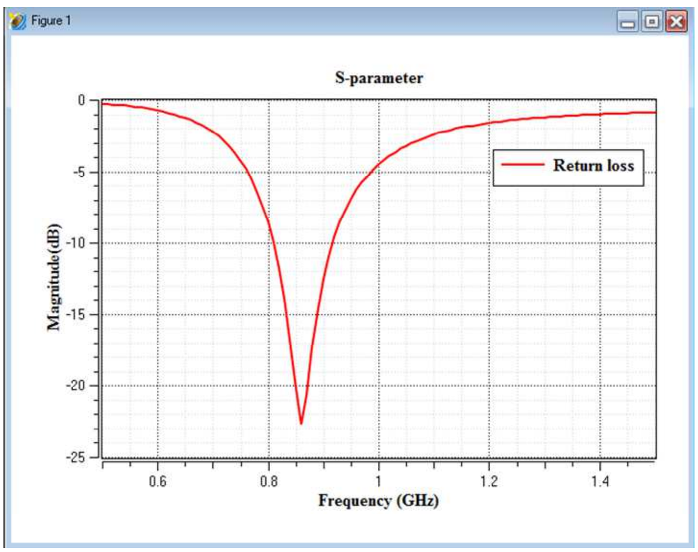
Click on the **Simulate** button in the **Simulate** window.



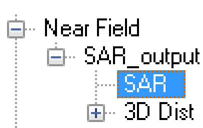
15.11 Result Visualization

Click on the **Data Visualization** button in the *GEMS Solver* window. Double-click on the **Port->excitation->FreqDomain->S-parameter->Magnitude(dB)** option.





Select the **Near Field->SAR output->SAR** option.



Select the **View in Table** option in the **Result** menu.

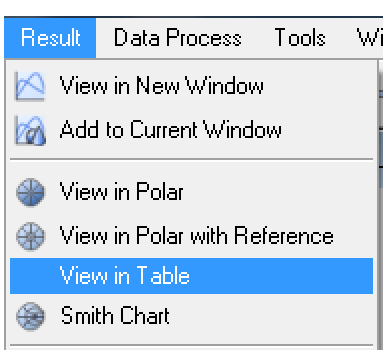


Table 3

	Frequency	Max SAR(w/kg)	Ave SAR(w/kg)	mess(kg)	1g SAR(w/kg)	10g SAR(w/kg)
0	8e+08	16.4157	0.0533769	16.2	11.0418	7.06112
1	9e+08	15.9084	0.0517744	16.2	10.6799	6.82223
2	1e+09	15.1315	0.0503415	16.2	10.1797	6.50403

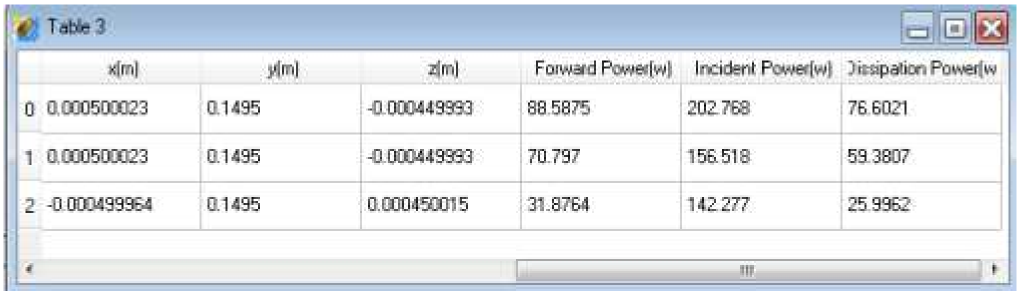
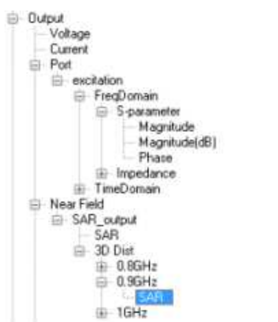


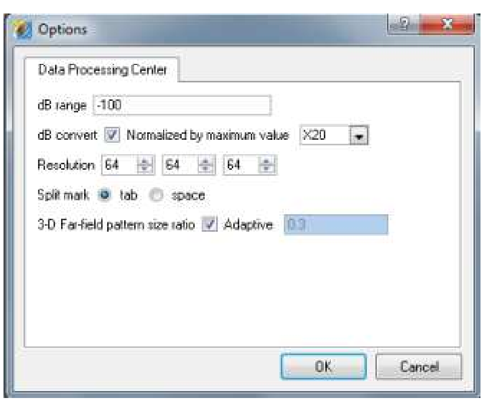
Table 3

	x(m)	y(m)	z(m)	Forward Power(w)	Incident Power(w)	Dissipation Power(w
0	0.000500023	0.1495	-0.000449993	88.5875	202.768	76.6021
1	0.000500023	0.1495	-0.000449993	70.797	156.518	59.3807
2	-0.000499964	0.1495	0.000450015	31.8764	142.277	25.9962

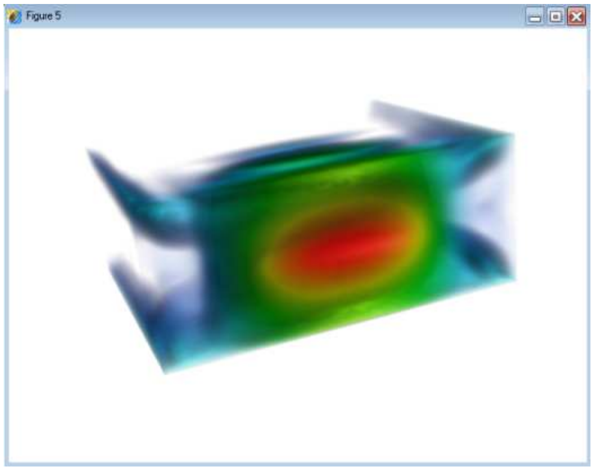
Double-click on the **SAR** option in the **Output->Near Field->SAR_output->3D Dist->0.9GHz->SAR** folder.



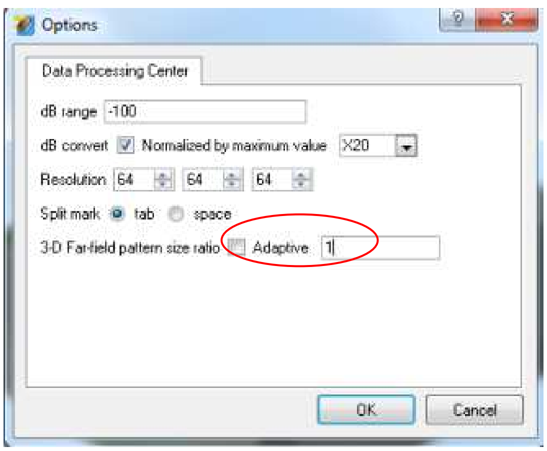
Select the **Tools->Default** option in the toolbar.



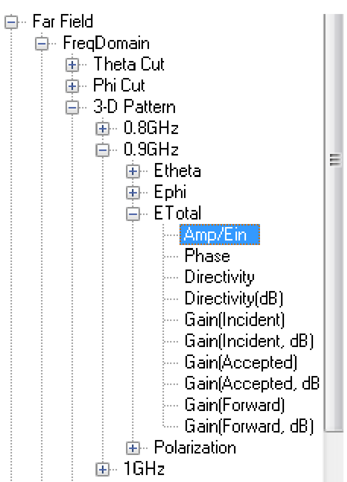
Double-click on the **SAR** option in the **Output->Near Field->SAR_output->3D Dist->0.9GHz->SAR** folder.



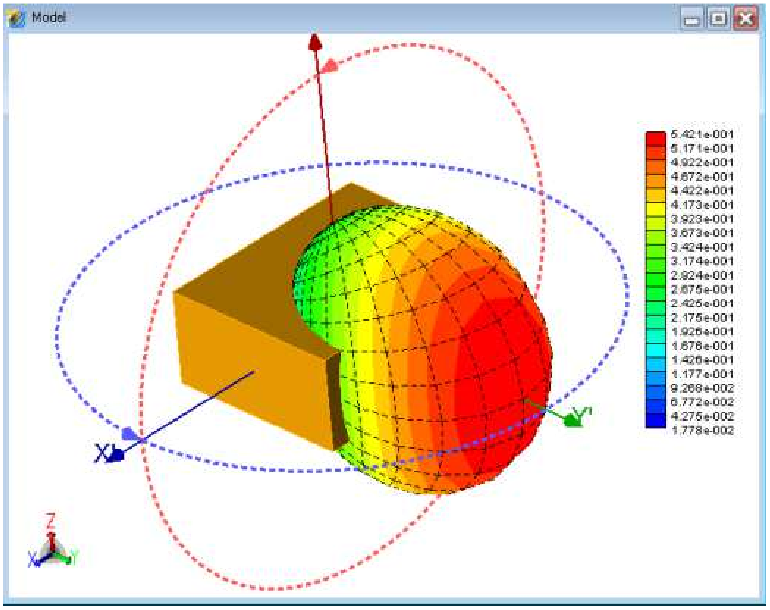
Select the **Tools->Default** option in the toolbar.

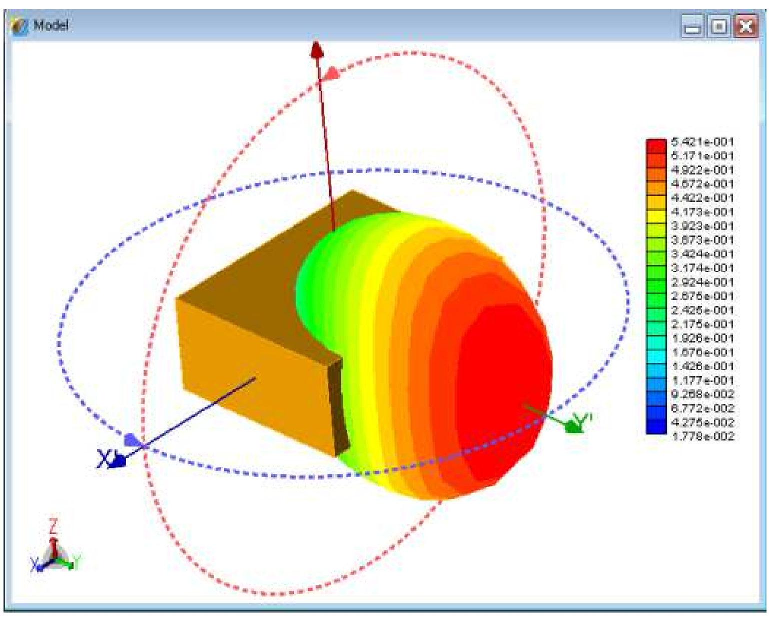


Double-click on the **AMP/Ein** option in the **Output->Far Field->3D Pattern->0.9GHz** folder. The far field is normalized to the voltage measured at the lumped port.



Double-click on the **AMP/Ein** option in the **Output->Far Field->3D Pattern->0.9GHz** folder. The far field is normalized to the voltage measured at the lumped port.





16

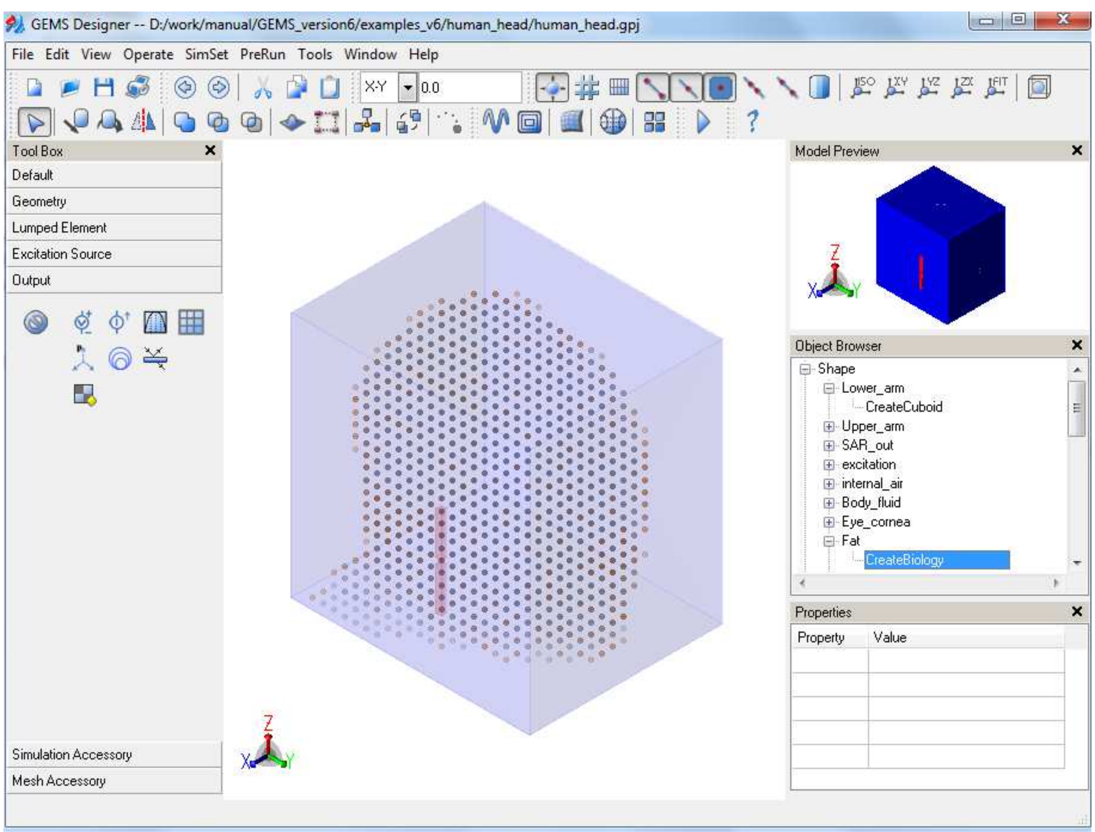
Example 16. Bio-Electromagnetics

Description: A biologic human head with 23 layers of dispersive media is excited by a dipole antenna. The output parameters are the SAR distribution and far field pattern.

Keywords: 3D SAR, Voxel file, far field pattern, and human head.

16.1 Problem Configuration

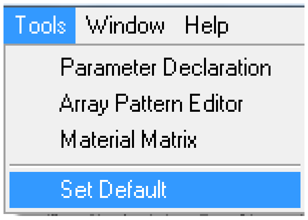
The human head is in the Voxel format. A dipole is located nearby the human head. We calculate the 3D SAR distribution and far field pattern.

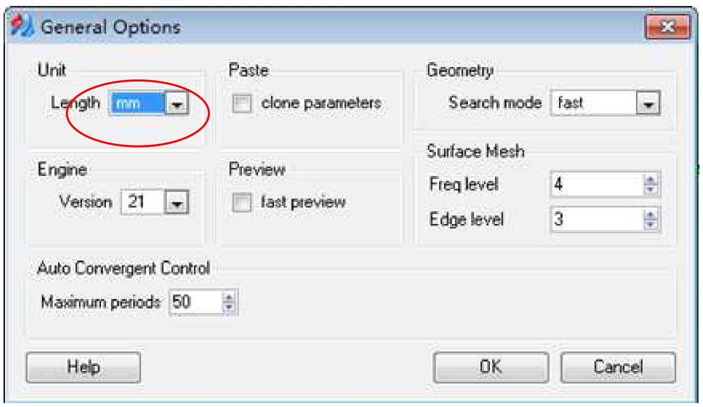


16.2 Create Project Model

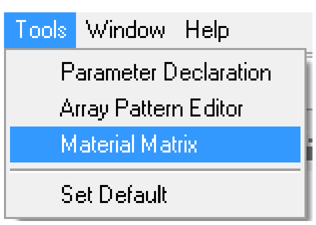
Follow the steps below to import the head model:

(1) Open GEMS Designer
(2) Select the **Set Default** option in the **Tools** menu



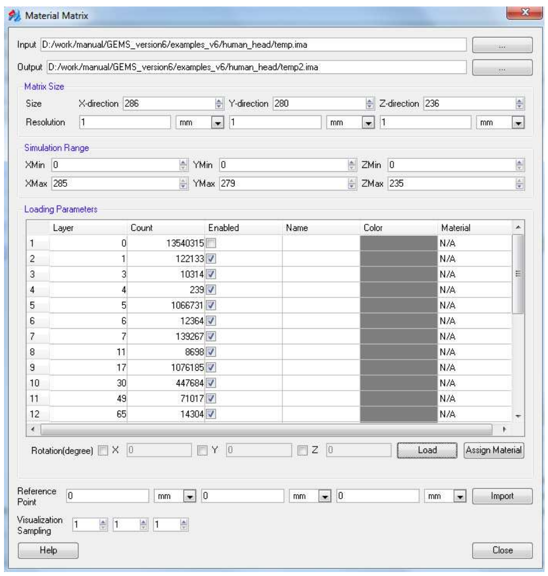


(3) Select the **Material Matrix** option in the **Tools** menu.



(4) Select the search button to find the input file (*.ima, Voxel format).
(5) Select the search button to find the output file (*.ima, it can be same as the input file).
(6) Type "286" in the X-direction (the number of sampling points in the x-direction in the *.ima file); "280" in the Y-direction (the number of sampling points in the y-direction in the *.ima file); "236" in the Z-direction (the number of sampling points in the z-direction in the *.ima file).
(7) Type "1" in the x-direction (resolution in the x-direction); "1" in the y-direction (resolution in the y-direction); "1" in the z-direction (resolution in the z-direction).
(8) Type the range we like to simulate.

(9) Type the rotation angle around the x-, y-, and z-directions.
(10) Click on the **Load** button to load the data.
(11) Check the layers you like to load in the **Enabled** column.
(12) Change the color you like to display the layer in the **Color** column.
(13) Specify the name of the layers in the **Material** column.
(14) Specify the reference point.
(15) Specify the visualization resolution for display only.
(16) Click on the **Import** button to display the model.

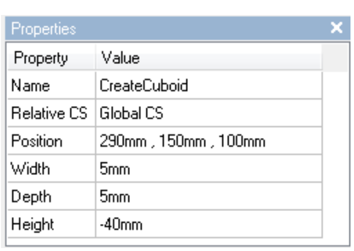


(17) Click on the **Assign Material** button to load the material file to the "*.ima". The File format is in the following format:

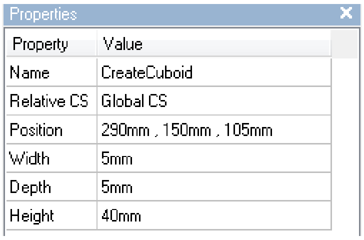
```
//The line beginning with "//" is comment line.
//"Name"    "ID"    "epsilon"   "sigma"  "mu"   "musigma" "density(kg/m^3)"
  skin       143     41          0.7      1.0    0.0       1000
```

Follow the steps below to create the dipole model:

(1) Draw a lower arm of the dipole with a rectangular cross section.

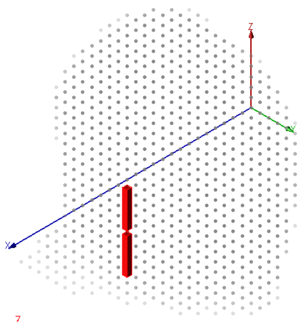


(2) Draw an upper arm of the dipole with a rectangular cross section.



Property	Value
Name	CreateCuboid
Relative CS	Global CS
Position	290mm , 150mm , 105mm
Width	5mm
Depth	5mm
Height	40mm

(3) Change their names to the “Lower_arm” and “Upper_arm” options in the **Object Browser** box.

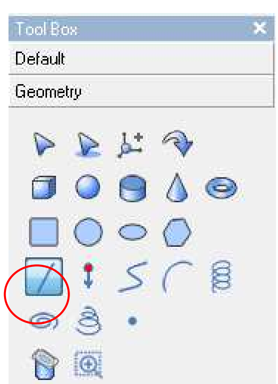


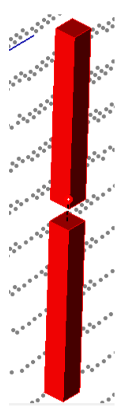
(4) Select the “Lower_arm” and “Upper_arm” options in the **Object Browser** box, and check the **PEC** box in the **Properties** box.

16.3 Add Excitation

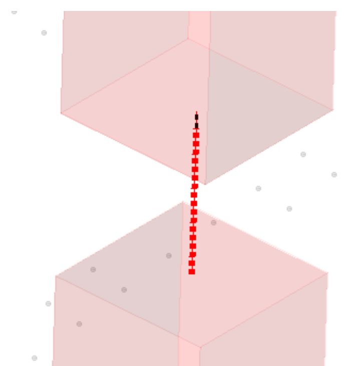
Follow the steps below to add a lumped port:

(1) Select the **Line** icon in the **Tool Box->Geometry** box.



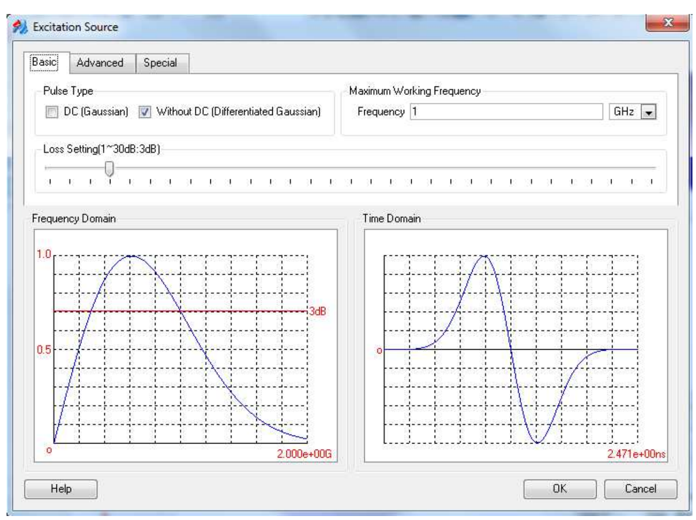


(2) Snap the mouse icon to the center of the top surface of the "lower arm", press the left mouse button.
(3) Move and snap the mouse icon to the center of the lower surface of the "upper arm", press the left mouse button.
(4) Select the line option in the **Object Browser** box, and change its name to "excitation" in the **Properties** box.
(5) Select the "excitation" option in the **Object Browser** box, and click on the **Lumped port** icon in the **Tool Box->Simulation Accessory** box.



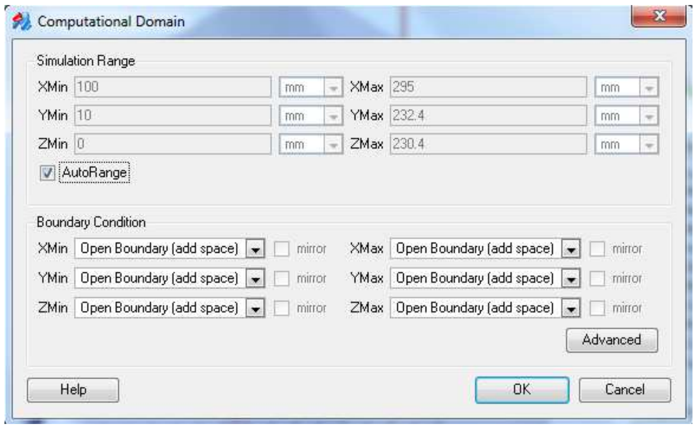
16.4 Set Excitation Pulse

Click on the **Set excitation source** button in the toolbar.



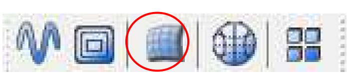
16.5 Set Domain and Boundary Condition

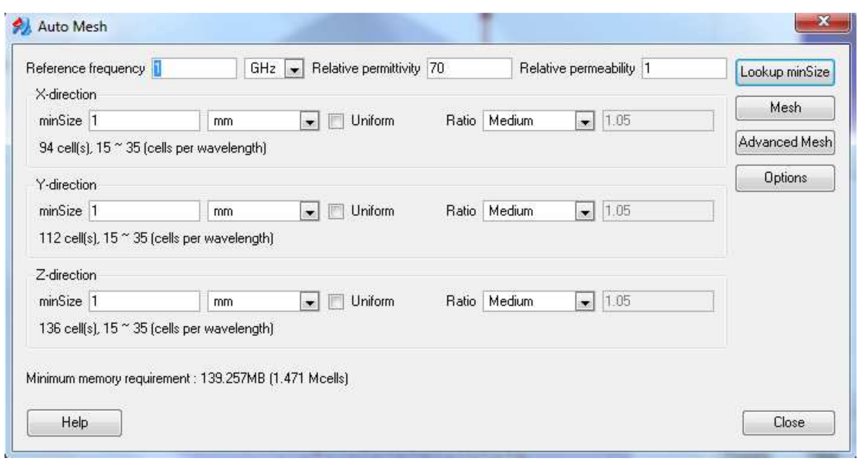
Click on the **Set boundary condition** button in the toolbar.



16.6 Design Mesh Distribution

Click on the **Auto Mesh** button in the toolbar. Type "1" in the "minSize" boxes and then click on the **Mesh** button.

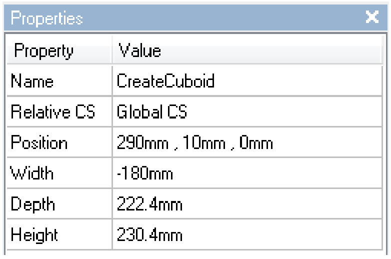




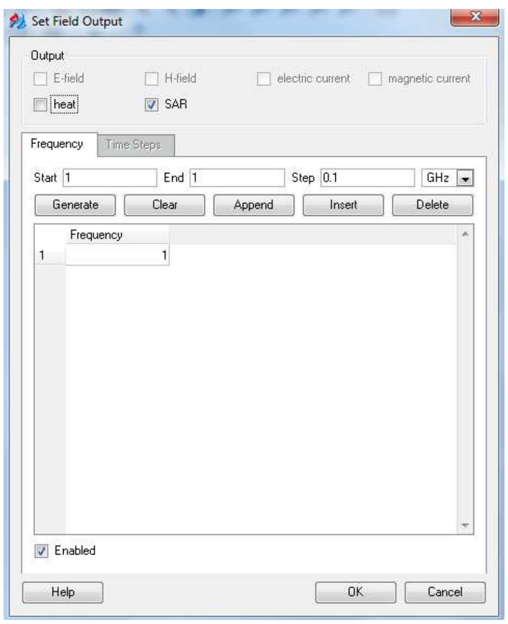
16.7 Set SAR Output

Follow the steps below to add a box for the SAR output:

(1) Click on the **Cuboid** icon in the **Tool Box->Geometry** box and draw a box.



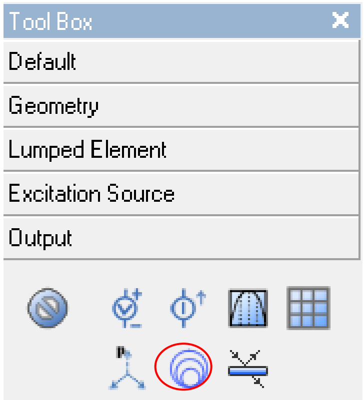
(2) Select the box option in the **Object Browser** box and change its name to “SAR_out” in the **Properties** box.
(3) Select the “SAR_out” option in the **Object Browser** box and click on the **Field** icon in the **Tool Box->Output** box.
(4) Check the **Enabled** box.
(5) Check the **SAR** box and specify the frequency list of interest.
(6) Click on the **OK** button to confirm the settings.



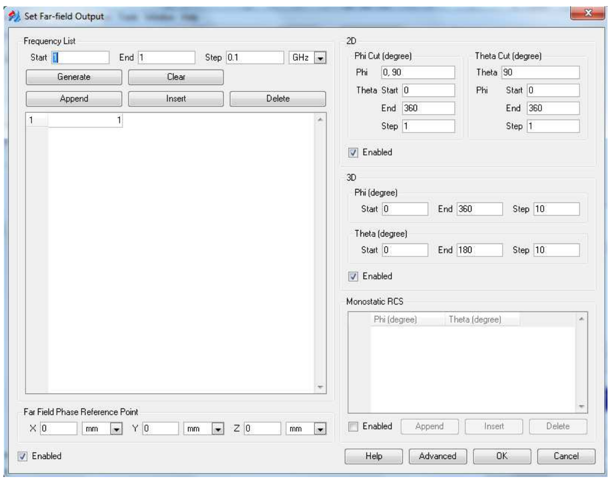
16.8 Set Far Filed Output

Follow the steps below to set the far field output:

(1) Click on the **Far field** icon in the **Tool Box->Output** box.



(2) Click on the **Enabled** box.



(3) Specify the frequency list of interest.
(4) Check the **Enabled** in the **3D** box.
(5) Click on the **OK** button to confirm the settings.

16.9 Save Project

Click on the **Save** button in the toolbar to save the project.

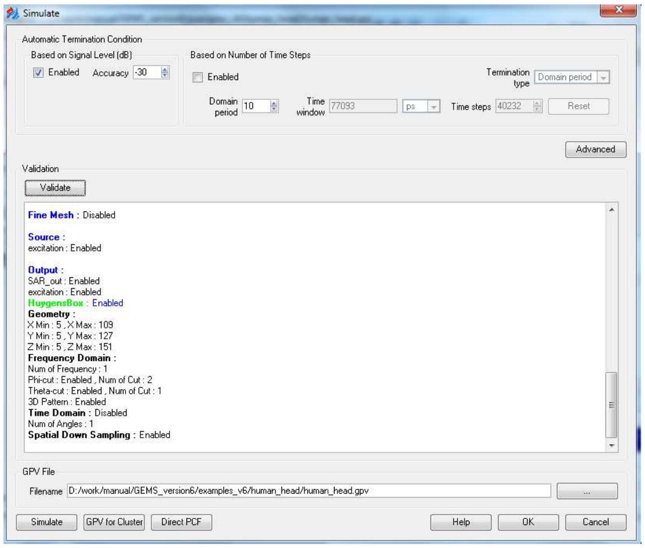
16.10 Generate Simulation File

Click on the **PreCalculate** button in the toolbar.

Click on the **Validate** button to validate the project settings.

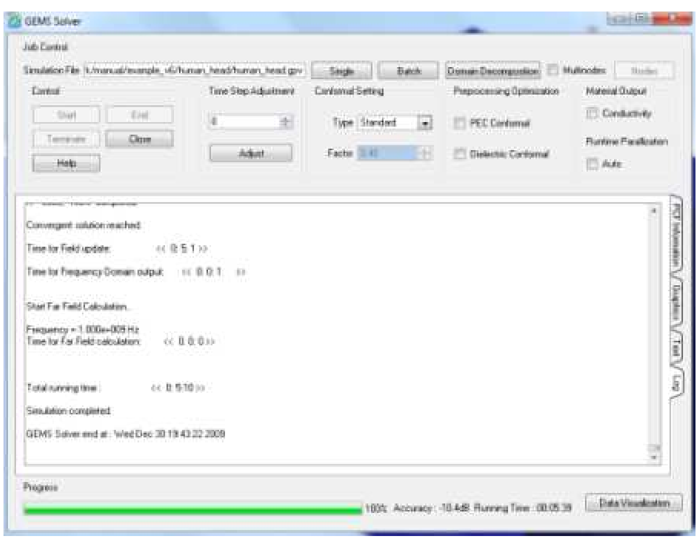
Click on the **Simulate** button to open the *GEMS Solver* window.

If the project is saved as from another project, you need to pay special attention on the folder and name in the **Filename** box.



16.11 Simulate Project

Click on the **Simulate** button in the *Simulate* window.

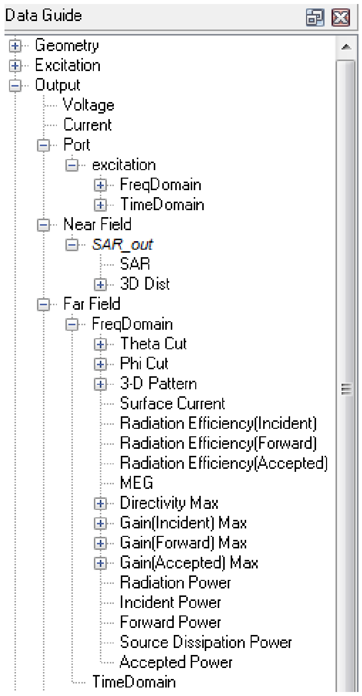


Click on the **Start** button to start the simulation.

16.12 Result Visualization

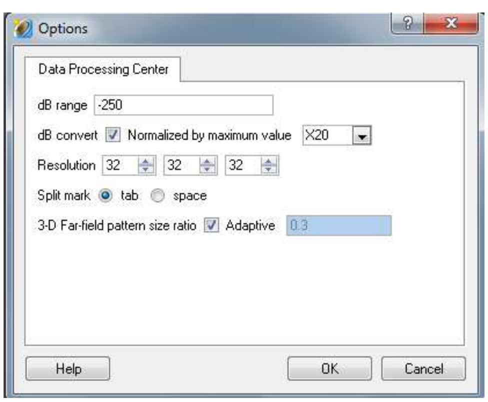
Click on the **Data Visualization** button in the *GEMS Solver* window.

The results include the 3D SAR distribution and far field pattern.

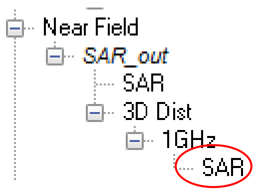


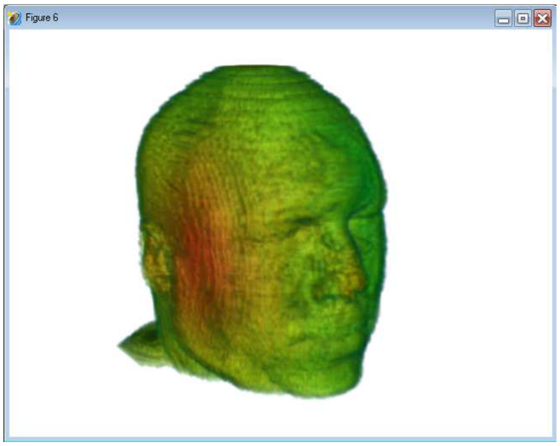
Select the **Options** option in the **Tools** menu.

Specify the dB range for the better view and the resolution in the x-, y-, and z-directions to get the better display appearance.

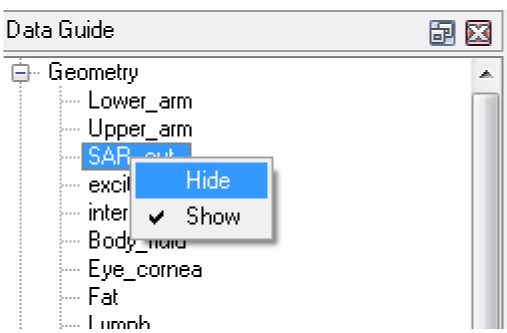


Double-click on the **Near Field->SAR_out->3D Dist->1GHz->SAR** button in the result tree.



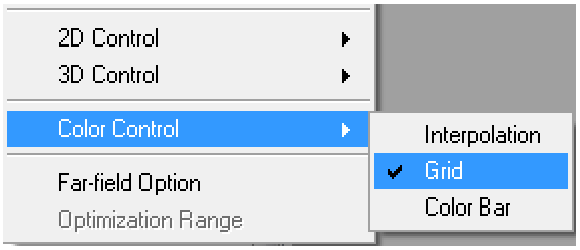


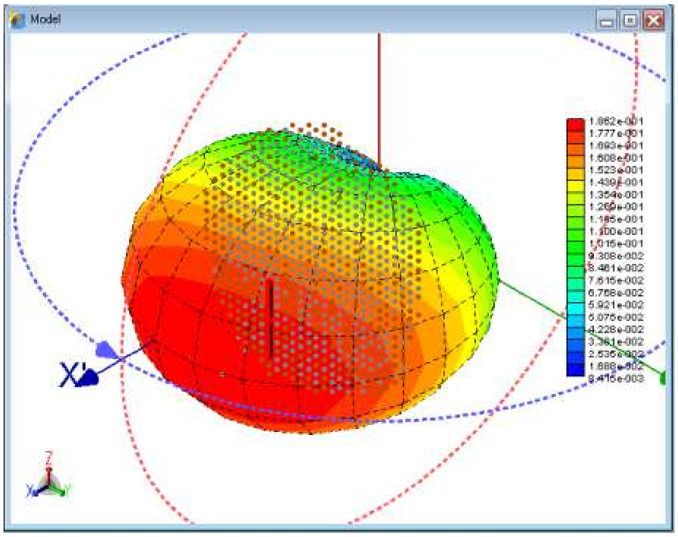
When clicking the **View** model button in the toolbar, the human head is blocked by the 3D box. You can hide it by selecting the **Hide** option in the **Geometry** folder.



Select the 3D far field pattern option in the result tree, and click on the **Add to current window** button in the toolbar, and on the **Play** button in the toolbar.

Select the **Grid** option to add the grid in the 3D far field pattern.


2D Control
3D Control
Color Control
Interpolation
Grid
Color Bar
Far-field Option
Optimization Range


Model

17

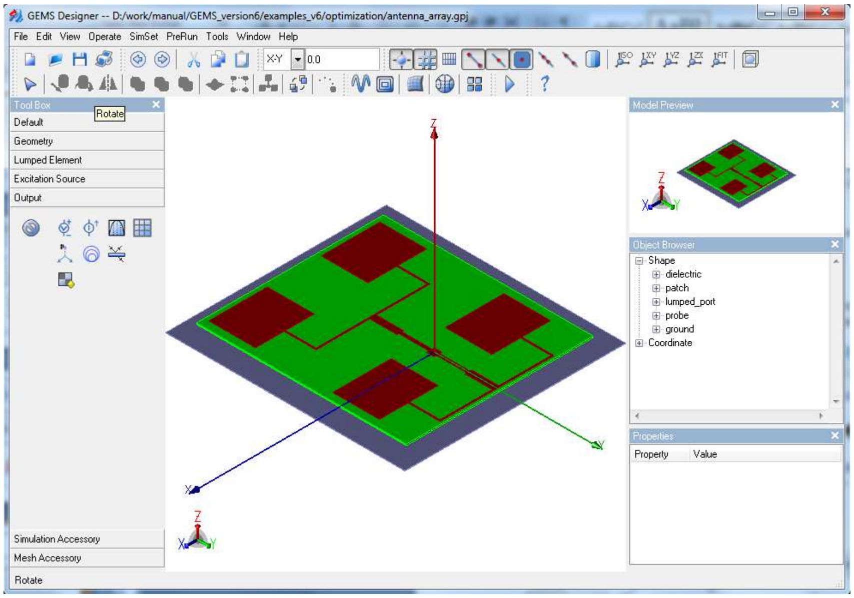
Example 17. Antenna Array

Description: An antenna array includes four rectangular patches and feed network system. The output parameters are the far field and return loss.

Keywords: Antenna array, S-parameters, far field pattern, and parametric design.

17.1 Problem Configuration

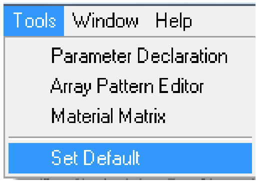
The antenna system includes four patches and feed network. In this project, we will simulate the antenna with the feed network together.

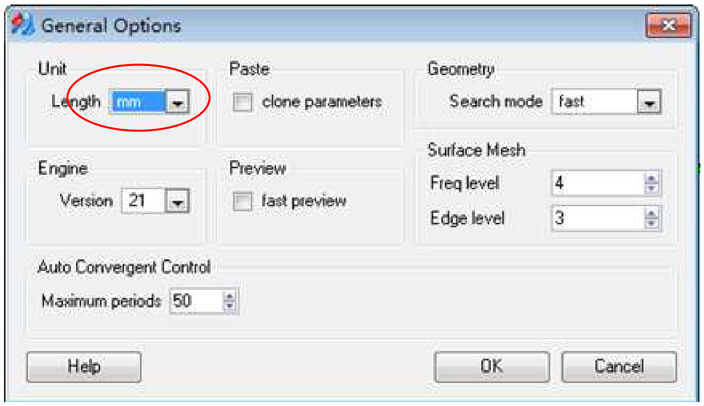


17.2 Create Antenna Model

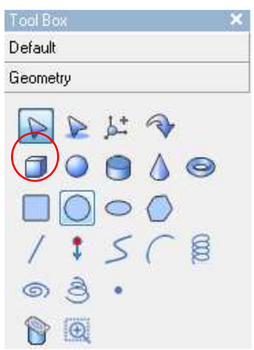
Follow the steps below to create the substrate structure:

(1) Open GEMS Designer
(2) Select the **Set Default** option in the **Tools** menu





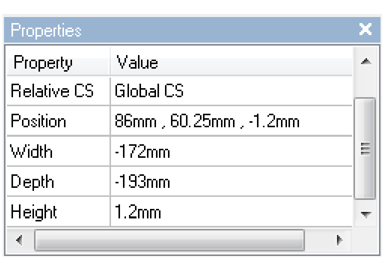
(3) Click on the **Cuboid** icon in the **Tool Box->Geometry** box.



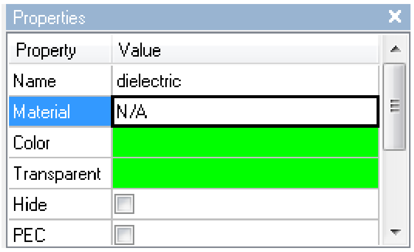
(4) Draw a box inside the figure region.

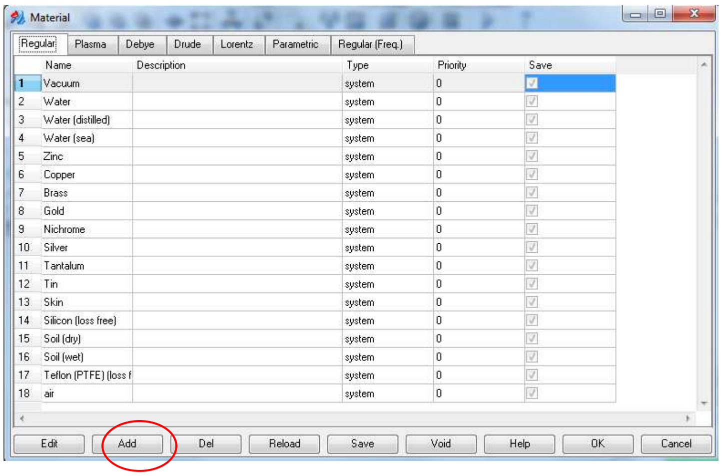
(5) Select the box option in the **Object Browser** box and change its name to “dielectric”.

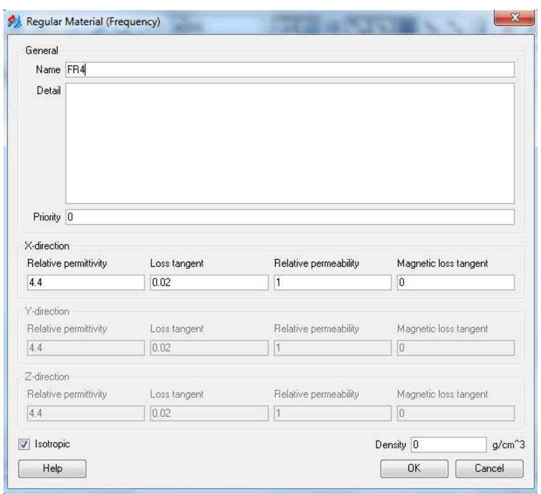
(6) Select the “CreateCuboid” option in the “dielectric” folder in the **Object Browser** box and change its dimensions to the reference point (86mm, 60.25mm, -1.2mm), width=-172mm, length=-193mm, and thickness=1.2mm.



(7) Select the **dielectric** option in the **Object Browser** box and click on the **Material** box.

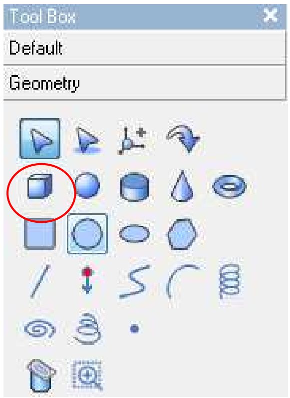






Follow the steps below to create the ground structure:

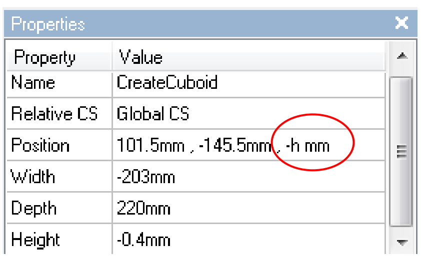
(1) Click on the **Cuboid** icon in the **Tool Box->Geometry** box.



(2) Draw a box inside the figure region.

(3) Select the box option in the **Object Browser** box and change its name to “ground”.

(4) Select the “CreateCuboid” option in the “ground” folder in the **Object Browser** box and change its dimensions to the reference point (101.5mm, 145.5mm, -h), width=-203mm, length=-220mm, and thickness=-0.4mm.


Properties

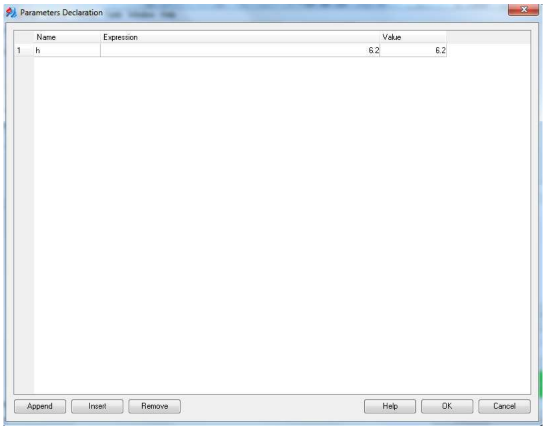
Property	Value
Name	CreateCuboid
Relative CS	Global CS
Position	101.5mm , -145.5mm , -h mm
Width	-203mm
Depth	220mm
Height	-0.4mm

If the thickness of the ground is fixed, we change the distance between the ground and substrate through changing position of the ground. We set the position of the ground as a variable “h” that will change during the simulation.

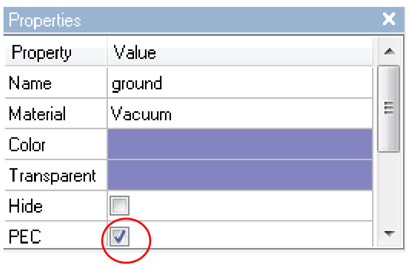
If we type a variable as a dimension in the **Properties** box, the **Parameters Declaration** widow will popup.

Click on the **Expression** box in the **Parameters Declaration** window, and specify the value of the variable.

You can specify the new variables or expressions here and use them in the project modeling.

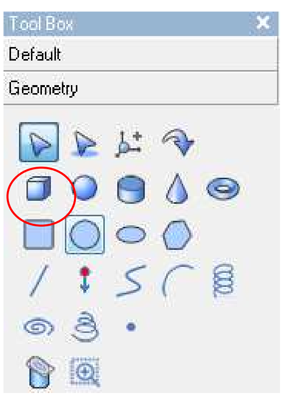


(5) Select the **ground** option in the **Object Browser** box and check the **PEC** box.



Follow the steps below to create the excitation probe:

(1) Click on the **Cuboid** icon in the **Tool Box->Geometry** box



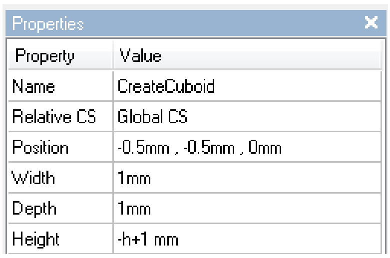
(2) Draw a box inside the figure region

(3) Select the box option in the **Object Browser** box and change its name to “probe”.

(4) Select the “CreateCuboid” option in the “probe” folder in the **Object Browser** box and change its dimensions to reference point (-0.5mm, -0.5mm, 0), width=-1mm, length=-1mm, and thickness=(-h+1)mm.

The probe is a part of feed structure, and its thickness should be as small as possible, however, a small structure requires a small cell size, in turn, it will significantly increase the simulation burden. Therefore, we should set its cross section equal to or larger than the fine structure of interest to avoid using the small cell size for this probe. For the same reason, you need to use the rectangular cross section for the probe if possible.

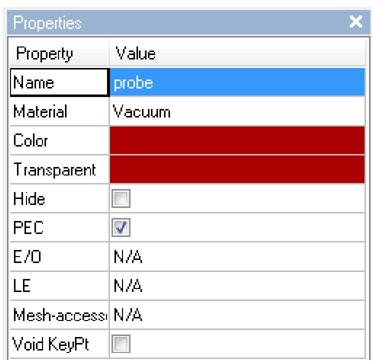
Since the variable “h” has been in the variable list, we can directly use it.



Properties

Property	Value
Name	CreateCuboid
Relative CS	Global CS
Position	-0.5mm , -0.5mm , 0mm
Width	1mm
Depth	1mm
Height	-h+1 mm

(5) Select the **probe** option in the **Object Browser** box and check the **PEC** box.

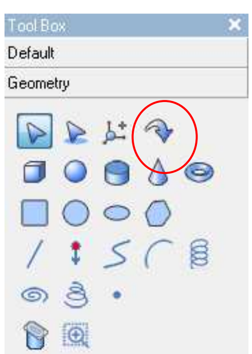


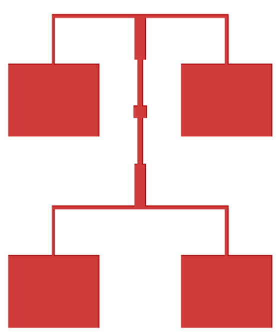
Properties

Property	Value
Name	probe
Material	Vacuum
Color	
Transparent	
Hide	☐
PEC	☑
E/O	N/A
LE	N/A
Mesh-access	N/A
Void KeyPt	☐

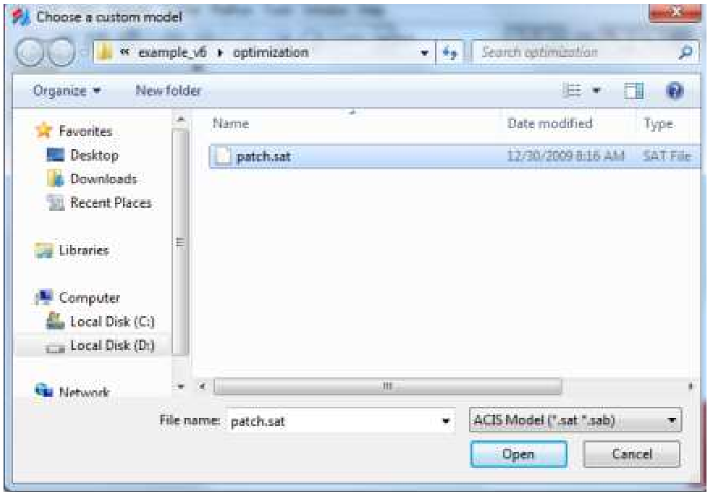
Follow the steps below to import the patch and feed network model:

(1) Click on the **Import model** icon in the **Tool Box->Geometry** box.

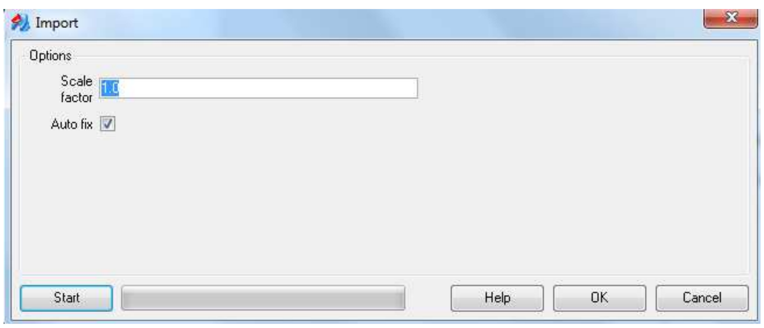




(2) Search for and select the “patch.sat” file, and then click on the **Open** button.



Click on the **Start** button to start the processing, and then click on the **OK** button after the processing is complete.

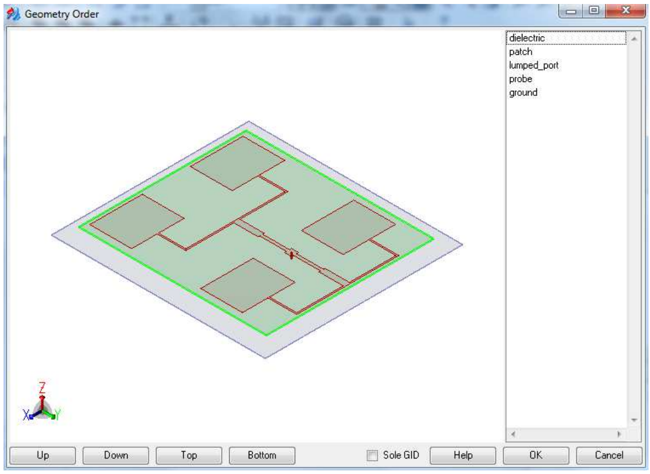


(3) Select the patch option in the **Object Browser** box and change its name to “patch” in the **Properties** box.

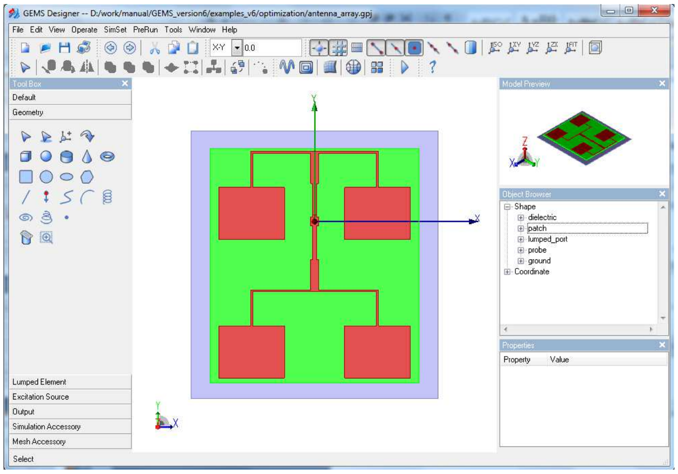
(4) Select the **patch** option in the **Object Browser** box and change its color to red in the **Properties** box.

GEMS allows you to simulate a thin PEC structure without thickness. However, the object order is very important if the thin PEC touches partially or completely other objects. Please ensure the order is correct if a project includes any thin PEC structure.

There is a simple way to check or adjust the order by clicking on the **Order adjust** button in the toolbar.



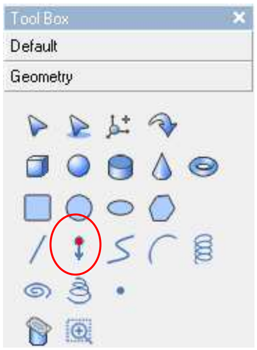
(5) Select the **patch** option in the **Object Browser** box and check the **PEC** box.



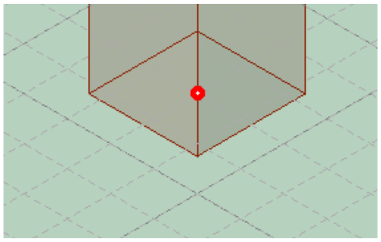
17.3 Set Excitation Source

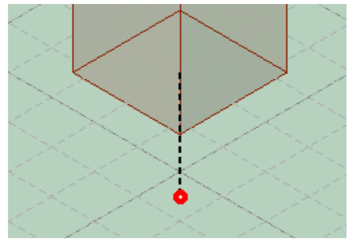
Follow the steps below to set the excitation source:

(1) Click on the **Plumb line** icon in the **Tool Box->Geometry** box.



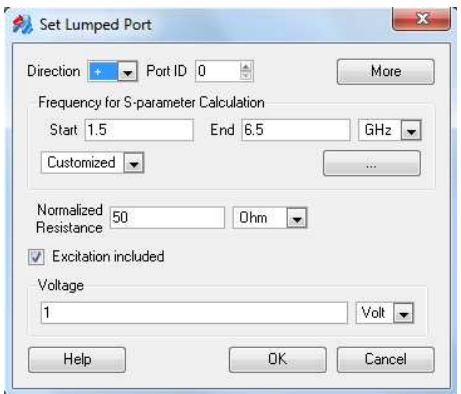
(2) Snap the mouse icon to the center of the bottom surface of the probe, move the mouse icon to stop at the ground.



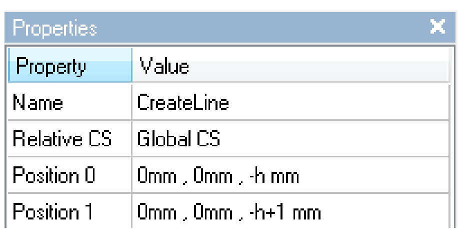


(3) Select the line option in the **Object Browser** box and change its name to “lumped_port” in the **Properties** box.

(4) Select the **lumped_port** option in the **Object Browser** box and click on the **Lumped port** icon in the **Tool Box->Simulation Accessory** box.

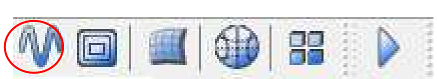


(5) Select the “CreatePlumb” in the **lumped_port** folder in the **Object Browser** box and change the height from “–h” to “–h+1” in the **Properties** box.

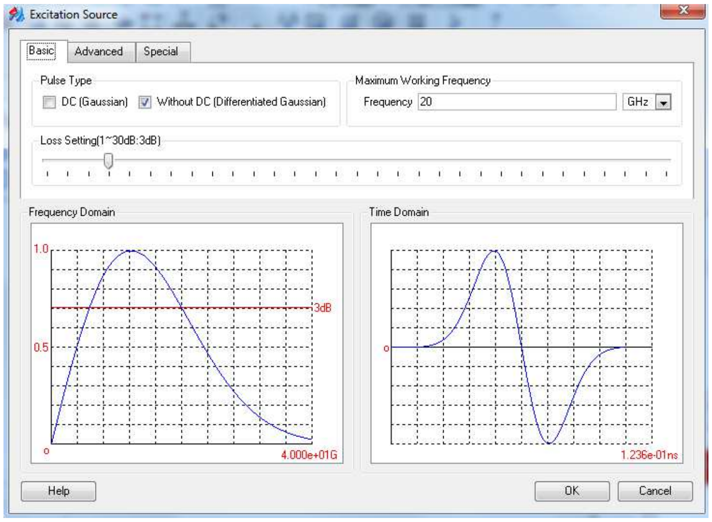


17.4 Set Excitation Pulse

Click on the **Set excitation source** button in the toolbar.

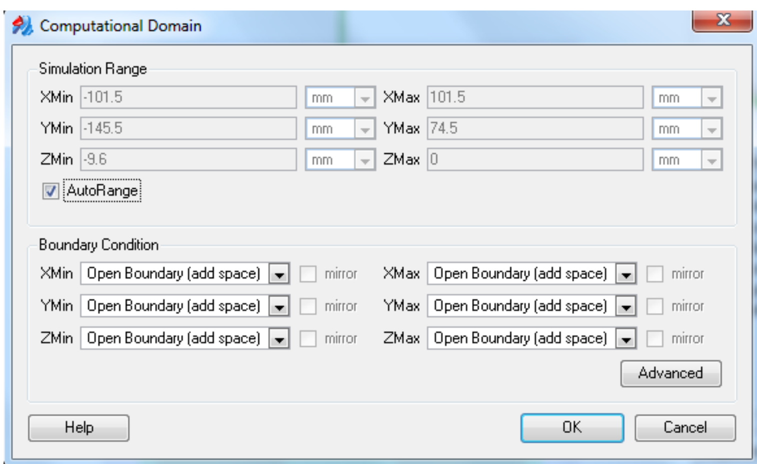


Input “0.4” in the **Maximum Working Frequency->Frequency** box



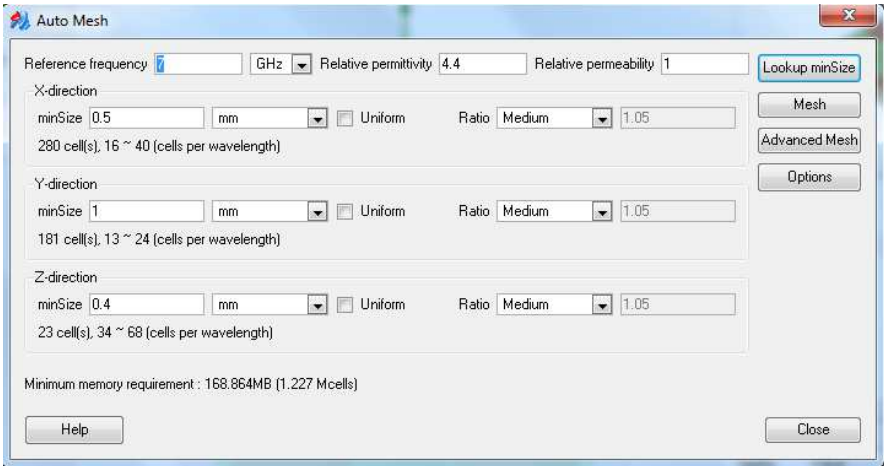
17.5 Set Domain and Boundary Condition

Click on the **Set boundary condition** button in the toolbar.



17.6 Design Mesh Distribution

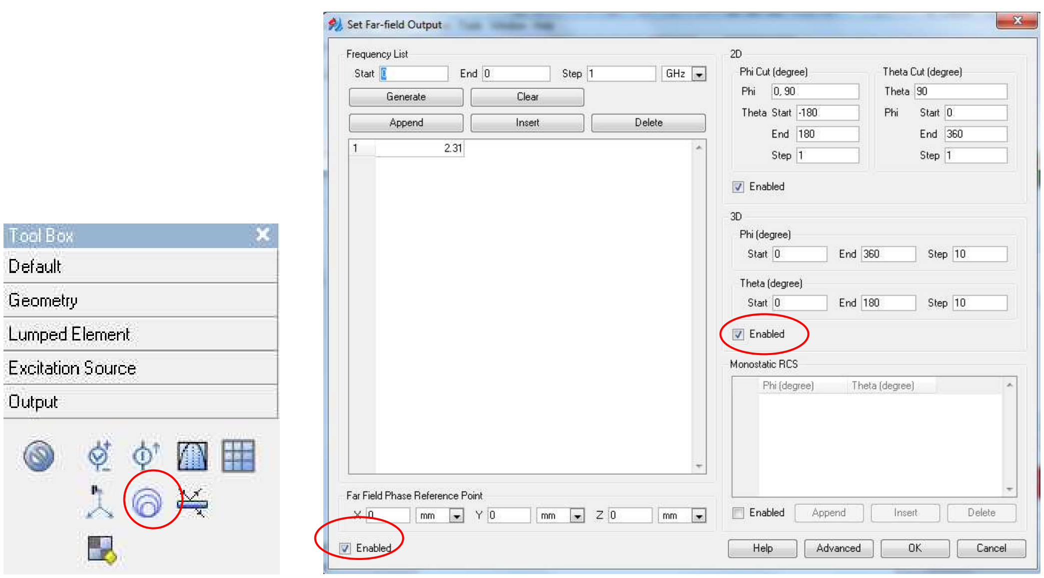
Click on the **Auto mesh** button in the toolbar.



17.7 Set Far Field Output

Follow the steps below to set the far field output:

(1) Click on the **Far field** icon in the **Tool Box->Output** box
(2) Check the **Enabled** box
(3) Specify the frequency range of interest
(4) Check the **Enabled** box in the **3D** box
(5) Check **OK** button to confirm the setting



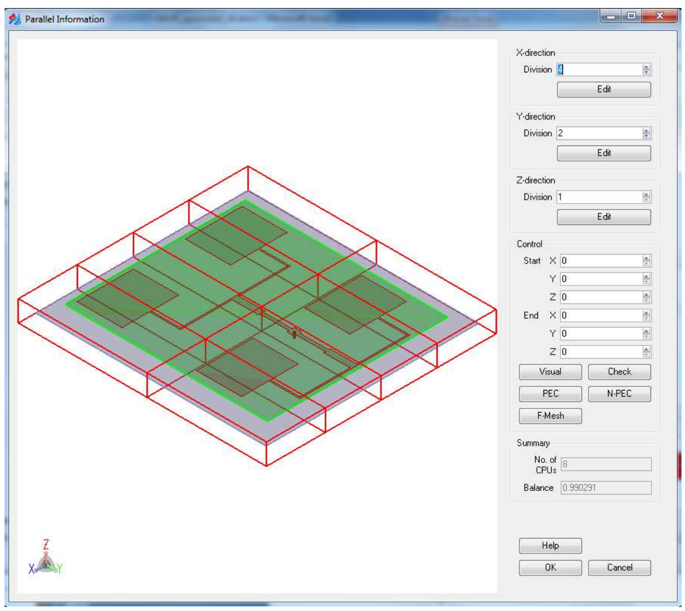
17.8 Set Parallel Processing

Two computers have total 8 cores, so we divide the domain to into 8 pieces.

Follow the steps below to design the core distribution for the parallel processing:

(1) Click on the **Set parallel info** button in the toolbar.

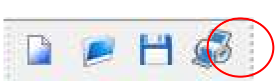
(2) Specify the number of cores in each direction to ensure the job balance. Due to the flat domain shape, we use 4 cores in the x-direction, and 2 cores in the y-direction.



The basic idea is ensure the minimum interface between the sub-domains to reduce the burden on the network system.

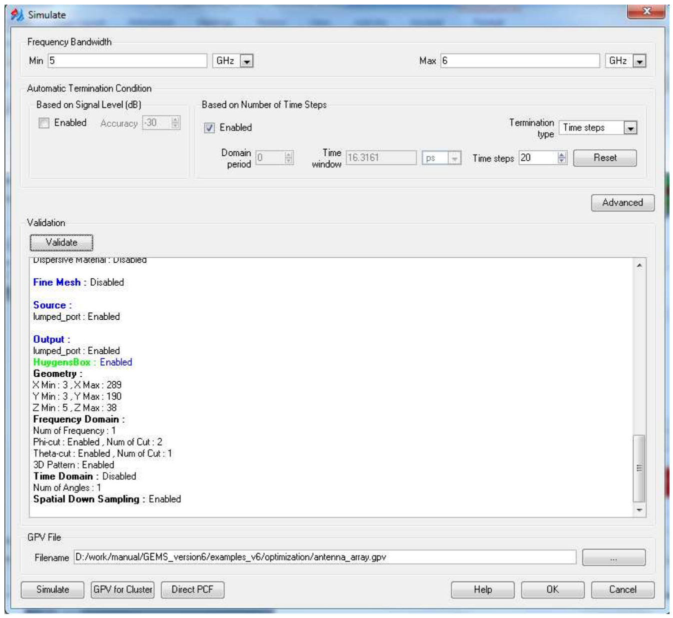
17.9 Save Project

Click on the **Save** button in the toolbar.



17.10 Generate Simulation File

Click on the **PreCalculate** button to generate the simulation file "*.gpv".



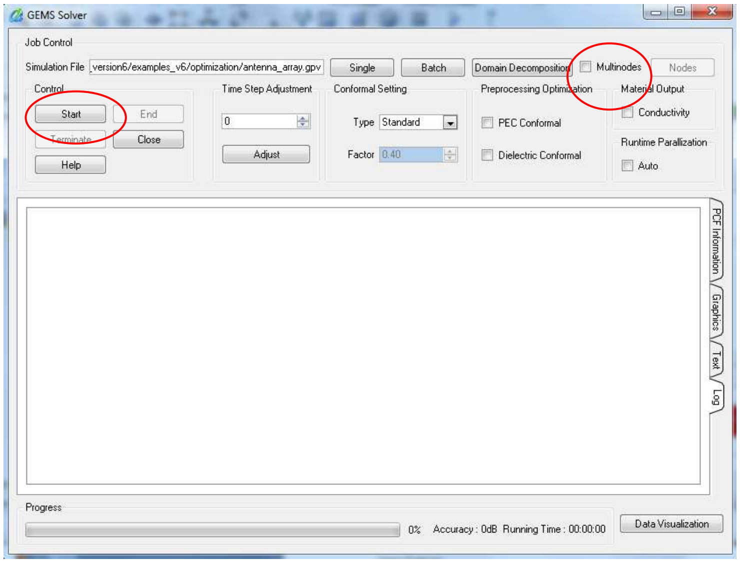
Click on the **Validate** button to validate the project setting.

Click on the **Simulate** button to open the *GEMS solver* window.

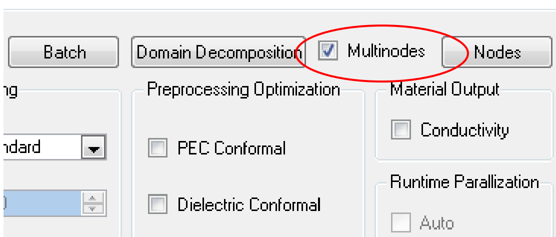
17.11 Simulate Project

We will use a network cable directly connect two computers to form a simple cluster, and use it to simulate this antenna array problem.

Click on the **Simulate** button in the *Simulate* window

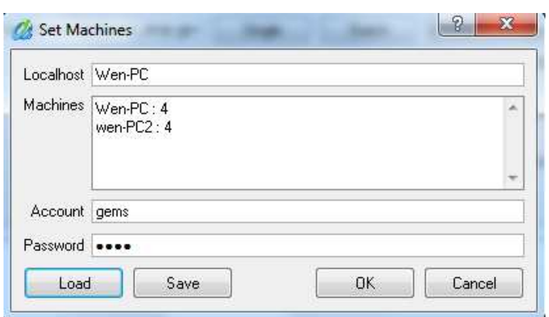


Check the **Multinodes** box.



Click on the **Nodes** button to open the *Set Machine* window.

List the computer names in the **Machine** box followed by the number of cores. The current computer (the computer you are using) should be at the first place. The results will be stored in the current computer.



Input the username and password, and then click on the **OK** button to confirm the parallel processing setting.

Click on the **OK** button in the *GEMS Solver* window to start the simulation.

Same as running the code on the single computer, the simulation results will be stored in the project folder. Follow the exactly same procedure to do the result display and data post-processing.

Though you use two computers, you do not need to read or write any data in another computer. What you need is to borrow the CPU and Memory from another computer for the simulation.

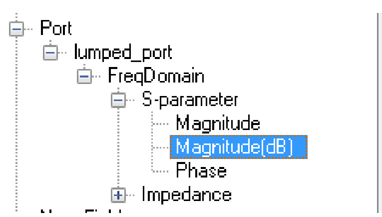
You need a network switch with the multiple ports to connect more than two computers together. The parallel processing procedure is no difference. GEMS does not care the types of operating systems, computer types, and CPU types.

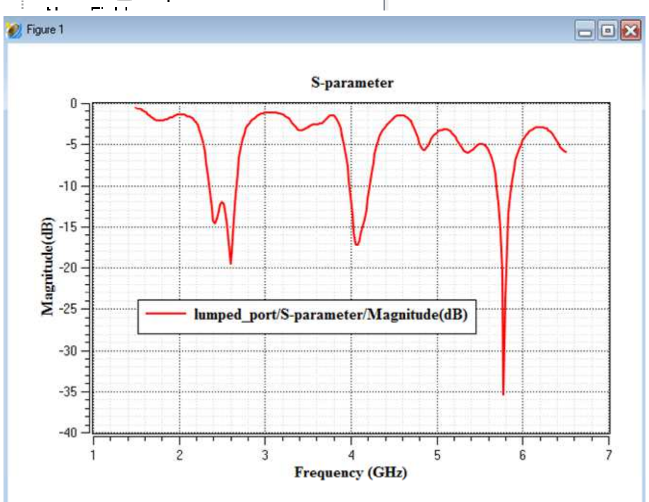
To use a PC cluster for GEMS simulation, you need to install GEMS software on each computer in the cluster.

17.12 Result Visualization

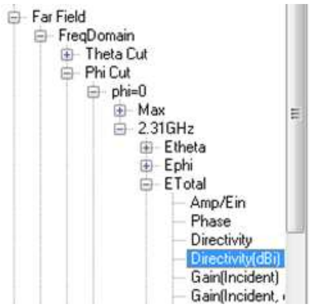
Click on the **Data Visualization** button in the *GEMS Solver* window to open the GEMS display window.

Double-click on the **Port->Lumped_port->FreqDomain->S-parameter->Magnitude(dB)** option.

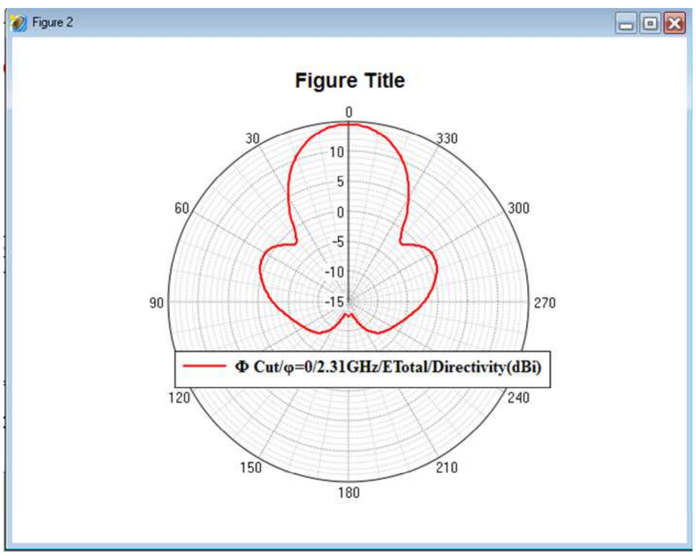




Select the **Output->Far Field->FreqDomain->Phi Cut->Phi=0->2.31GHz->Etoal->Amp/Ein** option.



Click on the **View in Polar** button in the toolbar.



18

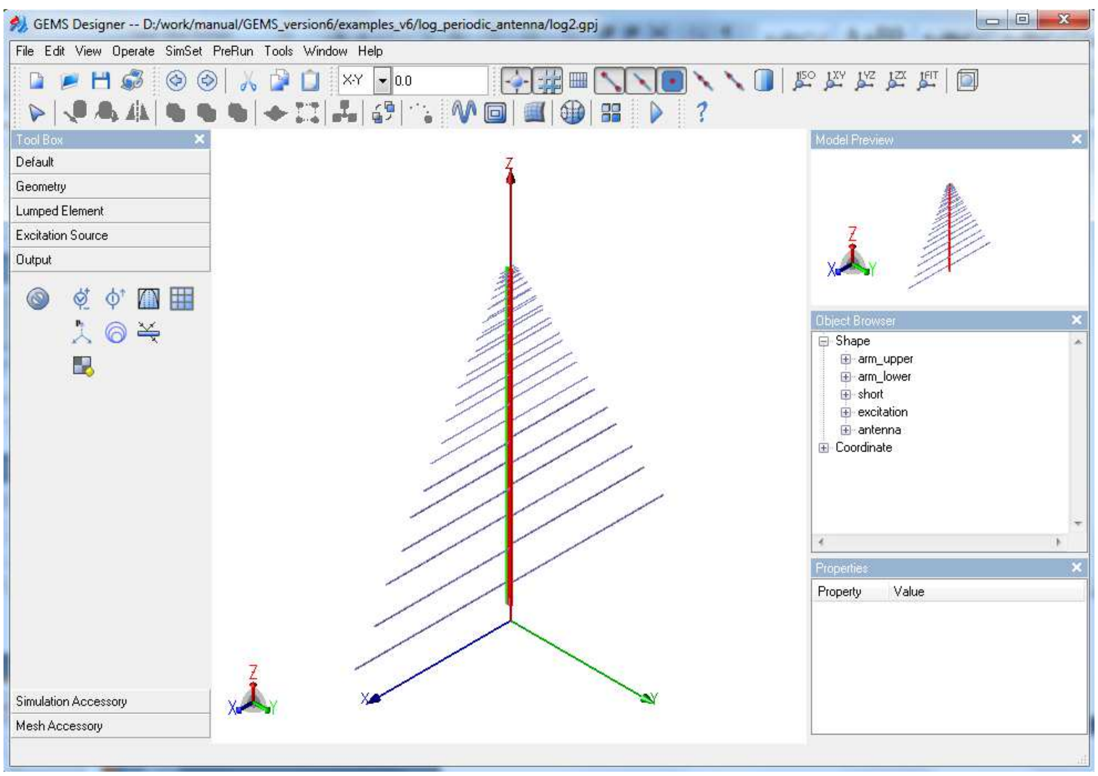
Example 18. Log-Periodic Antenna

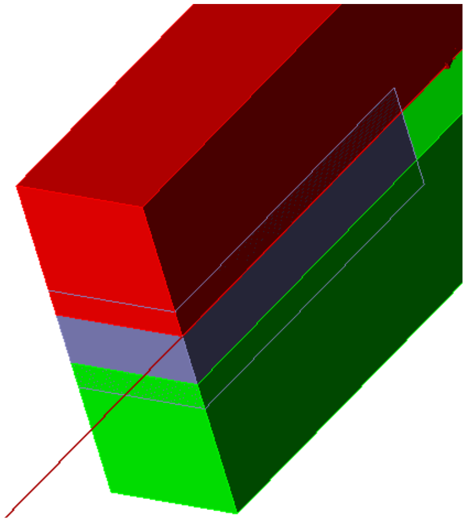
Description: A log-periodic antenna is excited by the lumped port. The output parameters are return loss and far field pattern.

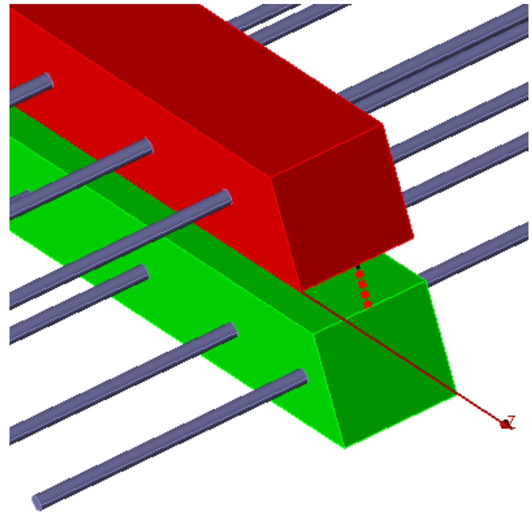
Keywords: Lumped port, return loss, and far field pattern.

18.1 Problem Configuration

The log-periodic antenna includes 15 pairs of arms with the circular cross section. The diameters of the thinnest and thickest arms are 5mm and 10mm, respectively. The lengths of the shortest and longest arms are 106.6mm and 2700mm, respectively. The frequency band of interest is 20MHz to 400MHz.







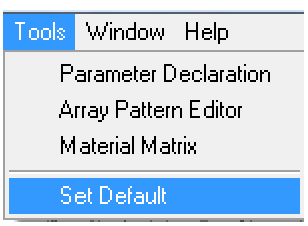
18.2 Create Antenna Model

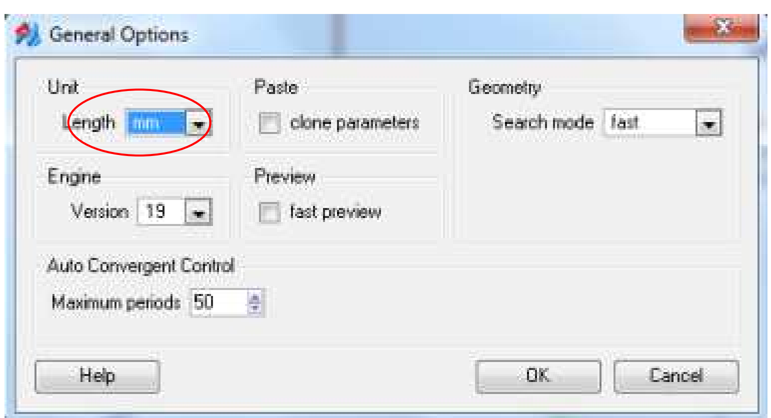
The model is in SAT format that can be imported to GEMS interface directly.

Follow the steps below to import the antenna model to GEMS interface:

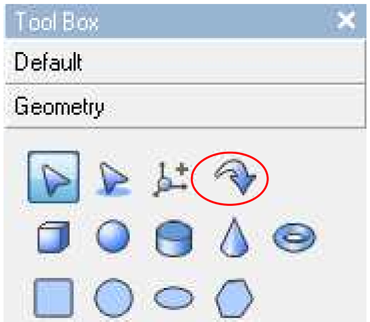
(1) Open GEMS Designer

(2) Select the **Set Default** option in the **Tools** menu



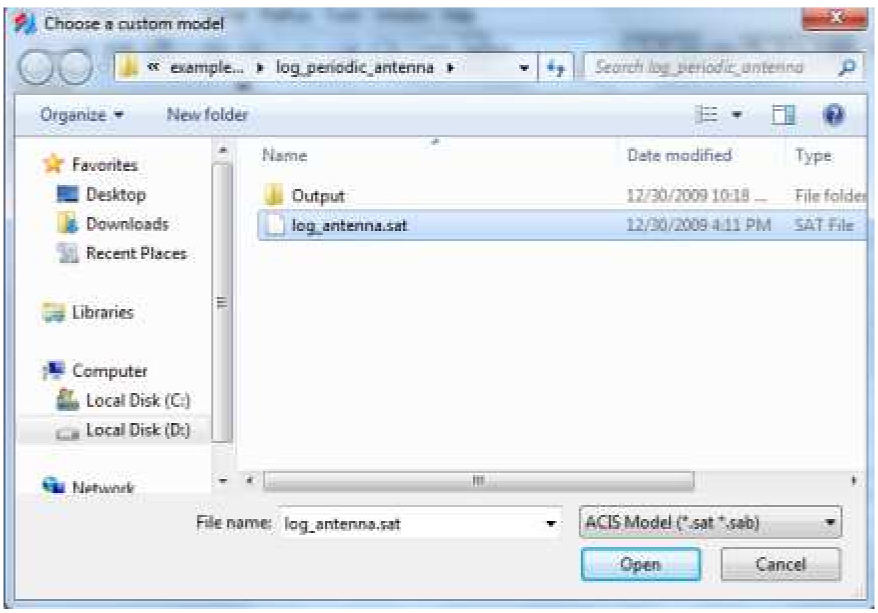


(3) Click on the **Import model** icon in the **Tool Box->Geometry** box.

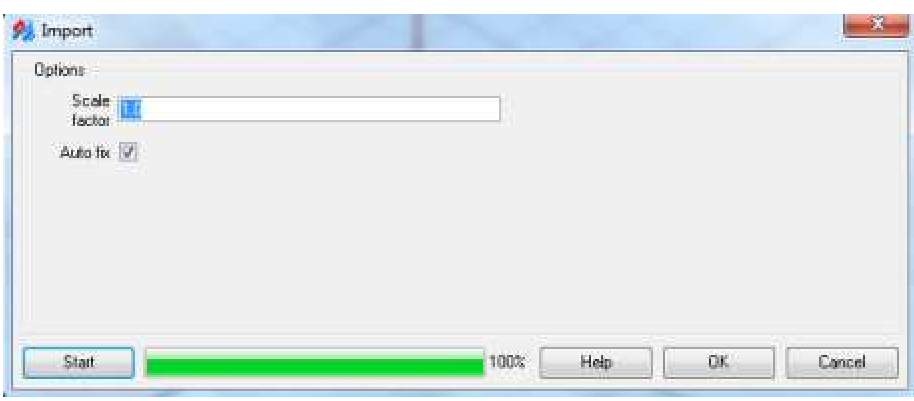


Select the file format “ACIS Model (*.sat *.sab)” in the format list.

Search and select the “log_antenna.sat” file, and then click on the **Open** button.

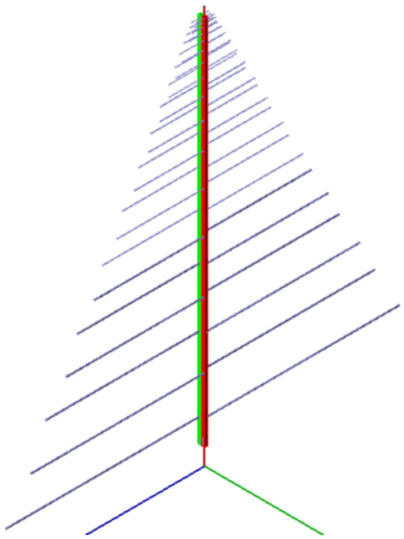


Click on the **Start** button to start the processing and then on the **OK** button after the processing is completed.



Click on the **FIT** button to fit the model inside the figure region.

Click on the **Select** button to switch to the view mode.



(4) Select the antenna options in the **Object Browser** box and change their names, and check the **PEC** box in the **Properties** box.

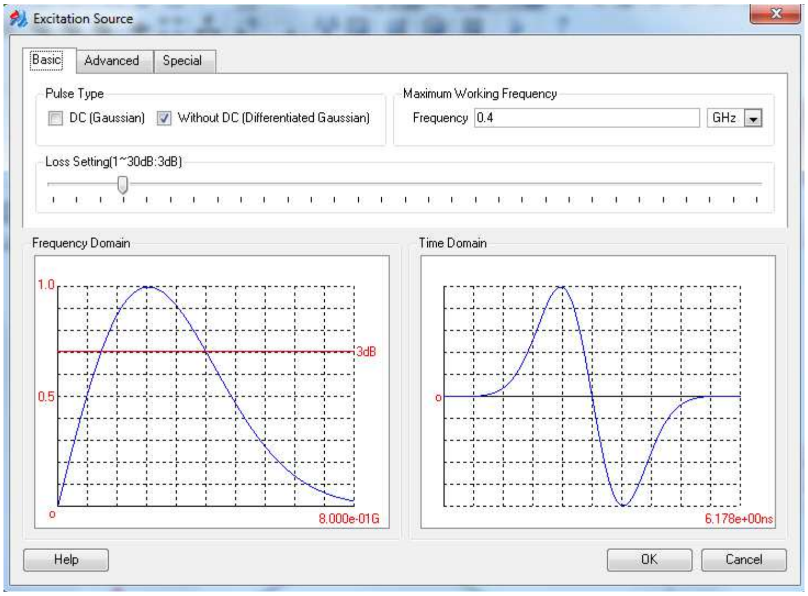
18.3 Set Excitation Source

Click on the **Set excitation source** button in the toolbar.

Since the maximum frequency of interest is 400MHz, so we input "0.4" in the **Maximum Working Frequency->frequency** box, and the default unit is GHz.

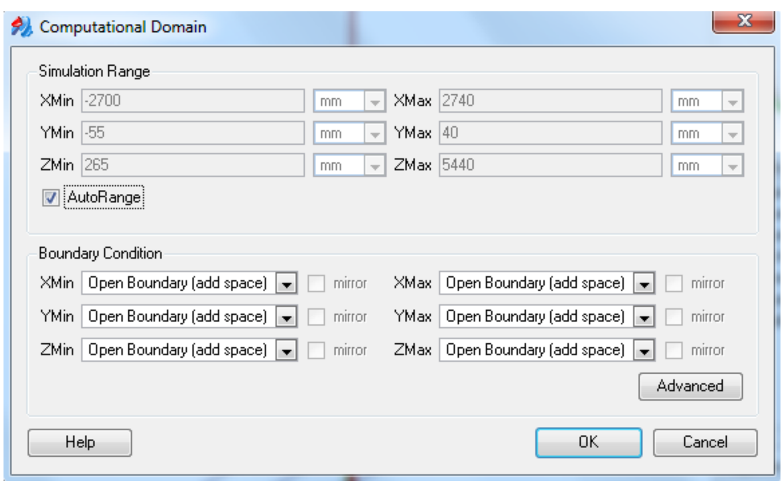
Except the plane wave source and mode excitation, in almost all the cases, it is no difference between the "DC (Pure Gaussian pulse)" and "Without DC (Differential Gaussian pulse)".

Click on the **OK** button to confirm the excitation pulse setting.



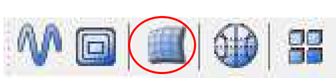
18.4 Set Domain and Boundary Condition

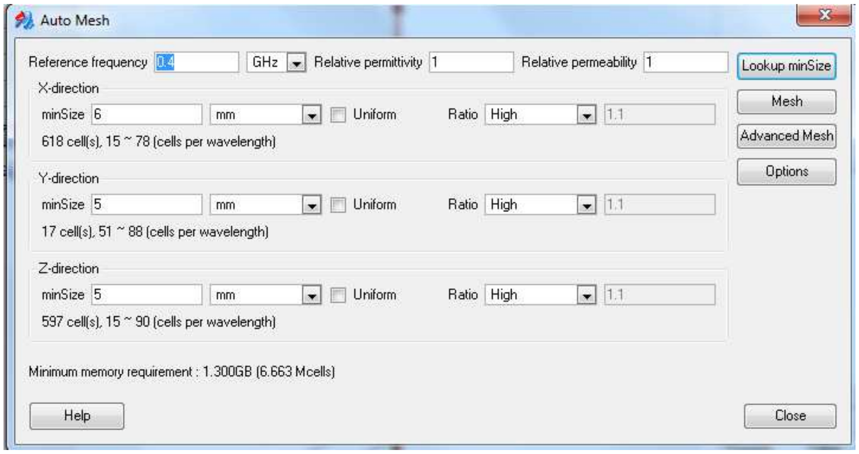
Click on the **Set boundary condition** button in the toolbar.



18.5 Design Mesh Distribution

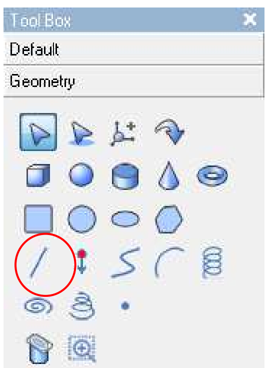
Click on the **Auto Mesh** button in the toolbar.



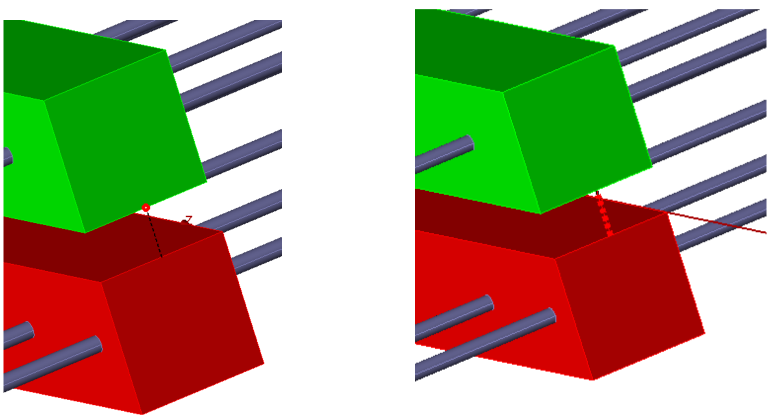


18.6 Set Excitation Port

Click on the **Line** icon in the **Tool Box->Geometry** box.



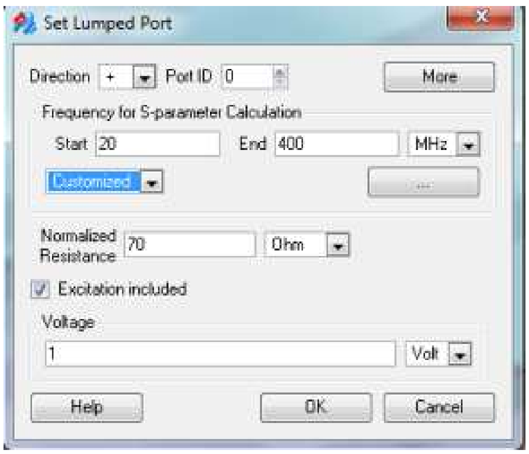
Snap to the center of the lower arm, press the left mouse button, and move to the center of the upper arm, press the left mouse button.

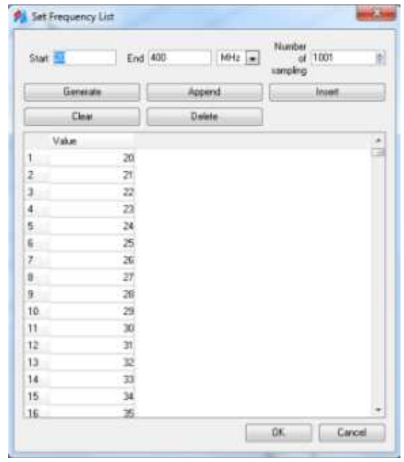


Select the line object in the **Object Browser** box and change its name to “excitation” in the **Properties** box.

Specify the frequency band of interest and click on the **Generate** button. Select the “Customized” option and click on the right button to specify the frequency list window.

Input the frequency resolution in the **Number of sampling** box.

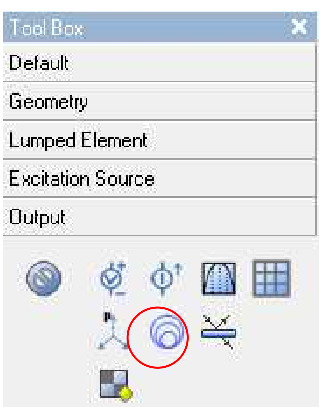




Click on the **OK** button to confirm the parameter settings.

18.7 Set Far Field Output

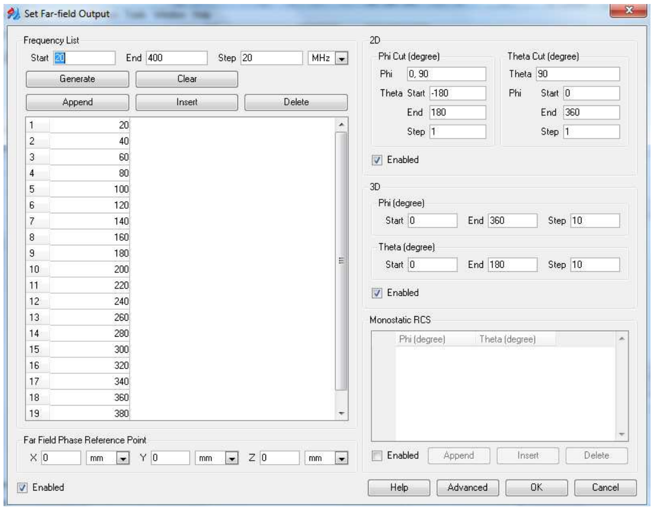
Click on the **Far field** icon in the **Tool Box->Output** box.



Check the **Enabled** box to enable the far field output options.

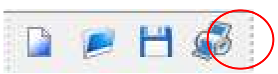
Specify a frequency list of interest, and click on the **Generate** button.

Check the **3D->Enabled** box to generate the 3D fat field pattern at the frequency list. Click on the **OK** button to confirm the far field pattern settings.



18.8 Save Project

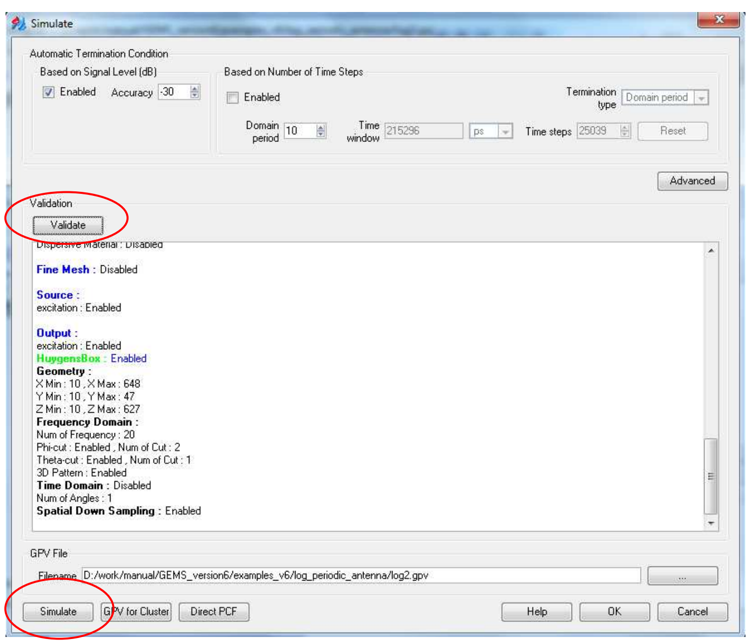
Click on the **Save** button in the toolbar.



18.9 Generate Simulation File

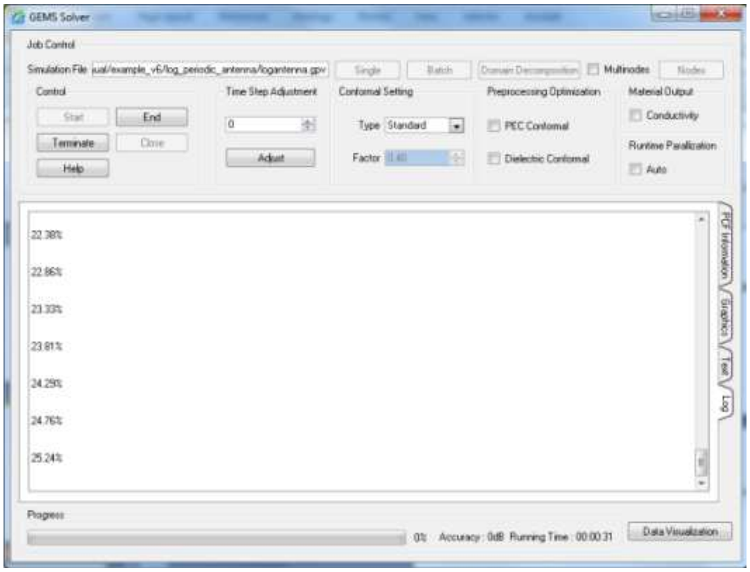
Click on the **PreCalculate** button in the toolbar.

Click on the **Validate** button to validate the project settings, and click on the **Simulate** button to open the *GEMS solver* window.



18.10 Simulate Project

Click on the **Start** button to start the simulation.



18.11 Result Visualization

Click on the **Data Visualization** button in the *GEMS Solver* window.

The direct results are listed in the result tree.

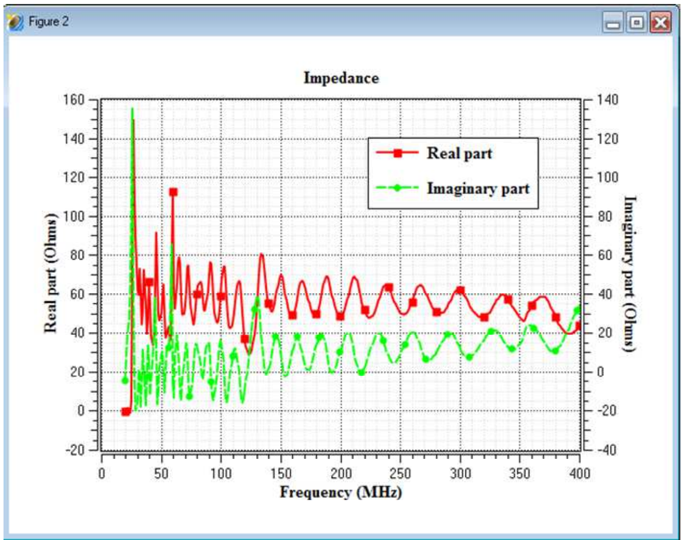
*You can also go to the project folder and double-click on the *.gpv file to open the GEMS display window and load the results in the result tree.*

*You can load multiple projects in the GEMS display window. Double-click on the project in the **Project** Box to make it active.*

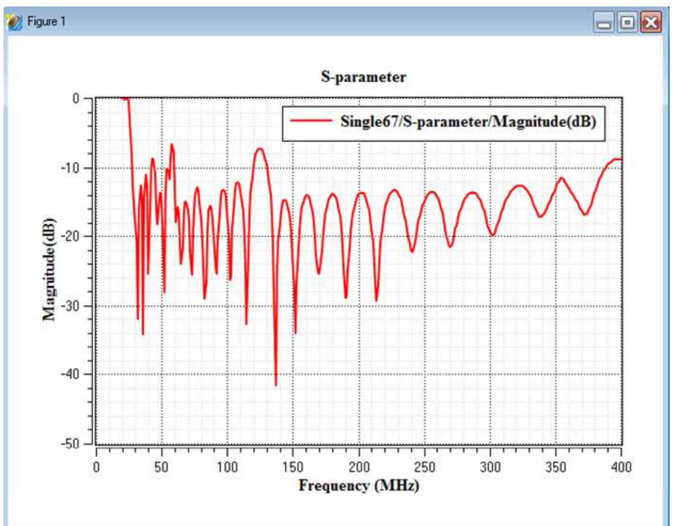
Double-click on the **Port->FreqDomain->Impedance->Real** option.

Select the **Port->FreqDomain->Impedance->Imaginary** option and then click on the **Add to current window** button in the toolbar.

The real part and imaginary part of the impedance will be plotted in the same figure.

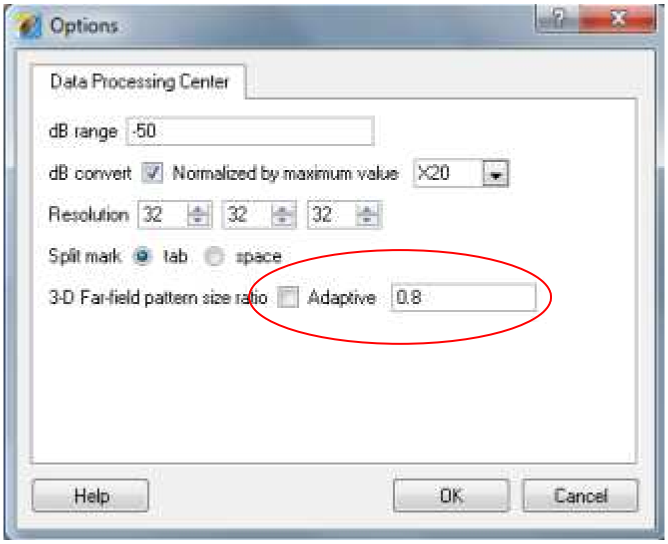


Double-click on the **Port->FreqDomain->S-parameter->Magnitude(dB)** option.

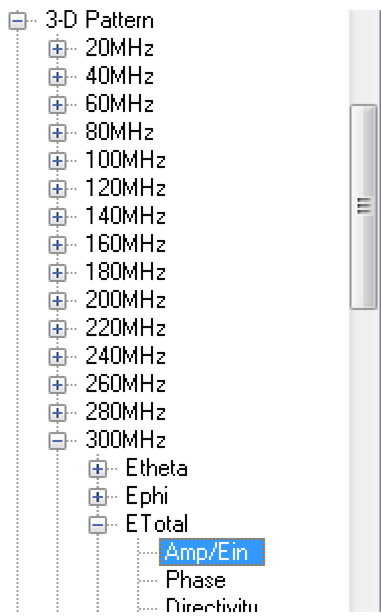


Click on the **View model** button and then on the **Transparent mode** button in the toolbar.

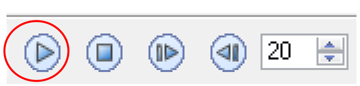
Select the **Default** option in the **Tools** menu in the toolbar. Uncheck and the **3-D far-field pattern size ratio** box and specify a proper number in the **Adaptive** box.

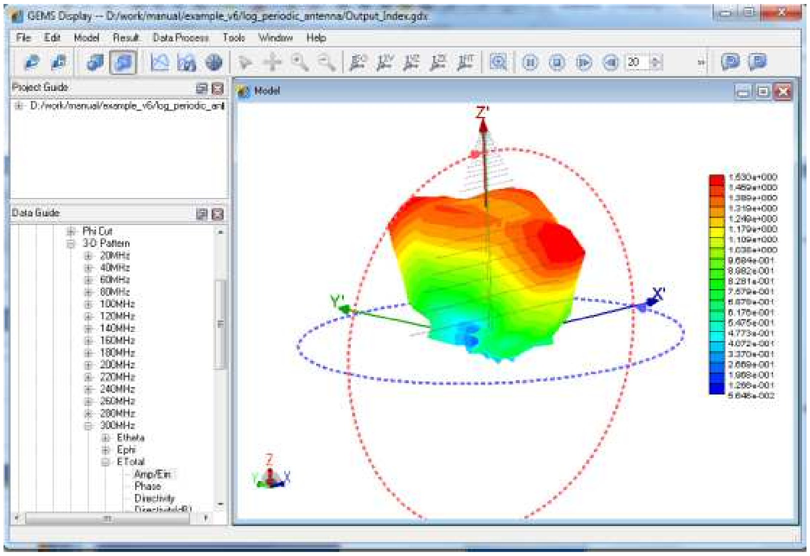


Select the **3-D Pattern->300MHz->Etotal->Amp/Ein** option in the result tree.



Click on the **Add to current window** button in the toolbar, and then on the **Play** button.




GEMS Display -- D:/work/manual/example_v6/log_periodic_antenna/Output_Index.gdx
File
Edit
Model
Result
Data Process
Tools
Window
Help
Project Guide
D:/work/manual/example_v6/log_periodic_ant
Data Guide
Phi Cut
3-D Pattern
20MHz
40MHz
60MHz
80MHz
100MHz
120MHz
140MHz
160MHz
180MHz
200MHz
220MHz
240MHz
260MHz
280MHz
300MHz
Etheta
Ephi
ETotal
Amp/Ein
Phase
Directivity
Model
Z'
Y'
X'

19

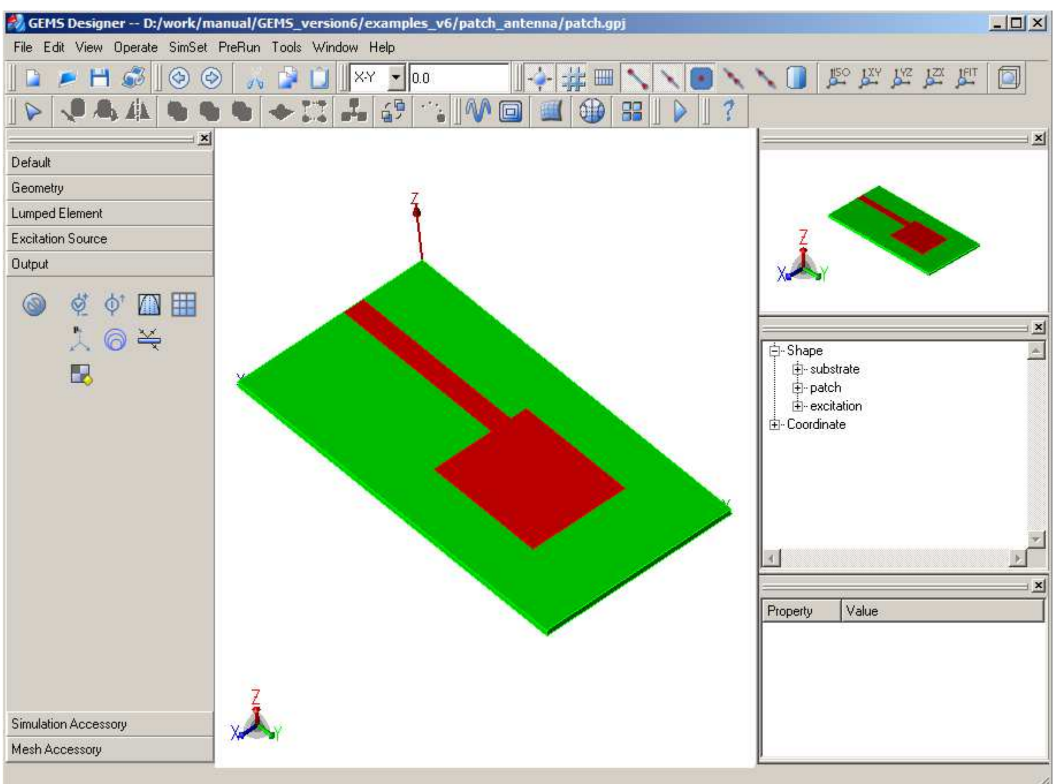
Example 19. Parameter Scan

Description: We set the length of a patch as a variable and investigate the variation of the return loss and far field pattern with the length of the patch.

Keywords: Parametric length, parameter scan, antenna.

19.1 Problem Configuration

The patch antenna includes a rectangular patch (thin PEC), microstrip feed line (thin PEC), substrate (relative permittivity=2.2), and ground plane (PEC). The mrcorstrip line is infinitely and terminated by the absorbing boundary.

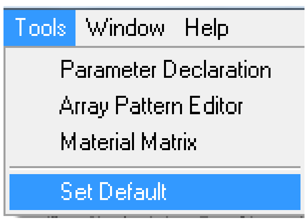


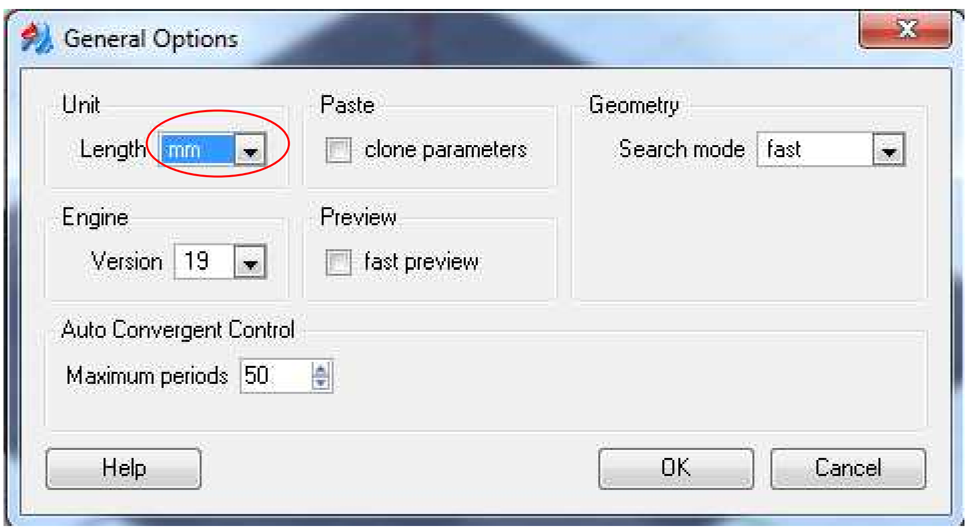
The frequency band of interest is from very low frequency to 20GHz. But the far field patterns are only at the several selected frequencies. Since the ground plane is infinitely large; hence, we can use the PEC boundary of computational domain to serve as the ground plane.

19.2 Create Antenna Model

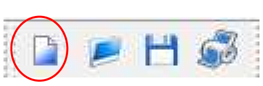
Follow the steps below to create the patch antenna model:

(1) Open the GEMS Designer
(2) Select the **Set Default** option in the **Tools** menu

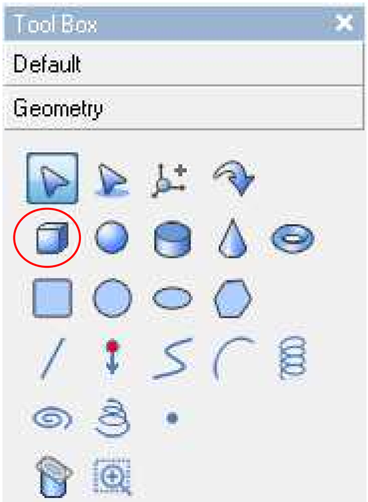


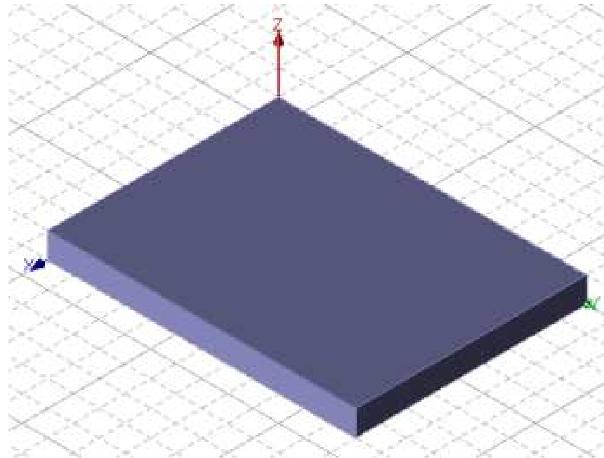


(3) Click on the **New** button in the toolbar to create a new project

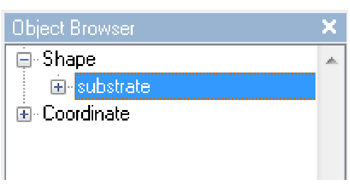


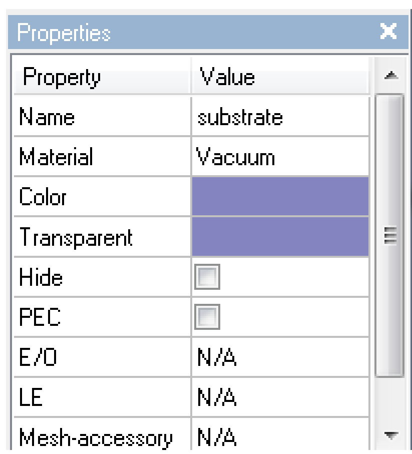
(4) Select the **Cuboid** icon in the **Tool Box->Geometry** box, and draw a box in the figure region.



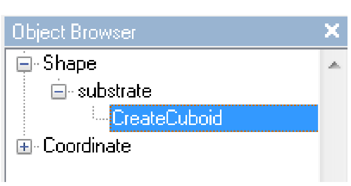


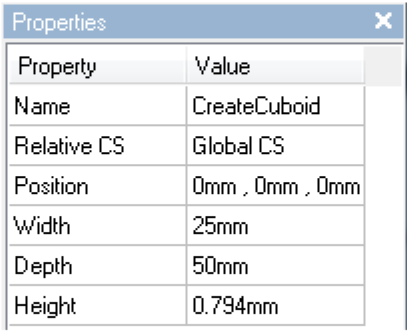
(5) Select the “Single0” in the **Object Browser** box, and change its name to “substrate” in the **Properties** box. Click on the **Color** bar in the **Properties** box and set the substrate color to be green.





(6) Select the **CreateCuboid** option in the **substrate** folder, and then change the dimensions of the substrate, the reference point (0, 0, 0), width=25mm, length=50mm, and thickness=0.794mm.

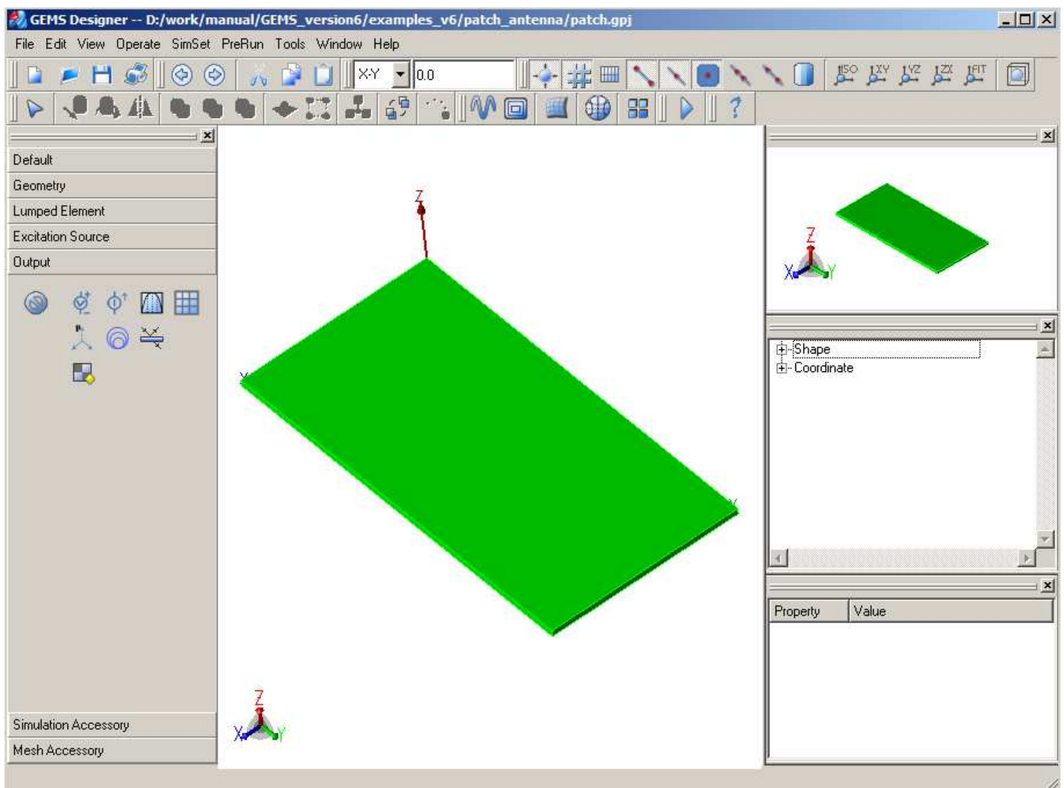




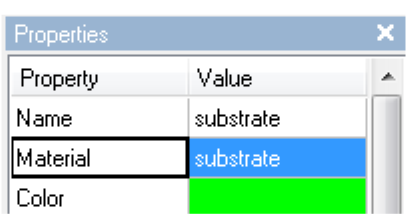
Properties

Property	Value
Name	CreateCuboid
Relative CS	Global CS
Position	0mm , 0mm , 0mm
Width	25mm
Depth	50mm
Height	0.794mm

Click on the **FIT** button in the toolbar to fit the substrate inside the figure region.



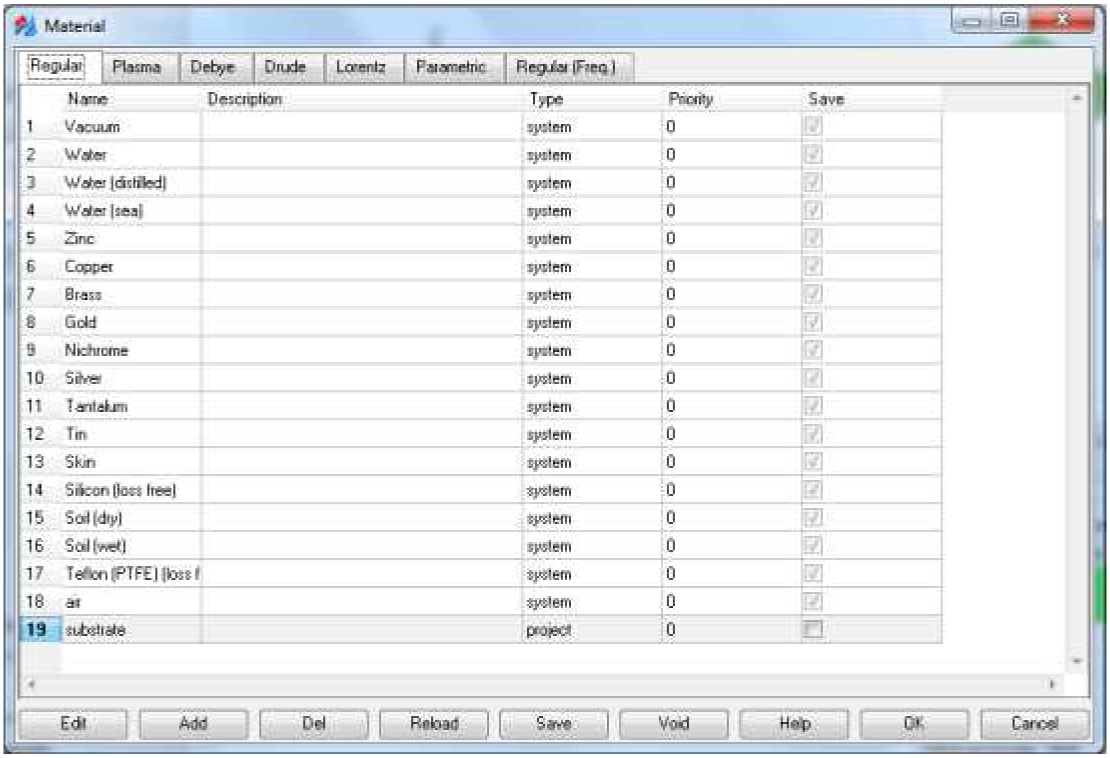
(7) Select the **substrate** option in the **Object Browser** box and click on the **Material** box in the **Properties** box. In this example, the substrate is lossless material, namely, its conductivity is zero.

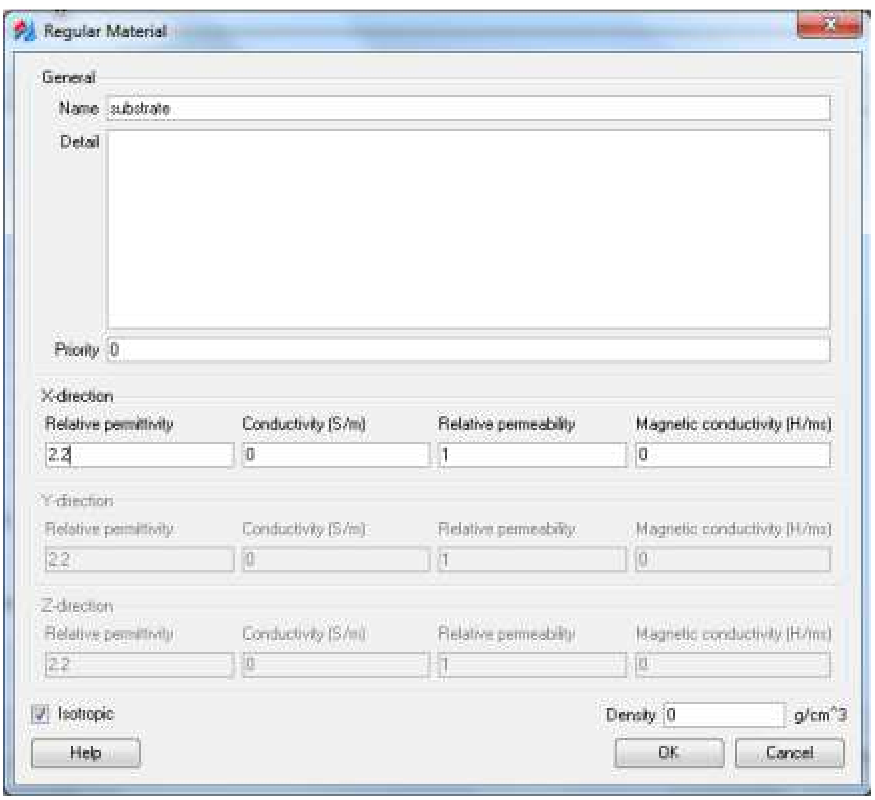


Properties

Property	Value
Name	substrate
Material	substrate
Color	

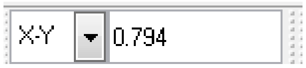
Click on the **Add** button to add a new material for this substrate. Specify a name in the **Name** box, and input the dielectric constant (2.2) in the **Relative permittivity** box.





Click on the **OK** button to confirm the dielectric setting.

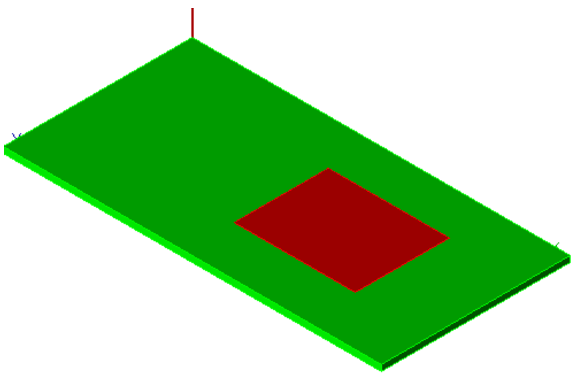
(8) Change the height of the drawing plane to be 0.794.



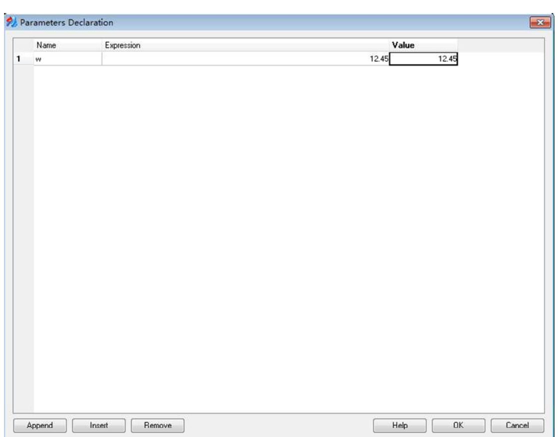
Click on the **Rectangle** icon in the **Tool Box->Geometry** box, and draw a rectangle on the surface of substrate.

Select the patch option in the **Object Browser** box, and change its name to "patch" and color to red, and check the **PEC** box in the **Properties** box.

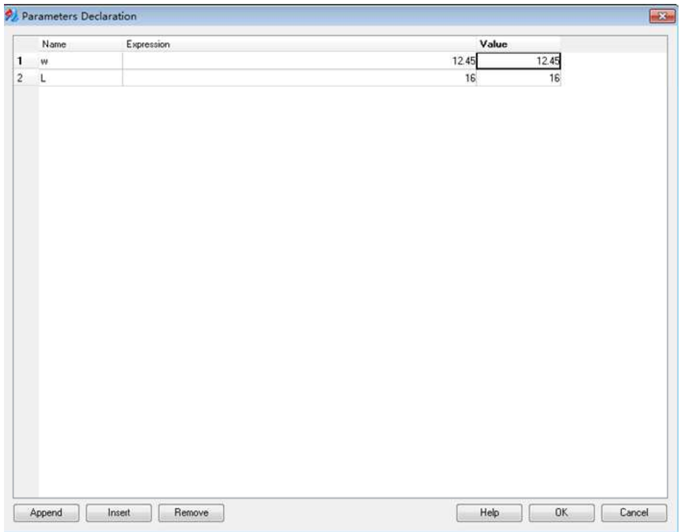
Select the **CreateRectangle** option in the **patch** folder in the **Object Browser** box, and change its coordinates to (6mm , 24mm , 0.794m), width= *w* mm the **Properties** box.



When the "*w*" is input in the "Width" option in the **Properties** box, a new window show up.

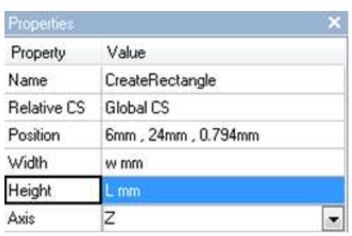


Click the box following the variable “*w*” option in the “Expression” column and specify an initial value, “12.45”. GEMS allows you to input an expression.



Click the “Append” button at the bottom, and click the box in the second row to add a new variable “L”. Click and input an intial value “16” in the expression box after the variable “L”.

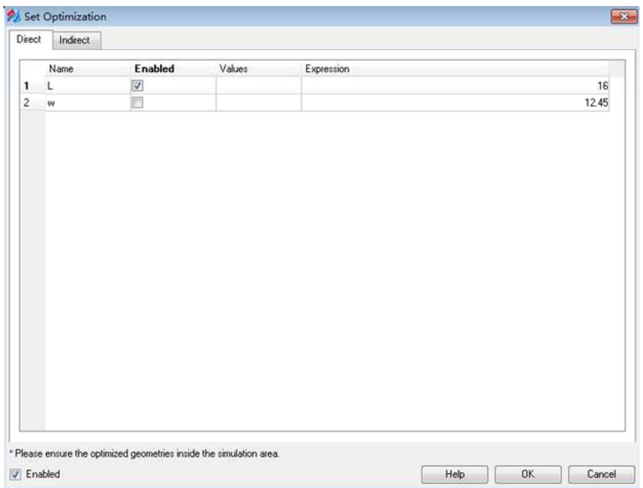
Click the box behind the “Height” option in the **Properties** box, and change the number to be “L”. The initial values of “w” and “L” are 12.45mm and 16mm, respectively.

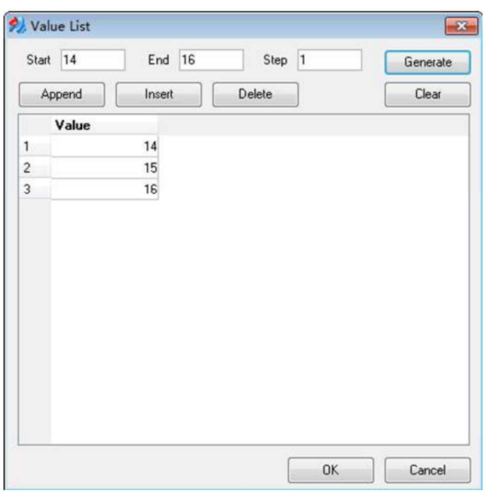


Select the “Optimization Setting” option in the **Simset** menu, and specify the variation range of the variable “L” in the popup window. Check the “Enabled” box at the bottom left corner, and then check the box after the variable “L”.

Click the box in the “Values” column and the “L” row, and specify the variation range and resolution in the popup window. Click the “Generate” button to generate the variable list. You can edit the list if necessary.

Click the **OK** button to confirm the variable value list.



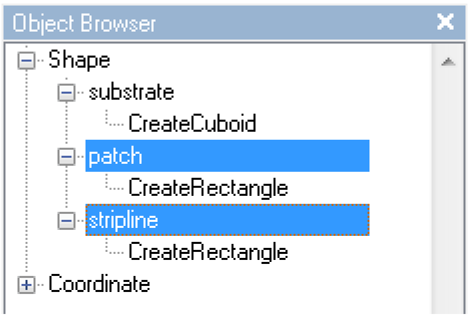


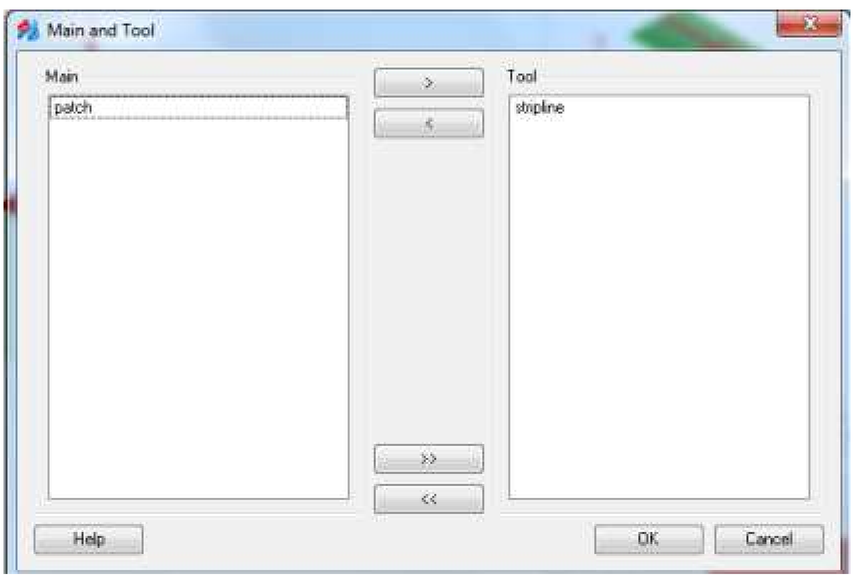
(9) Click on the **Rectangle** icon in the **Tool Box->Geometry** box, and draw a microstrip line on the surface of substrate.

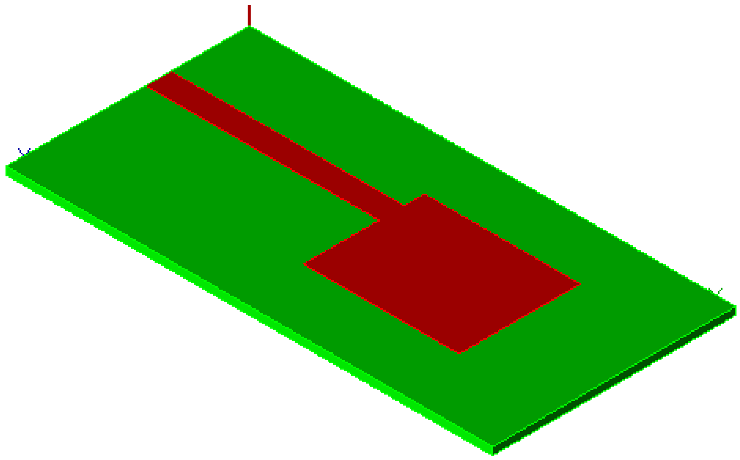
Select the microstrip line option in the **Object Browser** box, and change its name to "stripline" and color to red, and check the **PEC** box in the **Properties** box.

Select the **CreateRectangle** option in the **stripline** folder in the **Object Browser** box, and change its coordinates to (8.09mm , 0mm , 0.794m), width=2.56mm, and length=-24mm in the **Properties** box.

Select both the **patch** and **stripline** options in the **Object Browser** box, and click on the **Unite** button in the toolbar. The stripline and patch will be united as a single object.



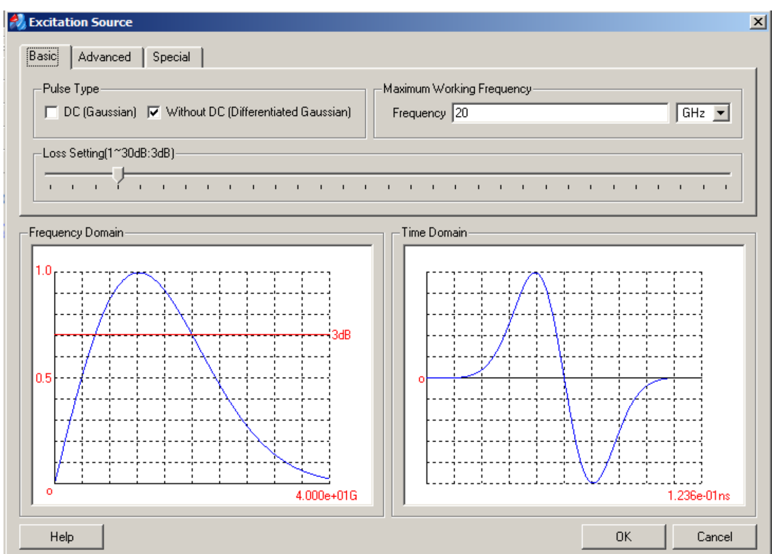




The order is very important for an object without thickness and it touches other objects. The wrong object order will erase the infinitely thin structure; in turn, the simulation result may be incorrect.

19.4 Set Excitation Pulse

Click on the **Set excitation source** button in the toolbar. Since the maximum frequency of interest is 20GHz, so we type "20" in the **Frequency** box (unit is GHz). Click on the **OK** button to confirm the excitation pulse setting.

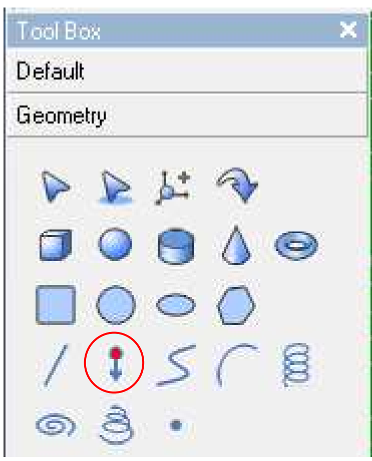


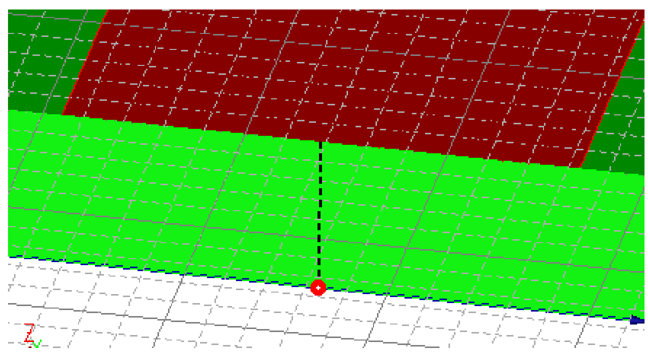
19.5 Create Lumped Port Excitation

Since the feed line is terminated at the edge of the substrate. A matched load is required when an excitation is added at this port. The lumped port is GEMS includes a voltage source from the ground plane to the signal line, and a resistance that will serves as internal resistor of excitation source and a reference for the return loss calculation.

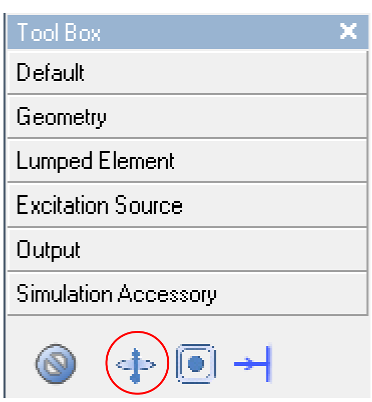
Click on the **Plumb line** icon in the **Tool Box->Geometry** box, and move the mouse icon to the center of the stripline, press the left mouse button, move the mouse icon along the vertical direction (the drawing plane is "X-Y"), stop when it meets the bottom of the substrate.

Select the **Plumb line** option in the **Object Browser** box and change the name of the plumb line to be "excitation" in the **Properties** box.





Select the "excitation" option in the **Object Browser** box and click on the **Lumped port** icon in the **Tool Box->Simulation accessory** box.

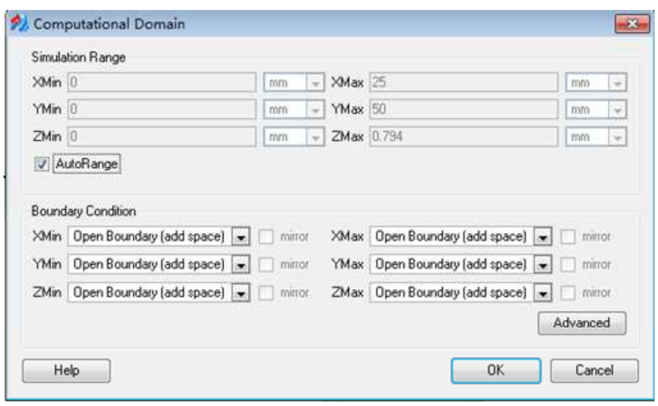


The black end should touch the signal conductor (higher potential). In this example, it should touch the stripline. It can be adjusted by the sign in the **Voltage Direction** box.

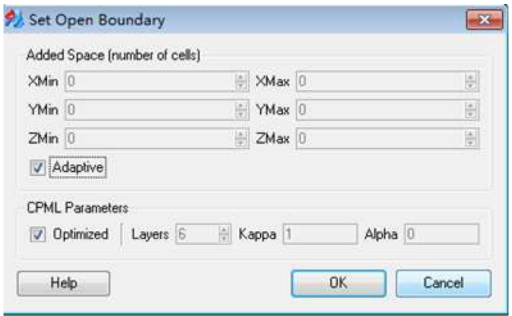
19.6 Domain and Boundary Settings

Click on the **Set boundary condition** button in the toolbar.

This is an open problem with a finite structure. The "Open Boundary (add space)" is used to truncate all the six walls of the computational domain.



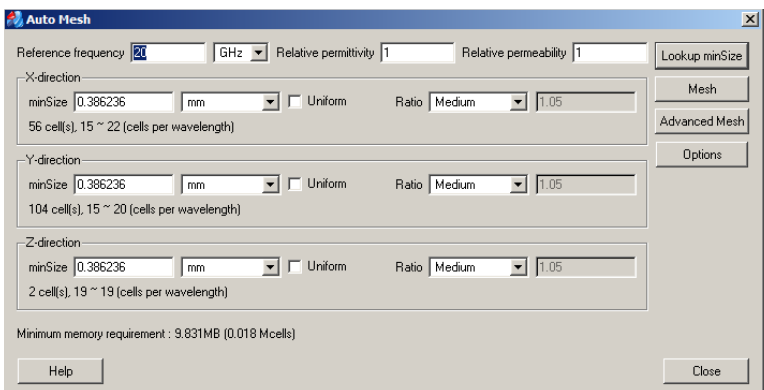
When the dimensions of problem geometry in two directions are much larger than that in the third direction, this geometry is referred as to the ill-conditioned domain. Click the "Advanced" button in the Computational Domain" window, and check the "Adaptive" box that allows the GEMS to adjust the domain to the "regular" domain automatically. In any case, you can uncheck the "Adaptive" button and adjust the white space between the objects and domain boundary manually.



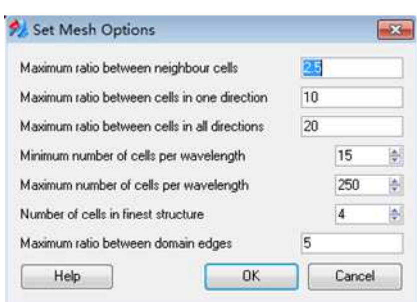
Checking the "Optimized" box allows the GEMS to select the proper PML parameters to achieve the better simulation result.

19.7 Mesh Generation

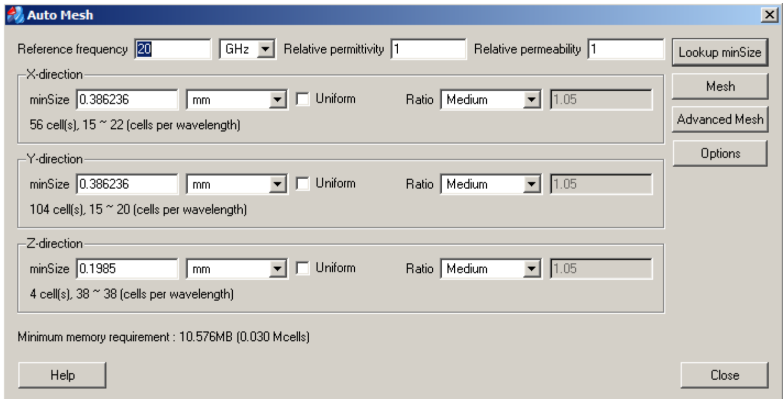
Click on the **Auto mesh** button in the toolbar.



For the simple case, you can adjust the parameters in the **Options** window to find the proper mesh size and ratio, and then generate a good mesh distribution. For example, if two cells are not sufficient to describe the field variation inside the substrate in the vertical direction, you can change the number from "2" to "4" in the **Number of cells in finest structure** box.



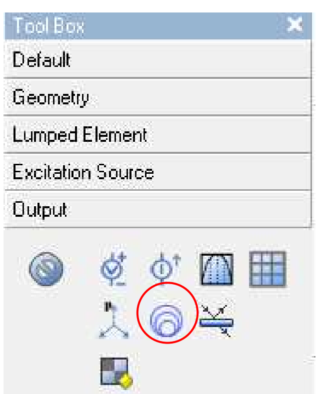
Press the **Lookup minSize** button again, and there will be 4 cells in the vertical direction. Click on the **Close** button to confirm the mesh design.

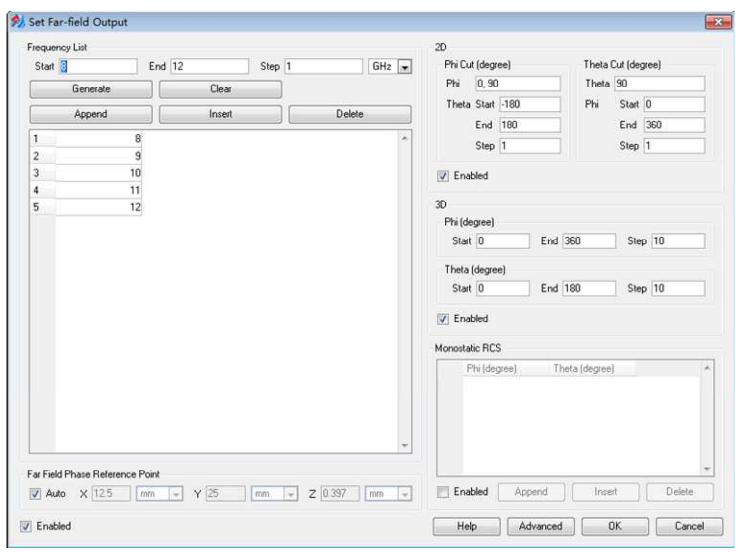


19.8 Far Field Output Setting

In this example, we cannot use a closed Huygens' surface for the far field pattern calculation since the size of substrate and ground plane is infinite. Because we are only interested in the far field in the upper hemisphere, the Huygens' surface will be degenerated into one surface at z=Zmax.

Click on the *Far-field* icon in the **Tool Box -> Output** box. Check the **Enabled** box to enable the far field output parameter settings. Specify a frequency list and enable the 2D and 3D far field options. Click on the **OK** button to confirm the far field parameter settings.





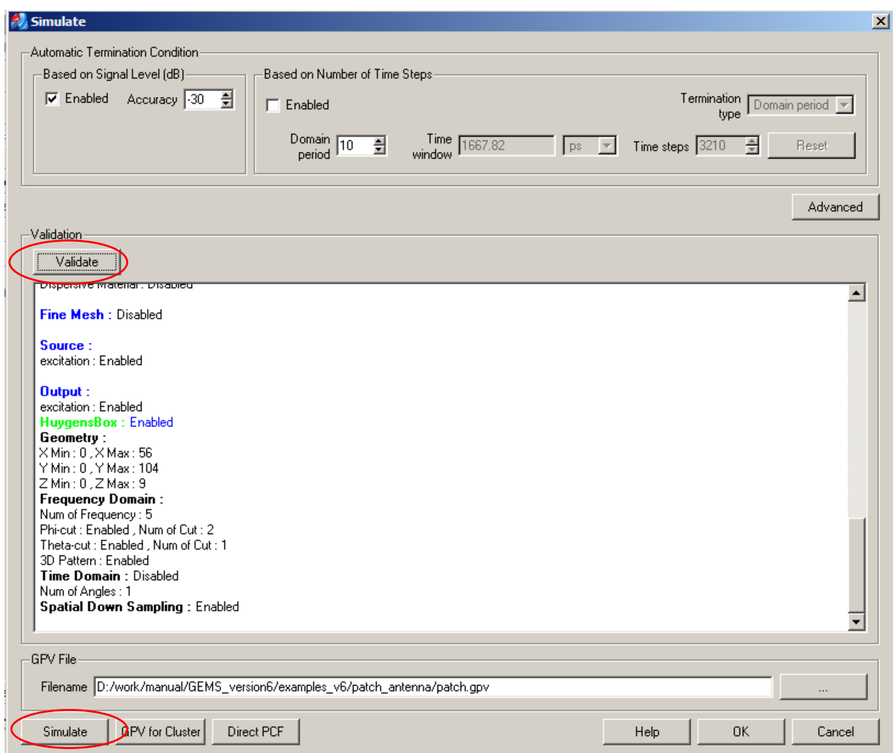
Check the "Auto" box and the plot of far field pattern will set the position of excitation source as the reference when the far field pattern is plotted together with the project model.

19.9 Save Project

Click on the **Save** button in the toolbar to save the project. The saved project has a default extension name “gpj”. It includes all the project information, and it can be loaded to the GEMS Designer later.

19.10 Generate Simulation File

Click on the **Precalculate** button in the toolbar to generate the simulation file.

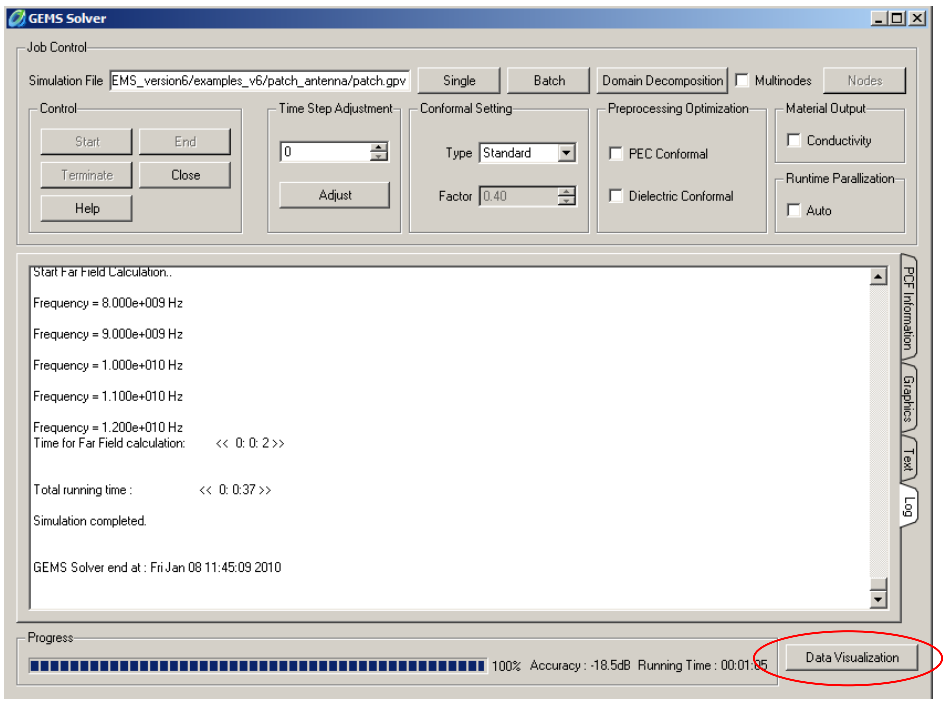


*Check the **Enabled** box and select a convergence criterion in the **Based on Signal level(dB)** box. GEMS simulation stops when the convergence criterion is achieved.*

Click on the **Validate** button to validate the project settings. If there is no message in red, click on the **Simulate** button to start the project simulation.

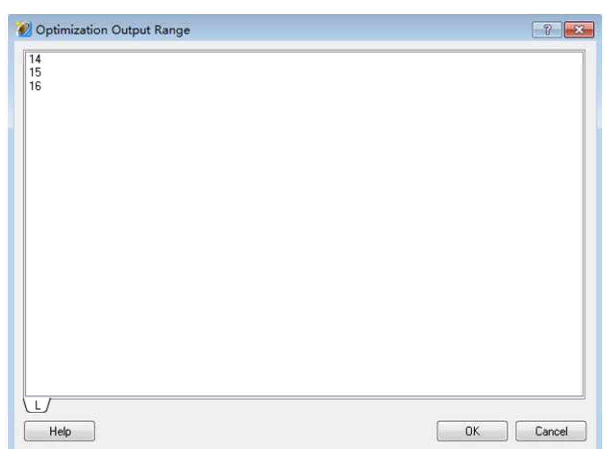
19.11 Simulate Project

Click on the **Simulate** button in the **Simulate** window to open the **GEMS Solver** window. Click on the **Start** button to start the simulation. The simulation status will also be displayed in the progress bar. The simulation information is summarized and stored in the **GEMSSummary.txt** file. Click on the **Data Visualization** button to open the display window.

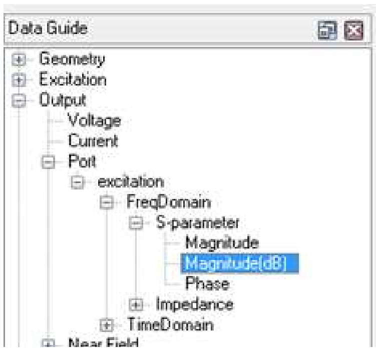


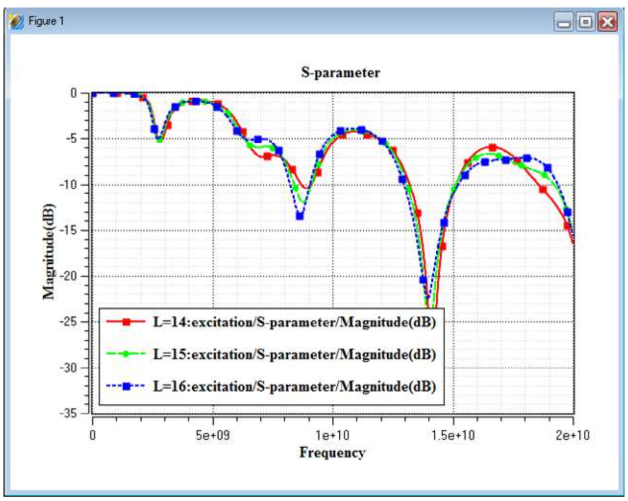
19.12 Result Visualization

Click on the **Data Visualization** button to open the *GEMS display* window. The direct simulation results are listed inside the result tree. Select an "Optimization" option in the Result menu. Select the three numbers "14", "15", and "16" in the popup window.

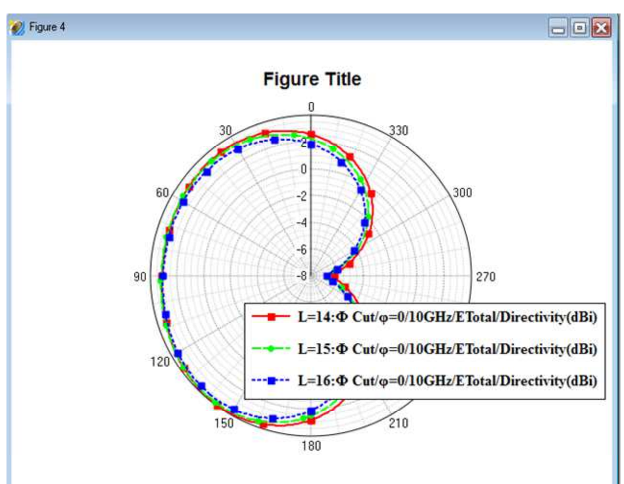


Select and double-click the "Output->Port->excitation->FreqDomain->S-Parameter-> Magnitude(dB)" option,





Three curves correspond to the three dimensions of the patch size. Select the "Output->Far Field->FreqDomain->PhiCut->10GHz->Dicrectivity(dB)" option, and click the **Polar** button in the toolbar.,



20

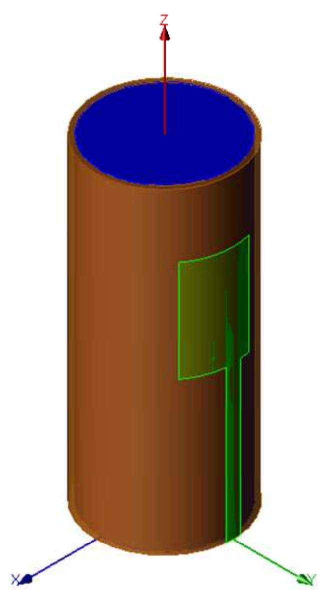
Example 20. Conformal Patch Antenna

Description: A patch is projected on to a surface of a cylinder. The conformal antenna includes a conformal patch, dielectric layer and a PEC cylinder inside.

Keywords: Conformal patch antenna, lumped port, return loss and far field pattern.

20.1 Problem Configuration

A patch is project on to the surface of a cylinder. The patch length and width are 16 mm and 12.45 mm, respectively. The radius and length of the PEC cylinder are 10 mm and 50 mm, respectively. The thickness and dielectric constant of the cylindrical dielectric coating are 0.794mm and 2.2, respectively.



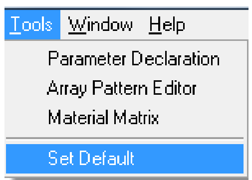
20.2 Create Problem Model

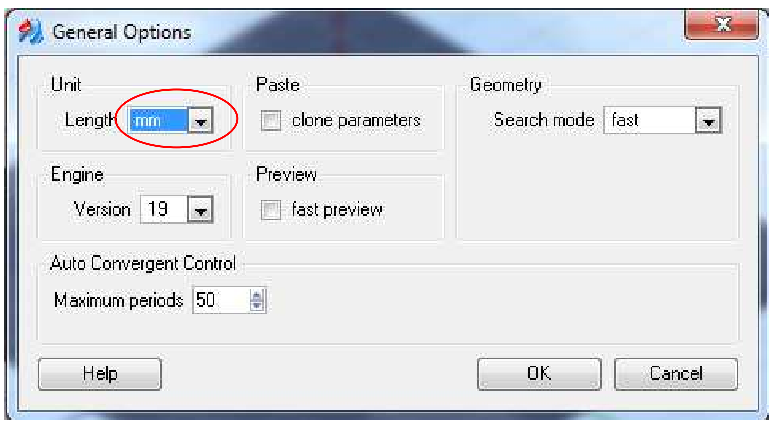
Following the steps below to draw a PEC cylinder with the radius = 10.794mm and length = 50mm:

(1) Open the GEMS designer

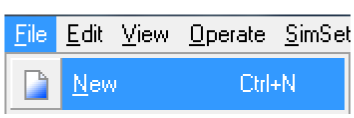
(2) Specify the project unit, which cannot be changed during the project modeling though you can input a variable in any units; however, the default unit can be only specified once at the beginning.

Select the **Tools->Set Default** option, and then select "mm" in the **Unit->Length** box as the project unit, click on the **OK** button to close the window.





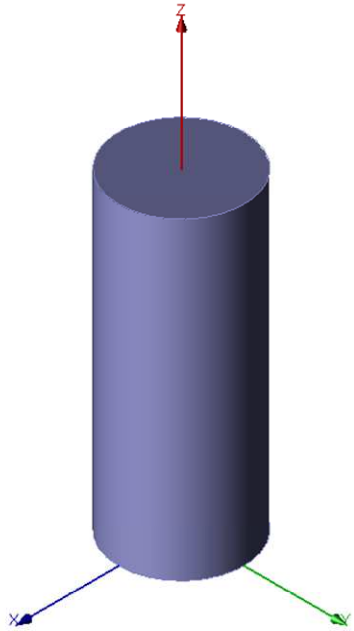
(3) Click on the **New** button in the toolbar or select the **New** option in the **File** menu.



(4) Click on the **Cylinder** icon in the "Tool Box->Geometry" box.

(5) Move the mouse icon to the origin point (0, 0, 0) and click the left mouse button, Move the mouse icon to draw a circle in the X-Y plane, and click the left nouse button at a proper radius. Move the mouse icon in the vertical direction to draw a cylinder and click the left mouse button at the proper height.

(6) Select the cylinder in the **Browser Object** box and change its name and color, and adjust the dimensions to the desired values in the **Properties** box.

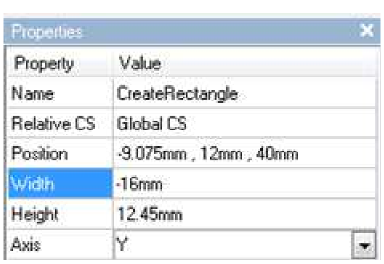


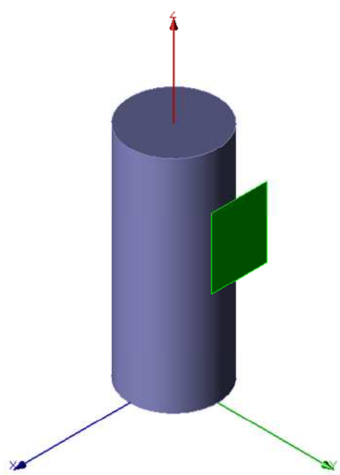
Following the steps below to draw a patch with the width = 12.45mm and length = 16mm:

(1) Select the drawing plane

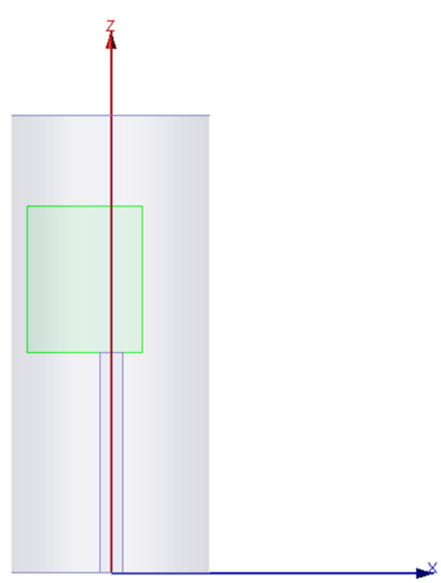
(2) Click the “ZX” button in the toolbar.

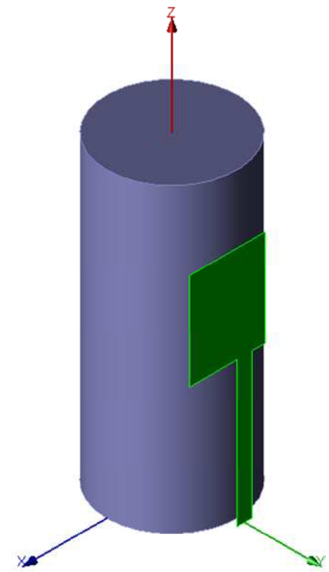
(3) Draw a rectangle with the length = 16mm and width = 12.45mm.





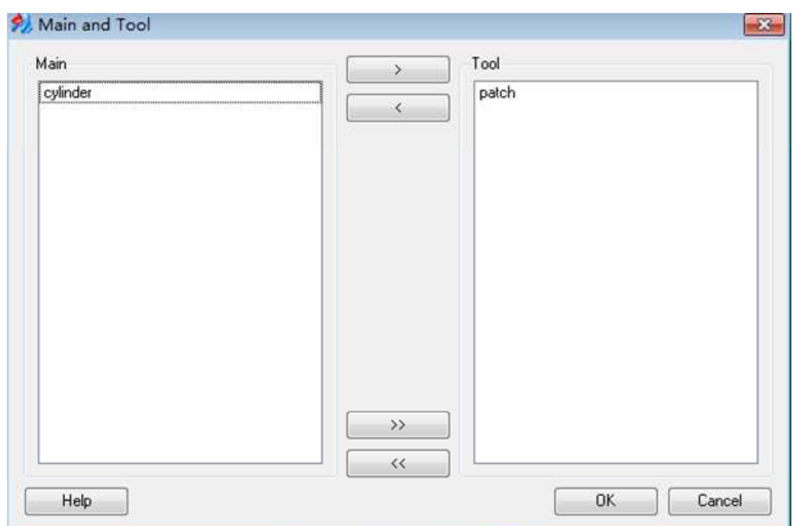
Following the similar steps to draw a feed line with the width = 2.45mm and length = 24mm. Select the rectangle and feed in the **Browser Object** box, and unite them as a single object.



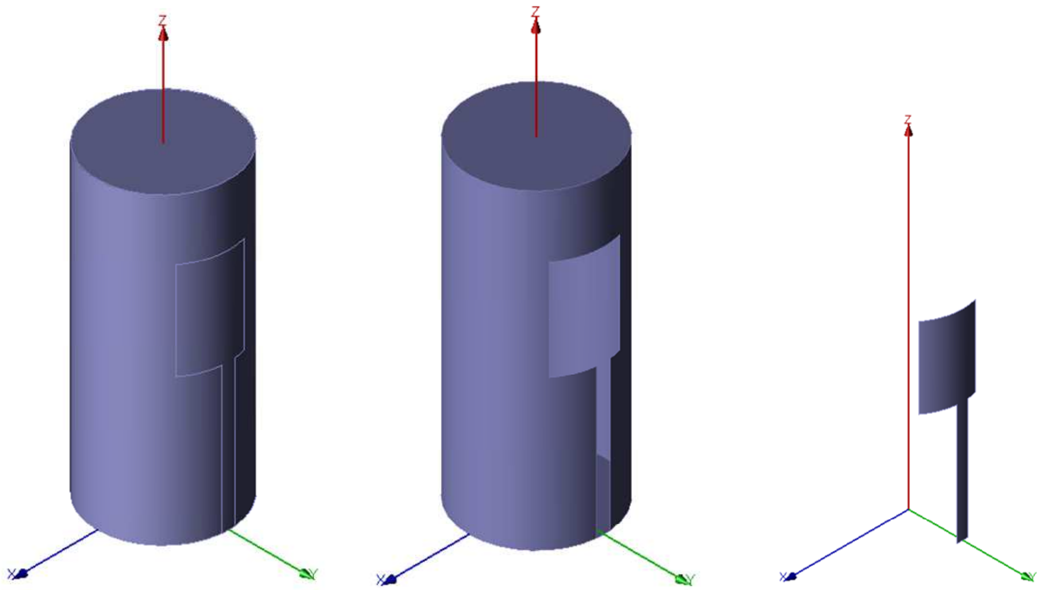


Following the steps below to project the patch to the cylinder:

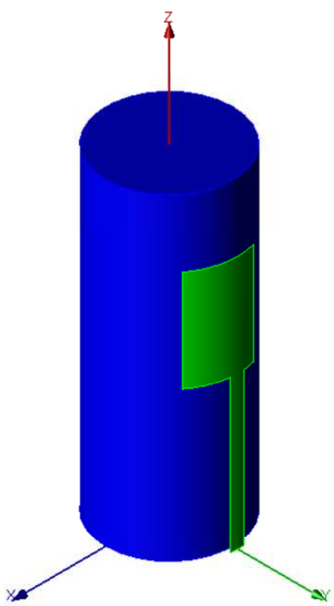
(1) Select the cylinder in the **Browser Object** box;
(2) Select the patch in the **Browser Object** box;
(3) Select the "Project" option in the **Operate** menu;



(4) Click the **OK** button, the patch will be printed on the surface of cylinder. Click the “Select face” icon in the “Tool Box->Geometry” box and select the surfaces of the cylinder except the patch. Press the delete key on the keyboard. Only patch is left. Specify the color of patch and select the PEC in the **Properties** box.



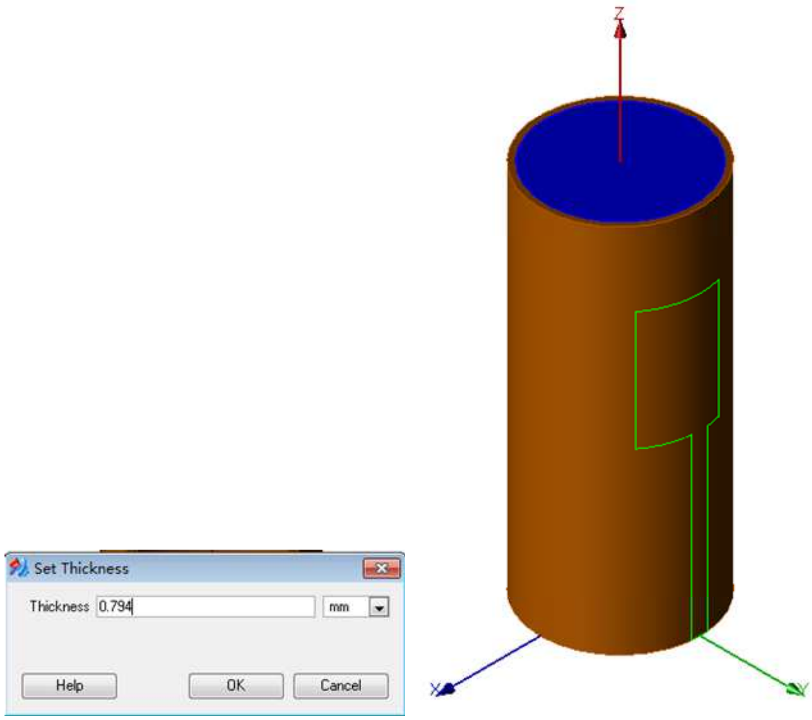
Draw a PEC cylinder with the radius = 10mm and length = 50mm. Specify the color of patch and select the PEC in the **Properties** box.



Following the steps below to generate a dielectric coating:

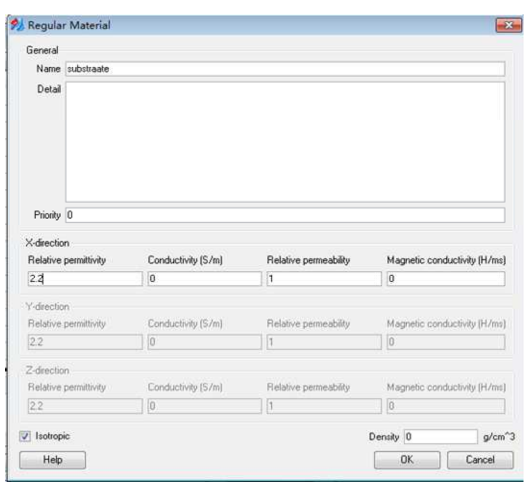
(1) Click the “Select face” icon in the “Tool Box->Geometry” box and select the surfaces of the cylinder. Press the “Ctrl + c” and then the “Ctrl + v” keys on the keyboard to generate a surface.

(2) Select the “face_0” in the **Browser Object** box and select the “Operate->Shaelling” option. Specify the thickness to be 0.794mm.



(3) Select the “face_0” in the **Browser Object** box and change its name to be “substrate” and color.

(4) Select the “substrate” in the **Browser Object** box and click the “Material” option in the **Properties** box. Specify the dielectric constant to be 2.2 and no loss.



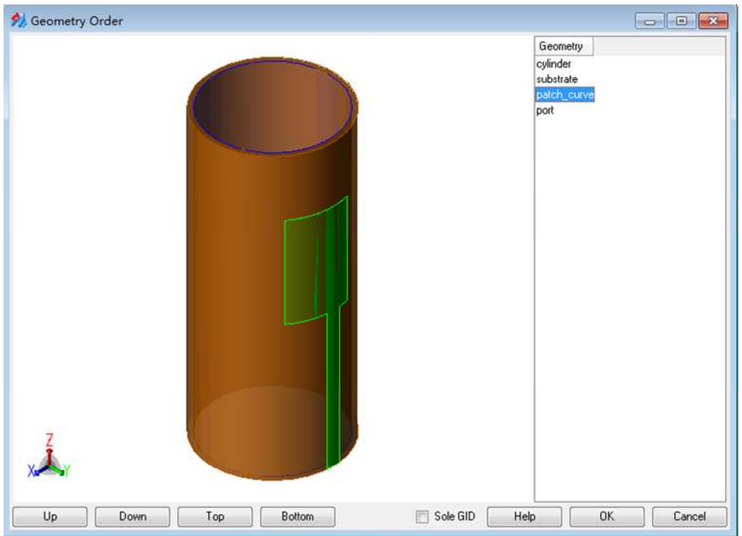
20.3 Adjust Object order

If any object inside the domain has no thickness, the order is very important. Improper order of objects may result in wrong results.

Following the steps below to adjust the object order:

(1) Click the order adjust button in the toolbar;

(2) The patch without thickness should be below the cylinder and substrate coating. Select the patch option and click the **Down** button to move the patch below the cylinder and substrate.

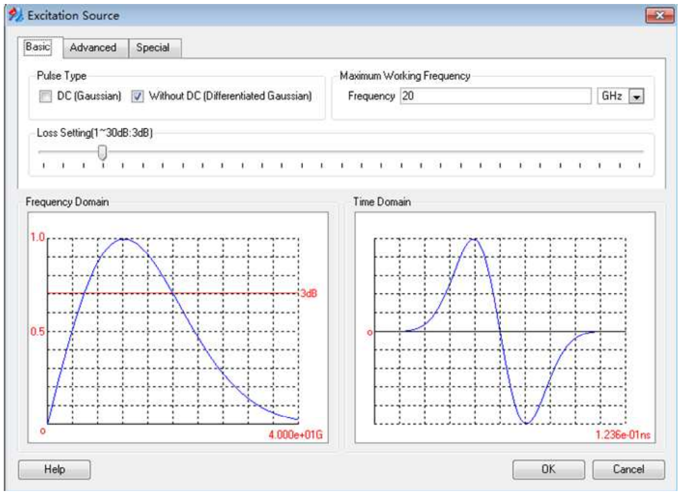


20.4 Set Excitation Pulse

Follow the steps below to set the excitation pulse:

(1) Click the "Set excitation source" button in the toolbar;

(2) Check the "Without DC" option and input "20" in the "Maximum working frequency" box. The frequency unit is GHz. The excitation pulse and its frequency spectrum are displayed in the window below.

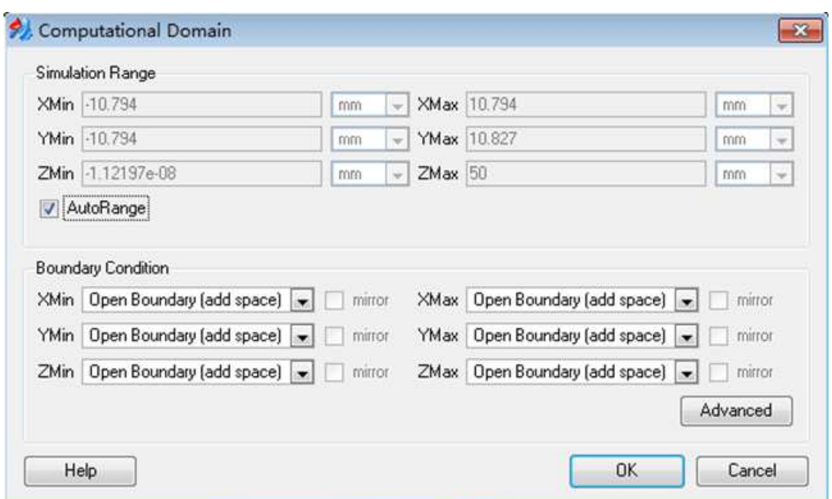


20.5 Set Boundary and Domain Size

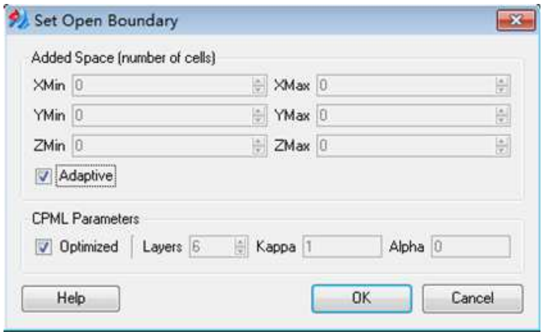
Follow the steps below to set the domain size and boundary condition:

(1) Click the "Set boundary condition" button in the toolbar;

(2) Set the domain and boundary parameters in the popup window. Checking the **AutoRange** box will allow GEMS to find the minimum box that encloses all the objects. For the open space problem, the "Open Boundary (add space)" is used to truncate all the six walls.



The window including the advanced options of boundary and domain is launched by clicking the **Advanced** button. Check the **Adaptive** box, and GEMS will add proper white space between the objects and domain boundary. Check the **Optimization** box, and GEMS will select the proper boundary parameters in the absorbing boundary condition.

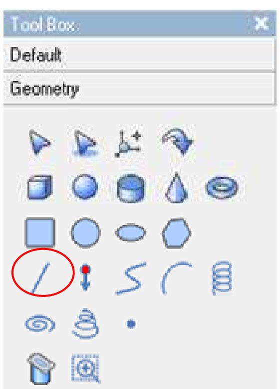


(3) Click the OK button to confirm the boundary and domain settings.

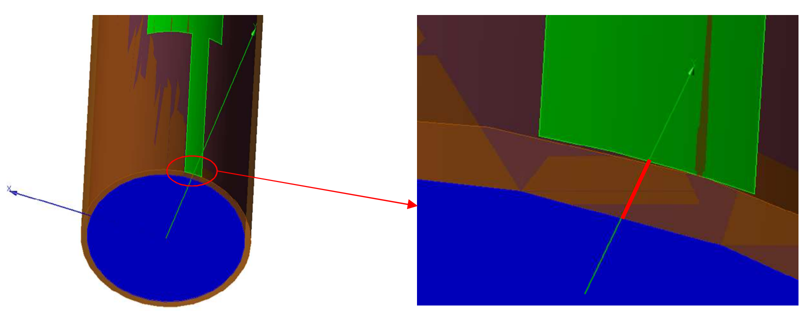
20.6 Add Excitation Port

Follow the steps below to add the excitation port:

(1) Click the “Line” icon in the “Tool Box->Geometry” box;

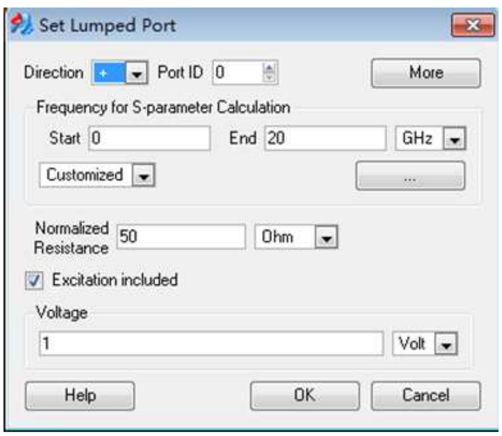


(2) Draw a line between the inner conductor and feed line.



(3) Select the line in the **Object Browser** box and click the “Lumped port” icon to assign the lumped port to the line.

(4) Specify the lumped port parameters in the popup window.



Direction: the black end points to the feed line
Port ID: it is meaning for single excitation source
Start: start frequency
End: end frequency
Normalized Resistance: internal resistance and normalized impedance for the return loss calculation.
Excitation included: check it for an excitation port, otherwise, it is for output port.
Voltage: excitation source level

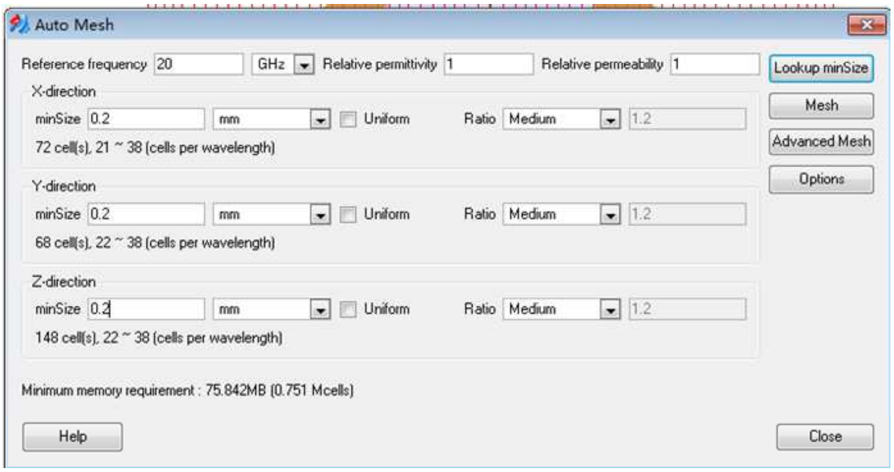
20.7 Design Mesh Distribution

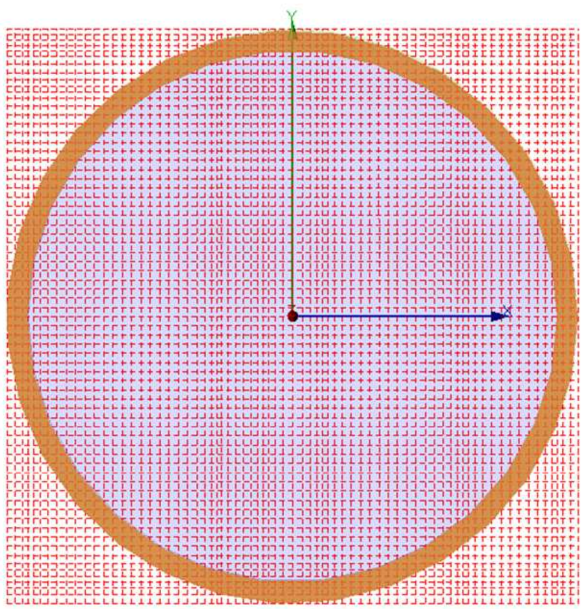
Follow the steps below to design the mesh distribution:

(1) Click the “Auto mesh” button in the toolbar;

(2) Find the fine structure of interest.

The fine structure is the thickness of the substrate, which is equal to 0.794mm. The minimum cell size should be less than 0.397mm. For the safety, it is taken to be 0.2mm.





If the curved patch and feed line have no thickness, they do not require the small cell size to describe the curved surface, instead, the large cell size can ensure that the surface is not leaking.

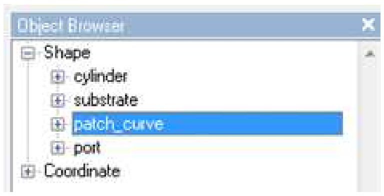
If the curved patch and feed line have a finite thickness, they do require the small cell size in the patch and feed line area to describe the curved surface. For example, the local minimum cell size should be smaller the half thickness of the patch thickness.

20.8 Set Output Parameters

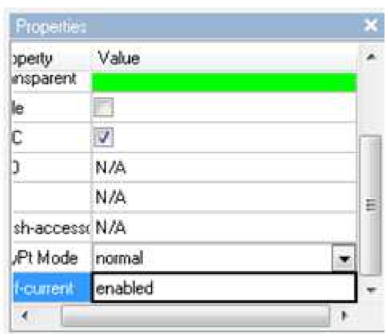
The output parameters in this project are return loss, surface current on the patch, and far field patterns. The return loss is calculated automatically when the lumped port is added.

Follow the steps below to set the surface current distribution:

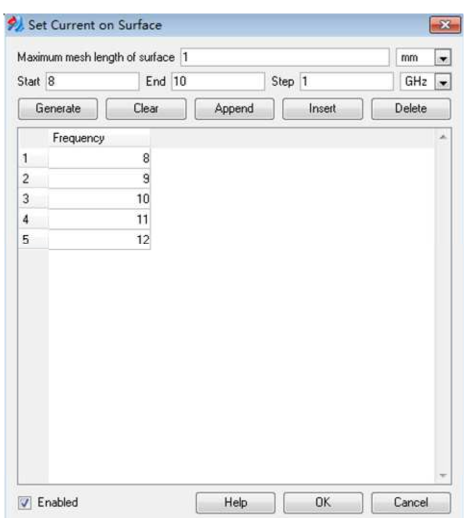
(1) Select the "patch_curve" option in the **Object Browser** box;



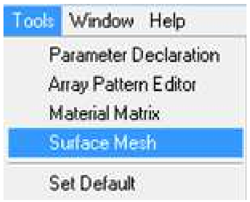
(2) Click the "Sur_current" option in the **Properties** box;

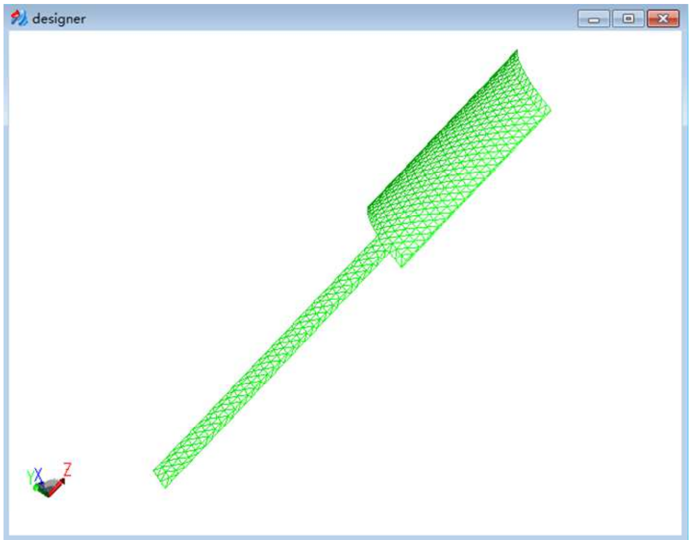


Specify the frequency list of interest, and the minimum length of the surface mesh in the popup window.



(3) View mesh by selecting the "Tools->Surface Mesh" option;

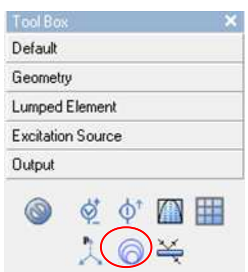




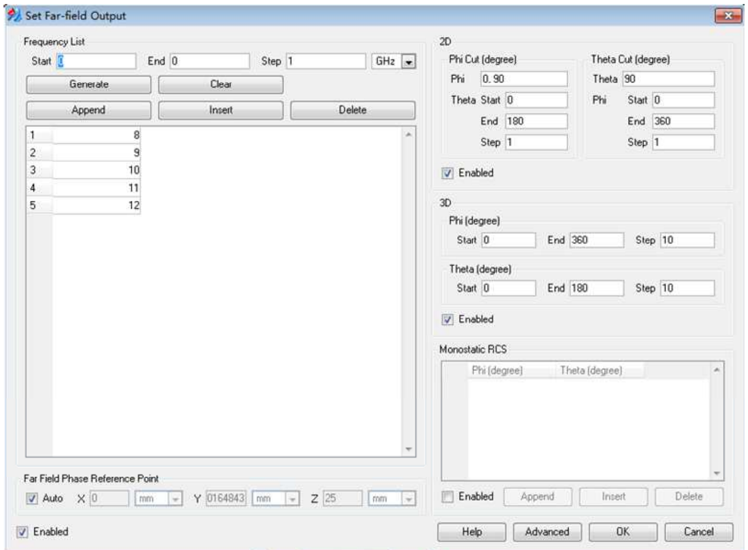
If the mesh is not sufficient for the description of the patch structure, you can adjust the number inside the "Maximum mesh length of surface" box in the **Set Current On Surface** window.

Follow the steps below to set the far field outputs:

(1) Select the "Far-field" icon in the "Tool Box->Output" box;



(2) Specify the frequency list and output options;

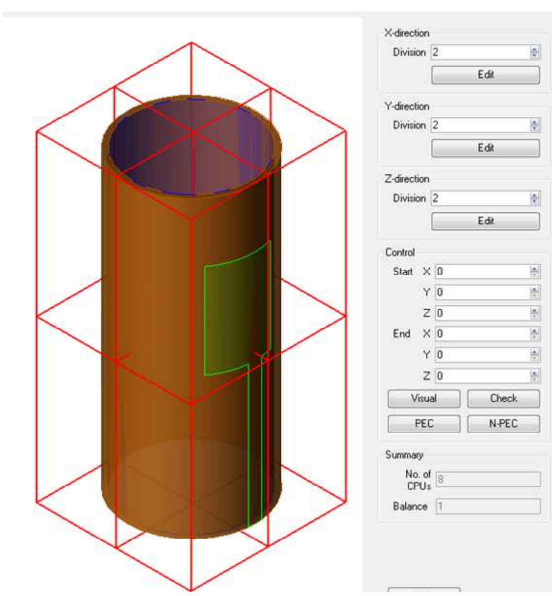


20.9 Parallel Processing Setting

If we have a workstation including 8 CPUs, we can split the domain into 8 sub-domains. Follow the steps below to set the parallel processing.

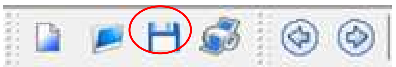
(1) Click the "Set parallel info" in the toolbar;

(2) First split the domain size that has more number of cells into 2. Then split the domain size that has more number of cells into 2 again. Then split the domain size that has more number of cells into 2 again.



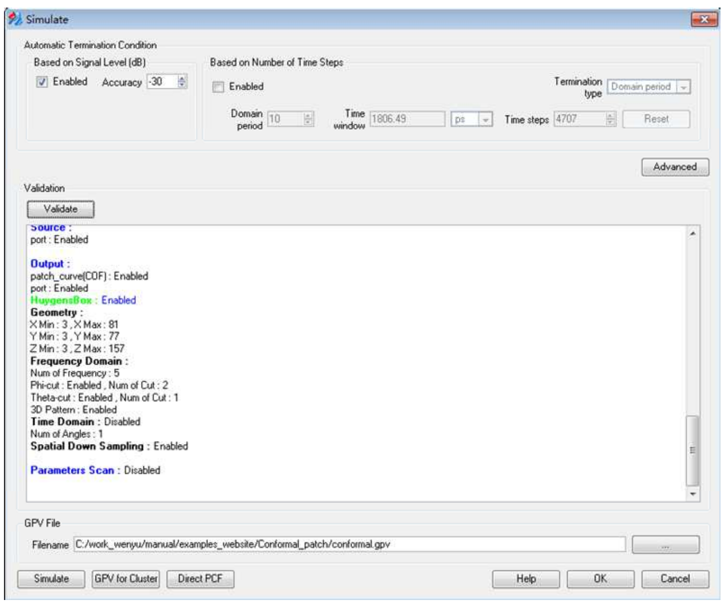
20.10 Save Project

Click the “Save” button in the toolbar to save the project.



20.11 Validate Project Settings

Click the “PreCalculate” button in the toolbar to validate the project settings.

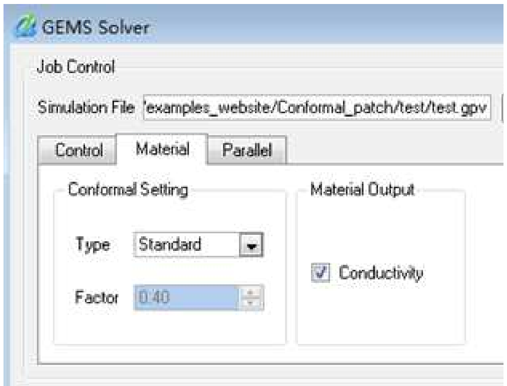


If any options in the list are in red color, you should check the reason.

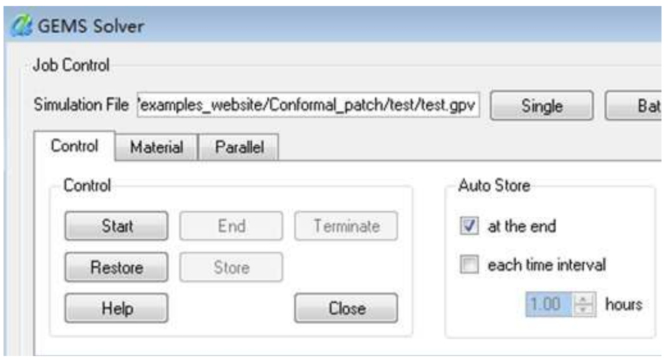
20.12 Simulate Project

Click the “Simulate” button in the **Simulation** window to open the GEMS solver window.

Click the “Simulate” button in the **Simulation** window to open the GEMS solver window. If you need to check the conductivity distribution inside the domain and see if the structure is damaged for the given mesh distribution, check the “Material->Conductivity” box and then the conductivity will be generated at the beginning of the simulation. If it is not correct, you should stop the simulation and redesign mesh or check if the model is ill-conditioned.



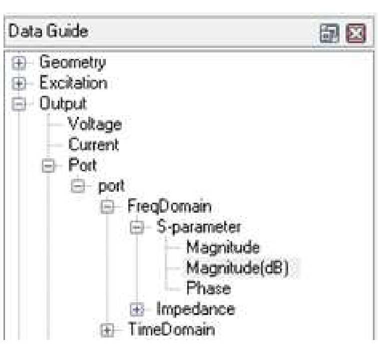
Checking the **at end** box requires GEMS to save the intermediate result for the resuming simulation later if it is necessary.

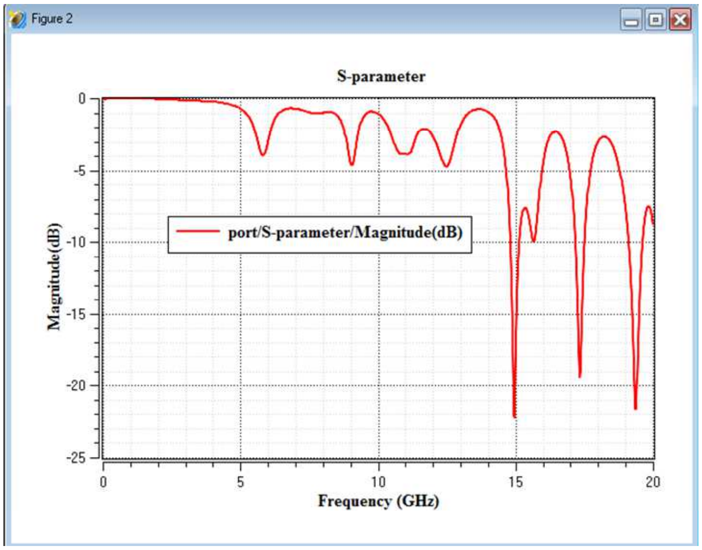


20.13 Result Visualization

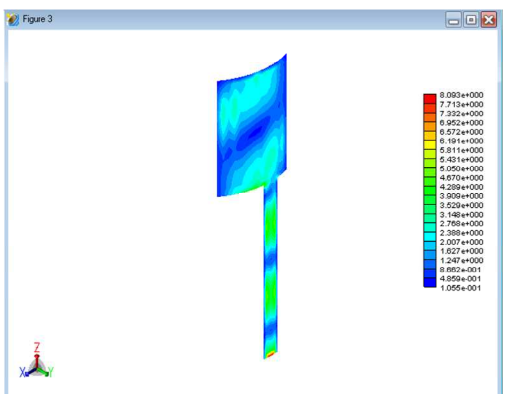
During the simulation, click the "Data View" button in the **GEMS Solver** window to open the **GEMS display** window and generate the results in frequency domain. When the **GEMS display** window is open, selecting the **Refresh** option in the File menu will generate the new frequency domain results.

Select and double-click the "Output->Port->FreqDomain->S-Parameter->Magnitude(dB)" option.

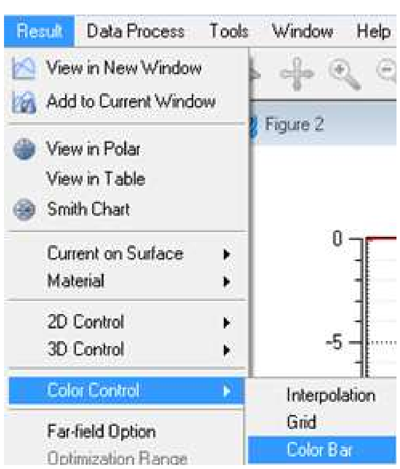


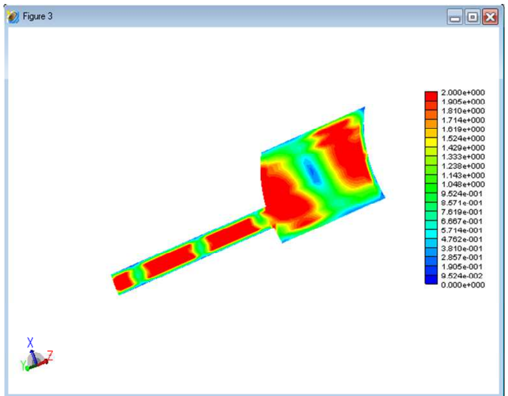


Select and double-click the "Output->Near FIeld->Patch_curve->CurrentOnSurface->11GHz" option.

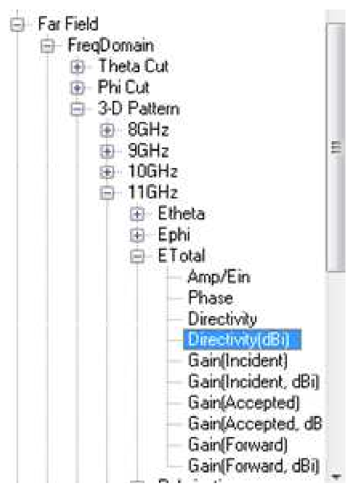


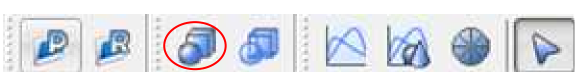
Select the "Result->Color Control->Color bar" option to adjust the color display range.

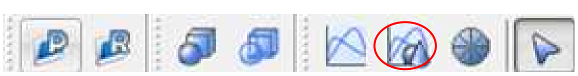


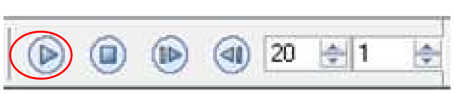


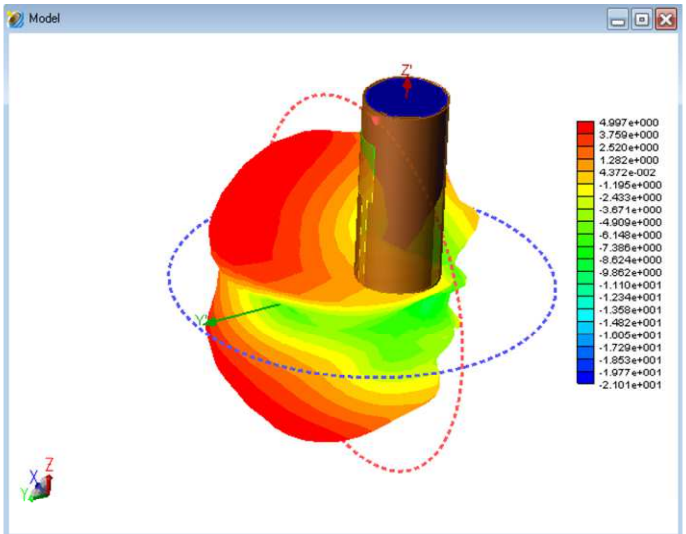
Click the Project model button in the toolbar. Select the 3-D directivity option in the result tree and click the add to current window button in the tool bar to show the 3-D directivity pattern with the project model in the same figure.











Model
Z'
4.997e+000
3.759e+000
2.520e+000
1.282e+000
4.372e-002
-1.195e+000
-2.433e+000
-3.671e+000
-4.909e+000
-6.148e+000
-7.386e+000
-8.624e+000
-9.862e+000
-1.110e+001
-1.234e+001
-1.358e+001
-1.482e+001
-1.605e+001
-1.729e+001
-1.853e+001
-1.977e+001
-2.101e+001
X
Y
Z

21

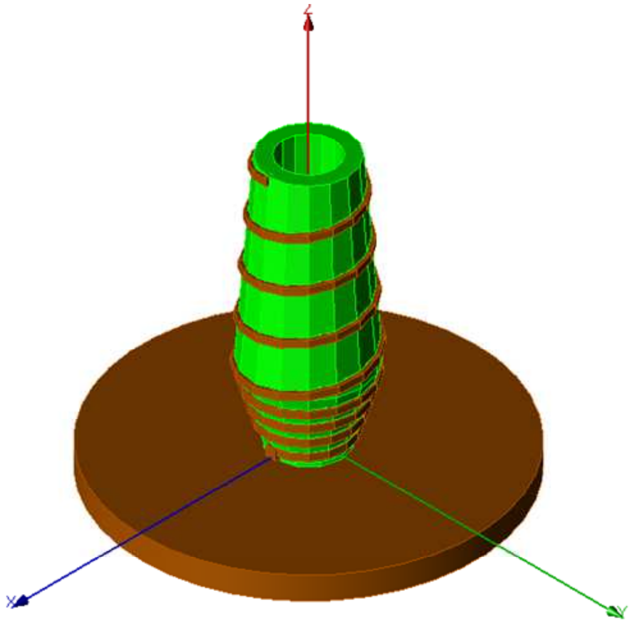
Example 21. Helix Antenna

Description: A helix with the dielectric support is mounted on the PEC base. The excitation feed is located between the base and helix. The helix model is in the SAT format, which can imported into GEMS interface.

Keywords: Return loss, surface current, and 3-D far field pattern.

21.1 Problem Configuration

The helix antenna is shown in the figure below. The size of helix cross section is 3mm. The dielectric constant and conductivity of the dielectric support are 3.15 and 0.007969 s/m, respectively.



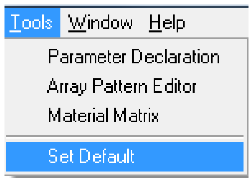
21.2 Create Project Model

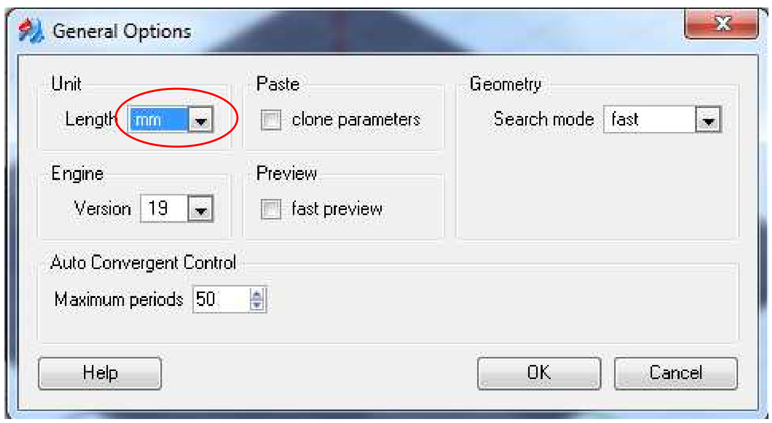
Following the steps below to draw a PEC base with the radius = 15mm and thickness = 2mm:

(1) Open the GEMS designer

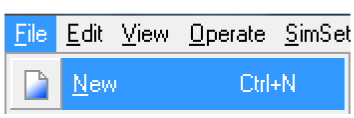
(2) Specify the project unit, which cannot be changed during the project modeling though you can input a variable in any units; however, the default unit can be only specified once at the beginning.

Select the **Tools->Set Default** option, and then select “mm” in the **Unit->Length** box as the project unit, click on the **OK** button to close the window.





(3) Click on the **New** button in the toolbar or select the **New** option in the **File** menu.

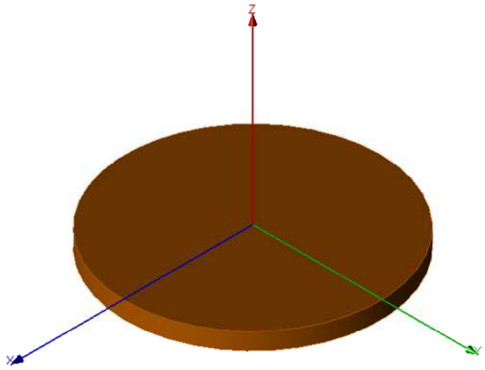


(4) Click on the **New** button in the toolbar or select the **New** option in the **File** menu.

(7) Click on the **Cylinder** icon in the “Tool Box->Geometry” box.

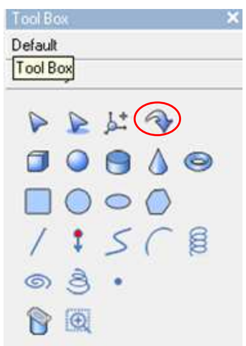
(8) Move the mouse icon to the origin point (0, 0, 0) and click the left mouse button, Move the mouse icon to draw a circle in the X-Y plane, and click the left nouse button at a proper radius. Move the mouse icon in the vertical direction to draw a cylinder and click the left mouse button at the proper height.

(9) Select the cylinder in the **Browser Object** box and change its name and color, and adjust the dimensions to the desired values in the **Properties** box.

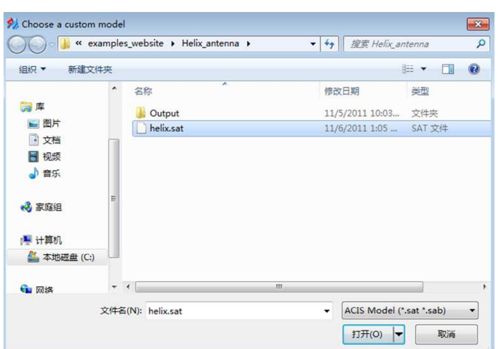


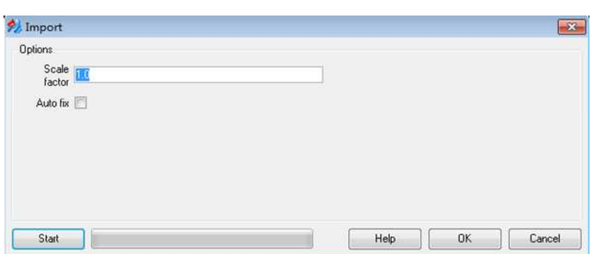
Following the steps below to import the helix into GEMS:

(1) Click the “Import Model” icon in the “Tool Box->Geometry” box;

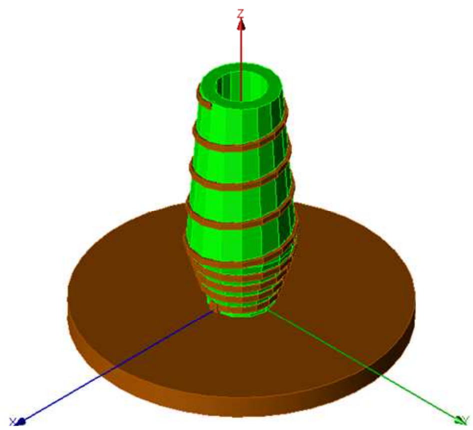


(2) Search for “helix.sat” file;



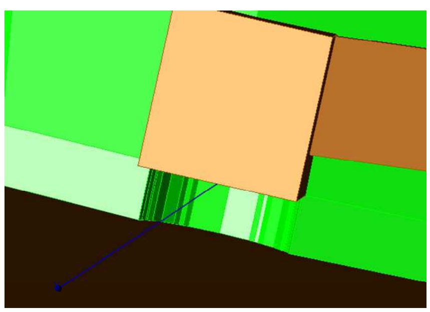


Click the **Start** button to start to import the helix model. Select the individual piece in the **Object Browser** box, change the color for each part, and assign the dielectric (relative constant = 3.15 and conductivity = 0.007969 s/m) to the dielectric support, and PEC to helix structure.

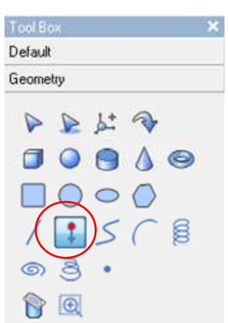


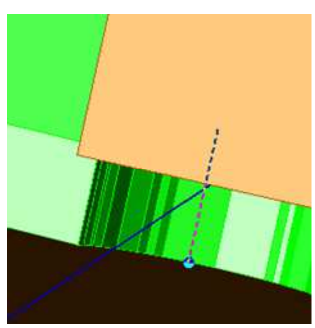
20.3 Set Excitation Port

The feed structure is shown in the figure below. The lumped port is from the base to the helix.

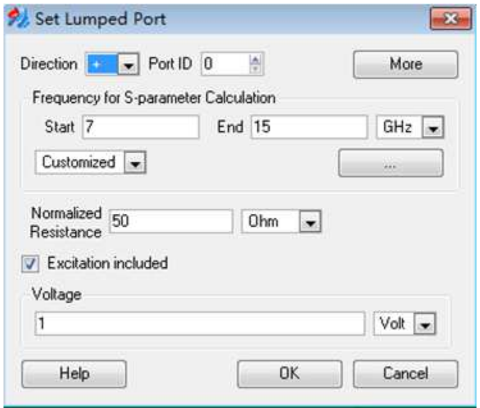


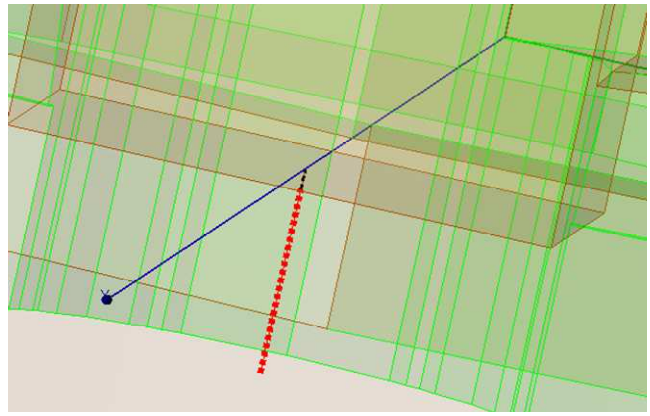
Click the “Plumb line” icon in the “Tool Box->Geometry” box, and snap the enter point of the bottom surface of the helix structure. The plumb line will stop on the surface of the PEC base.





Select the plumb line in the **Object Browser** box, and click the Lumped port icon in the “Tool Box->Simulation accessory” box.



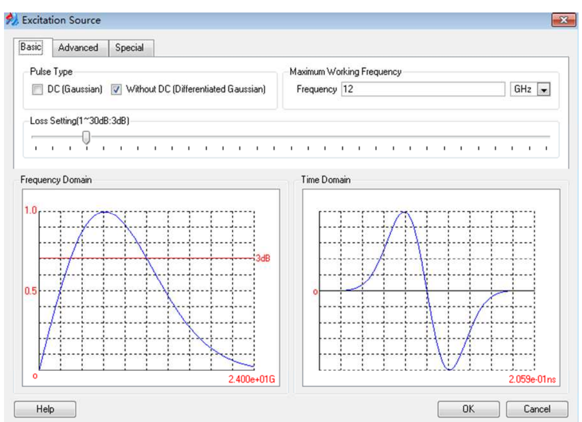


20.4 Set Excitation Pulse

Follow the steps below to set the excitation pulse:

(1) Click the “Set excitation source” button in the toolbar;

(2) Check the “Without DC” option and input “12” in the “Maximum working frequency” box. The frequency unit is GHz. The excitation pulse and its frequency spectrum are displayed in the window below.

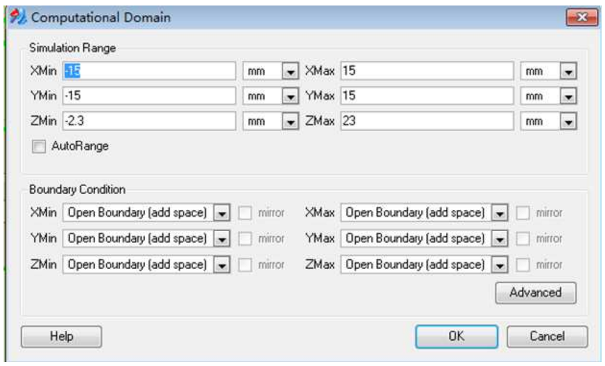


20.5 Set Boundary and Domain Size

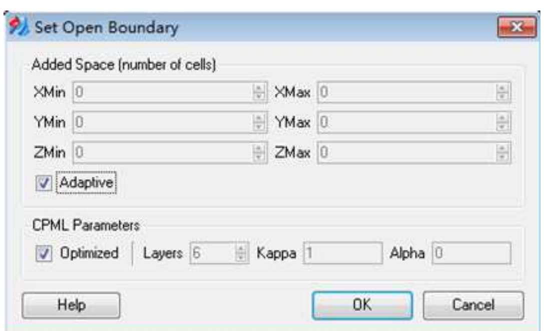
Follow the steps below to set the domain size and boundary condition:

(1) Click the “Set boundary condition” button in the toolbar;

(2) Set the domain and boundary parameters in the popup window. Checking the **AutoRange** box will allow GEMS to find the minimum box that encloses all the objects. For the open space problem, the “Open Boundary (add space)” is used to truncate all the six walls.



The window including the advanced options of boundary and domain is launched by clicking the **Advanced** button. Check the **Adaptive** box, and GEMS will add proper white space between the objects and domain boundary. Check the **Optimization** box, and GEMS will select the proper boundary parameters in the absorbing boundary condition.



(3) Click the **OK** button to confirm the boundary and domain settings.

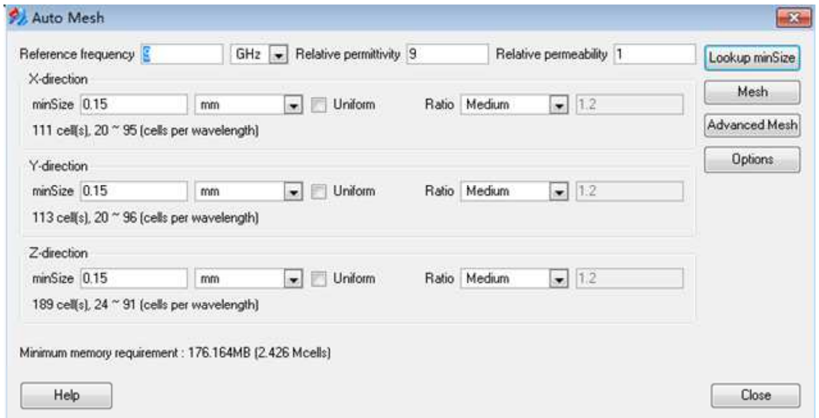
20.7 Design Mesh Distribution

Follow the steps below to design the mesh distribution:

(1) Click the “Auto mesh” button in the toolbar;

(2) Find the fine structure of interest.

The fine structure is thickness of the dielectric base and height of the feed gap, which is equal to 0.3mm. The minimum cell size should be less than 0.15mm.



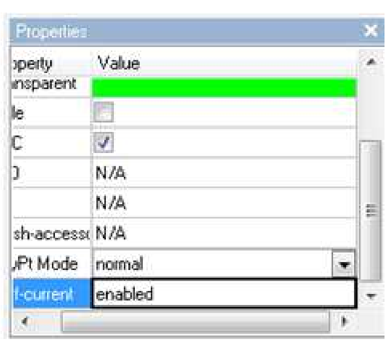
20.8 Set Output Parameters

The output parameters in this project are return loss, surface current on the conductors, and far field patterns. The return loss is calculated automatically when the lumped port is added.

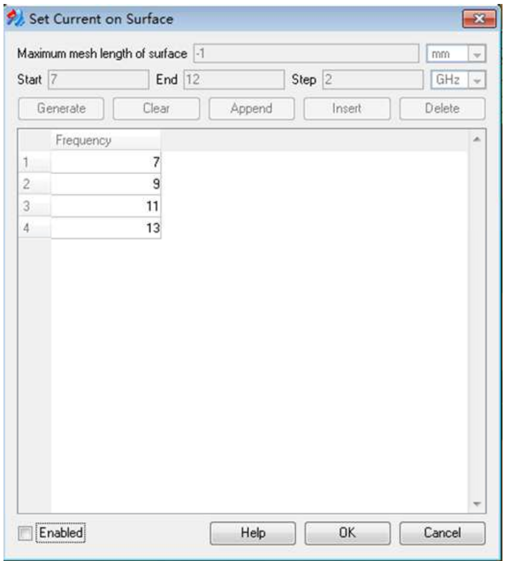
Follow the steps below to set the surface current distribution:

(4) Select the “helix” , “GND” and “PEC” options in the **Object Browser** box;

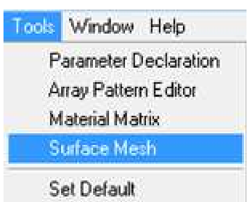
(5) Click the “Sur_current” option in the **Properties** box;

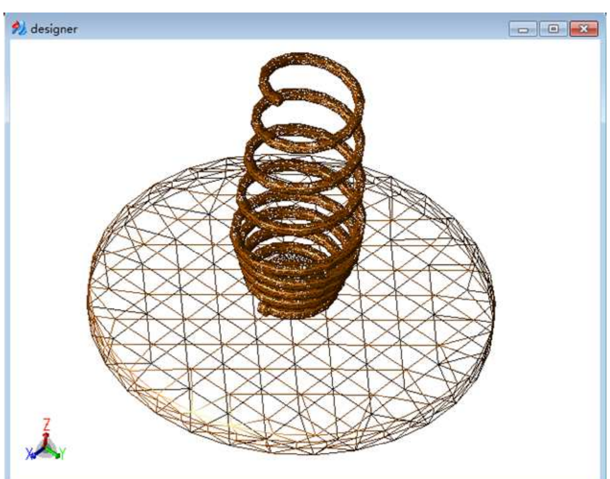


Specify the frequency list of interest, and the minimum length of the surface mesh in the popup window.



(6) View mesh by selecting the "Tools->Surface Mesh" option;

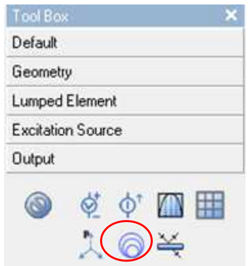




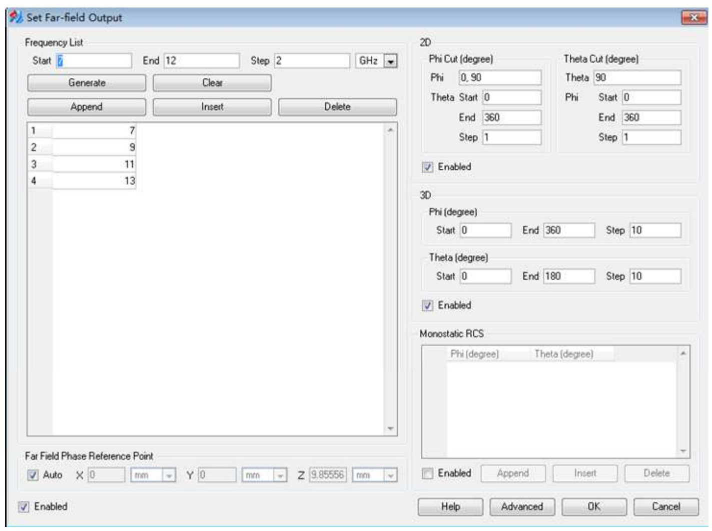
If the mesh is not sufficient for the description of the patch structure, you can adjust the number inside the "Maximum mesh length of surface" box in the **Set Current On Surface** window.

Follow the steps below to set the far field outputs:

(3) Select the “Far-field” icon in the “Tool Box->Output” box;



(4) Specify the frequency list and output options;

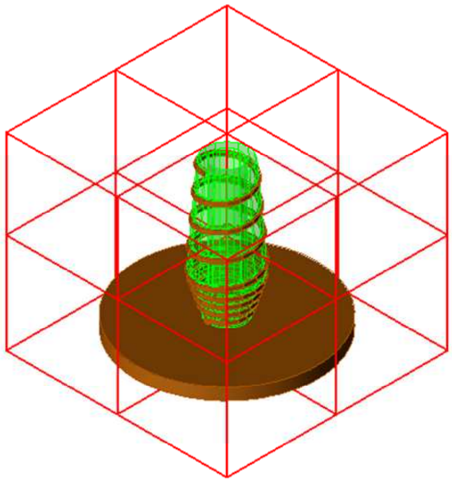


20.8 Parallel Processing Setting

If we have a workstation including 8 CPUs, we can split the domain into 8 sub-domains. Follow the steps below to set the parallel processing.

(1) Click the “Set parallel info” in the toolbar;

(2) First split the domain size that has more number of cells into 2. Then split the domain size that has more number of cells into 2 again. Then split the domain size that has more number of cells into 2 again.

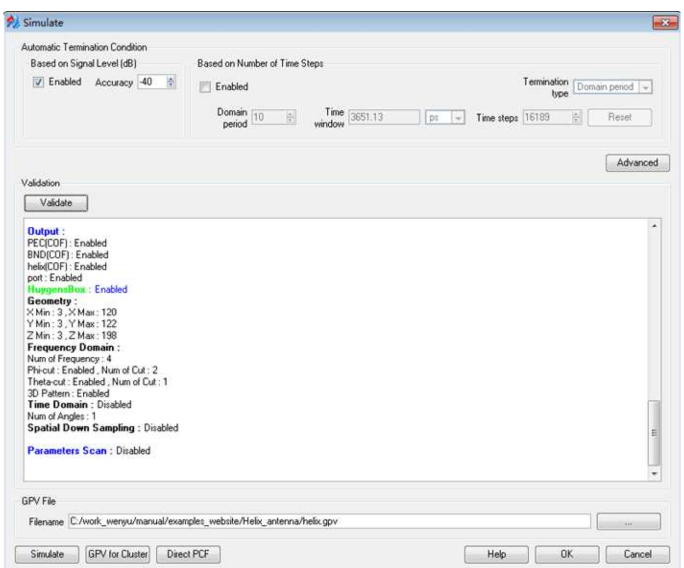


20.9 Save Project

Click the “Save” button in the toolbar to save the project.

20.10 Validate Project Settings

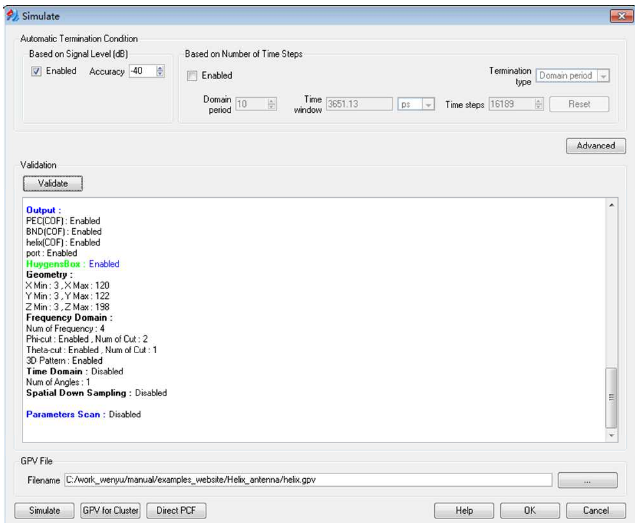
Click the “PreCalculate” button in the toolbar to validate the project settings.



If any options in the list are in red color, you should check the reason.

20.11 Simulate Project

Click the “Simulate” button in the **Simulation** window to open the GEMS solver window.

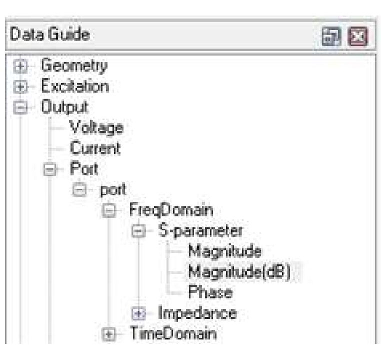


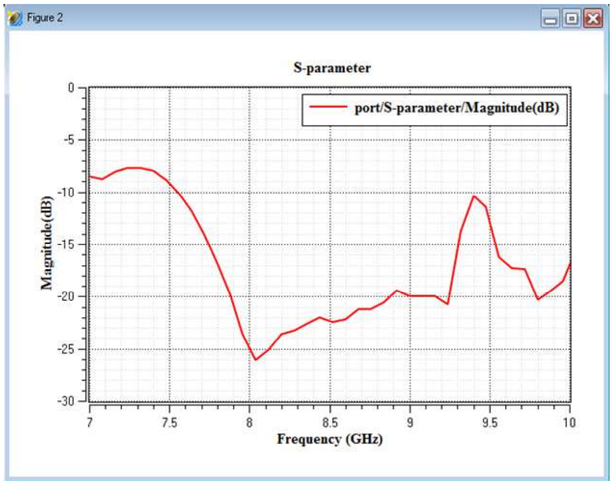
Click the “Simulate” button in the **Simulation** window to open the GEMS solver window. If you need to check the conductivity distribution inside the domain and see if the structure is damaged for the given mesh distribution, check the “Material->Conductivity” box and then the conductivity will be generated at the beginning of the simulation. If it is not correct, you should stop the simulation and redesign mesh or check if the model is ill-conditioned.

20.13 Result Visualization

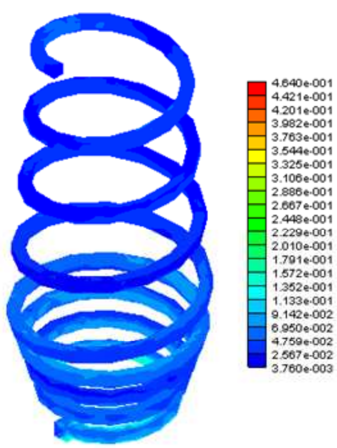
During the simulation, click the “Data View” button in the **GEMS Solver** window to open the **GEMS display** window and generate the results in frequency domain. When the **GEMS display** window is open, selecting the **Refresh** option in the File menu will generate the new frequency domain results.

Select and double-click the “Output->Port->FreqDomain->S-Parameter->Magnitude(dB)” option.

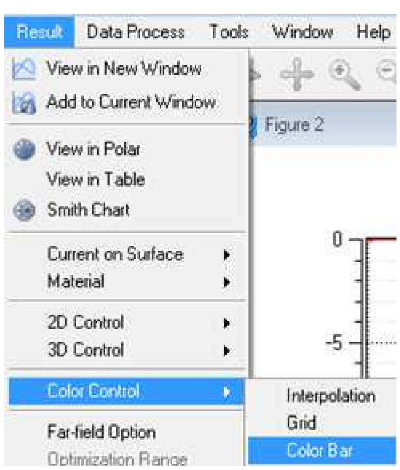


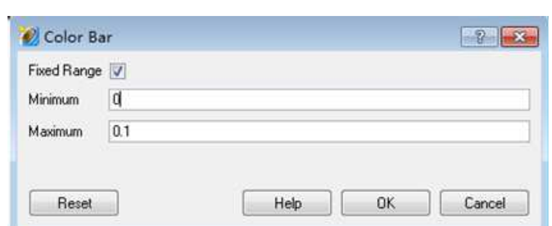


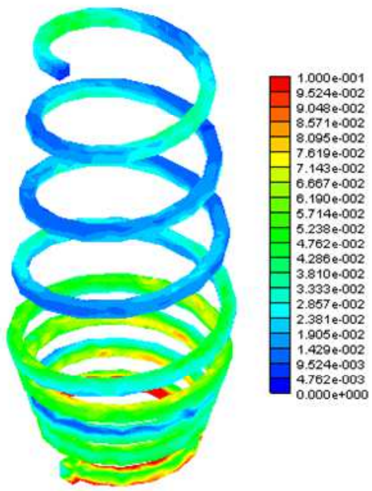
Select and double-click the “Output->Near FIeld->Patch_curve->CurrentOnSurface->9GHz” option.



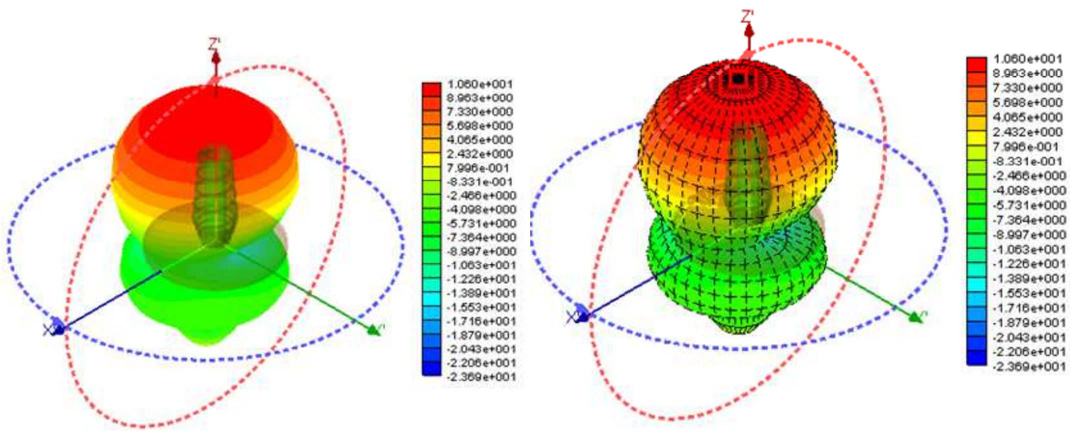
Select the “Result->Color Control->Color bar” option to adjust the color display range.







Click the “View model” button in the toolbar. Select the 3-D directivity option in the result tree and click the “add to current window” button in the tool bar to show the 3-D directivity pattern with the project model in the same figure. Select the “Result->Color Control->Grid” option to add the grid in the 3-D directivity pattern.



22

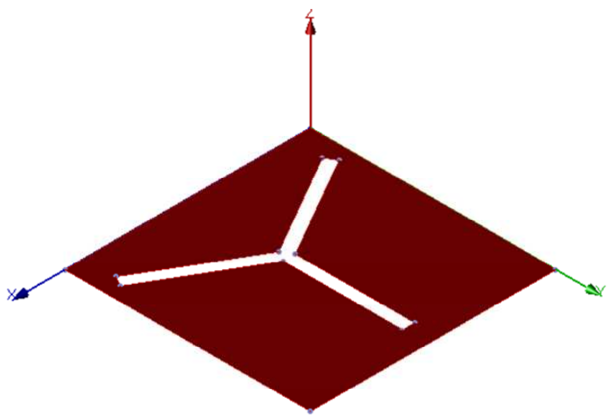
Example 22. FSS (Frequency Selective Surface)

Description: A triple slot forms a frequency selective surface screen. The output parameters are the reflection and transmission coefficients.

Keywords: Reflection and transmission coefficients, FSS, and plane wave.

22.1 Problem Configuration

The FSS screen is shown in the figure below. The width and length of triple arm are 0.5mm and 5.0mm, respectively.



The periodic element is shown in the figure below:

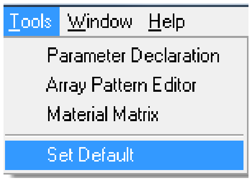
22.2 Create Project Model

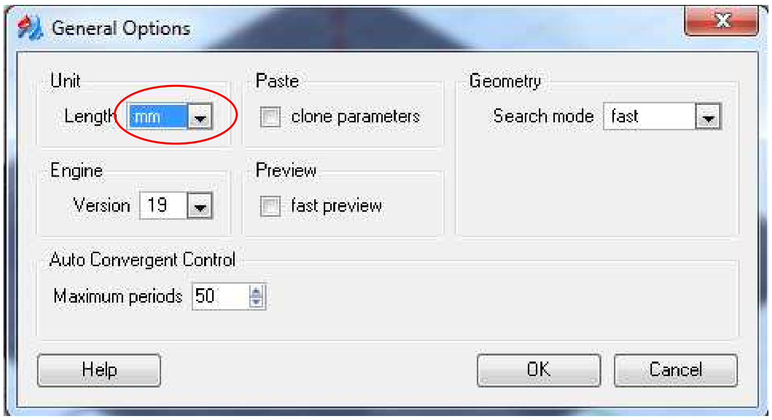
Follow the steps below to create FSS screen model:

(5) Open the GEMS designer

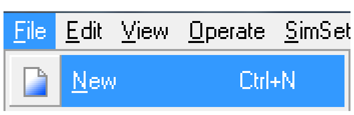
(6) Specify the project unit, which cannot be changed during the project modeling though you can input a variable in any units; however, the default unit can be only specified once at the beginning.

Select the **Tools->Set Default** option, and then select "mm" in the **Unit->Length** box as the project unit, click on the **OK** button to close the window.



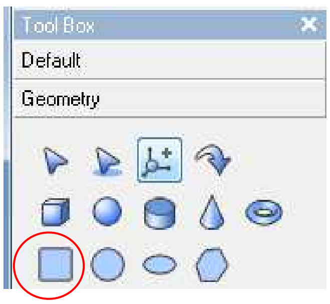


(7) Click on the **New** button in the toolbar or select the **New** option in the **File** menu.



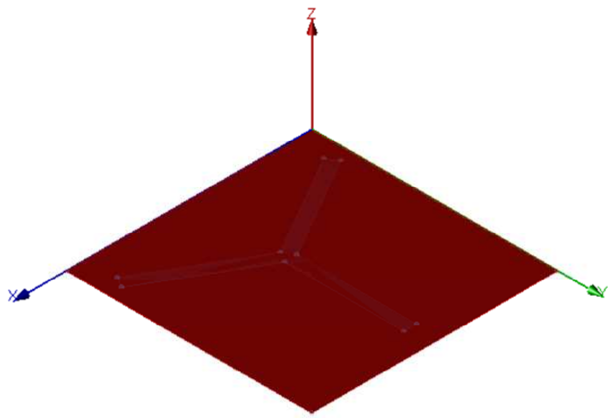
(8) Draw FSS Screen

Click on the *Rectangle* icon in the **Tool Box->Geometry** box.

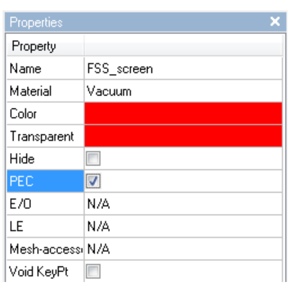


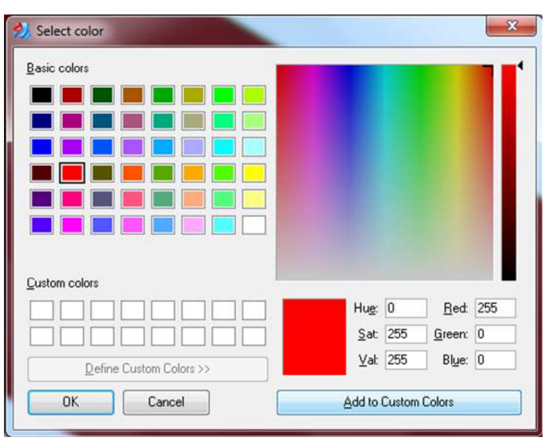
Select the drawing plane to be the "X-Y" plane and z=0 in the toolbar.

Move the mouse icon to the figure region, select the position (0, 0, 0) and press the left mouse button. Move the mouse icon to the position (10, 17.4) and press the left mouse button. Or draw a box with an arbitrary width and length, and then modify its dimensions in the **Property** box.

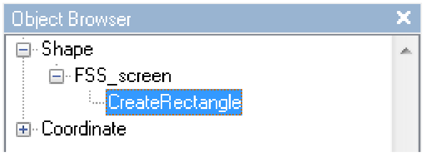


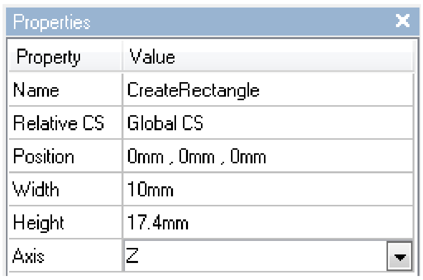
Select the rectangle option in the **Object Browser** box and change its name to “FSS_screen” and check the **PEC** box in the **Property** box. Click on the **Color** box and change its color to red in the color panel.





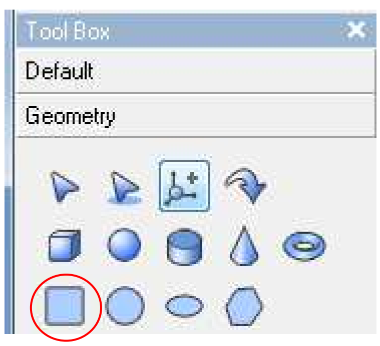
Select the "CreateRectangle" option in the "FSS_screen" folder in the **Object Browser** box and change the width and length to 10mm and 10mm, respectively, in the **Property** box.



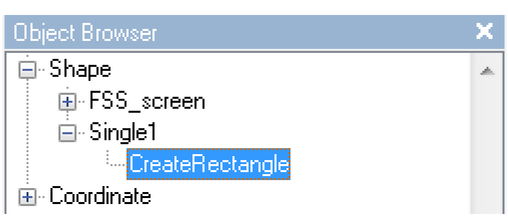


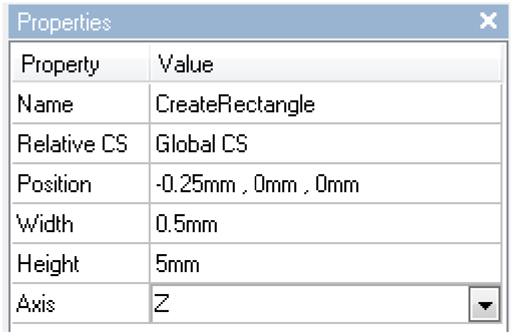
(9) Draw a triple

Click on the *Rectangle* icon in the **Tool Box->Geometry** box.

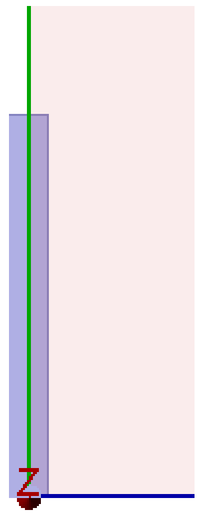


Follow the same procedure to draw a box at (-0.25mm, 0, 0), width=0.5mm, length=5mm.





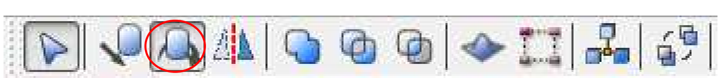
Property	Value
Name	CreateRectangle
Relative CS	Global CS
Position	-0.25mm , 0mm , 0mm
Width	0.5mm
Height	5mm
Axis	Z



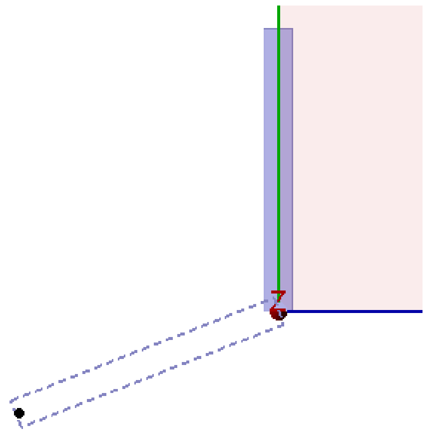
Select the “Single1” in the **Object Browser** box and press and hold the keys “Ctrl + c” and then “Ctrl + v” to duplicate a rectangle.

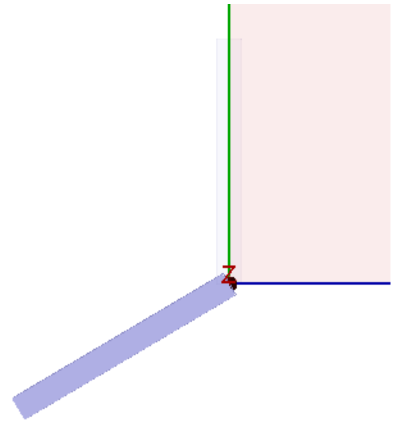
Click on the “XY” plane button in the toolbar.

Select the duplicated box “Single1_0” in the **Object Browser** box and click on the **Rotate** button in the toolbar.

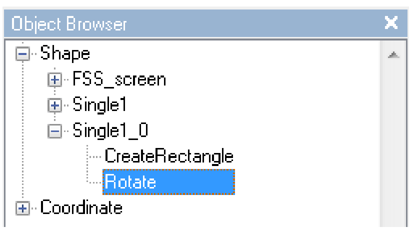


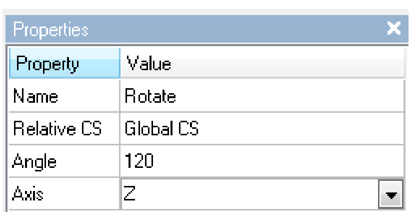
Select one point in the “Single1_0” object and move the mouse icon to another point around the z-axis, and press the left mouse button.



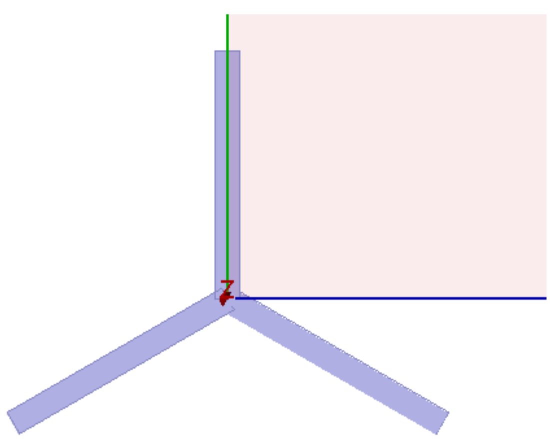


Select the **Rotate** option in the “Single1_0” folder in the **Object Browser** box. Change the angel to “120” degree in the **Properties** box.

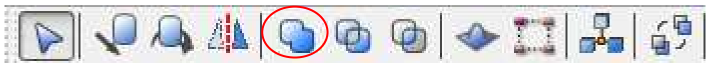


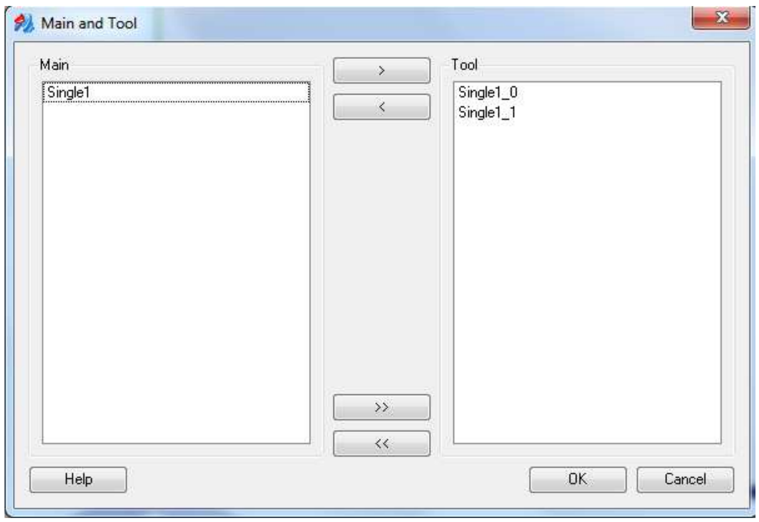


Follow the same procedure to generate the third arm of the triple.



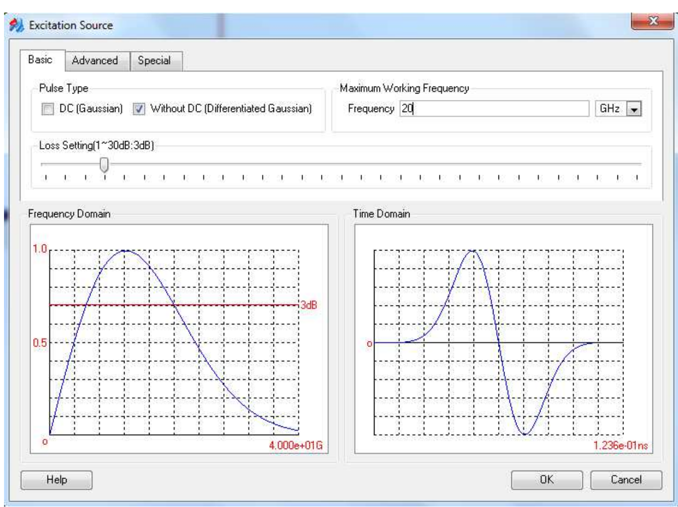
Select the three arms in the **Object Browser** box, and click on the **Unite** button in the toolbar to unite them as a single object.





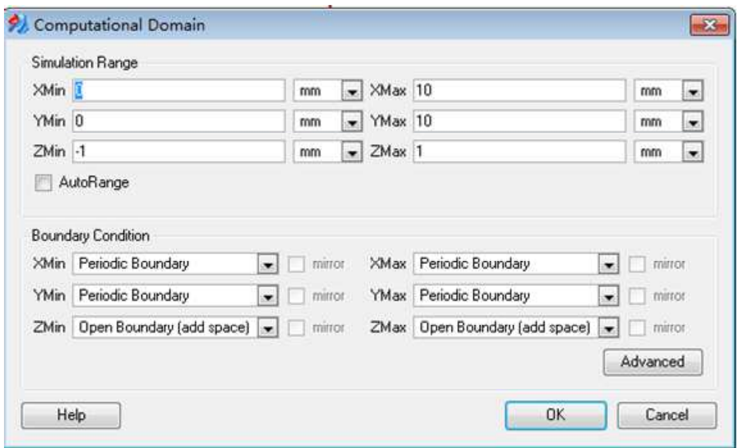
22.3 Set Excitation Pulse

Click on the **Excitation source** button in the toolbar. Check the "Without DC" option in the **Basic** tab. Since the maximum frequency of interest is 20GHz, so we type "20" in the **Frequency** box (unit is GHz). Click on the **OK** button to confirm the excitation pulse setting.



22.4 Set Domain size and Boundary Condition

Click on the **Set boundary condition** button in the toolbar.

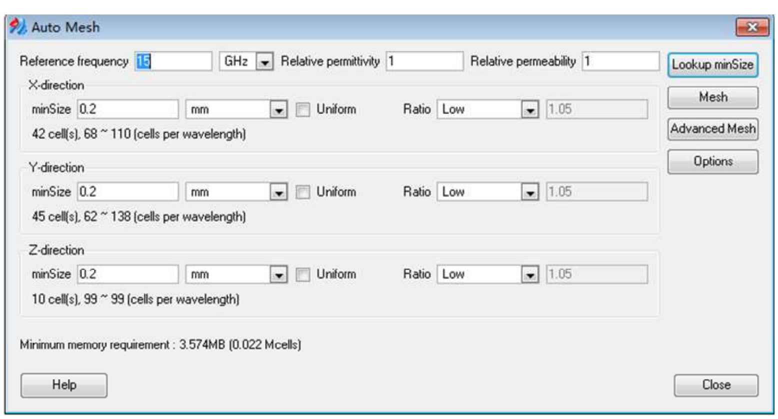


The boundary in the X-Y plane is "Periodic Boundary" and the "Open Boundary (add space)" is used in the z-direction. Add a small dimension in the vertical direction, for example, 2mm from -1mm to 1mm.

Click on the **OK** button to confirm the boundary and domain setting.

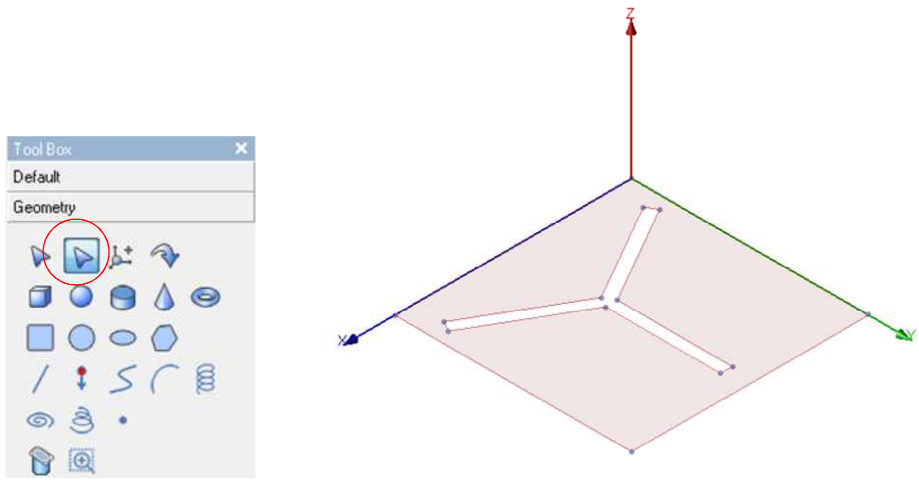
22.5 Design Mesh Distribution

Click on the **Auto Mesh** button in the toolbar.



The width of triple arm is 0.5mm, so we select the minimum cell size to be 0.2mm in the horizontal directions. There is no thickness of the FSS screen, we select the minimum cell size to be 0.2mm too.

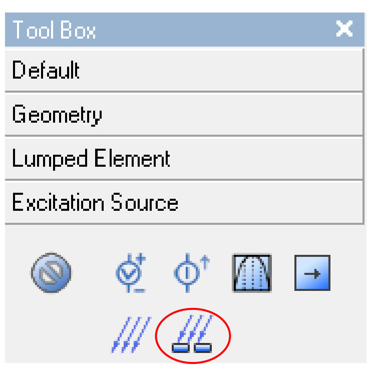
For the slanted slots, the cell size should be sufficiently small to describe the field variation inside the slots. We can add some key points at the corners of the slots. For example, click the "Select face" icon in the "Tool Box->Geometry" box, and then select the FSS screen surface. Press and hold the "Ctrl + L" and then "Ctrl + v" on the keyboard. The points are listed inside the "gkeypoint_0" folder inside the **Browser Object** box.



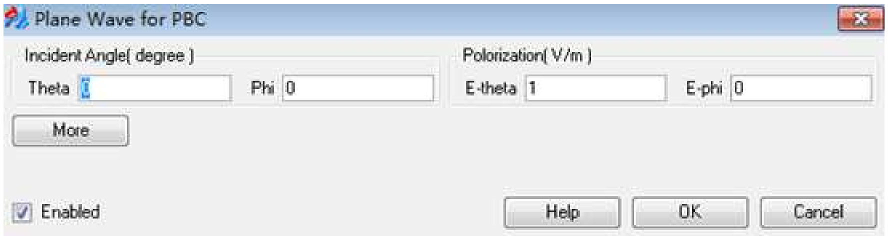
The grids will pass these key points in the mesh distribution.

22.6 Set Plane Wave Excitation

Click on the **PBC plane wave** button in the **Tool Box->Excitation Source** box.



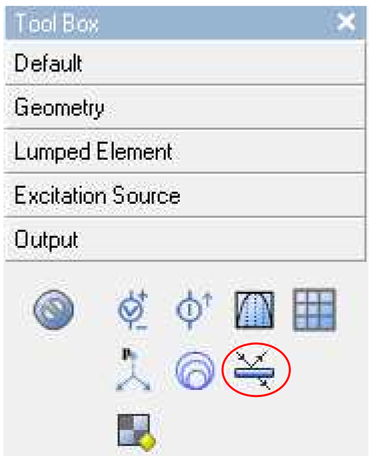
Check the "Enabled" box, specify the incident angle and polarization in the *Plane Wave for PBC* window. For normal incident case, the angle theta is equal to 0 degree.



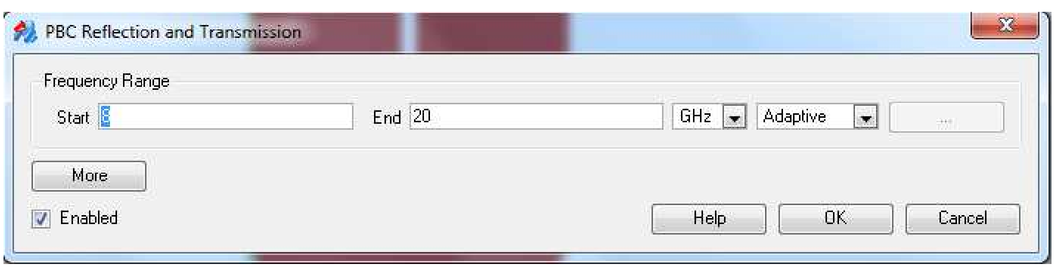
Click on the **OK** button to confirm the setting.

22.7 Set Output Parameters

Click on the **PBC reflection&transmission** icon in the **Tool Box->Output** box.

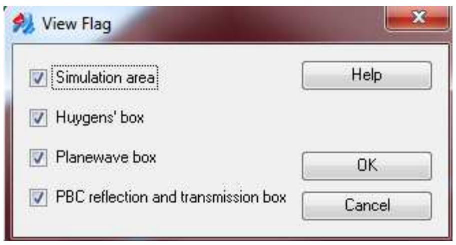


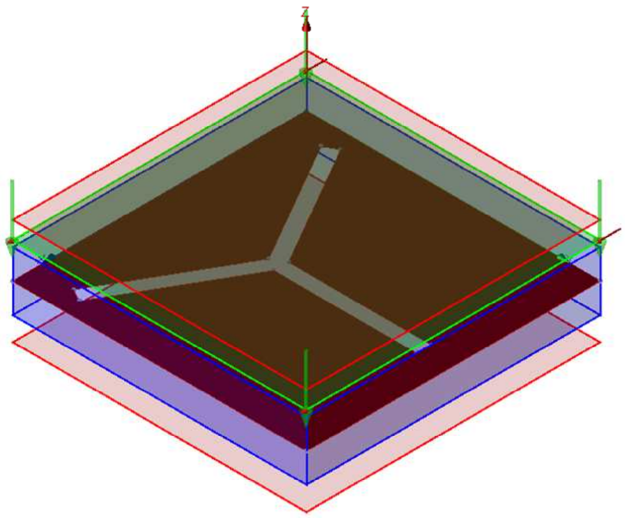
Check the **Enabled** box and specify the frequency range of interest. Click on the **OK** button to confirm the output setting.



Click on the **View defined box** button in the toolbar.

Check the four boxes in the *View Flag* window and then click on the **OK** button.





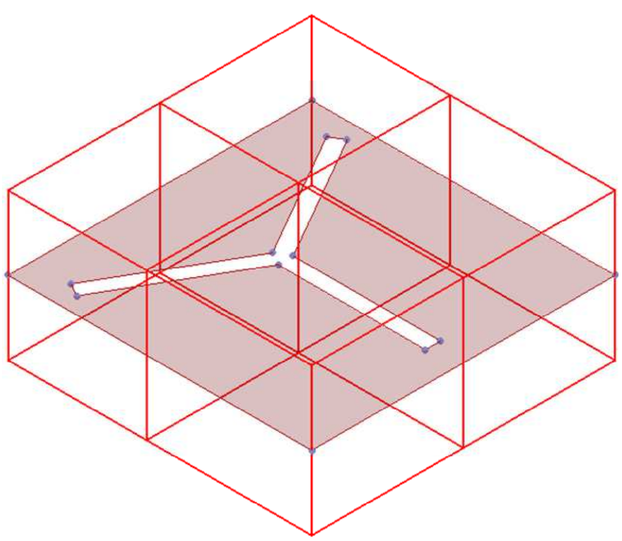
The two arrows indicate the propagation direction of the plane wave and polarization.

22.8 Parallel Processing Design

Click on the **Set parallel info** button in the toolbar to design the parallel division for parallel processing.

Now, we use two computers to simulate this problem. Each computer has 4 physical cores and 8 logic cores. So, we can divide the problem into 16 sub-problems.

For a flat geometry, the parallel division is in the x- and y-directions.



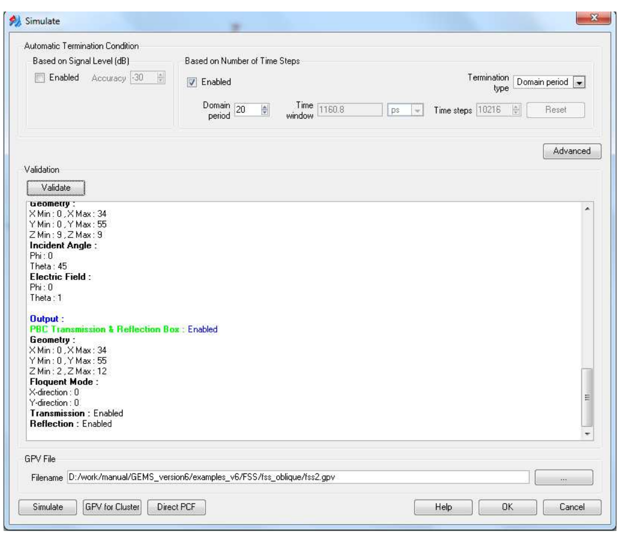
22.9 Save Project

Click on the **Save** button in the toolbar to save the project. The saved project has a default extension name “gpj”.

22.10 Generate Simulation File

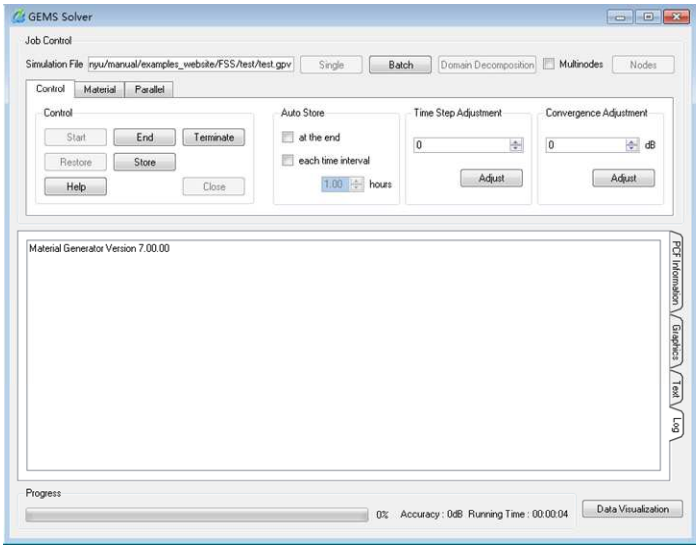
Click on the **Precalculate** button in the toolbar to generate a simulation file with an extension name “gpv”.

*If you know the number of domain periods, you can set the convergence criterion in the number of domain periods, for example, in this example, we know that 20 domain periods are sufficient. Therefore, we check the **Enabled** box in the “Based on number of time steps” box, and select “Domain period” in the “Termination type” list and then type a number “20” in the “Domain period” box.*



22.11 Simulate Project

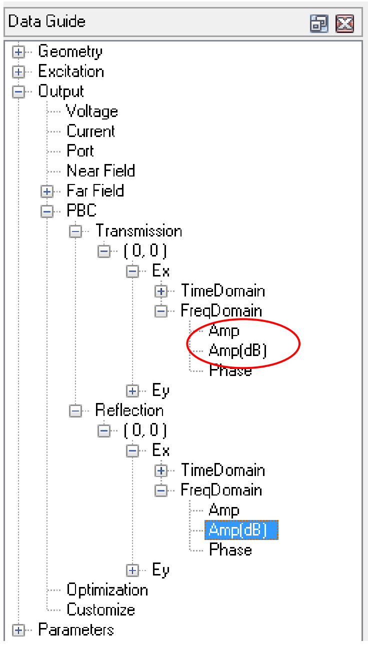
Click on the **Simulate** button in the *Simulate* window to open the simulation window. Click the “Start” button to start the simulation.



22.12 Result Visualization

Click on the **Data Visualization** button in the *GEMS Solver* window to open the GEMS display window.

Double-click on the “PBC->Transmission->(0, 0)->Ex->FreqDomain->Amp(dB)” option, the transmission coefficient will be plotted in the figure region.



Select the “PBC->Reflection->(0, 0)->Ex->FreqDomain->Amp(dB)” option, and then click on the “Add to current window” button in the toolbar to plot the reflection coefficient and transmission coefficient in the same figure.

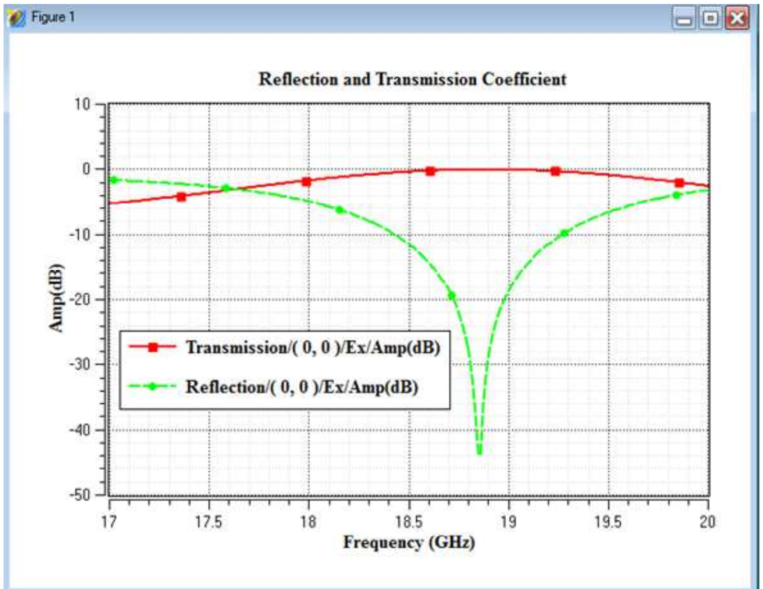
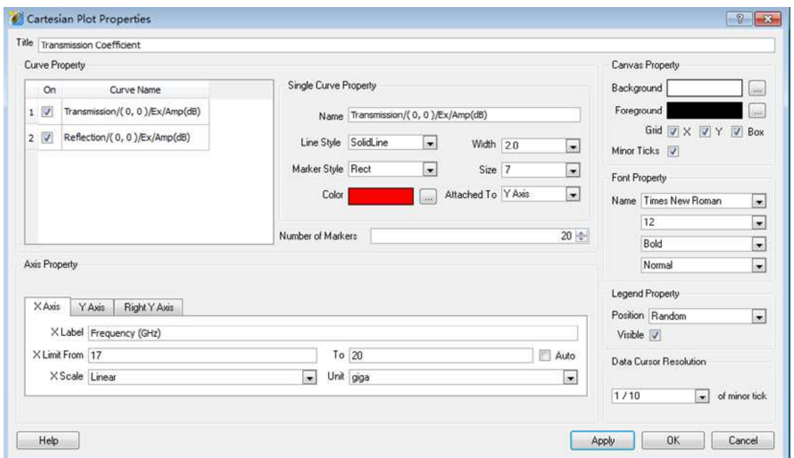


Figure 1
Reflection and Transmission Coefficient
Amp(dB)
Frequency (GHz)
Transmission/(0, 0)/Ex/Amp(dB)
Reflection/(0, 0)/Ex/Amp(dB)


Cartesian Plot Properties
Title Transmission Coefficient
Curve Property
Single Curve Property
Name Transmission/(0, 0)/Ex/Amp(dB)
Line Style SolidLine
Width 2.0
Marker Style Rect
Size 7
Color
Attached To Y Axis
Number of Markers 20
Canvas Property
Background
Foreground
Grid X Y Box
Minor Ticks
Font Property
Name Times New Roman
12
Bold
Normal
Legend Property
Position Random
Visible
Data Cursor Resolution
1 / 10 of minor tick
Axis Property
X Axis Y Axis Right Y Axis
X Label Frequency (GHz)
X Limit From 17 To 20 Auto
X Scale Linear Unit giga
Help Apply OK Cancel

23

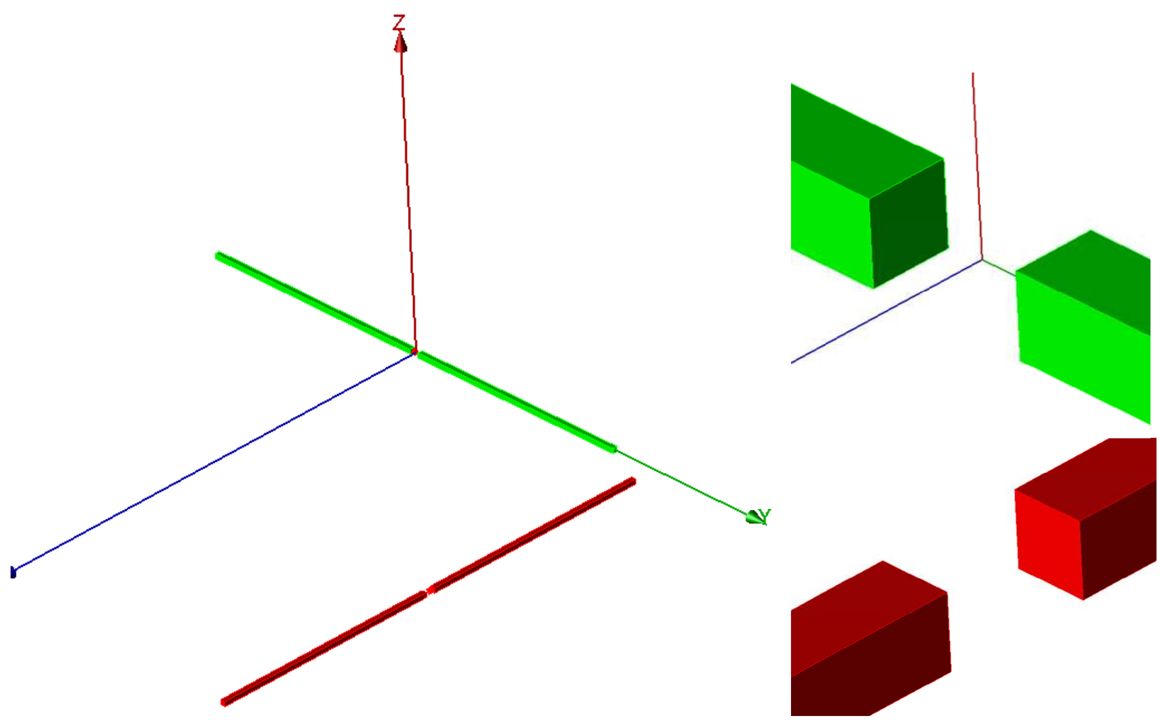
Example 23. Cross Dipole

Description: Two dipole antennas are set along the x- and y-directions, respectively. They have a distance of 50 mm in the z-direction, that is, one quarter wavelength for 1.5GHz. We calculate the far field pattern and axial ratio.

Keywords: dipole, far field pattern, and axial ratio.

23.1 Problem Configuration

The cross dipole configuration is shown in the figure below. The length of dipole is 100mm and its cross section is 1mm x 1mm.



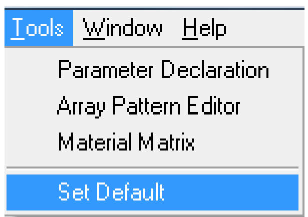
23.2 Create Project Model

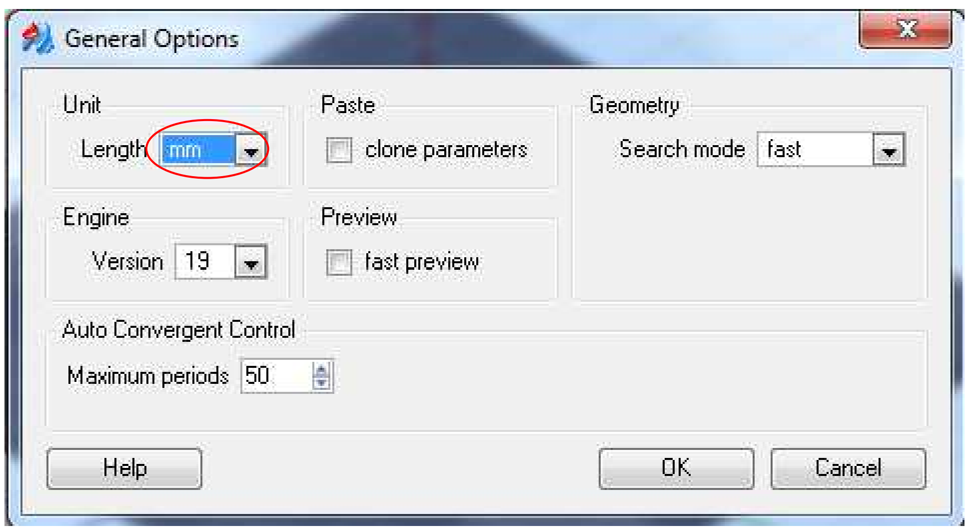
Follow the steps below to create cross dipole model:

(1) Open the GEMS designer

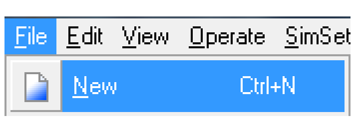
(2) Specify the project unit, which cannot be changed during the project modeling though you can input a variable in any units; however, the default unit can be only specified once at the beginning.

Select the **Tools->Set Default** option, and then select "mm" in the **Unit->Length** box as the project unit, click on the **OK** button to close the window.



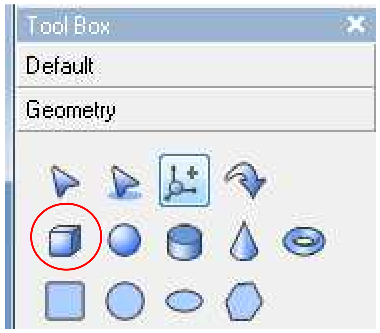


(3) Click on the **New** button in the toolbar or select the **New** option in the **File** menu.

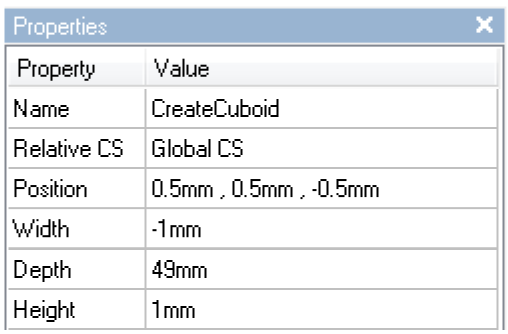


(4) Draw dipole arms

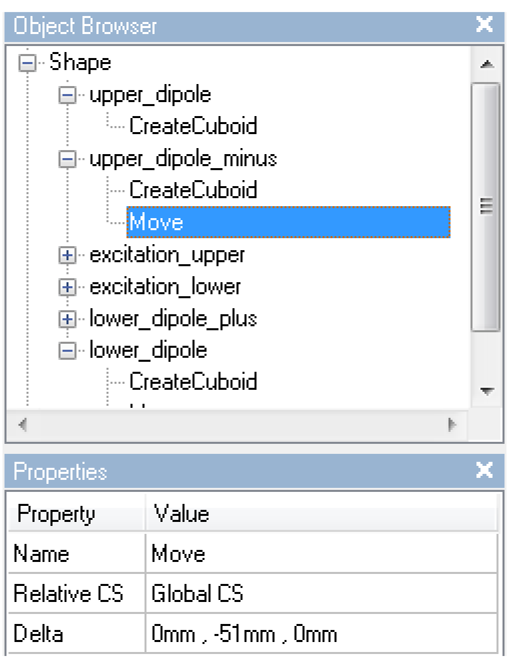
Click on the *Cuboid* icon in the **Tool Box->Geometry** box.



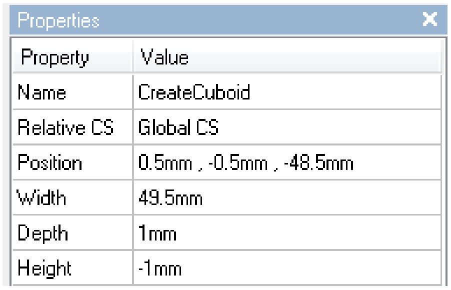
Draw the first arm (length = 49.5mm, cross section = 1 mm x 1mm). The corner coordinates are (0.5mm, 0.5mm, -0.5mm).



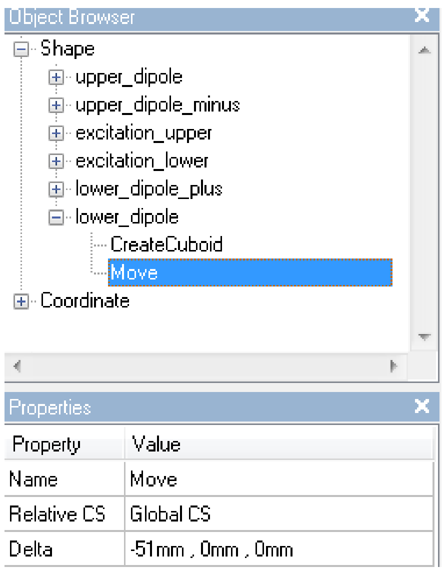
Duplicate the first arm and move it to (0mm, -51mm, 0mm).



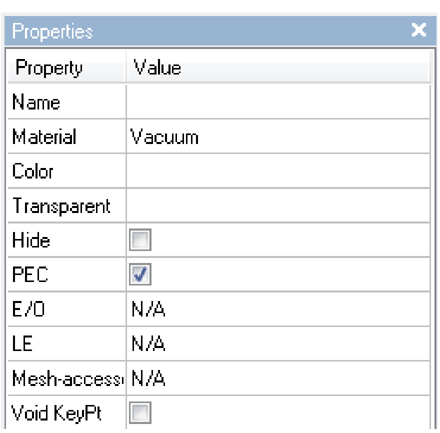
Draw the third arm (length = 49.5mm, cross section = 1 mm x 1mm). The corner coordinates are (0.5mm, -0.5mm, -48.5mm), that is,



Duplicate the third arm and move it to (0mm, -51mm, 0mm).

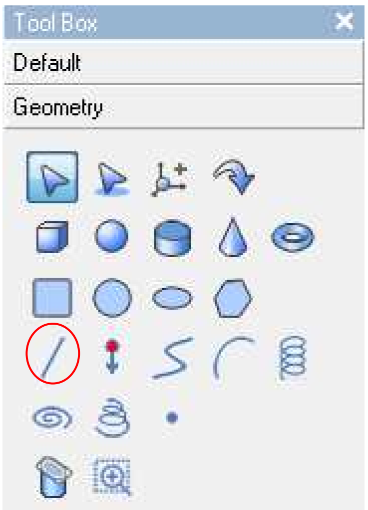


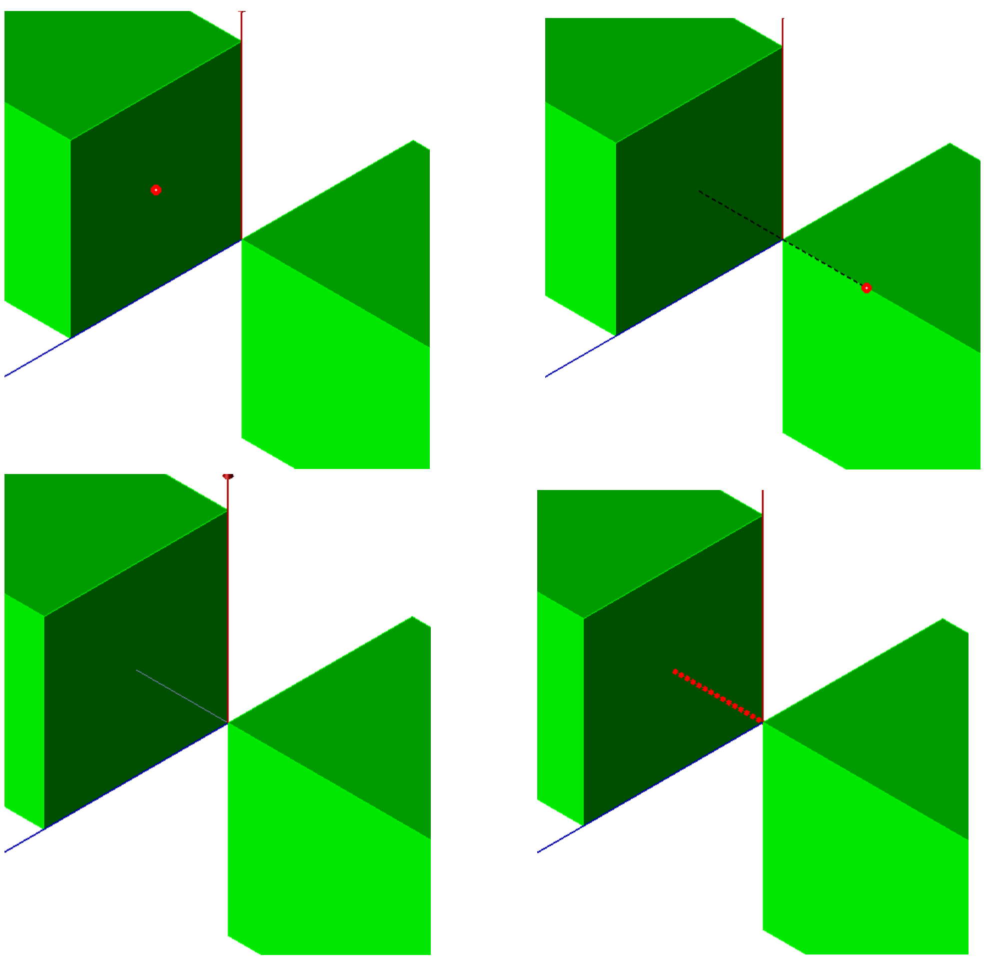
Select four arms in the **Object Browser** box and click on the **PEC** box in the **Properties** box.



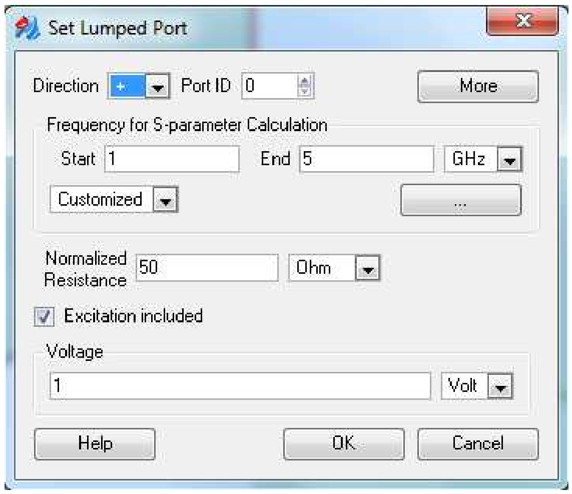
23.3 Set Excitation Port

Click on the **Line** icon in the **Tool Box->Geometry** box, and snap the mouse icon to the center of dipole arm, and then move the mouse icon to the center of another arm.





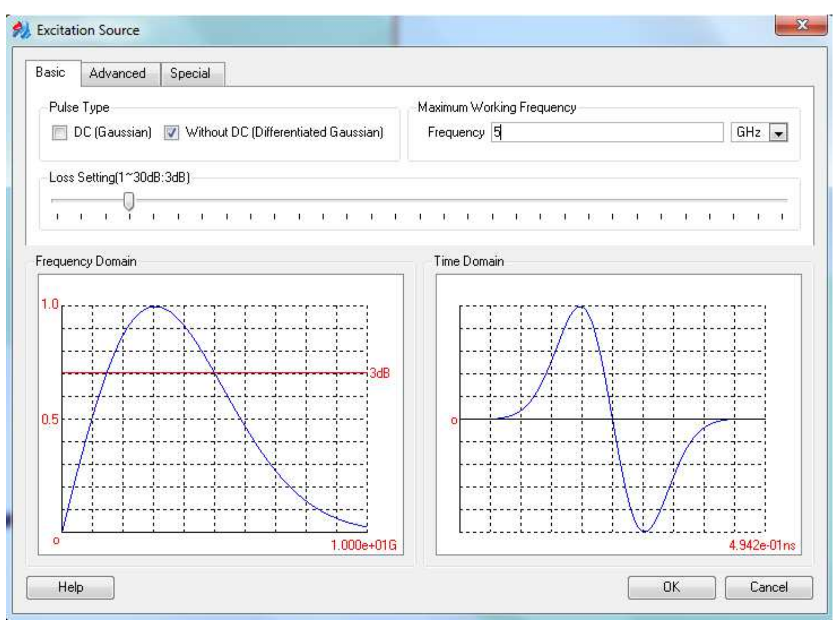
Select the line option in the **Object Browser** box and change its name to “excitation_upper” in the **Properties** box. Select on the “excitation_upper” option in the **Object Browser** box and click on the **Lumped port** icon in the **Tool Box->Simulation Accessory** box. Specify the frequency band of interest (1 to 5GHz), and click on the **OK** button to confirm the setting.



Following the similar procedure to draw an excitation in the feed gap of lower dipole.

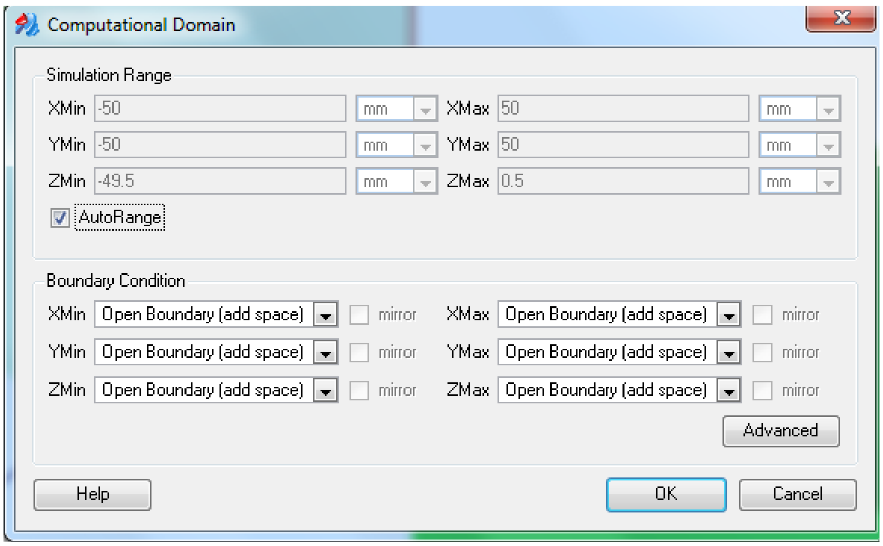
23.4 Set Excitation Pulse

Click on the **Set excitation** button in the toolbar.



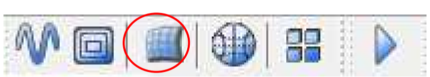
23.5 Set Domain and Boundary Condition

Click on the **Set boundary condition** button in the toolbar.

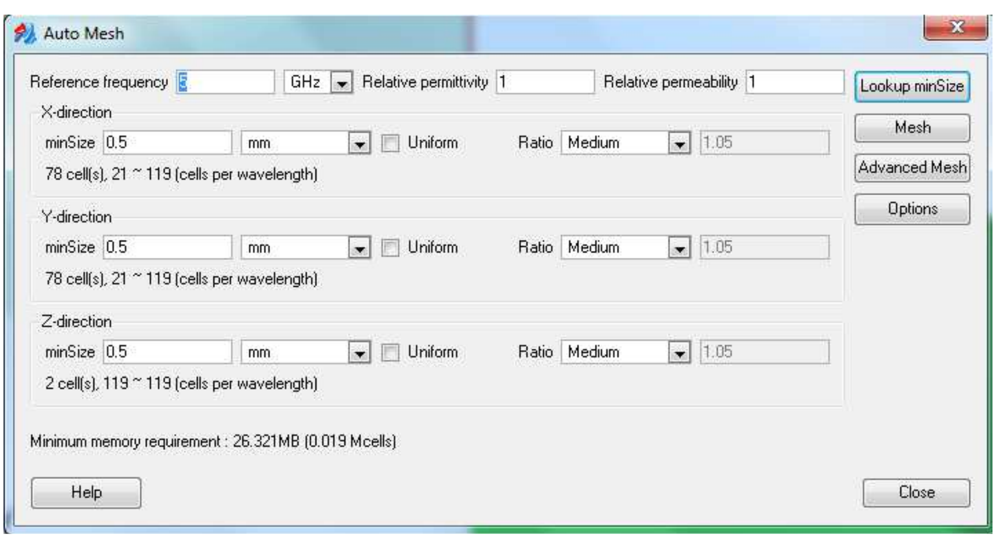


23.6 Design Mesh Distribution

Click on the **Auto mesh** button in the toolbar.



Click on the **Loopup miniSize** button and then on the **Mesh** button.

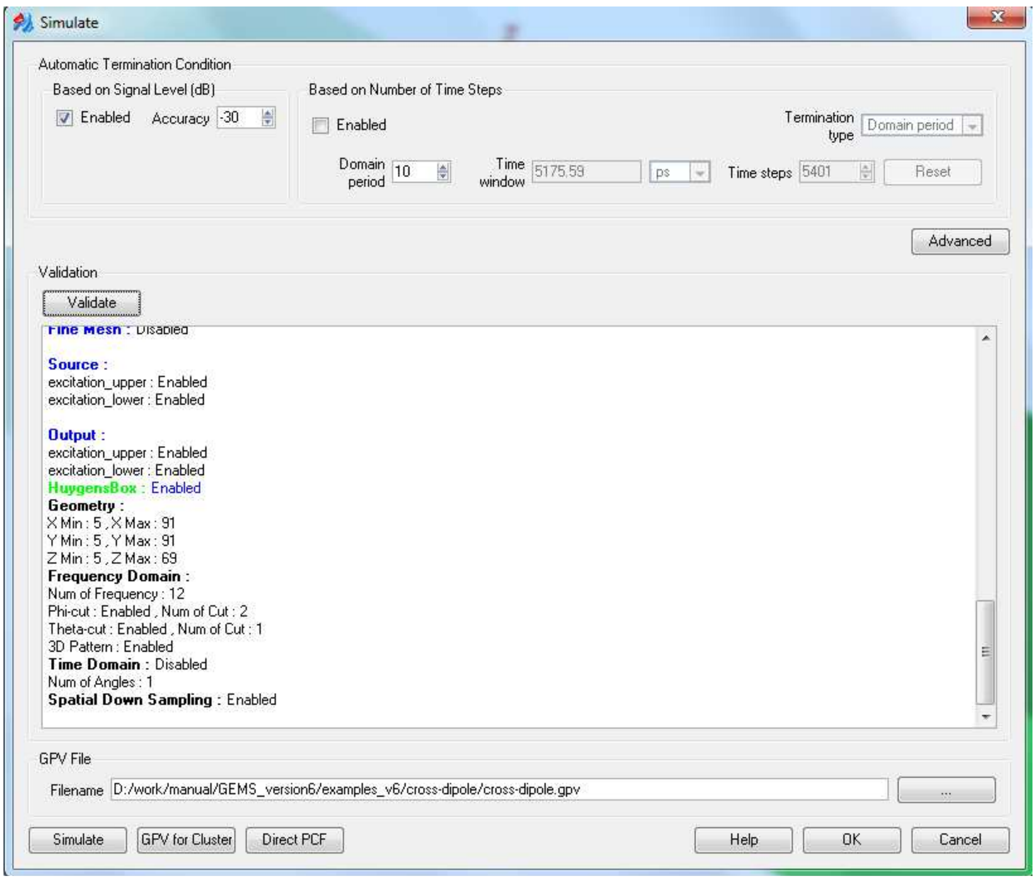


23.7 Save Project

Click on the **Save** button in the toolbar.

23.8 Generate Simulation File

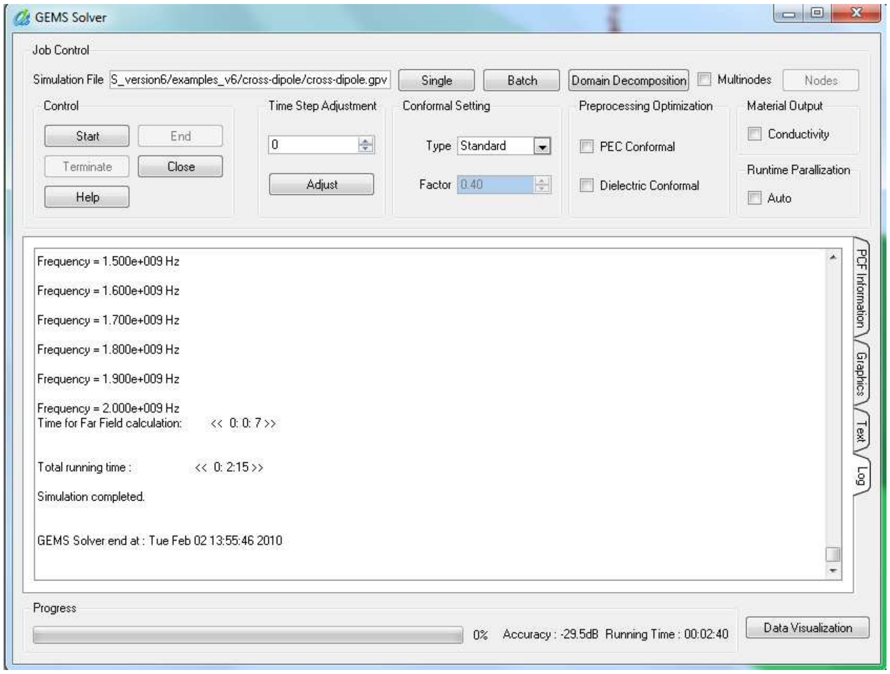
Click on the **PreCalculate** button in the toolbar.



Click on the **Validate** button to validate the project options. Click on the **Simulate** button to start the simulation.

23.9 Simulate Project

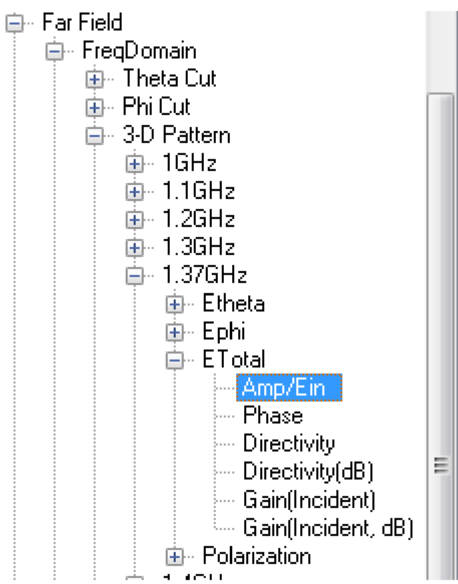
Click on the **Start** button to start the simulation.

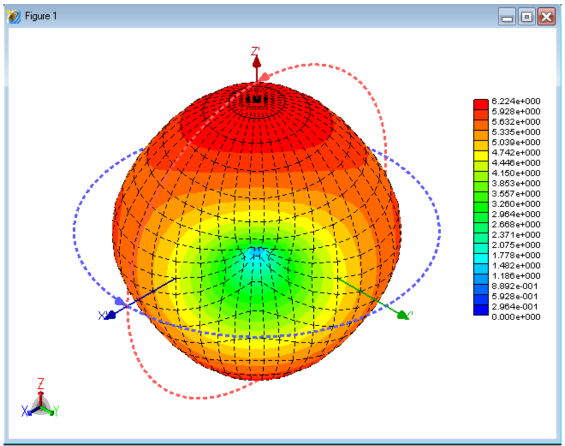


23.10 Result Visualization

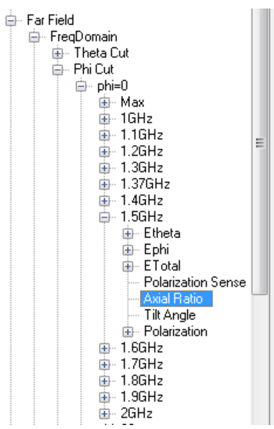
Click on the **Data Visualization** button to open the *GEMS Display* window.

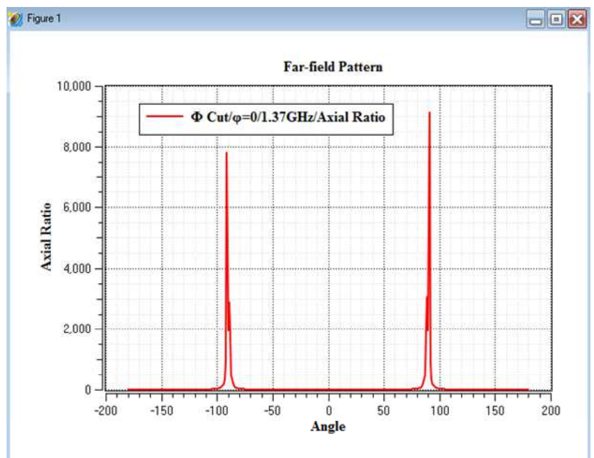
Double-click on the “AMP/Ein” option to display 3-D far field pattern at 1.37GHz.



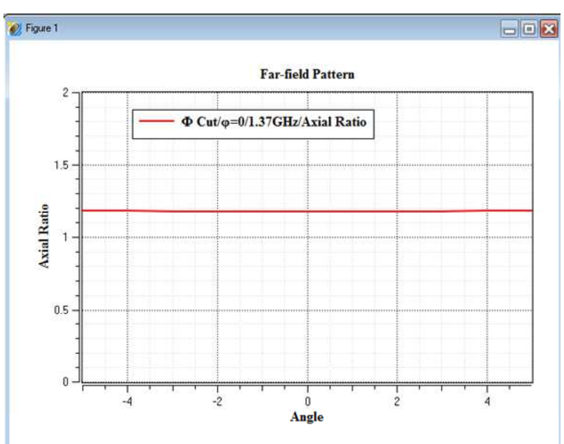


Double-click on the “Axial Ratio” option to plot the axial ration at 1.5GHz.





Double-click on the margin region in the figure and adjust the display range.



24

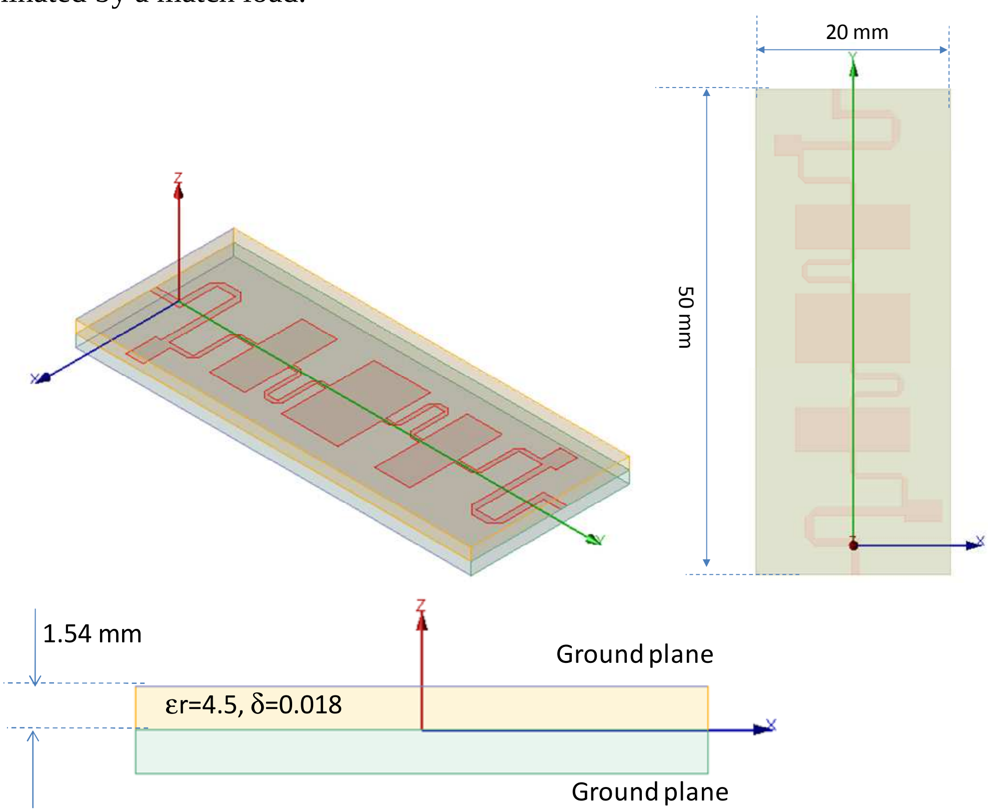
Example 24. Microstrip Filter

Description: One low pass filter in the sandwich structure includes top and bottom PEC ground, two layers of dielectric substrate. The output parameters include S-parameters, current distribution.

Keywords: filter, sandwich structure, surface current distribution.

24.1 Problem Configuration

Both the ground plane and substrate are finite with the length = 50 mm and width = 20 mm. The two ports in the filter are located at two ends of the filter. When one of them is excited, another is terminated by a match load.



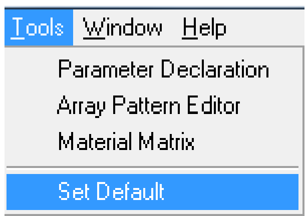
24.2 Create Project Model

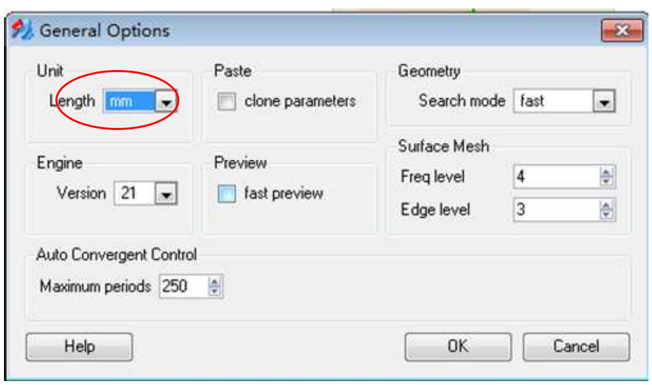
Follow the steps below to create substrates:

(1) Open the GEMS designer

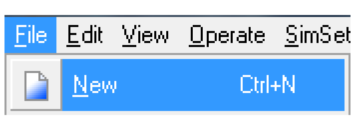
(2) Specify the project unit, which cannot be changed during the project modeling though you can input a variable in any units; however, the default unit can be only specified once at the beginning.

Select the **Tools->Set Default** option, and then select "mm" in the **Unit->Length** box as the project unit, click on the **OK** button to close the window.



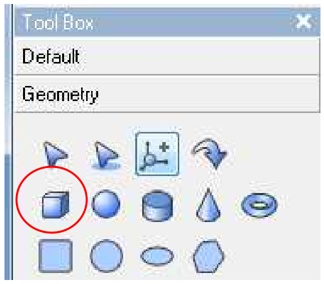


(3) Click on the **New** button in the toolbar or select the **New** option in the **File** menu.

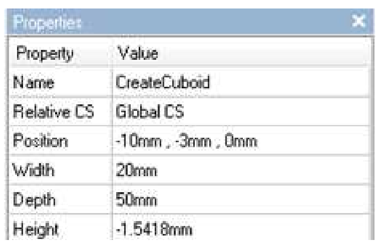


(4) Draw substrates

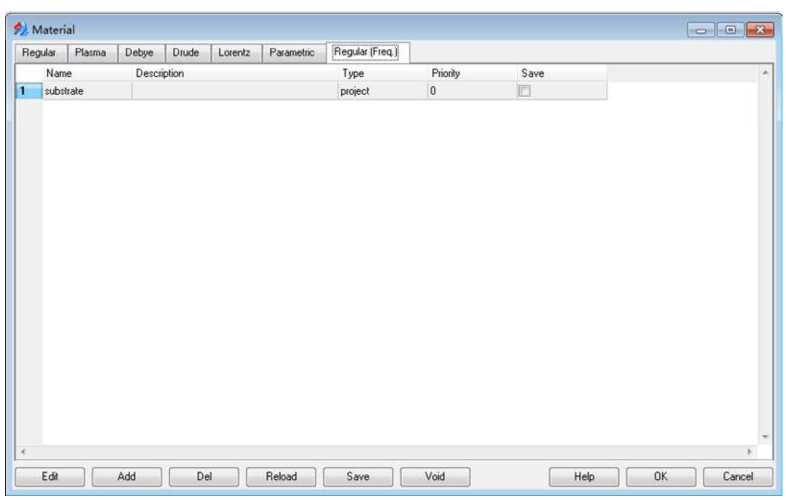
Click on the *Cuboid* icon in the **Tool Box->Geometry** box.



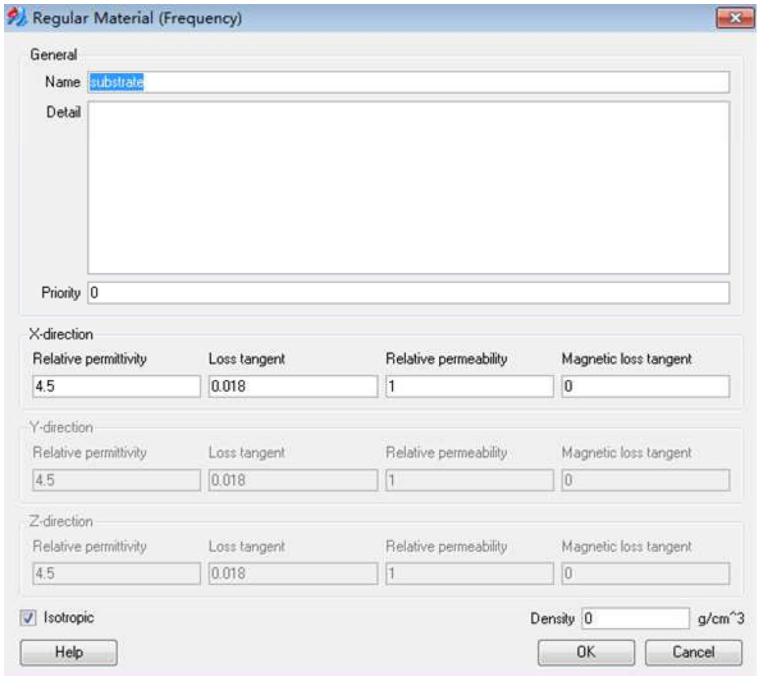
Draw the lower substrate (length = 50mm, width = 20mm, and height = 1.54mm). The corner coordinates are (-10mm, -3mm, 0mm).



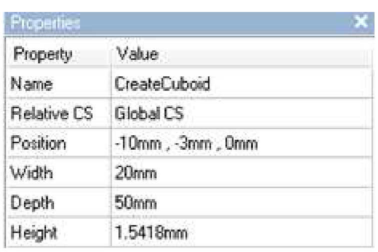
Select the substrate in the **Object Browser** box and click the **Color** bar in the **Properties** box to adjust the substrate color. Click the material option in the **Properties** box to specify the material type in the popup window. Click the Regular(Freq.) option and then the “Add” button.



Specify the material name in the “Name” box, input the description (Optional), and material parameters, relatively dielectric constant and loss tangent. For the isotropic material, check the “Isotropic” box and specify the material only in one direction. The density is designed for the SAR simulation, otherwise, it is meaning at all in the simulation.



Follow the similar procedure to draw the upper substrate (length = 50mm, width = 20mm, and height = 1.54mm). The corner coordinates are (-10mm, -3mm, 0mm). Since the substrates are same type of material, you select the upper substrate in the **Object Browser** box and then click the material option in the **Properties** box. Select the "substrate" material in the material list in the "Regular (Freq.)" option.



Properties

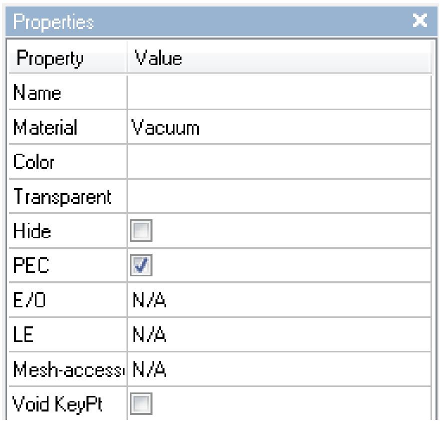
Property	Value
Name	CreateCuboid
Relative CS	Global CS
Position	-10mm , -3mm , 0mm
Width	20mm
Depth	50mm
Height	1.5418mm

(5) Draw ground planes

Click on the *Rectangle* icon in the **Tool Box->Geometry** box. Move the mouse icon to the one top corner of the upper substrate, and click the left mouse button to select the start point. Move the mouse icon to the cross corner and click the left mouse icon to select the end point of the rectangle.

Select the rectangle option in the **Object Browser** box, change its name in the **Properties** box to be "GND" and its color to the desired one, check the "PEC" box in the **Properties** box.

Follow the similar procedure to draw the ground underneath the lower substrate, and specify name and its color and check the “PEC” box in the **Properties** box.



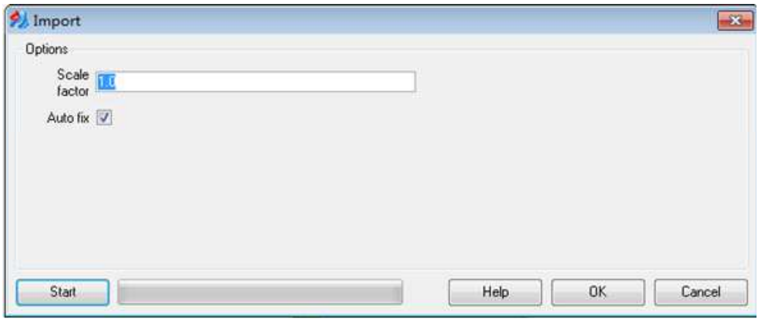
(6) Import the circuit structure

The circuit structure is generated in the AutoCAD software, and you need to save it in the 2004 format that GEMS can input in the current version. However, the new version of AutoCAD software can directly export the solid or filled planar structure in to the SAT format. In any time, if possible, you should get the model in the SAT format that is best fit to the GEMS software.

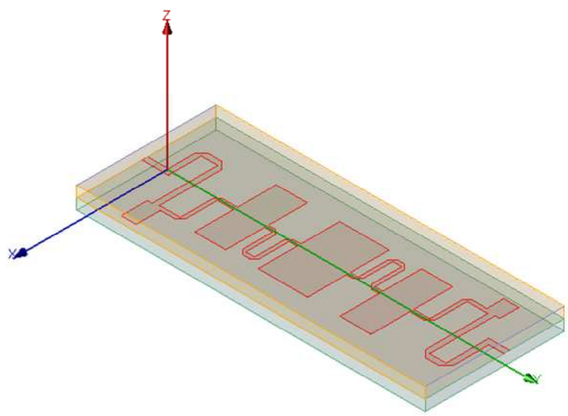
If the model is in the DXF format but not in the 2004 format, you can get help from the third party to convert it to the SAT format and then import it to GEMS interface.

To import the circuit structure in to GEMS, click the “Import model” icon in the **Geometry** box in the **Tool Box**. Search and select the SAT file in your computer.

In the popup window, you can make the model larger or smaller through the “Scale factor”. The “Auto fix” box is designed to fix some minor model problems, however, it will take time for the complex model. If the model is correct, unchecking this box can speed up the model import procedure.



Click the **Start** button to start the import procedure, and click the **OK** button after the import procedure is completed.

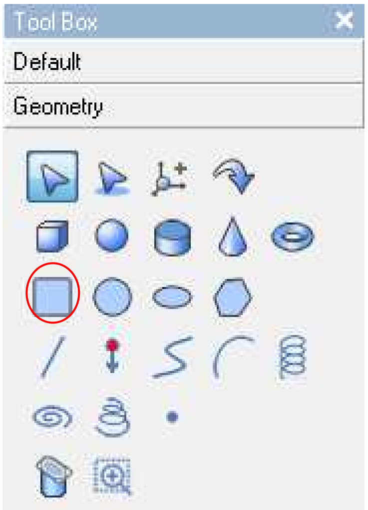


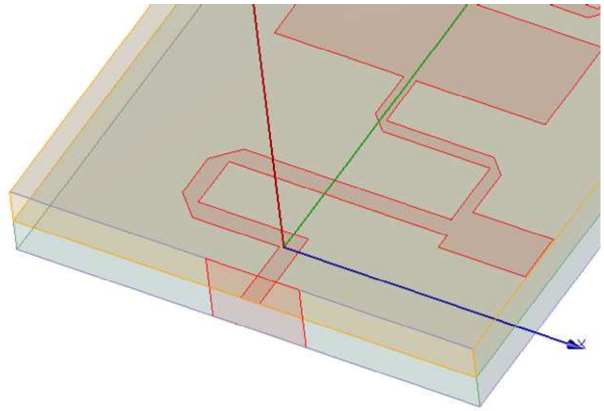
The circuit must be imported after the substrates to avoid being erased since the circuit has no thickness.

24.3 Draw Excitation geometry

There are several ways to excite the port in this filter problem. However, if the uniform feed structure is not long enough to avoid the higher mode in the measurement, it is a better choice to use the mode excitation.

To define a mode port, the first step is to draw a rectangle at the port location. Click on the **Point** icon in the **Tool Box->Geometry** box, and move the mouse icon in the figure region and use the snap feature to measure the coordinate of the excitation port. Set the drawing plane to be "X-Z" and the location of the drawing plane to be "-3" in the toolbar. Click on the **Rectangle** icon in the **Tool Box->Geometry** box. , and select a proper width in the horizontal direction, and the top and bottom sides touch the ground planes in the vertical direction.



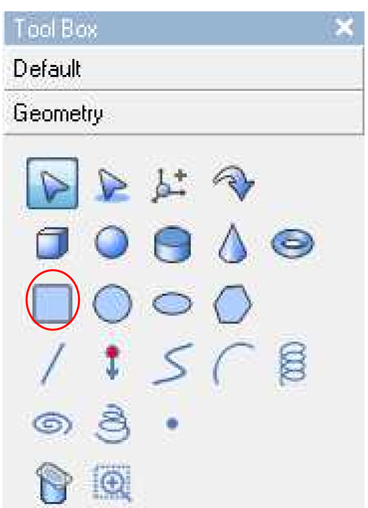


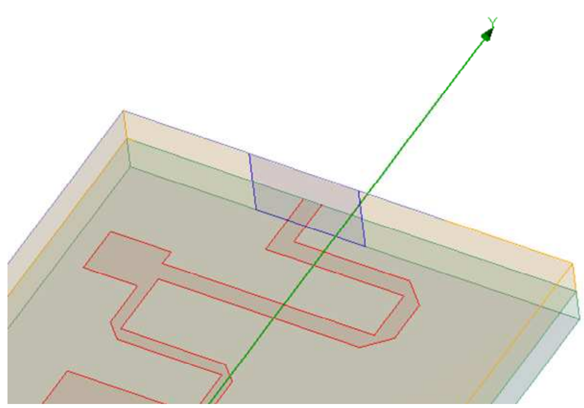
Its excitation property will be assigned after the mesh distribution is designed. Since the mode pattern is associated with the mesh in GEMS, the mode excitation requires that the mode pattern extraction must be redone once the mesh distribution is changed.

24.4 Draw Output Geometry

Follow the similar procedure to define an output port. Click on the **Point** icon in the **Tool Box->Geometry** box, and move the mouse icon in the figure region and use the snap feature to measure the coordinate of the excitation port. Set the drawing plane to be "X-Z" and the location of the drawing plane to be "47" in the toolbar.

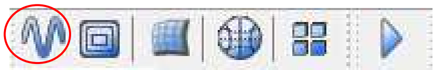
Cick on the **Rectangle** icon in the **Tool Box->Geometry** box, and select a proper width in the horizontal direction, and the top and bottom sides touch the ground planes in the vertical direction.

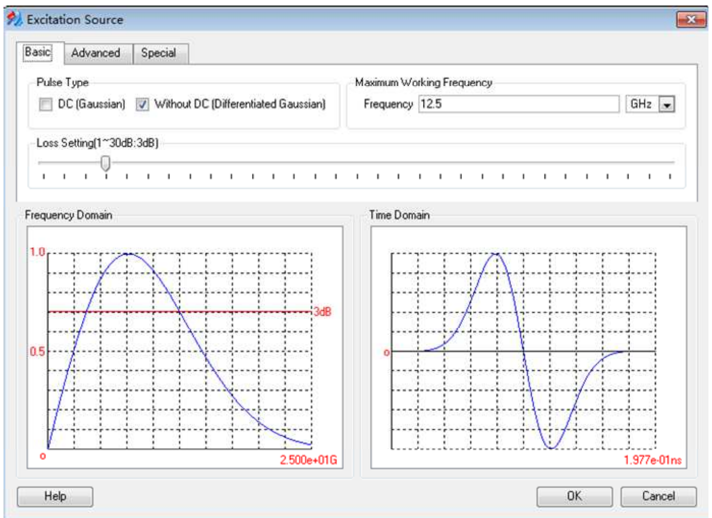




24.5 Set Excitation Pulse

Click on the **Set excitation** button in the toolbar.

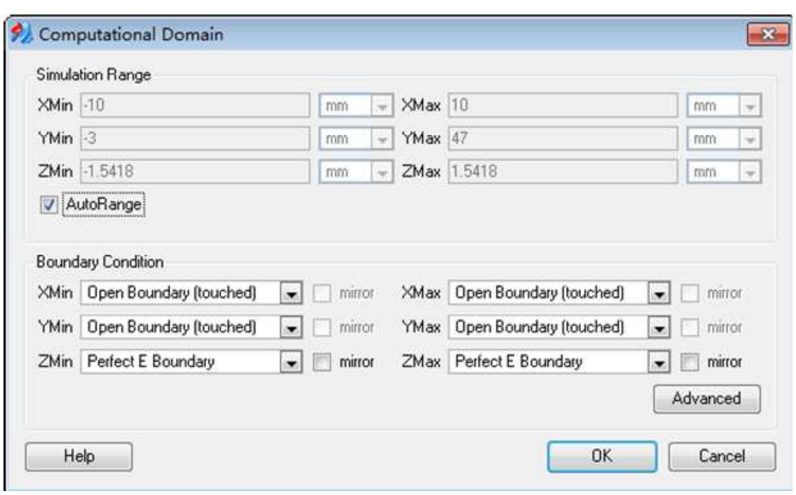




Since the highest frequency of interest is 12.5 GHz, check the "Without DC (Differential Gaussian)" box and select the frequency unit to be "GHz", and input the number "12.5".

24.6 Set Domain and Boundary Condition

Click on the **Set boundary condition** button in the toolbar.

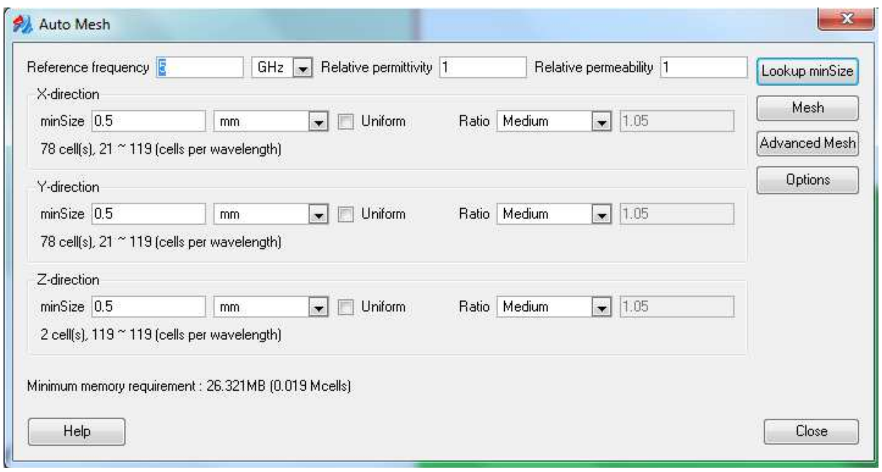


Since the filter structure is packaged inside two PEC grounds, we can use the “Perfect E Boundary” in the vertical directions to truncate the domain to reduce the simulation domain size. The “Open space (Touched)” in the horizontal directions is used to truncate the simulation domain, which will extended the ground planes, substrates and feed lines to the infinite.

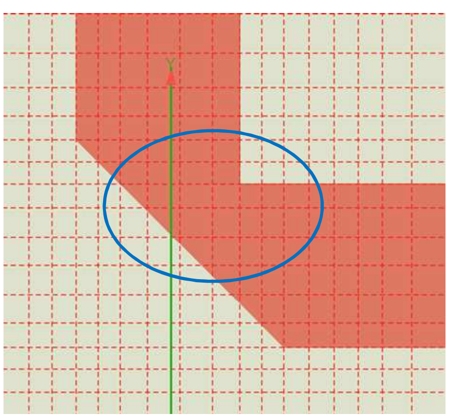
24.7 Design Mesh Distribution

Click on the **Auto mesh** button in the toolbar.

Click on the **Loopup miniSize** button and then on the **Mesh** button.



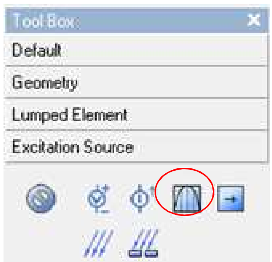
GEMS requires at least two cells inside the narrow circuit part, for example, the mesh distribution at the corner should be as shown in the figure below.

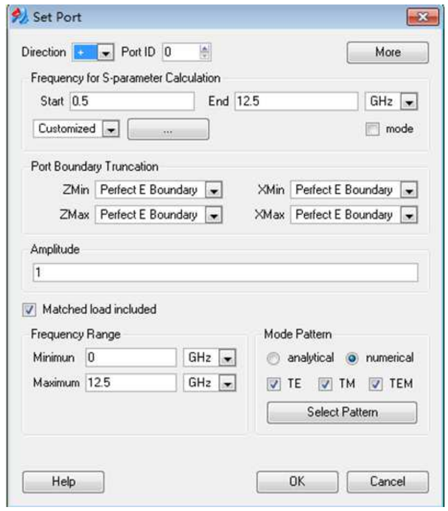


During the mesh generation, it is important that the port structure should not form the fine gap with the circuit or the output port. Otherwise, the port width should be adjusted to avoid it happens. To be safe, we can draw the port structure after the mesh generation, and the port structure can line up with the mesh.

24.8 Mode Extraction

Select the excitation rectangle in the **Object Browser** box, and click on the **Mode** icon in the "Tool Box->Excitation Source" box.

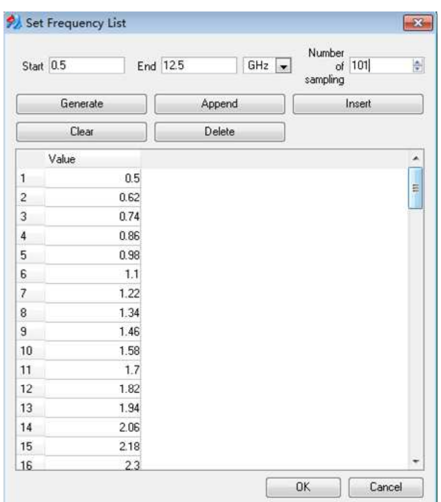




The "direction" selection makes the port direction point to the inside circuit. Specify the frequency range, click the "Customized" or "Adaptive" option and then click the button.

Option "Adaptive": GEMS will automatically generate 1000 frequency points inside the specified range.

Option "Customized": You will specify the number of frequency sampling points, and then click the Generate button.



Select the boundary type in the "Port Boundary Truncation", if the real model of the excitation port is a coaxial cable, we can select the "Perfect E Boundary" in the four directions.

If the number of excitation ports is more than one, they can have a different excitation level by specifying different amplitudes in the “Amplitude” box.

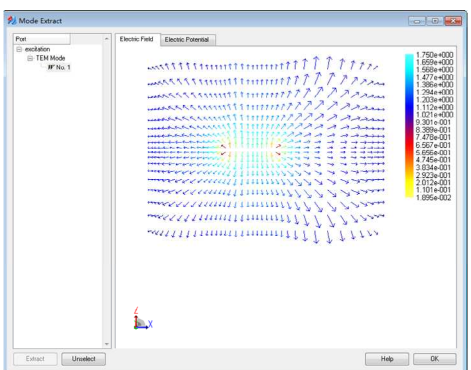
The option “Matched load included” will add a matched load behind the excitation port to ensure that the port is truncated properly. GEMS adds a uniform feed structure behind the excitation port. To improve the port performance, the excitation port will be moved to inside the uniform feed structure.

If a port structure has a cutoff frequency, you should point out the frequency range in the “Frequency Range” box so that GEMS only needs to extract the modes inside the frequency range.

If the excitation port structure is a circular or rectangular waveguide, you can check the “Analytical” radio button, and GEMS will use the analytic formula to calculate the mode patterns inside the specified frequency range. Otherwise, you need to check the “Numerical” radio button and GEMS will use the embedded frequency module to extract the mode patterns supported by the port structure.

After properly setting the parameters above, click the “Select pattern” button to extract the mode patterns and select the excitation mode in the popup window.

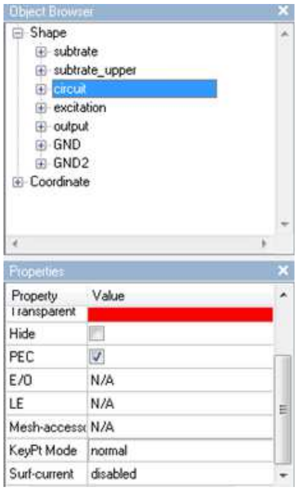
Inside the popup window, select the excitation port in the port list, and click the “Extract” button to start the modes. It takes a while for a large port structure. Select the mode you like to use for the excitation.

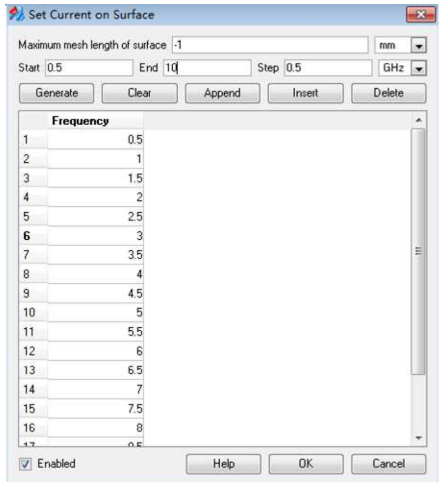


Follow the similar procedure to extract the modes for the output port, however, the number of output modes can be one.

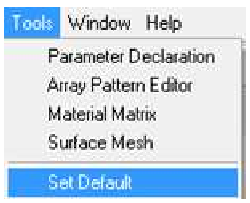
24.9 Set Surface Current Distribution Output

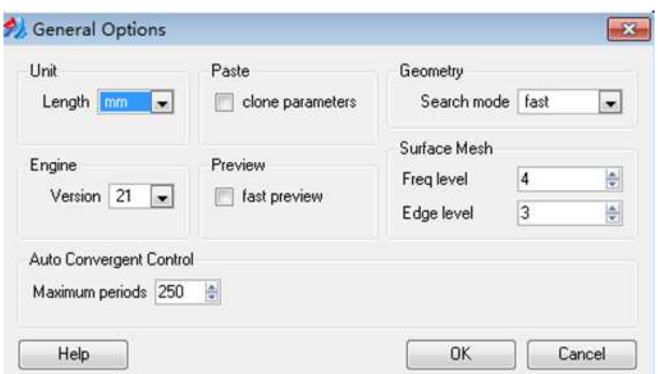
GEMS allows you to output the surface current distribution on the surface of an arbitrary conductor. To this end, select the “circuit” in the **Object Browser** box, and then click the “Surf_current” option in the **Properties** box. Check the “Enabled” box in the popup window and then on the “OK” button. GEMS will generate the surface current distribution on the circuit during the simulation.





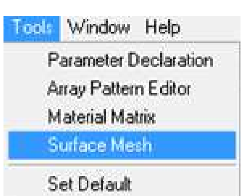
The surface current is generated based on a surface mesh. The default surface mesh settings are located inside the window:

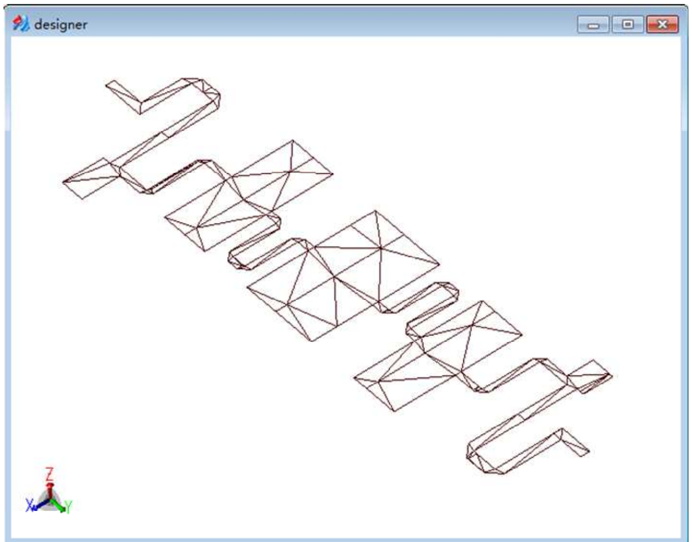




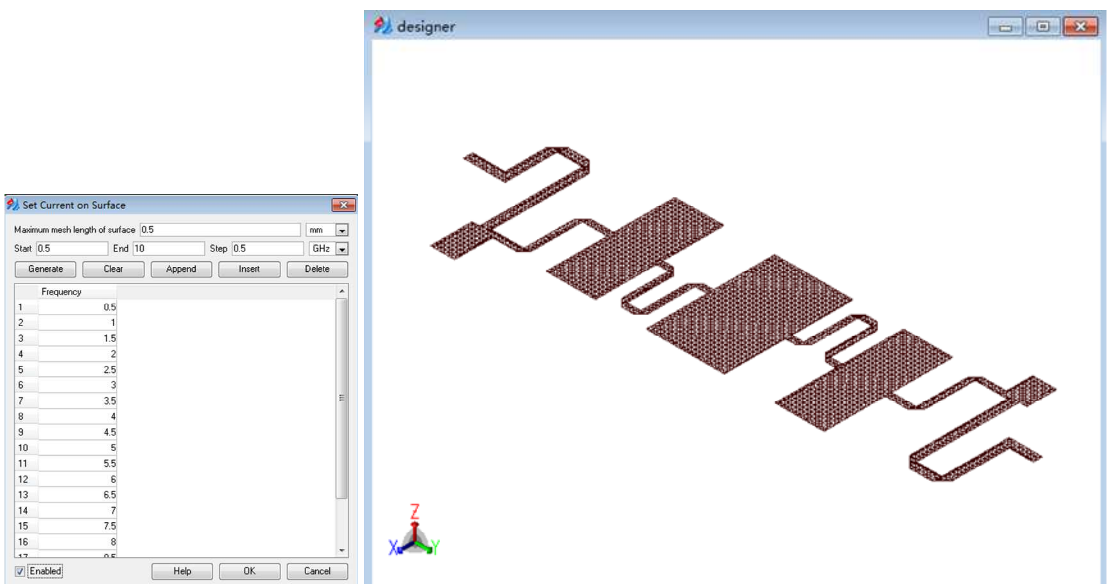
The option “Surface Mesh->Freqlevel” : the mesh size is 1/4 of the wavelength corresponding the highest frequency of interest.

The option “Surface Mesh->Edgelevel” : the mesh size is 1/3 length of the minimum edge.





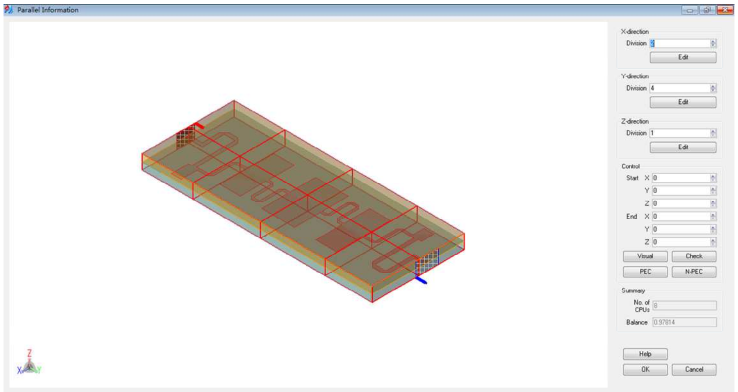
It is obvious that the mesh is too coarse. We can specify the mesh size in the “Set Current on Surface” window, for instance, change the “-1” to be “0.5” in the “Maximum mesh length of surface” box.



24.9 Parallel Processing Setting

For a multiple CPU workstation or a cluster platform, you need to specify the number of subdonains. GEMS will assign each subdomain to one CPU. For some AMD CPUs, each CPU may contains two physical CPUs. The number of subdomains should be two to use the two CPUs for simulation.

Job balance is important for the better simulation performance. We should always work on the side first that includes more cells.

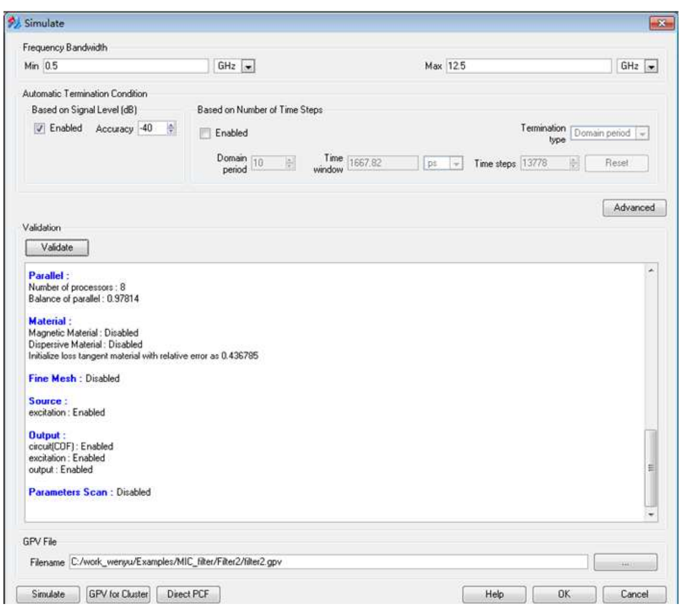


24.10 Save Project

Click on the **Save** button in the toolbar.

24.11 Generate Simulation File

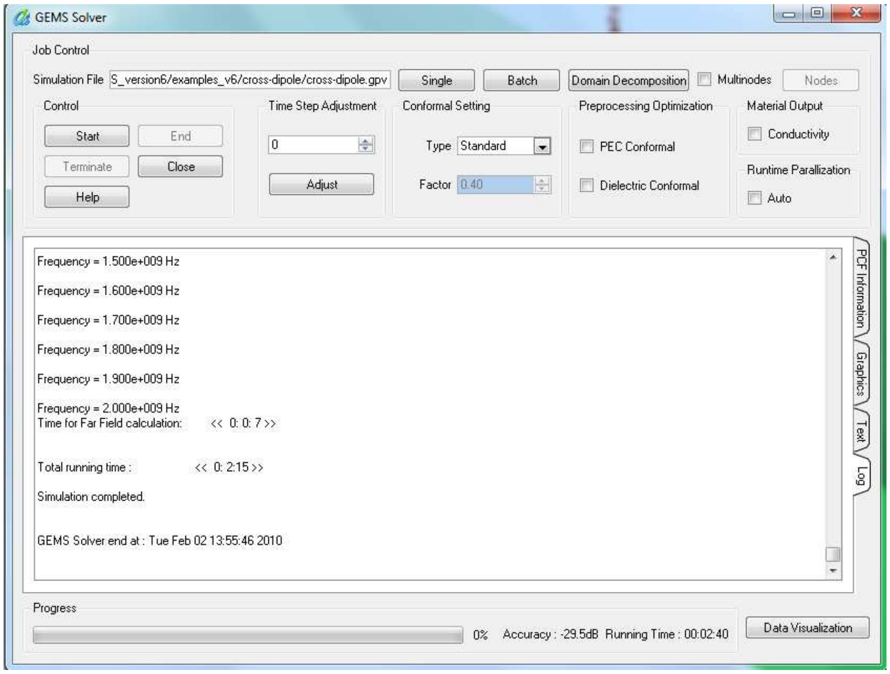
Click on the **PreCalculate** button in the toolbar.



Click on the **Validate** button to validate the project options. Click on the **Simulate** button to start the simulation.

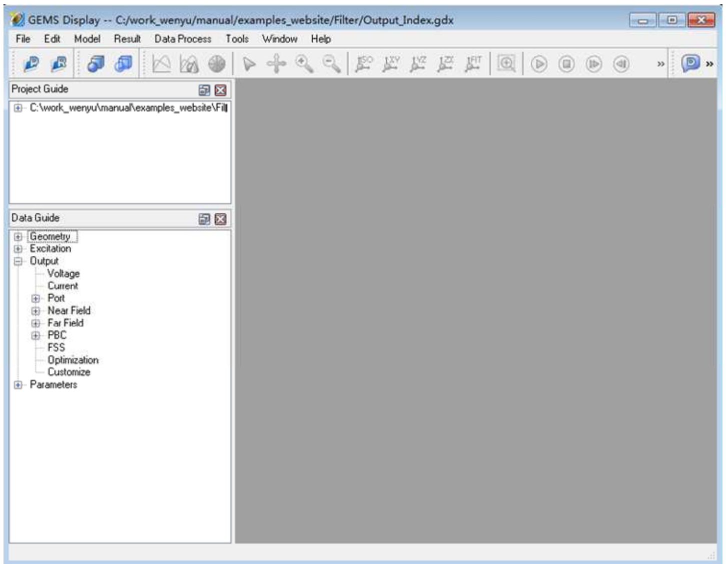
24.12 Simulate Project

Click on the **Start** button to start the simulation.

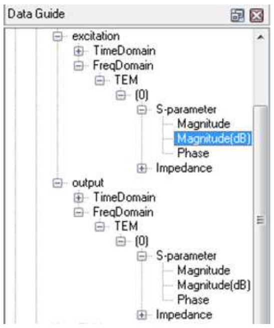


24.13 Result Visualization

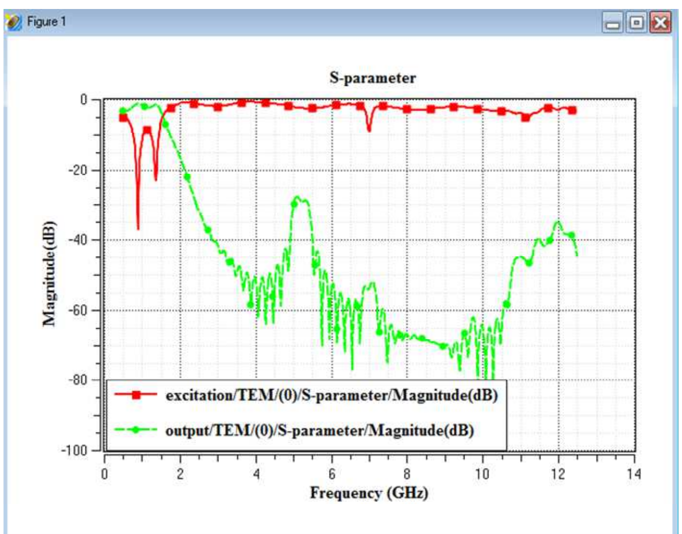
Click on the **Data Visualization** button to open the *GEMS Display* window.

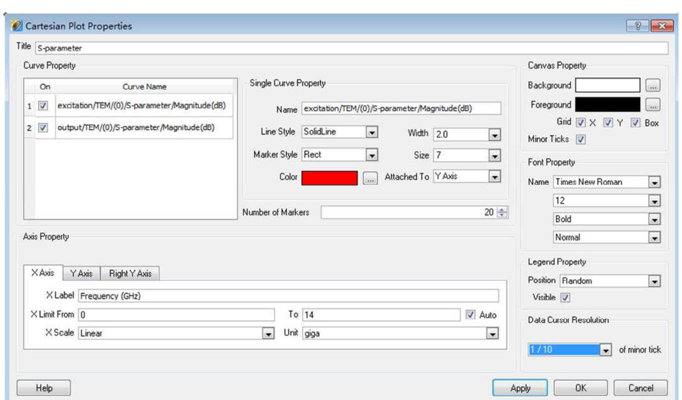


Select and double-click the “Output->excitation->FreqDomain->TEM->(0)->S-parameter-> Magnitude(dB)” option, and select the “Output->output->FreqDomain->TEM->(0)->S-parameter-> Magnitude(dB)” option, and then click the “Add to current window” button in the toolbar to add two curves in one figure.

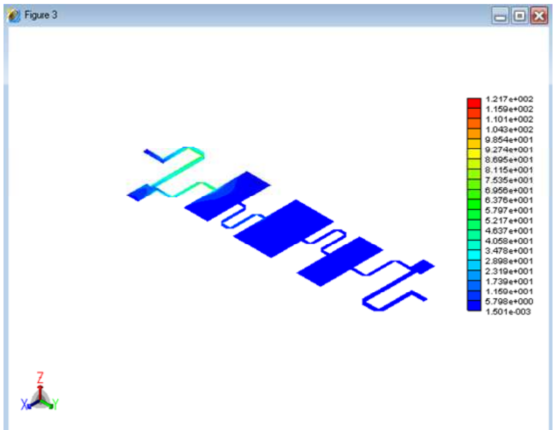


Double-click the figure region to specify the figure options in the popup window.

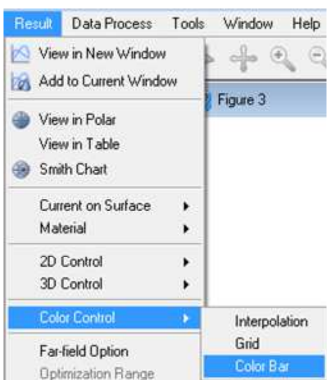


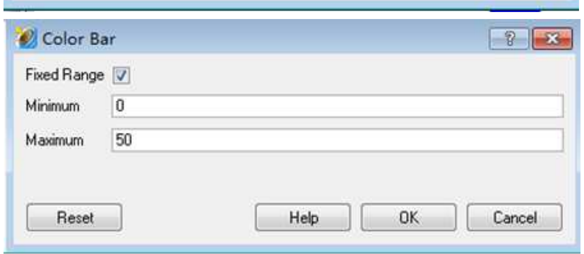


Open the “circuit->CurrentOnSurface” subfolder in the “Near Field” folder and select one frequency in the frequency list. For example, double-click the “3GHz” option to plot the surface current distribution on the circuit at 3GHz.



Select the “Result->Color Control->Color Bar” option, and check the “Fixed Range” box in the popup window, and then adjust the display range.





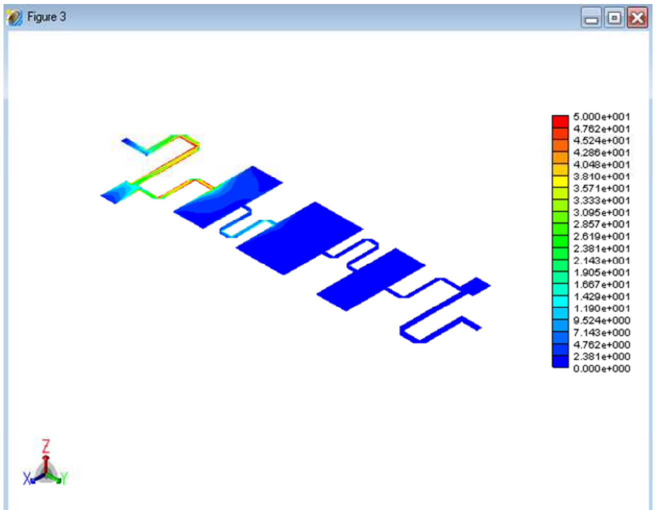


Figure 3
5.000e+001
4.762e+001
4.524e+001
4.286e+001
4.048e+001
3.810e+001
3.571e+001
3.333e+001
3.095e+001
2.857e+001
2.619e+001
2.381e+001
2.143e+001
1.905e+001
1.667e+001
1.429e+001
1.190e+001
9.524e+000
7.143e+000
4.762e+000
2.381e+000
0.000e+000
Z
X
Y

25

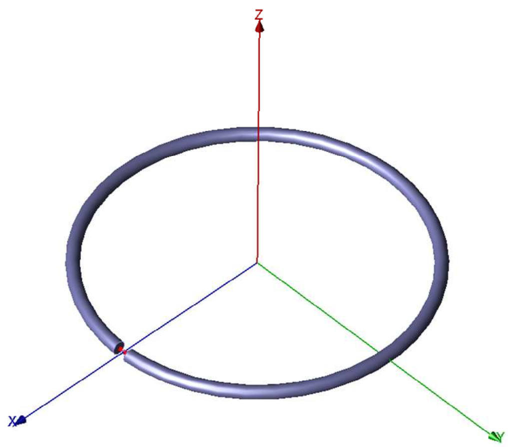
Example 25. Ring Antenna

Description: A ring antenna has a slot where a lumped port is placed for the excitation. The proper mesh distribution is generated by adding some key points around the ring.

Keywords: antenna, impedance, return loss and surface current distribution on ring.

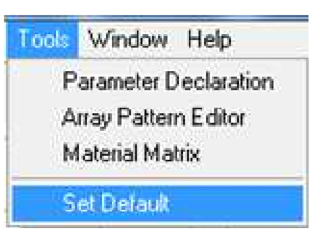
25.1 Problem Configuration

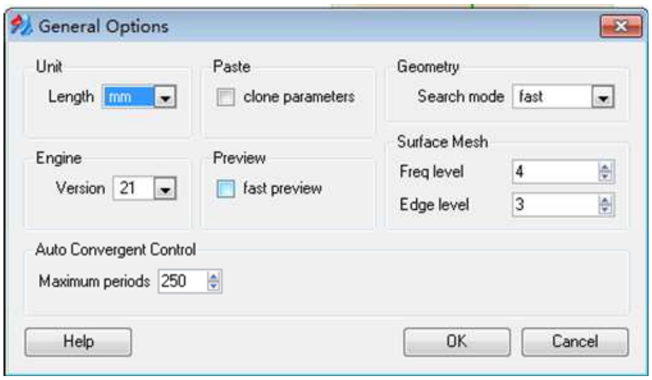
The antenna configuration is shown in the figure below. The main radius of ring is 5 mm and the wire radius is 0.2 mm. The feed gap is chosen to be 0.2 mm.



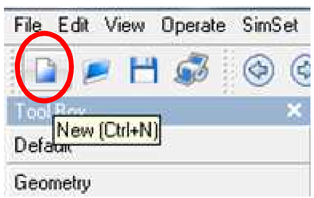
25.2 Start A New Project

GEMS requires that the unit specification must be done before you start a new project.





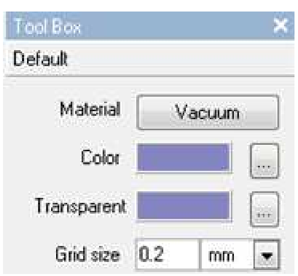
Start a new project by click the **New** button in the toolbar.



25.3 Create Project Model

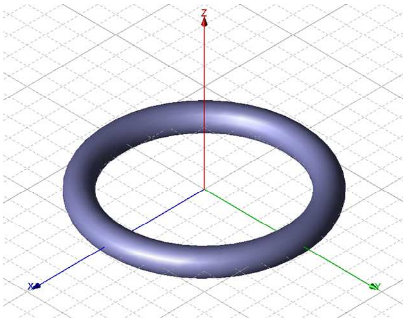
Follow the steps below to draw a ring:

(1) Select the background resolution.

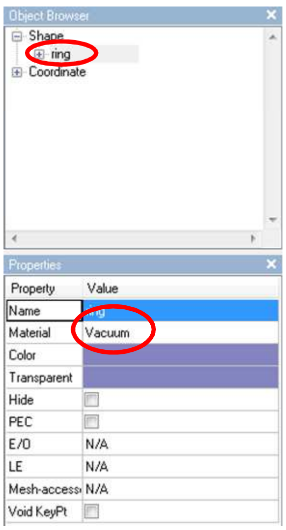


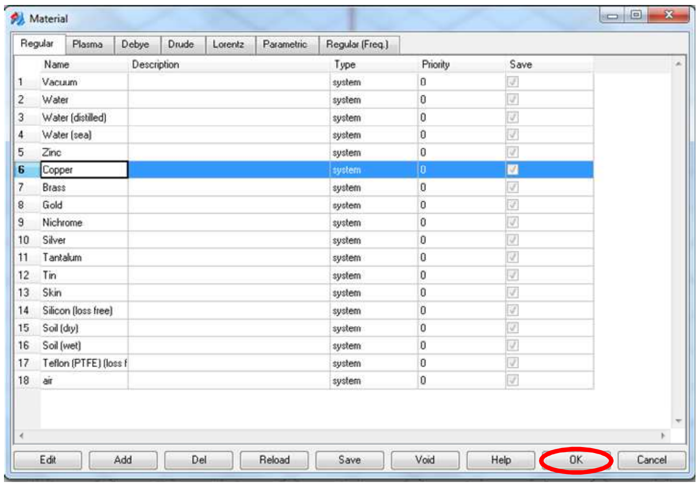
(2) Click the **Torus** icon in the **Tool Box->Geometry** box.

(3) Move the mouse icon to the figure region to draw a torus object.

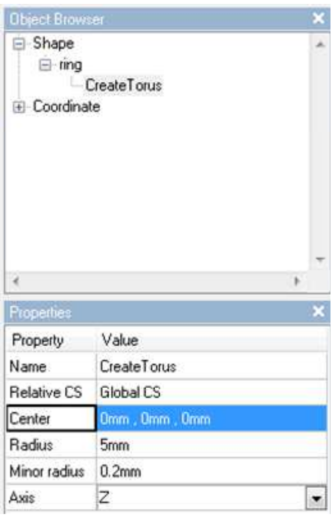


(4) Select the option in the **Object Browser** box and change its name and color, assign its material to be copper by clicking the **Material** box in the **Properties** box.





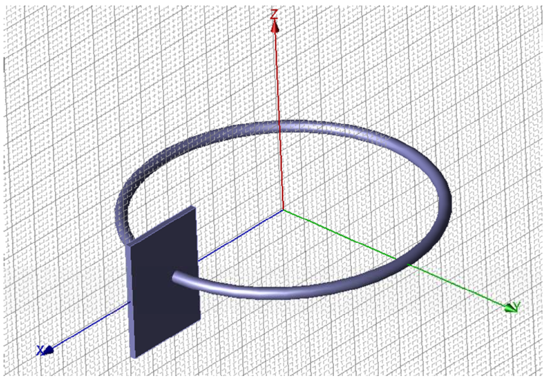
(5) Select the subentry under the option "ring" in the **Object Browser** box, and then change its geometry parameters to the desired values.



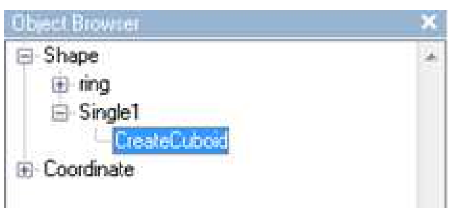
Follow the steps below to cut a feed gap:

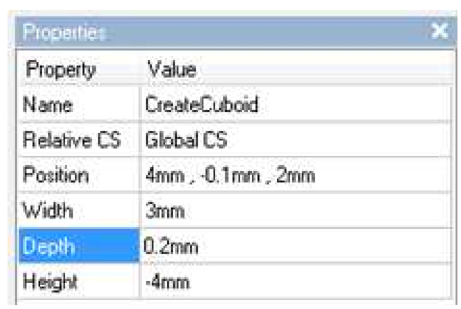
(1) Select the “X-Z” and Y=-0.1 as the drawing plane in the toolbar.

(2) Select the **Cuboid** icon in the **Tool Box->Geometry** box.

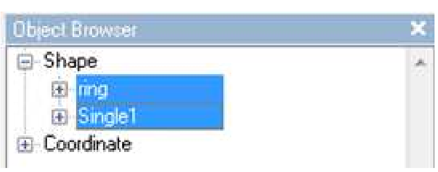


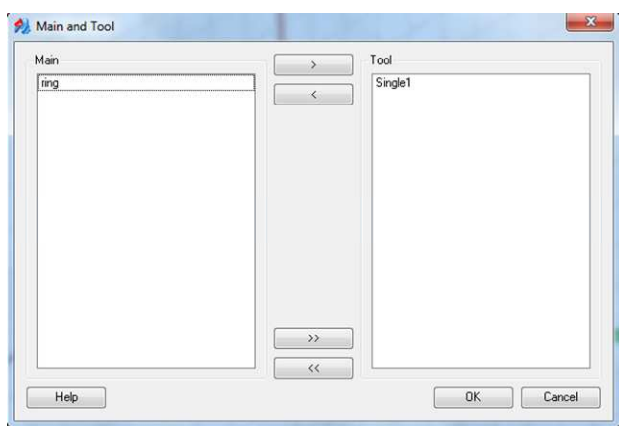
(3) Draw a cuboid and specify its height to be 0.2 mm.

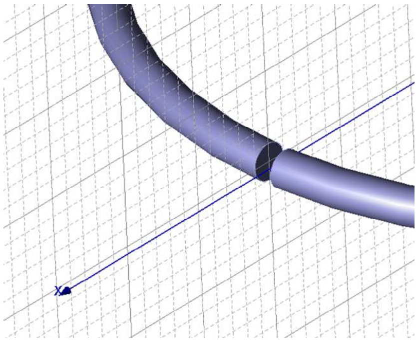




(4) Use the torus to subtract the cuboid to cut a feed gap on the ring.



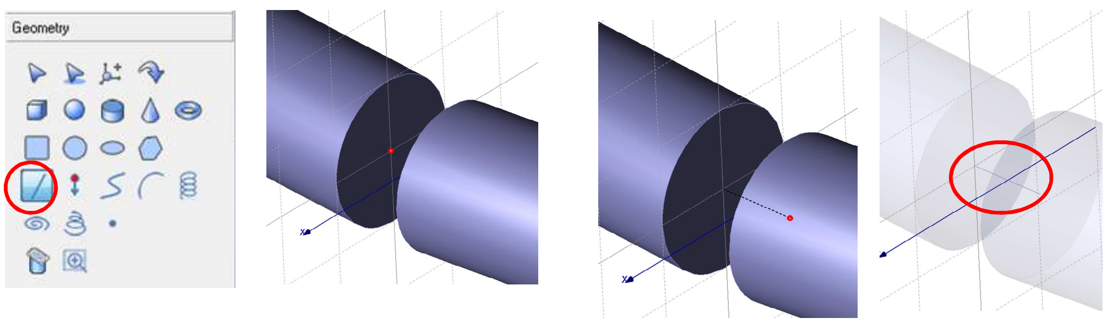




(5) Click the **Save** button in the toolbar to save the project.

Follow the steps below to draw feed path:

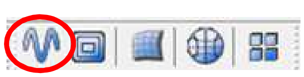
(1) Zoom in the feed gap.
(2) Click the Line icon in the **Tool Box->Geometry** box.
(3) Move the mouse icon to select the central point of one cross section, and then press the right mouse button.
(4) Move the mouse icon to look for another central point and then double-click the right mouse button.
(5) A line across the feed gap will appear.



25.4 Specify Excitation Pulse

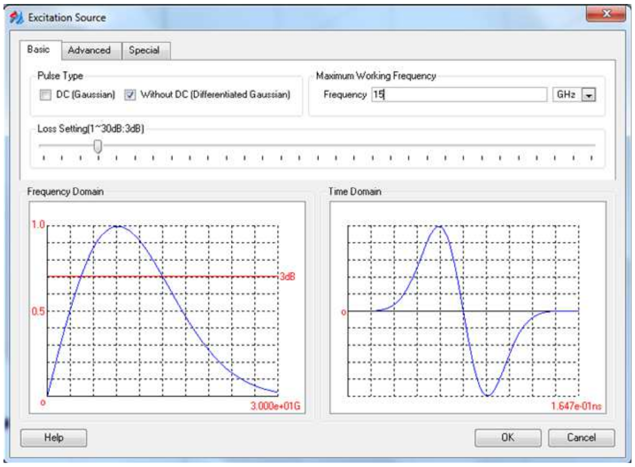
Follow the steps below to specify the excitation pulse:

(1) Click on the **Set excitation source** button in the toolbar.



(2) Set the Maximum working frequency to be “15” GHz. The left figure shows the frequency spectrum used in the simulation, and the right figure shows the time domain excitation pulse.

The excitation pulse will be applied to all the ports regardless of the port types. In the current version, one project only allows you to use one excitation pulse.

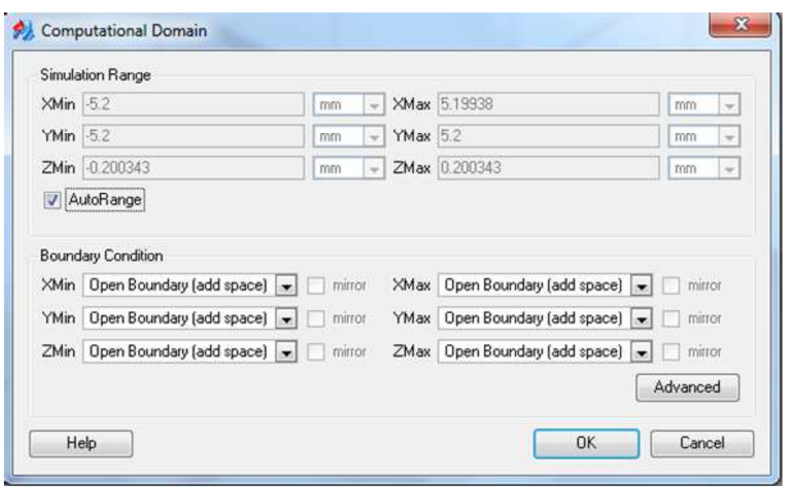


25.5 Set Domain Size and Boundary Condition

Follow the steps below to set the domain and boundary condition:

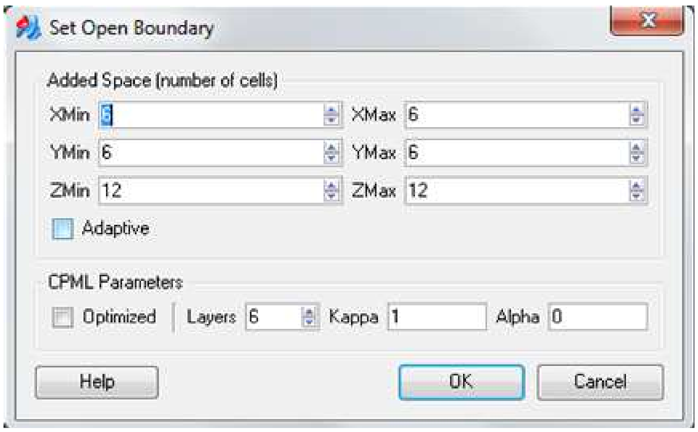
(1) Click on the **Set boundary condition** button in the toolbar.

(2) Specify the domain size.
  (a) Check the "AutoRange" box. GEMS finds the minimum domain that including all of the objects.
  (b) Uncheck the "AutoRange" box. You can define the domain size manually.



(3) For an open space problem, the option "Open Boundary (add space)" is used to all six walls.

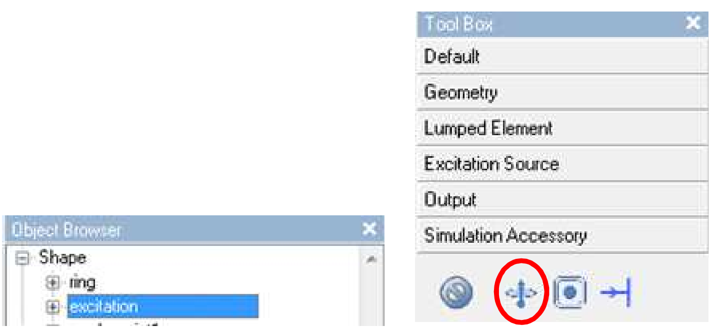
The domain dimension in the vertical direction is much smaller than those in the horizontal direction, which may result inaccurate results. Usually, we can use a larger domain in the vertical direction. For instance, we can uncheck the “AutoRange” button to modify the numbers in the **Zmin** and **Zmax** boxes to 20 to 30 percent of its horizontal dimension, say, Zmin=-0.75 and Zmax=0.75. Or, click the **Advanced** button to add more space in the vertical direction in terms of number of cells, for example, uncheck the **Adaptive** button and modify the numbers in the vertical direction, say, “12”.



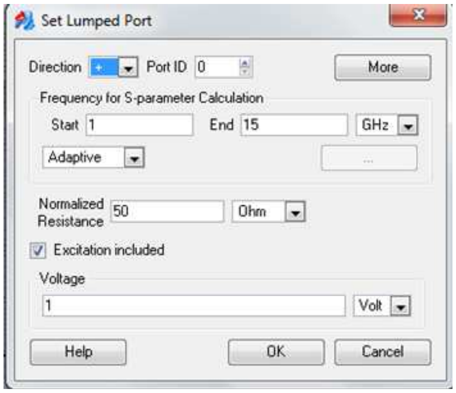
25.6 Define Excitation Port

Follow the steps below to define an excitation port:

(1) Select the line in the **Object Browser** box, which you have drawn inside the feed gap, and then click the **Lumped port** icon in the **Geometry** box inside the **Tool Box**.



(2) Specify the frequency range for the return loss and port impedance outputs. The “Adaptive” option will sample 1001 frequency points in the specified range, and the “Customized” option allows you to specify the number of sampling points. The “50” Ohms means that the internal impedance of the excitation source is 50 ohms and the return loss will be normalized to 50 ohms. Checking the “Excitation included” box means that this port is an excitation port; otherwise, it will be an output port. The excitation port will serve as the both excitation and output ports; namely, it will return the port impedance, return loss, time domain port voltage and current.

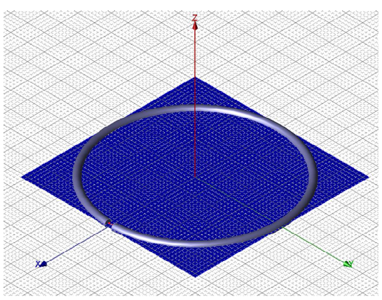


25.7 Set Field Distribution Output

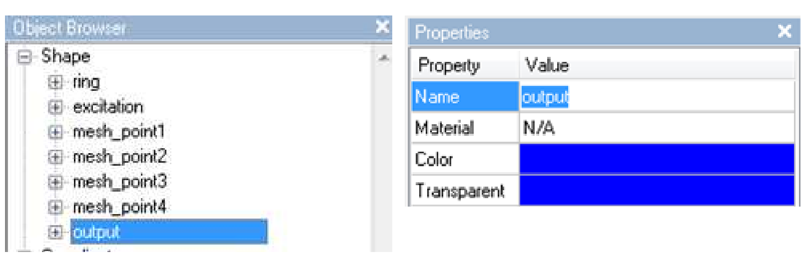
Follow the steps below to set the field distribution output:

(1) Click the **Rectangle** icon in the **Geometry** box inside the **Tool Box**.

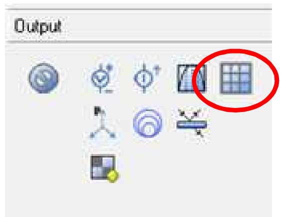
(2) Move the mouse icon to the figure region and draw a rectangle (-5.2, -5.2, 0), (5.2, 5.2, 0).



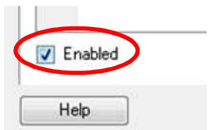
(3) Select the option in the **Object Browser** box and change its name to “output” in the **Properties** box.



(4) Click the **Field** icon in the **Output** box inside the **Tool Box**.

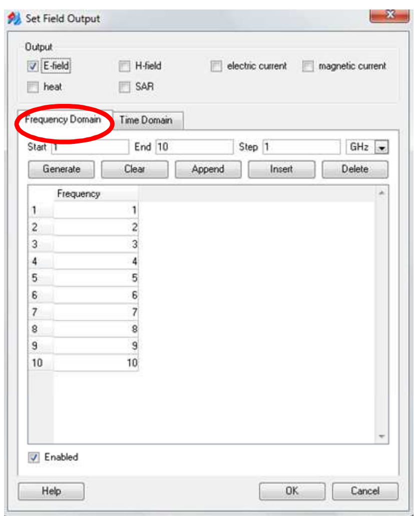


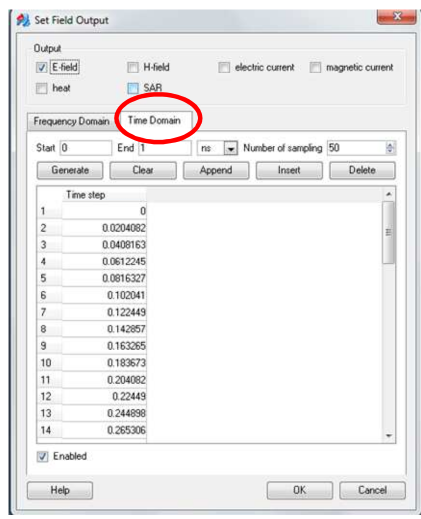
Check the “Enabled” checkbox before you specify the output parameters.



Uncheck the “H-field”, “magnetic current” and “electric current” boxes if you only need to output the electric field distribution on the selected surface.

Click the “Frequency domain” label and specify the frequency list for the field output, and click the “Time domain” label and specify the time list for the field output.

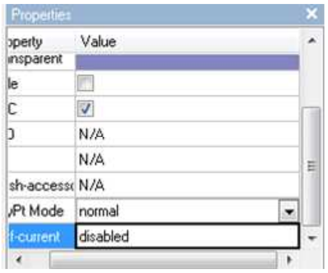




25.8 Set Surface Current Output

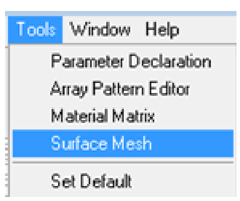
Follow the steps below to set the surface current output:

(1) Select the “Ring” option in the **Object Browser** box;
(2) Check the “Enabled” checkbox;
(3) Specify the frequency range and step size;
(4) Click the “Generate” button to generate the frequency list;

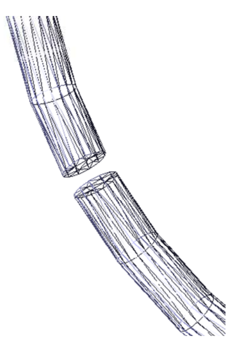


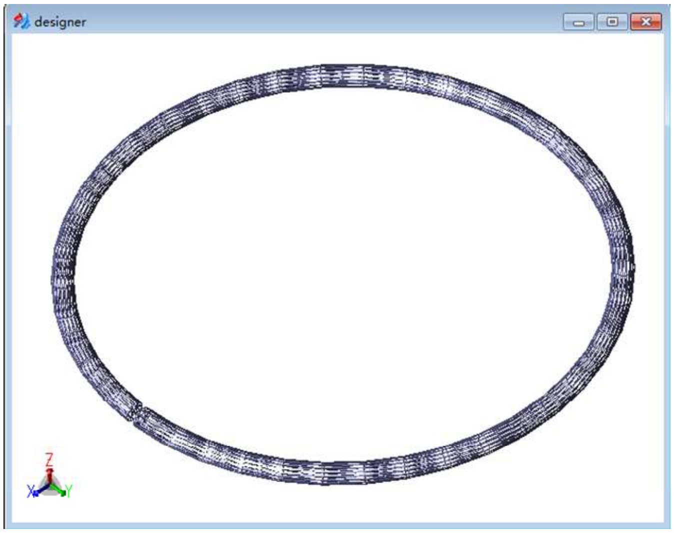
(5) The number “-1” represents that the surface mesh is generated based on the options in the window opened by selecting the “Tools->Ser Default” option.

(6) Select the “Surface Mesh” in the “Tools->Surface Mesh” option;



(7) View the surface mesh in the popup window;

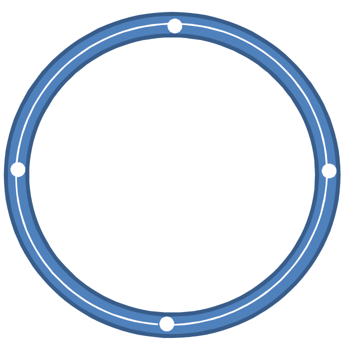




(8) If the mesh is too coarse, you can adjust the mesh size by giving a proper number in the "Maximum surface mesh of surface" box inside the "Set Current on Surface" window.

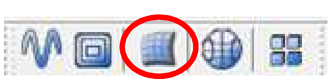
25.9 Design Mesh Distribution

Sometimes, you may not capture the smoothly thin structures in the domain, you can add some points at the key points to force the mesh pass them. For example, you can add four points at 0^o, 90^o, 180^o and 270^o along the central ring, which force the mesh to pass the center of the ring cross section.

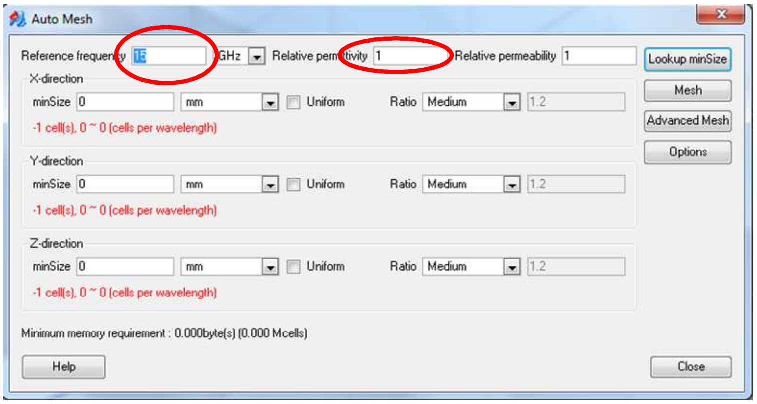


Follow the steps below to design the mesh distribution:

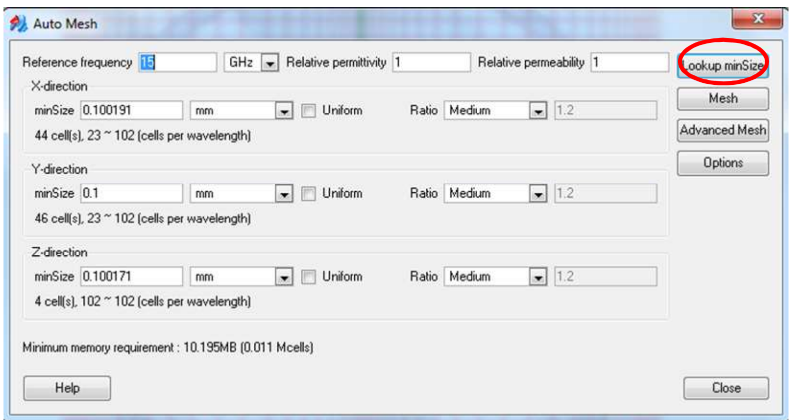
(1) Click on the **Auto mesh** button in the toolbar and then a new window will appear.



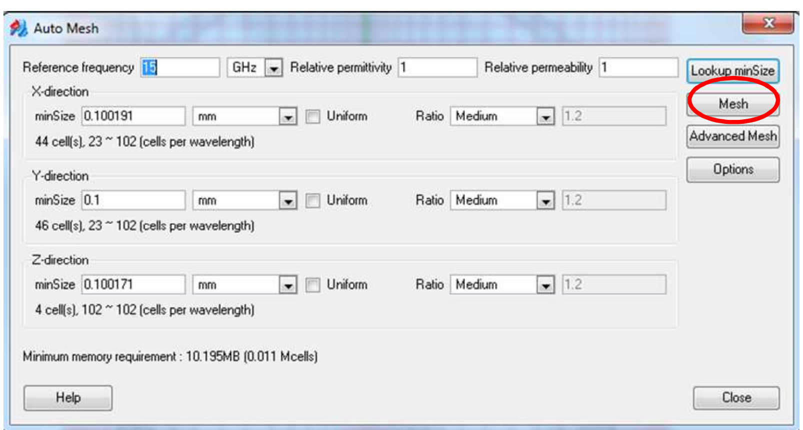
(2) Input the maximum frequency of interest, and maximum relative permittivity inside the domain.



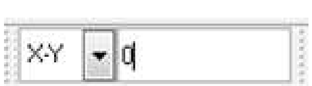
(3) Click on the **Lookup minSize** button to find the cell size automatically.



(4) Click on the **Mesh** button to generate the mesh distribution.

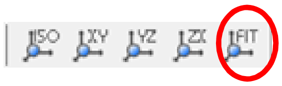


Select the "X-Y" and Z=0 as the display plane.

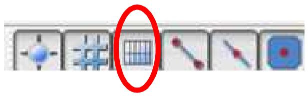


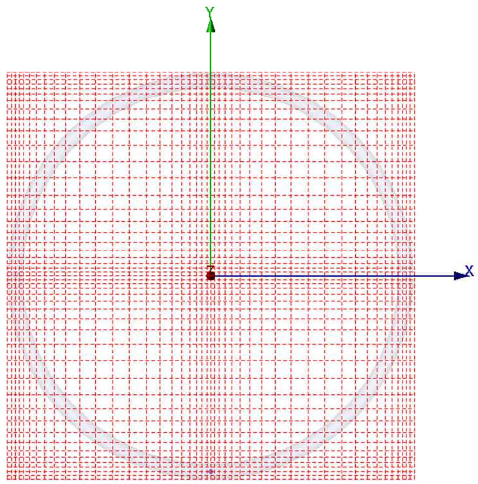
Click the "x-y" button to display the mesh in the X-Y plane.

Click the “Fit” button to fit the model to the display window.



Click the “Mesh mode” button to display the mesh distribution.



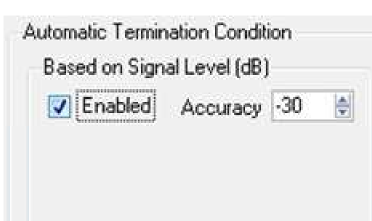


25.10 Generate Simulation File

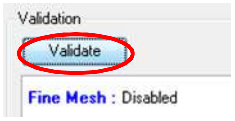
Follow the steps below to generate simulation file (*.gpv):

(1) Click the **Precalculate** button in the toolbar and then a new window will appear.

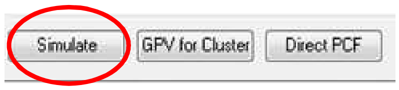
(2) Select the convergence style and level.



(3) Click the **Validate** button to validate the project settings, and pay attention on the message in the red color.

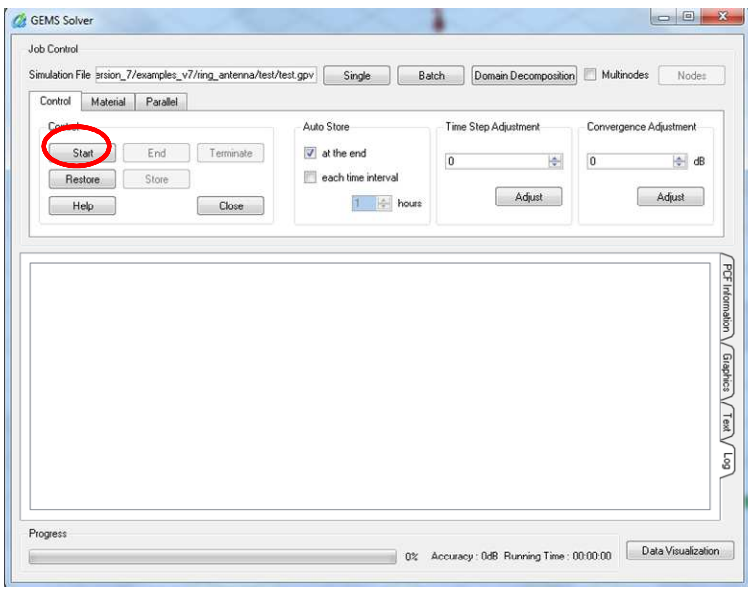


(4) Click the **Simulate** button to launch the simulation window.



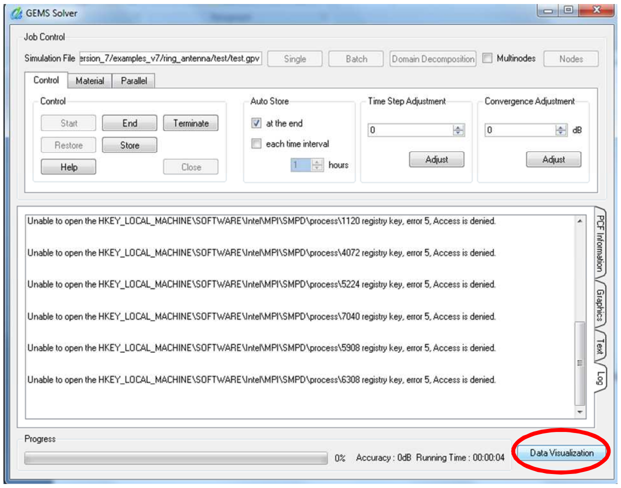
25.11 Simulate Project

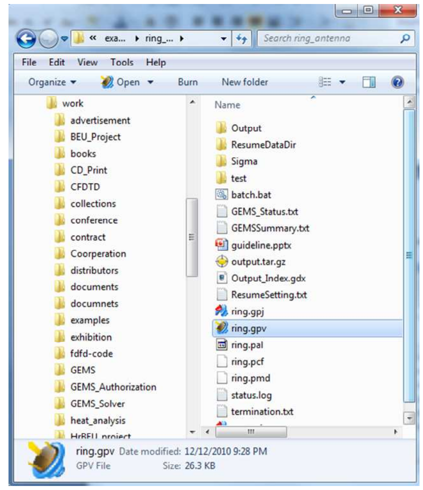
Click the **Start** button to start the simulation and the simulation progress will be shown in the **Log** window.



25.12 Result Visualization

Click the **Data Visualization** button or double-click the *.gpv file in the project folder to open the result display window.

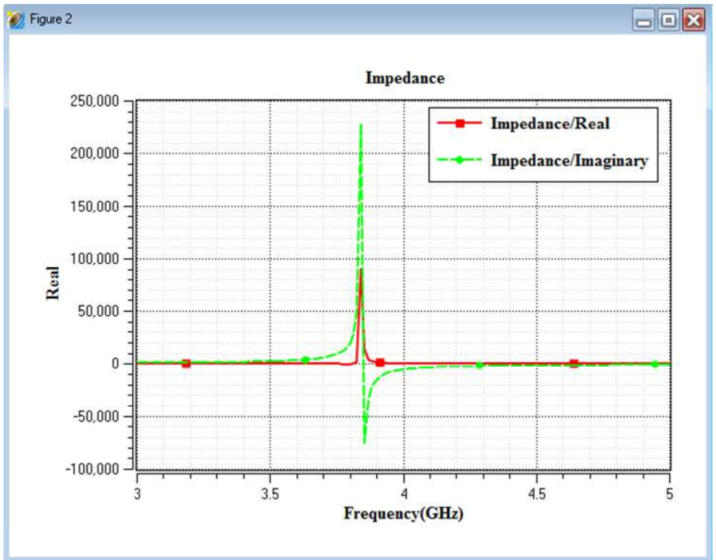


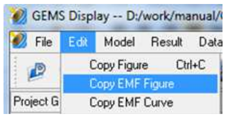


The Simulation results are listed in the Data Guide box. Double-click the **Port->Freq->Impedance->Real** option to plot the impedance inside the figure window.

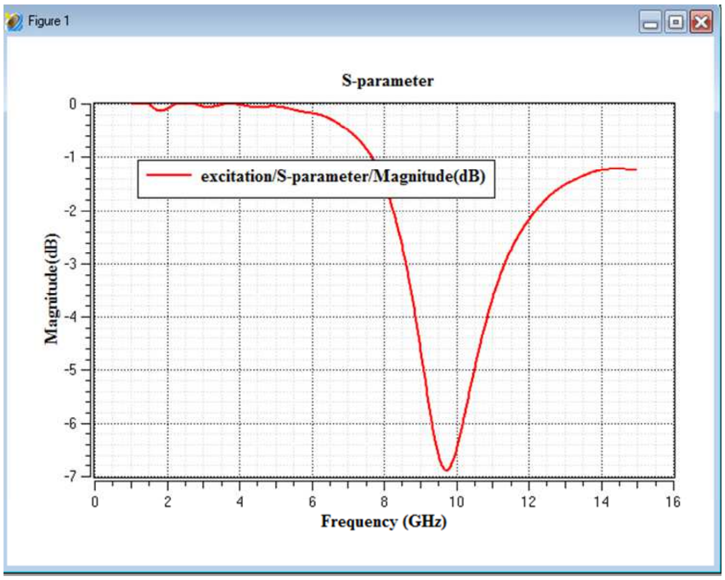
Select the **Port->Freq->Impedance->Imaginary** option and then click the **Add to current window** button in the toolbar.

Double-click on the figure title, labels, legend or on the figure margin to edit them if necessary. You can copy the figure in the vector format to other software for the editing.

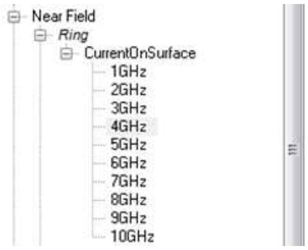




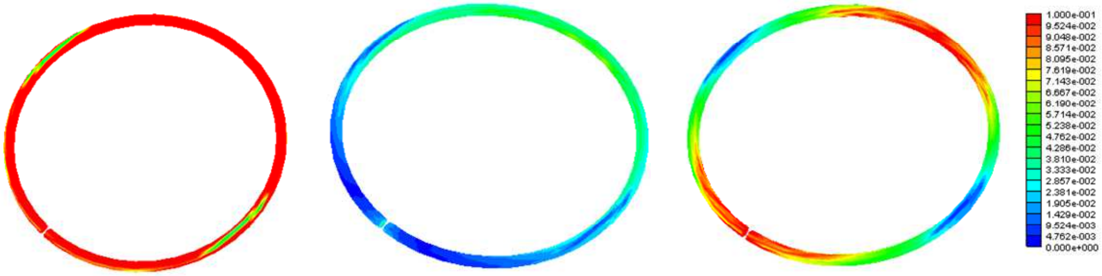
Click the **Port->Freq->S-Parameter->Magnitude** option to show the return loss.



Double-click one of the frequencies in the "**Near Field->Ring**" folder to show the surface current distribution on the ring conductor.

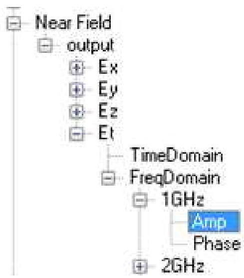


Frequency = 2GHz Frequency = 2GHz Frequency = 10GHz



Follow the steps below to plot the field distribution on the specified surface at 1GHz:

(1) Select the option in the result tree.

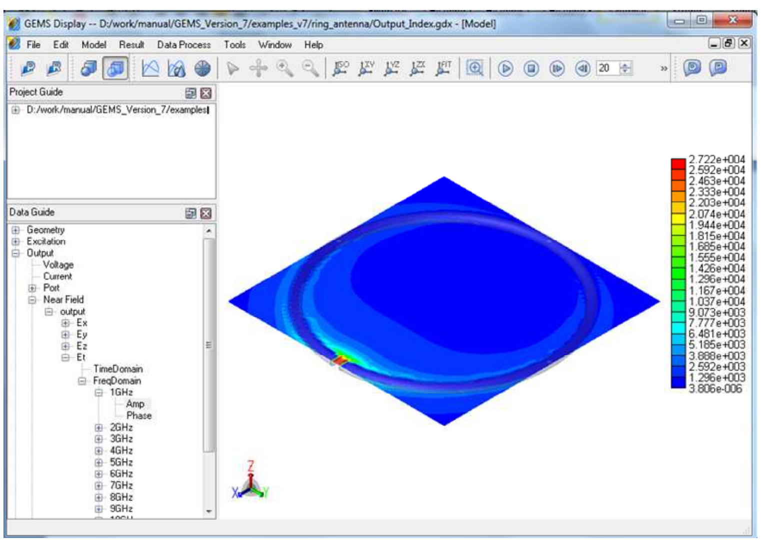


(2) Click the **View model** and then **Transparent** model buttons in the toolbar.

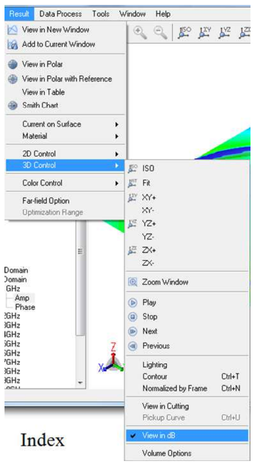
(3) Click the **Add to current window** button in the toolbar.

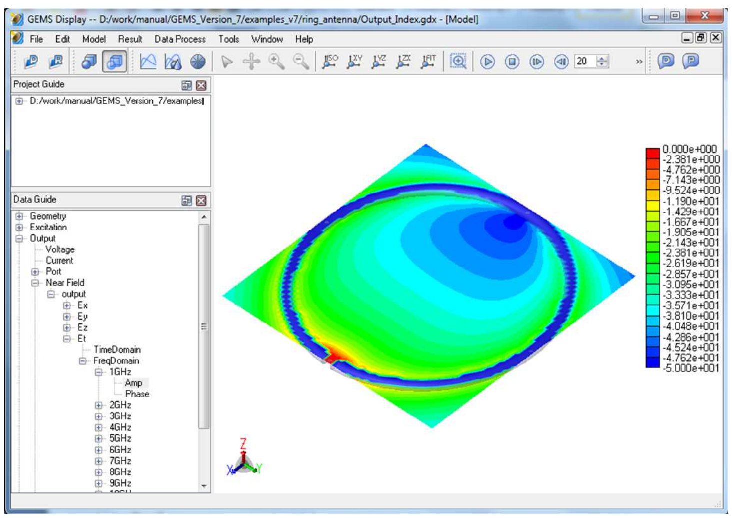
(4) Click the **Play** button in the toolbar.

(5) The field distribution appears in the figure window.

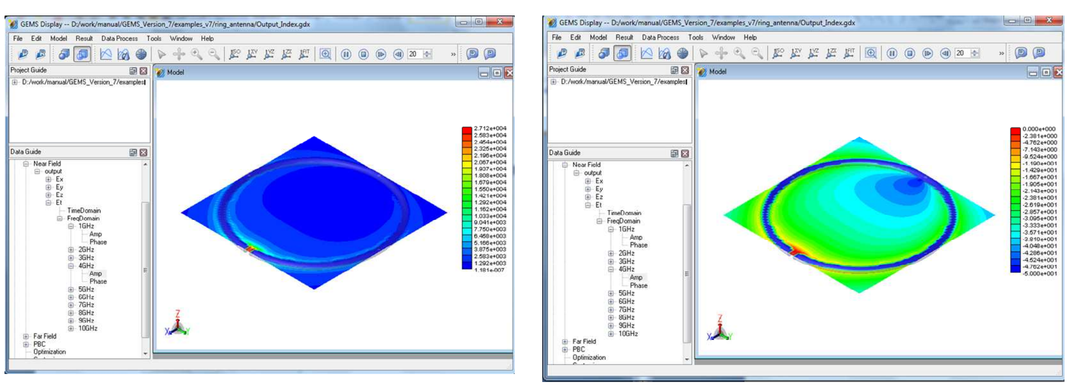


(6) Plot it in the dB scale

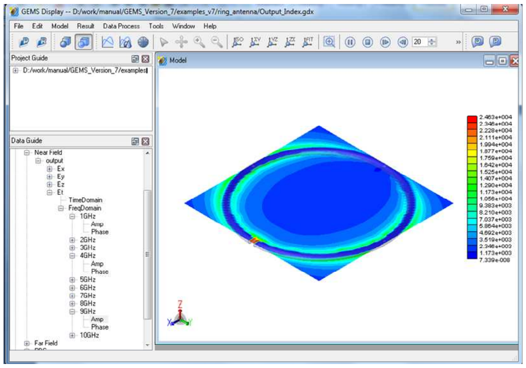


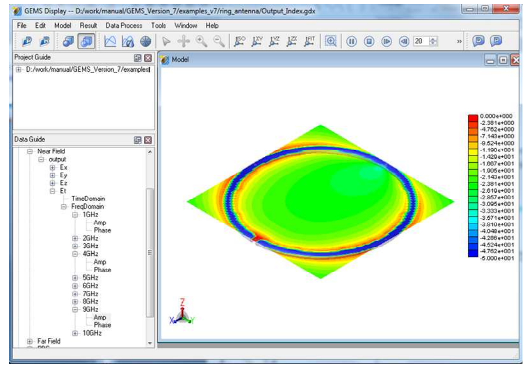


The field distribution in the linear and dB scales at 4 GHz.

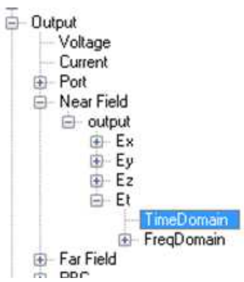


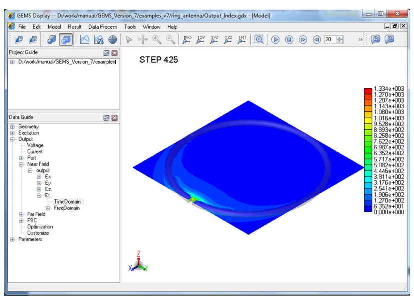
The field distribution in the linear and dB scales at 9 GHz.

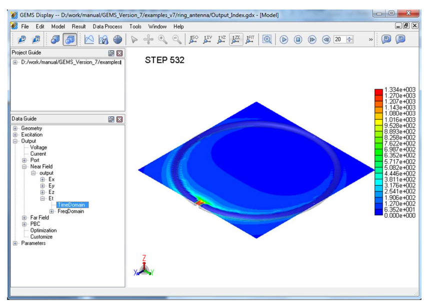


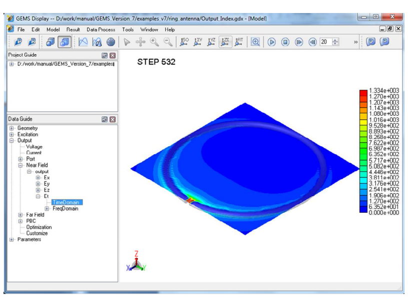


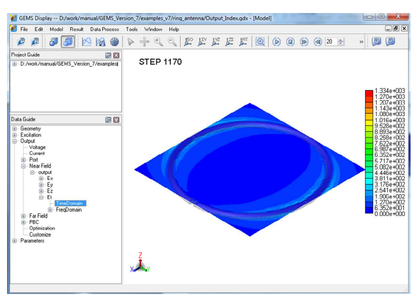
Plot the field distribution in the time domain.











26.13 Double Rings

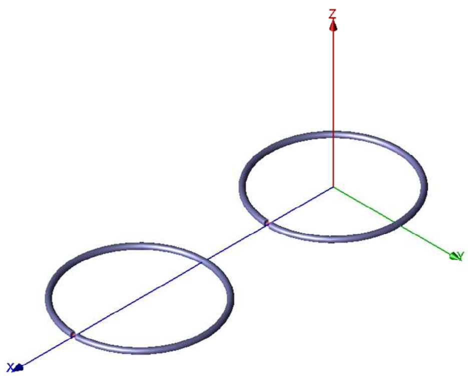
The two rings are separated by 15mm and one are excited by the lumped ports. The output parameters are impedance, S-parameters, far field, and surface current distribution on the rings. There are ways to duplicate the ring:

(1) Select the ring, excitation port and key points inside the **Object Browser** box;
(2) Select the “Move” button in the toolbar;

(3) Select one point on the ring, click the mouse left button;
(4) Move the mouse icon to desired position;
(5) Open the “Ring” folder and select the “Move” option inside the **Object Browser** box;
(6) Modify the moving distance inside the **Properties** box.

Or

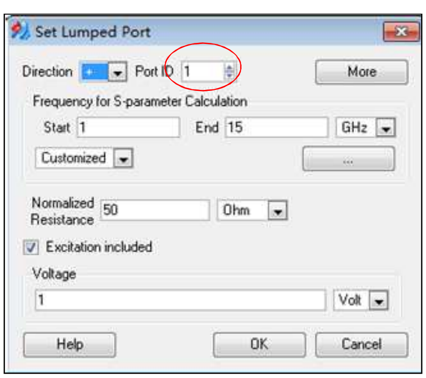
(1) Select the ring, excitation port and key points inside the **Object Browser** box;
(2) Press the “Ctrl + c” and “Ctrl + v” keys on the keyboard;
(3) Change the name of each duplicated option in the **Object Browser** box.



Modify the range of the output option so that it can cover two rings.

Select the **excitation** and then **excitaton2** options in the **Object Browser** box, and then click the **E/O** box in the **Properties** box.

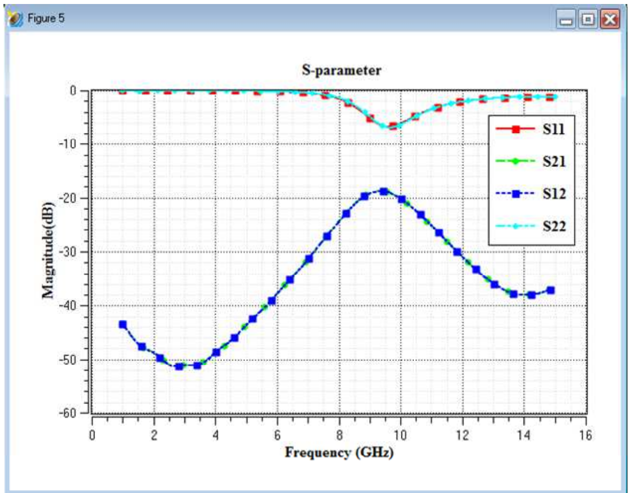
Excitation port 1



Excitation port 2

GEMS simulation will run three times: (i) The port 1 is excited and the port 2 is terminated a match load; (ii) The port 2 is excited and the port 1 is terminated a match load; (iii) Bothe ports are excited.

Follow the similar procedure to design the mesh distribution, boundary and so on. Now we show the simulation results. S-parameters are shown in the figure below.

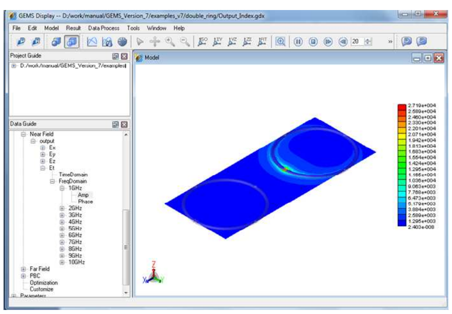


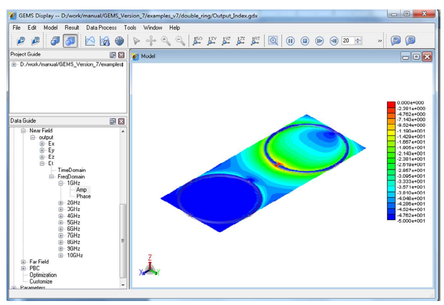
The field distributions at different frequencies.

Frequency=1GHz

Field distribution in linear scale

Field distribution in dB scale

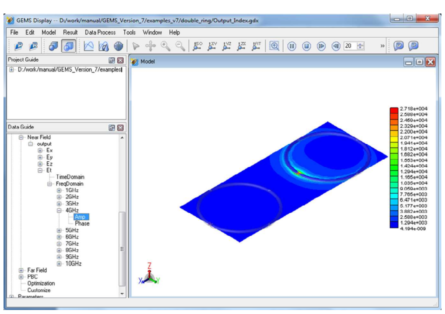


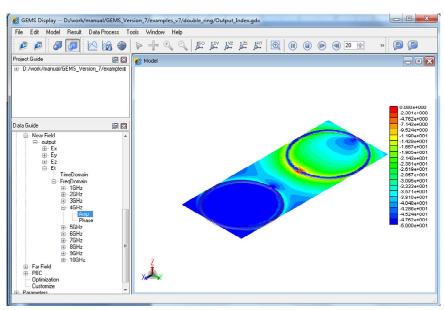


Frequency=4GHz

Field distribution in linear scale

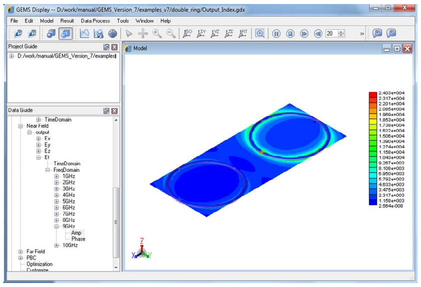
Field distribution in dB scale



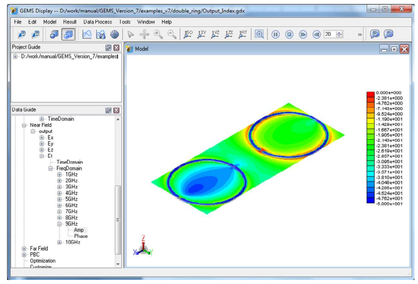


Frequency=9GHz

Field distribution in linear scale



Field distribution in dB scale



Index