LAB

Antenna CAD Software: CST Microwave Studio

Objective

The objective is to get acquainted with CAD software for antennas and particularly with *CST Microwave Studio.*

Lab-work

1. Follow the tutorial to design a microstrip patch antenna resonating at 2.45 GHz. Notice that you are using a new version of the software and some options may be a little different. In particular:
   - After creating the coaxial feeding the software detects the intersection between the coaxial and the ground plane. You will be prompted with a new "Shape intersection" menu. Choose "cut away highlighted shape". This will remove the part of the ground plane that intersects with the coaxial feeding.
   - The solver you have to choose is the "Time domain solver".

2. Once you have followed the tutorial instructions modify the patch geometry to obtain the resonant frequency of 2.45 GHz and an input impedance of 50Ω at that frequency.
   - The minimum return losses are obtained at frequency below 2.4 GHz. Use the marker functions (double click on the curve) to obtain that frequency.
   - Modify the dimension of the patch length to move that minimum to 2.45 GHz. HINT: you can use math expressions in the parameter list. The change is proportional to the ratio between frequencies.
   - Once you have obtained the desired frequency watch the S11 into the Smith Chart (the display options are on the top right). Obtain the value of the impedance at 2.45 GHz (double click on the curve and move the marker).
   - The real part of the impedance is not 50Ω. You can change it by moving the feeding point. Moving it to the center lowers the impedance, moving it outwards increases the impedance. Modify the variable CoaxPos to obtain a 50Ω impedance.
   - Estimate the antenna Q by measuring the 10 dB Return Loss fractional bandwidth.
3. Now double the height of the patch antenna and readjust it and obtain the Q. What can you say about the dependence of the Q and the patch height?

LAB

Design of a dual band compact antenna

Objective

Once you have mastered the use of an antenna CAD software it is time you apply all your knowledge to the design of a real antenna. In this lab sessions you will have to choose between these two designs:

- A dual band PIFA antenna to cover the 2.45 and 5 GHz WIFI band [1].
- A dual band GSM/DCS/PCS/UMTS printed antenna [2].

Pre- Work

- Select the design you want to do. In case you want to do an alternative design check with your instructor.
- Identify the frequency bands that your antenna design must cover to support the wireless systems they are intended for.

Lab-work

Use CST studio to analyze your antenna design. Follow these guidelines :

- You will use as a coaxial excitation.
- In the solver settings set the accuracy to – 40 dB.
- Follow the guidelines given in the paper and try to adjust the antenna response to the design objectives.
- For the final design specify the bandwidth and directivity.
- One final aspect is how the manufacturing tolerances will affect the antenna response. Introduce small variations in the antenna dimensions and identify the most critical dimensions and try to give manufacturing tolerance for them.

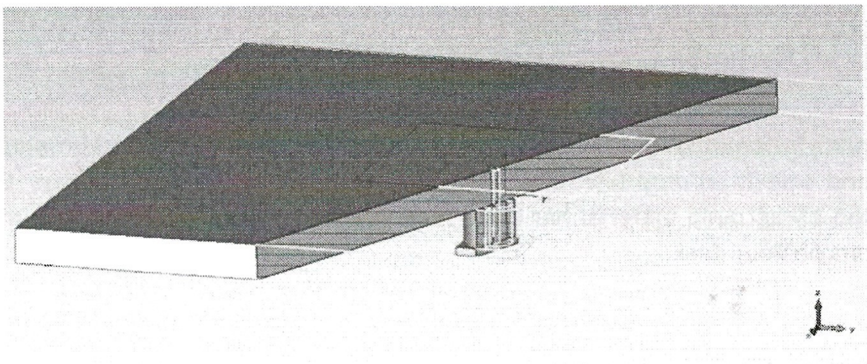
[1] "P.Salonen et al.,Single-Feed Dual-Band Planar Inverted F Antenna with U-shaped Slot", IEEE Transactions on Antennas and Propagation, vol. 48. No. 8, August 2000" .

[2] C. H. Ku, H. W. Liu and S. Y. Lin, "Folded Dual-Loop Antenna for GSM/DCS/PCS/UMTS Mobile Handset Applications," in *IEEE Antennas and Wireless Propagation Letters*, vol. 9, no. , pp. 998-1001, 2010. doi: 10.1109/LAWP.2010.2089037

CST Microwave Studio
Patch Antenna Tutorial

In this tutorial you will learn how to simulate basic antenna devices. We will design and simulated one of the most widely used antenna architectures, the rectangular patch antenna. The following explanations can be applied to other antennas as well.

Although, CST MICROWAVE STUDIO® can provide a wide variety of results, this tutorial will concentrate mainly on the S-parameters and farfield results.

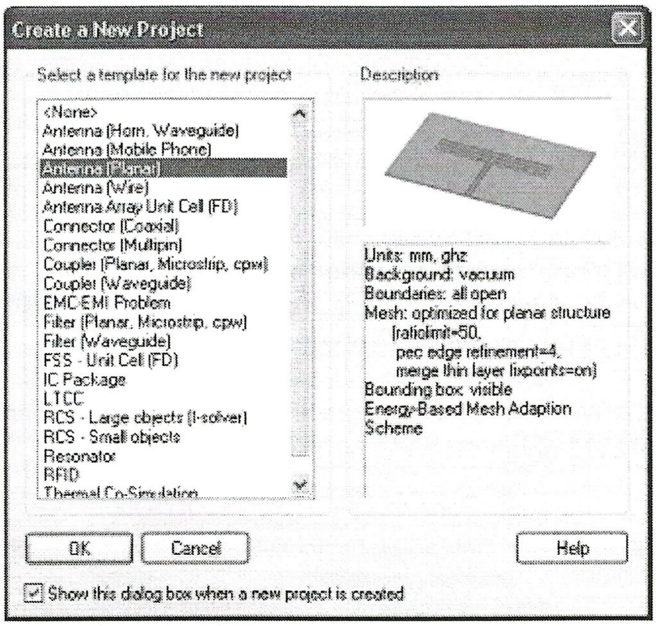


Index

1. Project and Template

After you have started CST DESIGN ENVIRONMENT™ and have chosen to create a new CST MICROWAVE STUDIO® project, you are requested to select a template that best fits your current device. Here, the Antenna (on Planar Substrate) template should be chosen. If the following dialog box does not occur automatically, select *File ⇨Select template...* from the main menu.

This document contains parts, sometimes verbatim, from the original antenna tutorial included in CST help documentation



This template automatically sets the units to mm and GHz, defines the background material to vacuum and selects appropriate boundary conditions (see chapter Boundary Conditions). Because the background material has been set to vacuum, the structure can be modeled just as it appears on your desk.

2. Geometric Model

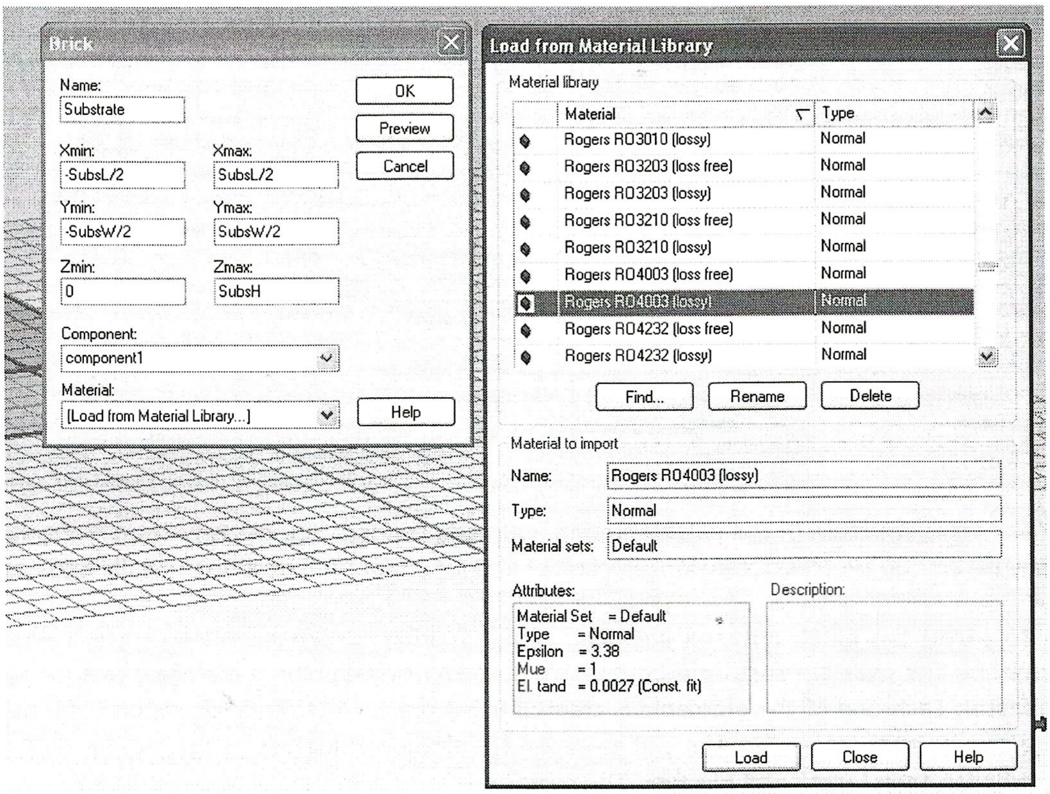
Now, we start creating the 3D model of the patch antenna. One of the most powerful characteristics of CST it that the 3D models are parametric. This is particularly useful when optimizing the antenna since it is enough to change the value of these parameters to change the dimensions of the antenna model.

Lets define our first three parameters. These parameters will determine the length, width and heigth of the antenna substrate. They can be directly introduced in the *Parameter List* located on the lower part of the screen. If this list is not visible, it can be displayed at *View ⇨ Parameter list.* Remeber that the default length units for this template are 'mm'. You can see the unit default values in the lower-left corner of the screen.

Name	Value	Description	Type
SubsH	1.5		Undefined
SubsL	100		Undefined
SubsW	100		Undefined

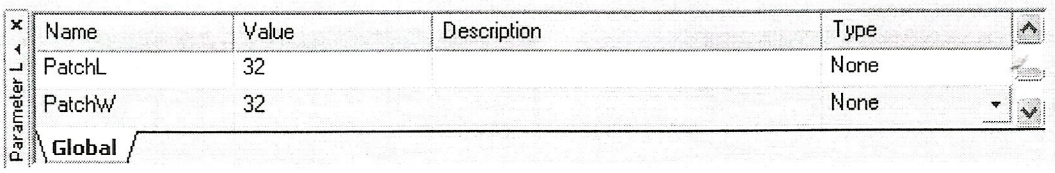
Parameter List — Global

Now we can create the antenna **substrate** using the brick creation function (*Objects ⇨ Basic Shapes ⇨ Brick,*). Double-click in the design area to define the corners of the brick. Immediately a new window will appear where you can specify the position and dimensions of the brick.



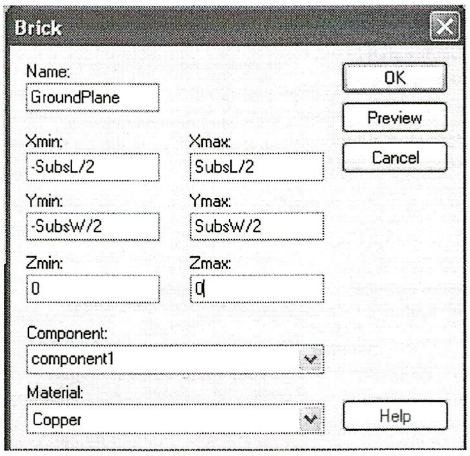
We specify the name of the brick: 'Substrate'. The coordinates of the brick can be specified using either numerical dimensions or parameters. In this case, we will use the previously defined parameters. The material can be directly selected from a *Material Library.* We will use a standard substrate *Rogers RO4003(lossy).*

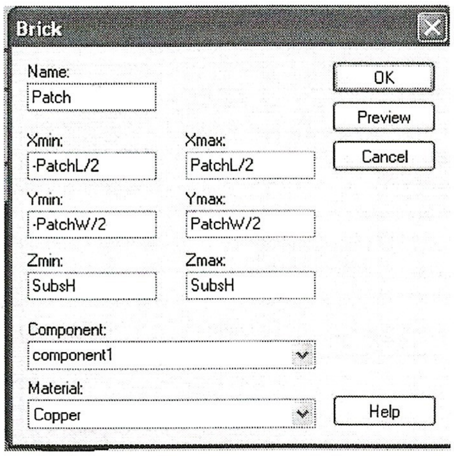
We can follow the same process to include the **ground plane** and the **patch antenna**. For the patch antenna, we will need to additional parameters to define its length and width:



Notice that 32mm corresponds to half a wavelength at **2.5 GHz** considering the RO4003 permitivity.

We'll choose equal Zmin and Zmax and CST will model the brick as a 2D surface. Due to the small height of copper metallization (35um typically) a 2D modelling is accurate and faster than simulating a 3D model. The material used in this case will be *copper.*

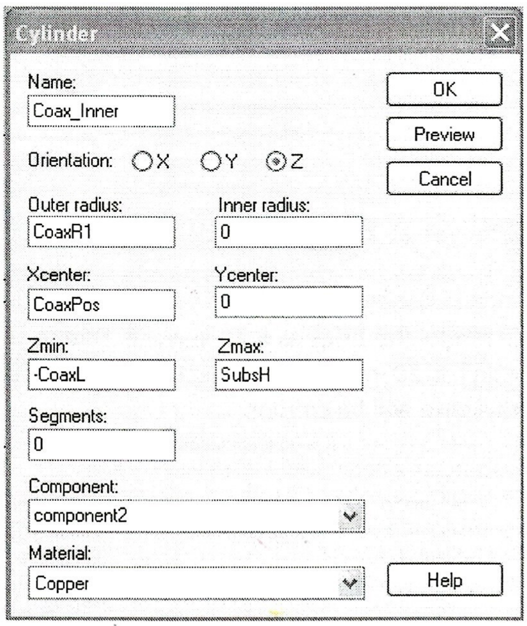


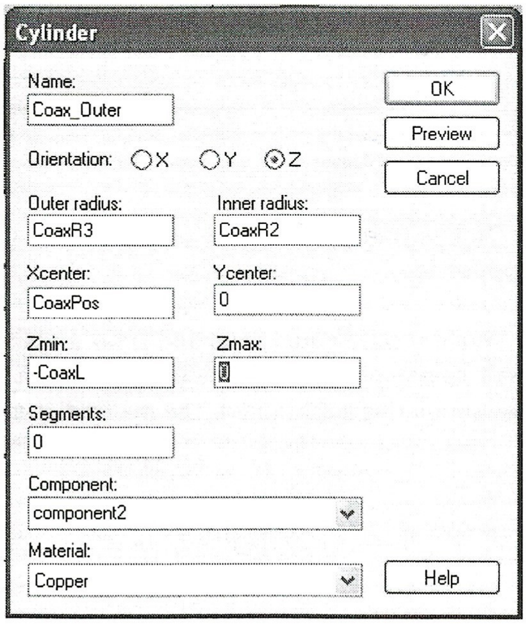


The basic structure of the patch antenna is already created. You can now modify the value of any parameter (*PatchL*, for instance), click *F7* and the model will be automatically updated.

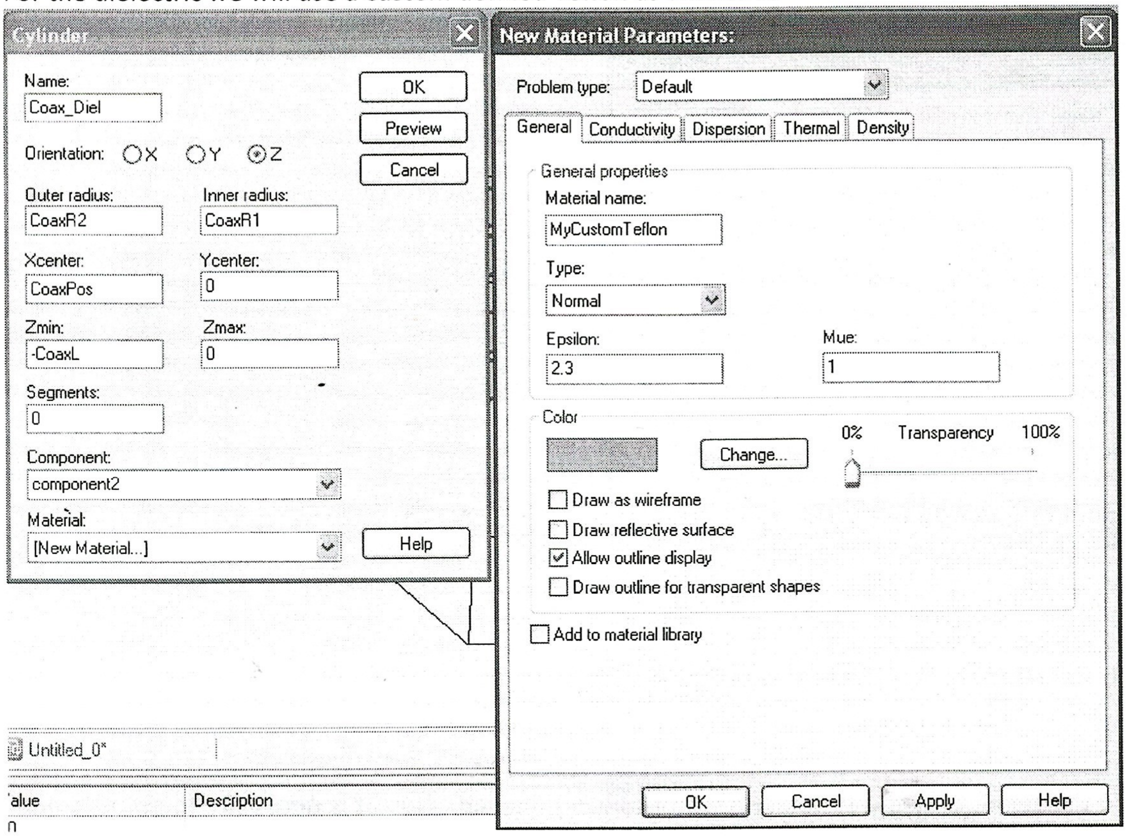
To excite the patch antenna we need also a feeding structure. We will use a **coaxial excitation.** The coaxial structure is composed of an inner conductor, a dielectric and an outer conductor. Each one of the elements is created using the cylinder creation mode (*Objects* ⇨ *Basic Shapes* ⇨ *Cylinder*,). We will need the following parameters to specify the radius of the cylinder, their length and position. The correspond to a commercial SMA connetor.

Name	Value	Description
CoaxBack	1	
CoaxL	5	
CoaxPos	5	
CoaxR1	0.635	
CoaxR2	2.3	
CoaxR3	4	

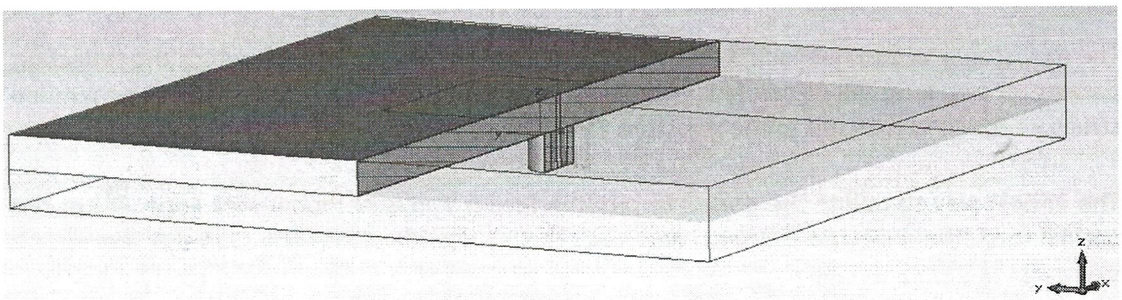




For the dielectric we will use a custom defined Teflon as dielectric:

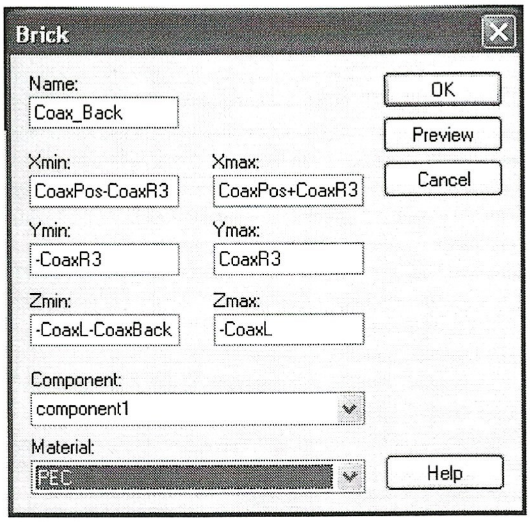


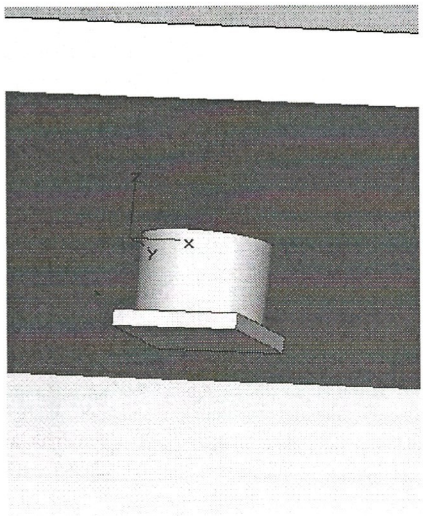
After creating the coaxial feeding the antenna model should look like the following:



To finish the 3D model, the last thing to do consists on removing the part of the ground plane that is shortcircuiting the coaxial feed. Select the ground plane and click *Objects ⇨ Boolean ⇨Insert*). Then select the 3 elements of the coaxial feed (inner conductor, dielectric and outer conductor) and press enter. Now the ground plane has a circular hole to prevent blocking the RF signal travelling from the coaxial and the patch antenna model is complete.

Some solvers require that back of the RF-ports is covered by a block of Perfect Electric Conductor PEC. This can be easily done with the brick tool.





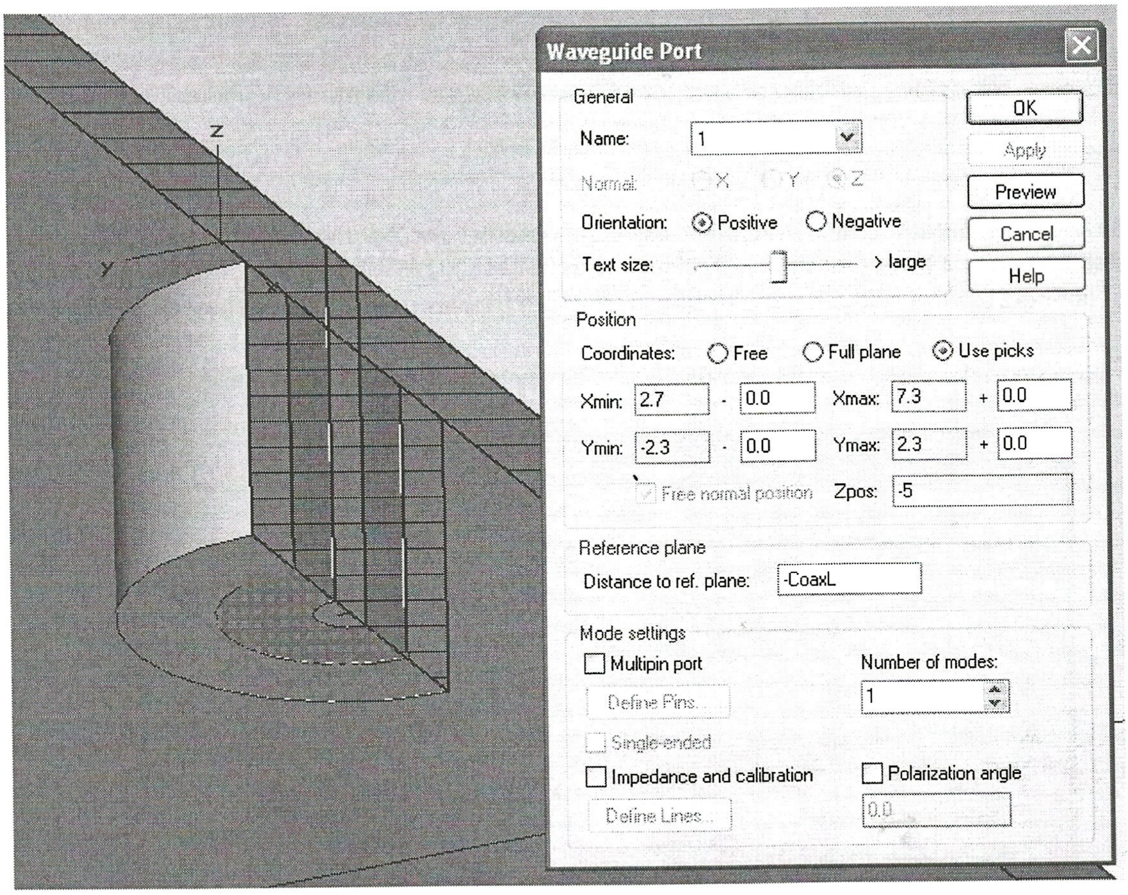
3. Excitation, frequency range, boundaries and monitors.

To this point, only the structure itself has been modeled. Now it is necessary to define some solver-specific elements. For an S-parameter calculation you must define input and output ports. Furthermore, the simulation needs to know how the calculation domain should be terminated at its bounds.

Excitation

The next step is to add the excitation port to the patch antenna device, for which the reflection parameter will later be calculated. The port simulates an infinitely long coaxial waveguide structure that is connected to the structure at the ports plane.

The easiest way to define the port is to pick the face (*Objects ⇨Pick ⇨Pick Face,*) of the coaxial feed (the dielectric element) and open the waveguide dialog box (*Solve ⇨Waveguide Ports,*) to define the port:



Here, you have to choose how many modes should be considered by the port. For a simple coax port with only one inner conductor, usually only the fundamental TEM mode is of interest. Thus, you should simply keep the default setting of one mode.

We can also specify the position of the reference plane. In this case, we want the reference plane to be located at the antenna ground plane and therefore the distance to the ref. plane is *–CoaxL*. The position of this ground plane basically affects the phase of the reflection coefficient.

Frequency range

In this example, the S-parameters are to be calculated for a frequency range between 2 and 3 GHz. Open the frequency range dialog box (*Solve ⇨ Frequency*,) and enter the range from 2 to 3 (GHz) before pressing the *OK* button (the frequency unit has previously been set to GHz by the selected template and is displayed in the status bar).

Boundaries

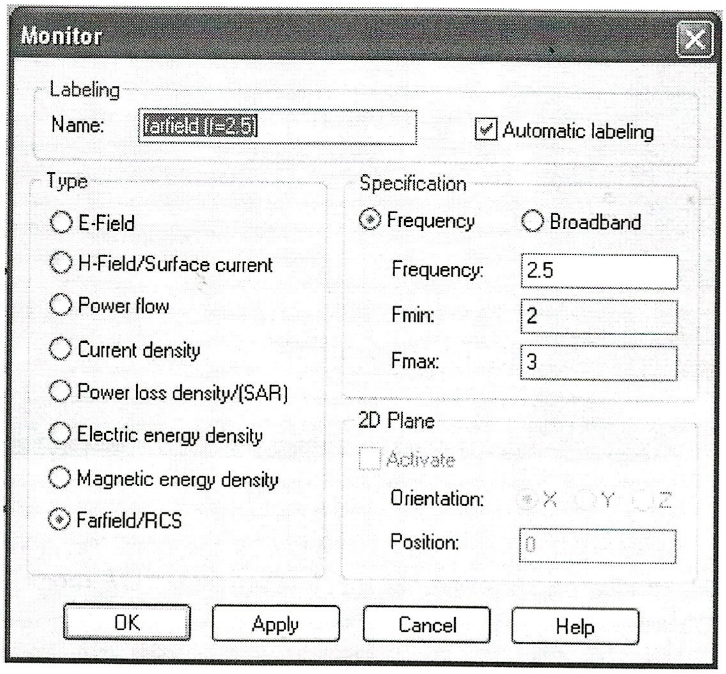
Because the calculation domain is only a limited volume it is necessary to define boundary conditions that incorporate the influences of the outer space. Please open the Boundary Conditions dialog box by selecting *Solve ⇨ Boundary Conditions* ().

In this window, different boundary conditions such as electric, magnetic or periodic boundaries can be defined, as well as symmetry planes that can speed up the simulation. In this case, the default options are used (open boundaries and no symmetry planes).

Monitors

Besides the S-parameters, the main result of interest for antenna devices is the farfield distribution at a given frequency. The solvers in CST MICROWAVE STUDIO® offer the possibility of defining several field monitors to specify at which frequencies the field data shall be stored.

Please open the monitor definition dialog box by selecting *Solve ⇨ Field Monitors* ():



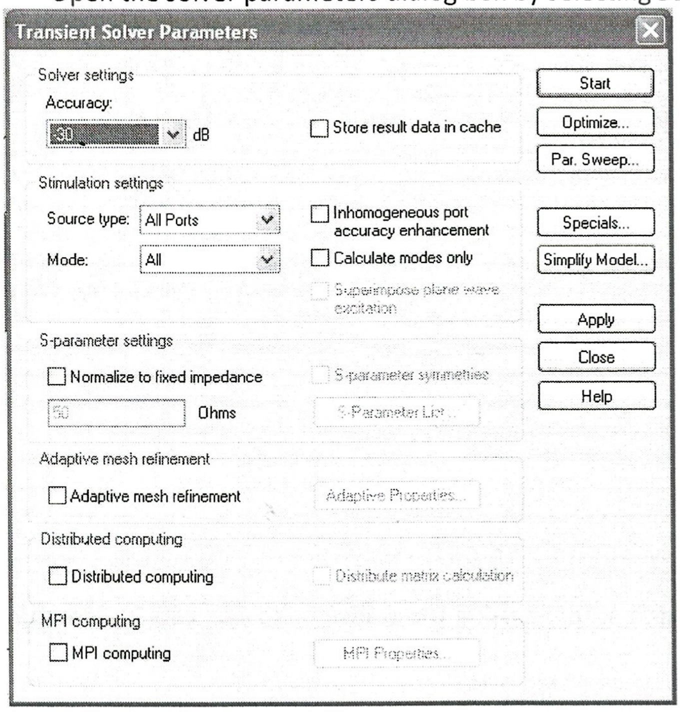
In this dialog box, you should first select the Type *Farfield/RCS* before specifying the frequency for this monitor in the *Frequency* field. Afterwards, press the *Apply* button to store the monitors data. Please define a monitor at the frequency of 2.4 (with GHz being the currently active frequency unit). However, you may define additional monitors at other frequencies, each time pressing the *Apply* button to confirm the setting and add the monitor in the *Monitors* folder in the navigation tree. After the monitor definition is complete, please close this dialog box by pressing the *OK* button.

4. Solver

CST MICROWAVE STUDIO® has several solvers. The performance of each solver depends on the type of antenna problem (electrical size, presence of dielectrics, bandwidth...) .
For this patch antenna we will use the **transient solver**. This is a very flexible time domain simulation module that is capable of solving any kind of S-parameter or antenna problem. It will stimulate the structure at a previously defined port using a broadband signal. Broadband

stimulation enables you to receive the S-parameters for your entire desired frequency range and, optionally, the electromagnetic field patterns at various desired frequencies from only one calculation run.

Open the solver parameters dialog box by selecting *Solve ⇨ Transient solver ( ).*



We recommend using reasonably large bandwidths of 20% to 100% for the transient simulation. In this example, the S-parameters are to be calculated for a frequency range between 2 and 3 GHz. With a center frequency of 2.5 GHz, the bandwidth (3 GHz - 2 GHz = 1 GHz) is 40% of the center frequency, which is inside the recommended interval. Thus, you can simply choose the frequency range as desired between 2 and 3 GHz.

The accuracy of the method is basically determined by two factors:

(1) The accuracy in the processing of the time signals. It is basically determined by the duration of the time simulation and is specified by the *Accuracy [dB].*

(2) The quality of the mesh. The finer the mesh, the more accurate will be the resuly, but the longer the simulation time will be. For an accurate simulation, it is convenient to turn on the *adaptive mesh refinement* option. This way the mesh will be automatically refined until converge is reached.

In this example we use the default values which provide a good accuracy for this simple structure:

Accuracy = -30 dB Adaptive mesh = OFF

The simulation of this structure using the previous solver parameters should take 1 or 2 minutes.

Transient Analysis: Processing excitation,

During the simulation, the message window will display additional information:

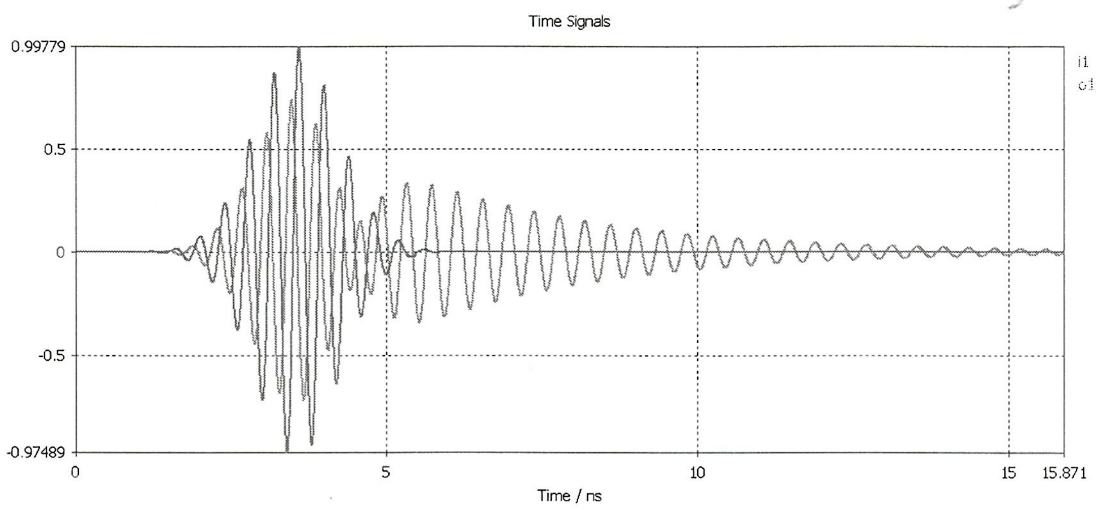
```
Starting adaptive port meshing.
Mode calculation for port 1:
Pass 1 successfully completed.
Pass 2 successfully completed. Delta (Line Imp.): 5.449897%
Pass 3 successfully completed. Delta (Line Imp.): 5.030836%
Pass 4 successfully completed. Delta (Line Imp.): 0.402989%
Adaptive port meshing finished.
Solver Settings:
Excited port mode: 1 (1)
Excitation duration: 7.109097e+000 ns
Steady state accuracy limit: -30 dB
Maximum number of time steps: 94640
Time step width:
    without subcycles: 1.502335e-003 ns
    used: 1.502335e-003 ns
Number of threads used: 2
```

5. Results

You have simulated the rectangular patch antenna using the transient solver. Lets review the results.

Port Signals

First, observe the port signals. Open the *1D Results* folder in the navigation tree and click on the *Port signals* folder. It is possible to observe the progress of the results during the computation. In order to get the complete information, however, wait until the solver has finished.

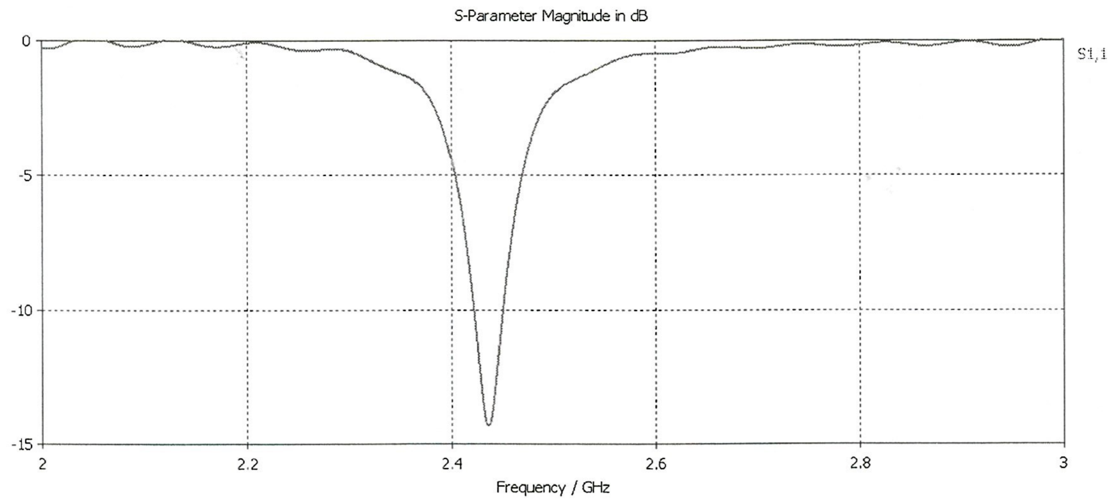


This plot shows the incident and reflected wave amplitudes at the waveguide port versus time. The incident wave amplitude is called i1 (referring to the port name: 1) and the reflected wave amplitude is o1,1. As evident from the above time-signal plot, the patch antenna has a strong resonance that leads to a slowly decreasing output signal.

By comparing the Fourier the transform of these two signals, the S-parameters of the antenna can be computed.

S-parameters

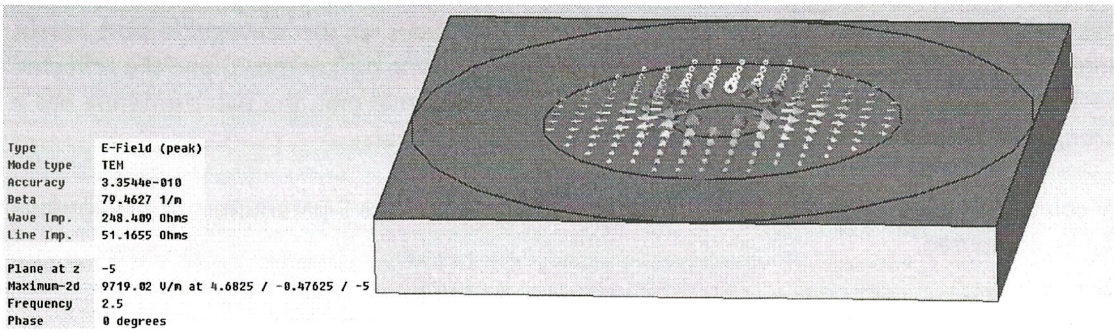
A primary result for the antenna is the S11 parameter that will appear if you click on the *1D Results* ⇨ *|S| dB* folder from the navigation tree. The following screenshot shows the reflection parameter:



It is possible to precisely determine the operational frequency for the patch antenna. Activate the axis marker by pressing the right mouse button and selecting the *Show Axis Marker* option from the context menu. Now you can move the marker to the S11 minimum and pinpoint a resonance frequency for the patch antenna of about 2.45 GHz.

Port Field

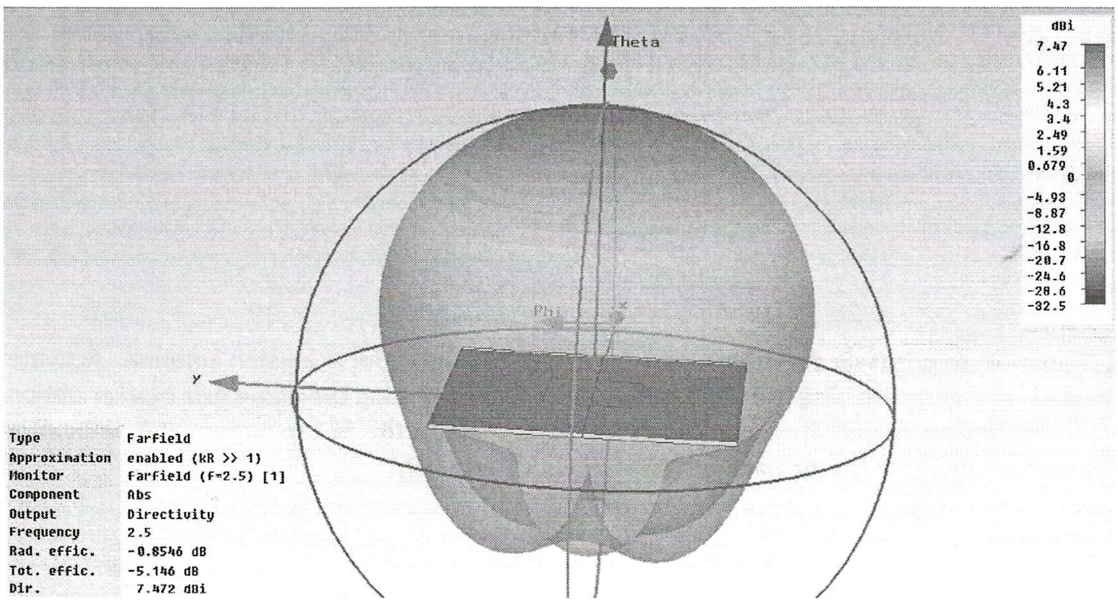
It always important to check the E-fields on the port. You can easily display this field by opening the *2D/3D Results* ⇨*Port Modes* ⇨*Port1* folder from the navigation tree. To visualize the electric field of the port mode, please click on the *e1* folder.



In this case, as expected, the field corresponds to the dominant mode in a coaxial waveguide. The plot also shows some important properties of the mode such as mode type, propagation constant and line impedance. Here, you can see that the port impedance is approximately 50 Ohms, which is agrees with the specifications of the SMA connector.

Radiation pattern

The farfield is another important parameter in antenna design. The farfield solution of the antenna device can be shown by selecting the corresponding monitor entry in the *Farfields* folder from the navigation tree. For example, the farfield at the frequency 2.4 GHz can be visualized by clicking on the *Farfields ⇨farfield (f=2.4) [1]* entry, showing the directivity over the phi and theta angle (the position of the color ramp can be modified by clicking on the *Results ⇨Plot Properties ⇨Color ramp* button):



You can see that the maximum power is radiated in the positive z-direction. Note that there are several other options available to plot a farfield: the Polar plot, the Cartesian plot and the 2D plot.

You can also obtain the maximum directivity, which is 7.5dB in this case, as well as the radiation efficiency of the structure, -0.85dB (82%).