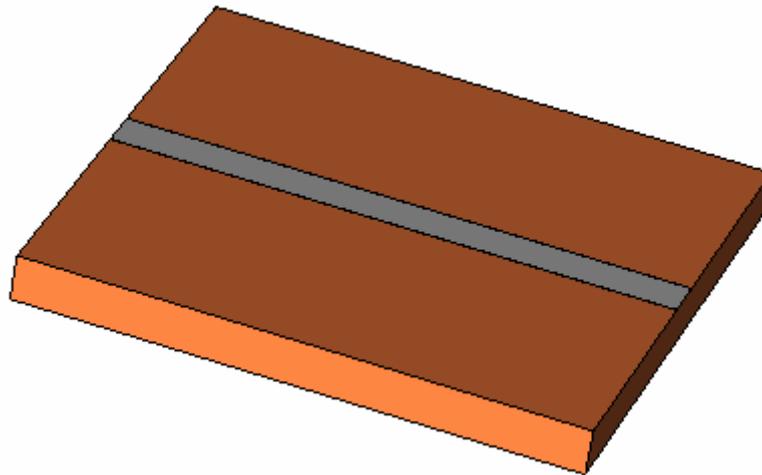


# ELE3310 Basic Electromagnetics

## Lab Session 1

Gao Xin

By modifying CST MICROWAVE STUDIO® 2006 tutorials

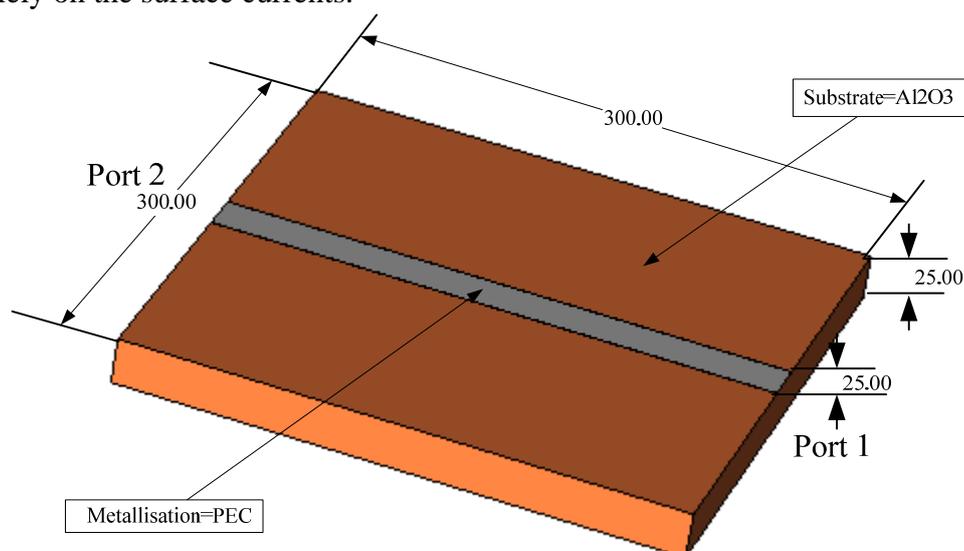


## Geometric Construction and Solver Settings

### Introduction and Model Dimensions

In this tutorial you will learn how to simulate planar devices. As a typical example for a planar device, you will analyze a Microstrip Line. The following explanations on how to model and analyze this device can be applied to other planar devices, as well.

CST MICROWAVE STUDIO® can provide a wide variety of results. This tutorial, however, concentrates solely on the surface currents.



All dimensions are given in milli-inches(mil). Thickness of the metallisation is 0.118 mil.

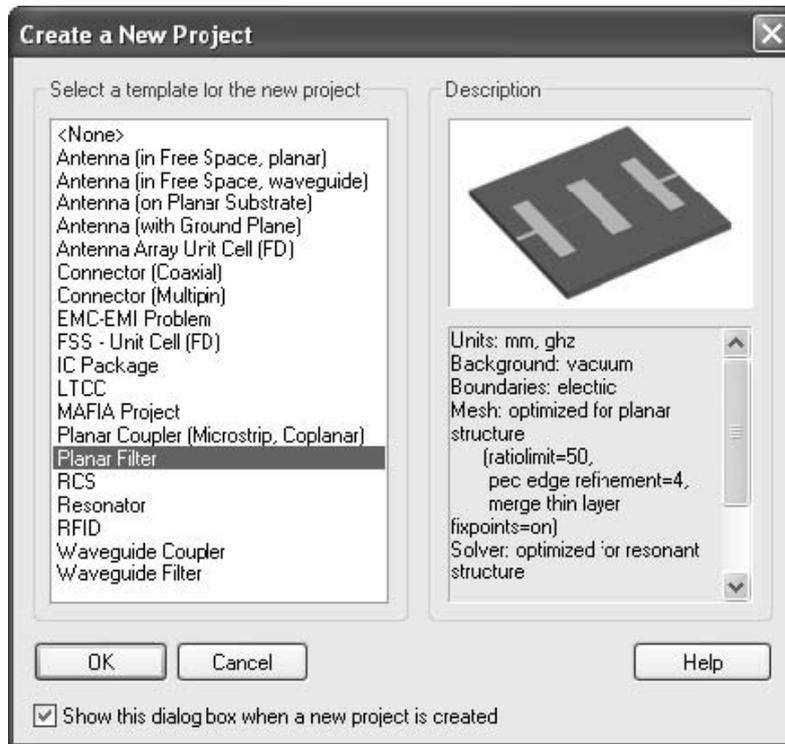
The structure depicted above consists of two different materials: The aluminum oxide substrate ( $\text{Al}_2\text{O}_3$ ) and the stripline metallization. There is no need to model the ground plane since it can easily be described using a perfect electric boundary condition.

# Geometric Construction Steps

The tutorial will take you step by step through the construction of your model, and relevant screen shots will be provided so that you can double-check your entries along the way.

- Select a Template

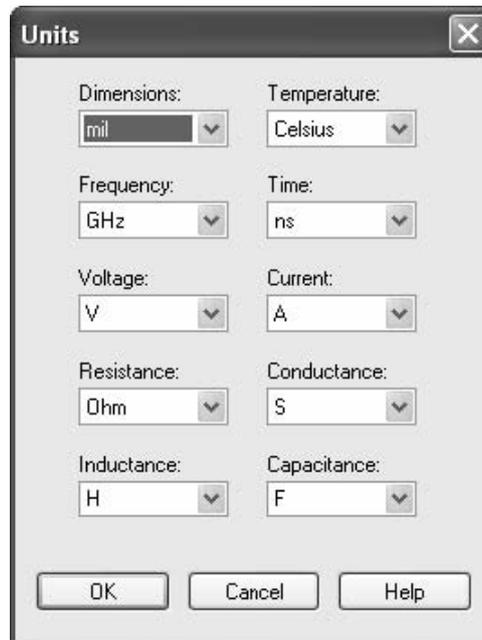
Once you have started CST DESIGN ENVIRONMENT™ and have chosen to create a new CST MICROWAVE STUDIO® project, you are requested to select a template that best fits your current device. Here, the “Planar Filter” template should be selected.



This template automatically sets the units to mm and GHz, the background material to vacuum and all boundaries to perfect electrical conductors. Because the background material has been set to vacuum, the structure can be modeled just as it appears. Furthermore, the automatic mesh strategy is optimized for planar structures and the solver settings are adjusted to resonant behavior.

- Set the Units

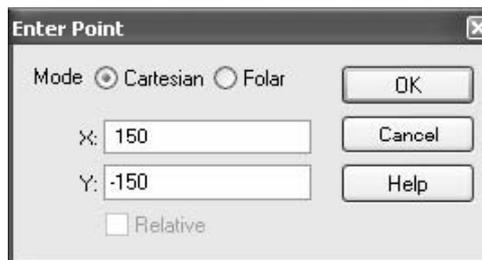
As mentioned, the template has automatically set the geometrical units to mm. However, since all geometrical dimensions are given in mil for this example, you should change this setting manually. Therefore, open the units dialog box by selecting *Solve -> Units* from the main menu:



- Draw the Substrate Brick

The first construction step for modeling a planar structure is usually to define the substrate layer. This can be easily achieved by creating a brick made of the substrate's material. Please activate the brick creation mode (*Objects -> Basic Shapes -> Brick*, ) .

When you are prompted to define the first point, you can enter the coordinates numerically by pressing the *Tab* key that will open the following dialog box:

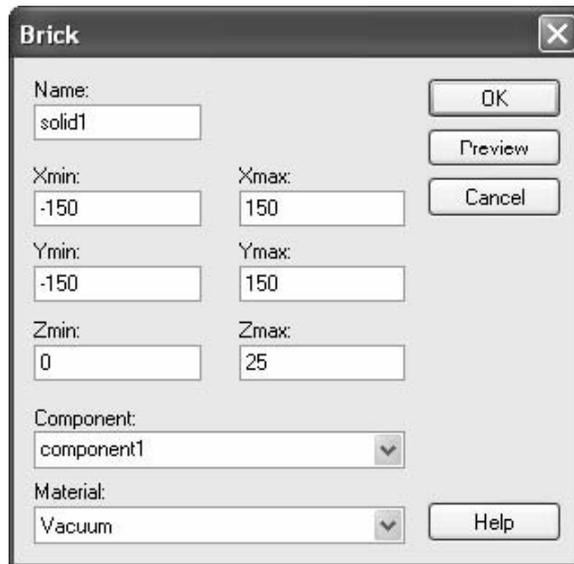


In this example, you should enter a substrate block that has an extension of 300 mil in each of the transversal directions. The transversal coordinates can thus be described by  $X = -150$ ,  $Y = -150$  for the first corner and  $X = 150$ ,  $Y = 150$  for the opposite corner, assuming that the brick is modeled symmetrically to the origin. Please enter the first point's coordinates  $X = -150$  and  $Y = -150$  in the dialog box and press the *OK* button.

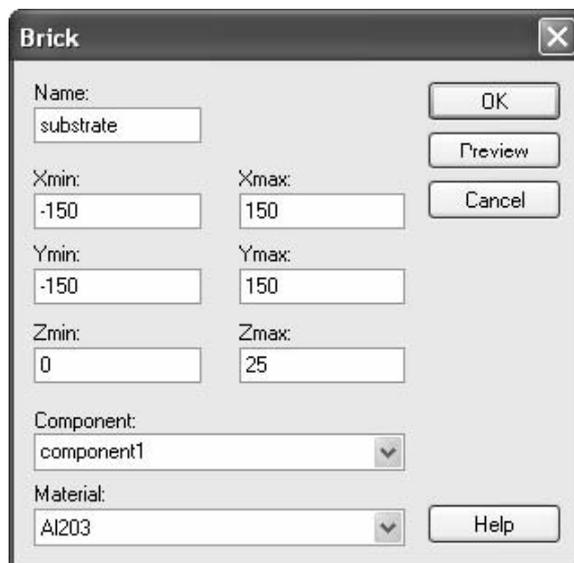
You can repeat these steps for the second point:

- 1 Press the *Tab* key
- 2 Enter  $X = 150$ ,  $Y = 150$  in the dialog box and press *OK*.

Now you will be requested to enter the height of the brick. This can also be numerically specified by pressing the *Tab* key again, entering the *Height* of 25 and pressing the *OK* button. Now the following dialog box will appear showing you a summary of your previous input:



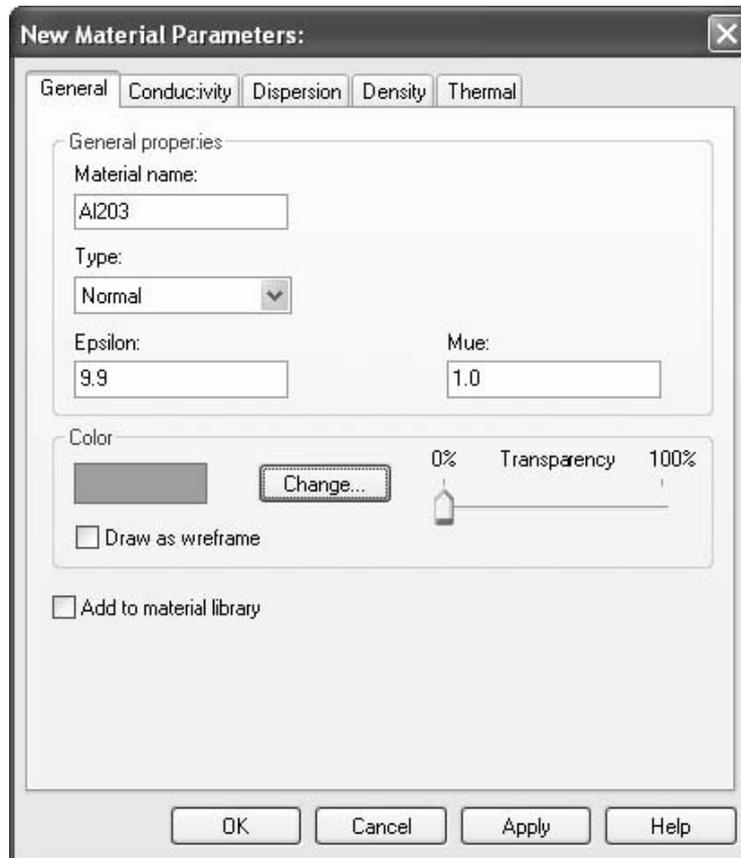
Please check all these settings carefully. If you encounter any mistake, please change the value in the corresponding entry field.



You should now assign a meaningful name to the brick by entering e.g. “substrate” in the *Name* field. Since the brick is the first object you have modeled thus far, you can keep the default settings for the first *Component* (“component1”).

**Please note:** The use of different components allows you to combine several solids into specific groups, independent of their material behavior.

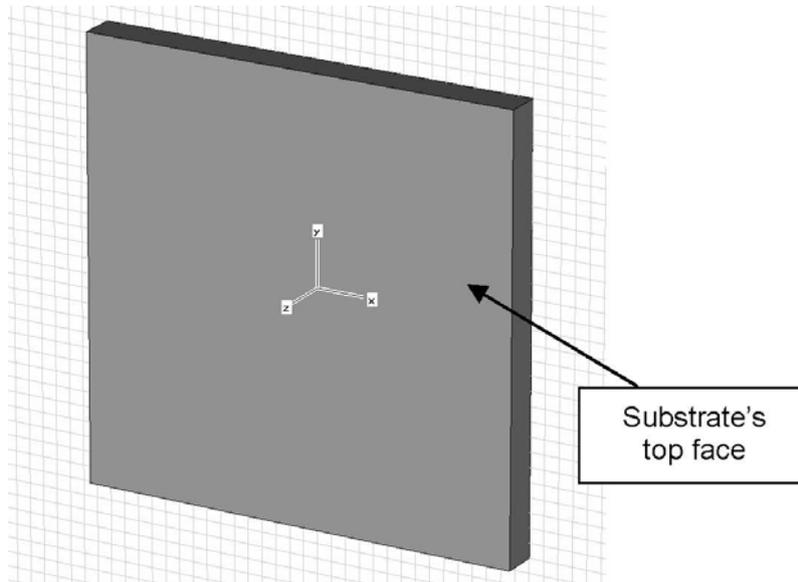
The *Material* setting of the brick must be changed to the desired substrate material. Because no material has yet been defined for the substrate, you should open the layer definition dialog box by selecting “[New Material...]” from the *Material* dropdown list:



In this dialog box you should define a new *Material* name (e.g. Al<sub>2</sub>O<sub>3</sub>) and set the *Type* to a *Normal* dielectric material. Afterwards, specify the material properties in the *Epsilon* and *Mue* fields. Here, you only need to change the dielectric constant *Epsilon* to 9.9. Finally, choose a color for the material by pressing the *Change* button. Your dialog box should now look similar to the picture above before you press the *OK* button.

**Please note:** The defined material “Al<sub>2</sub>O<sub>3</sub>” will now be available inside the current project for the creation of other solids. However, if you also want to save this specific material definition for other projects, you may check the button *Add to material library*. You will have access to this material database by clicking on *Load from Material Library* in the *Materials* context menu in the navigation tree.

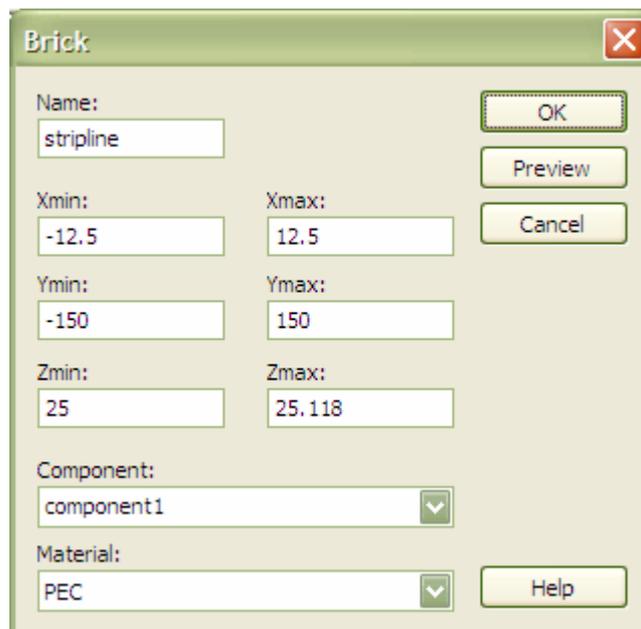
Back in the brick creation dialog box you can also press the *OK* button to finally create the substrate brick. Your screen should now look as follows (you can press the *Space* key in order to zoom the structure to the maximum possible extent):



- **Model the Stripline Metallization**

The next step is to model the stripline metallization on top of the substrate. This can also be easily achieved by creating a brick made of the PEC material.

Please check all the settings carefully as shown below before click *OK* button.



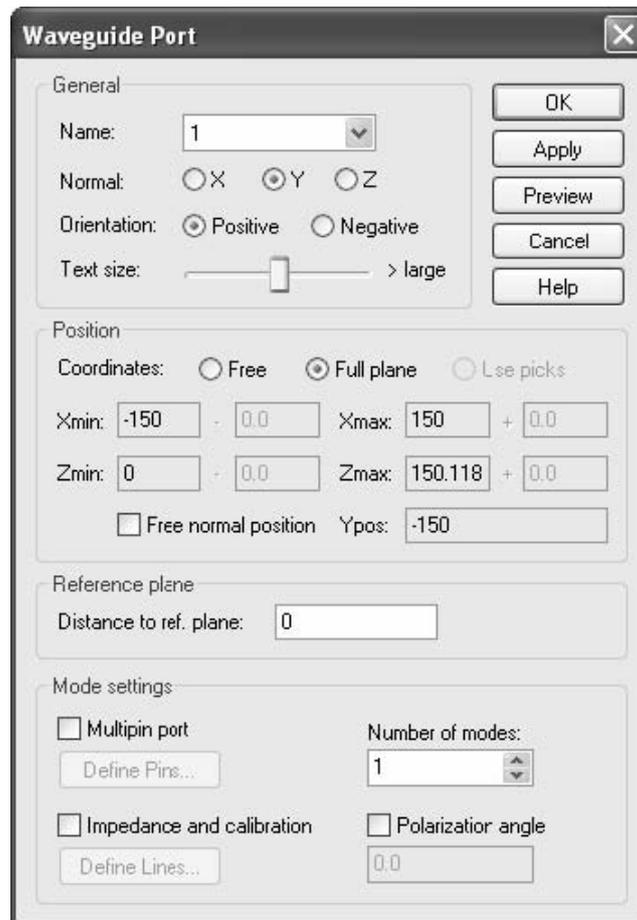
- **Define Port 1**

The next step is to add the ports to the microstrip device. Each port will simulate an infinitely long waveguide (here stripline) structure that is connected to the structure at the port's plane. Waveguide ports are the most accurate way to calculate the microstrip devices and should thus be used here.

A waveguide port extends the structure to infinity. Its transversal extension must be large enough to sufficiently cover the microstrip mode. On the other hand, it should not be chosen excessively large in order to avoid higher order mode propagation in the port. A good choice for the width of the port is roughly ten times the width of the stripline. A proper height is about five times the height of the substrate.

Applying these guidelines to the example here, you find that the optimum port's width is roughly 250 mil and that its height should be about 125 mil. In this example, the whole model has a width of 300 mil and a height of 150 mil. Because these dimensions are close to the optimal port size you can simply take these dimensions and apply the port to the full extension of the model.

Please open the waveguide port dialog box (*Solve -> Waveguide Ports*, ) to define the first port:



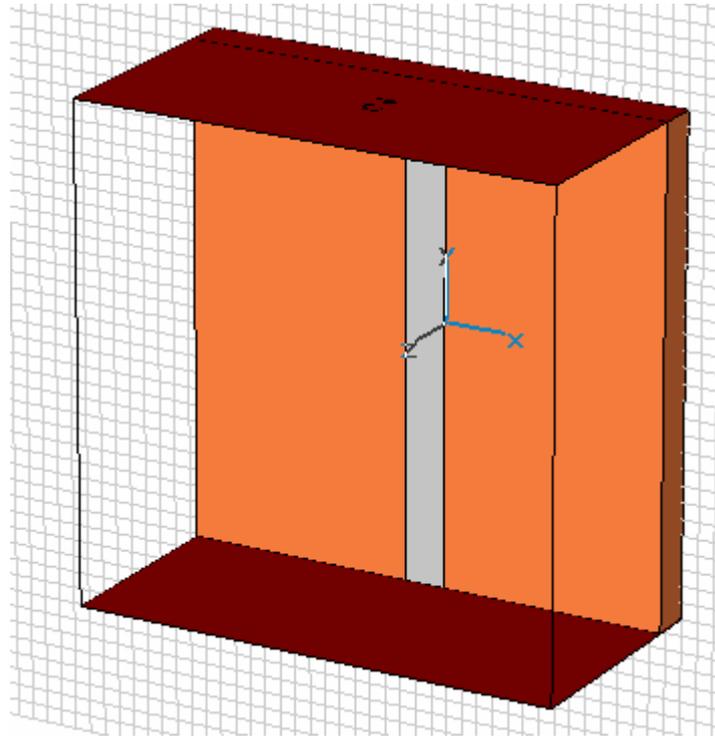
Here, you should set the *Normal* of the port's plane to the *Y*-direction and its *Orientation* in the positive *Y*-direction (*Positive*). Because the port should extend across the entire boundary of the model, you can simply keep the *Full plane* setting for the transversal position. Without the *Free normal position* check button activated, the port will be allocated as default on the boundary of the calculation domain.

The next step is to choose how many modes should be considered by the port. For microstrip devices, a single mode usually propagates along the line. Therefore, you should keep the default setting of one mode.

Please finally check the settings in the dialog box and press the **OK** button to create the port:  
You can now repeat the same steps for the definition of the opposite port 2:

- 1 Open the waveguide port dialog box. (*Solve* -> *Waveguide Ports*, )
- 2 Set the *Normal* to *Y*.
- 3 Set the *Orientation* to *Negative*.
- 4 Press *OK* to store the port's settings.

Your model should now look as follows:



- Define the Boundary Conditions

In this case, the structure is not embedded within a perfect electrically conducting enclosure, Therefore you need to change the boundary conditions to *Open* for all six boundaries by clicking *Solve -> Boundary Conditions*.

- Define the Frequency Range

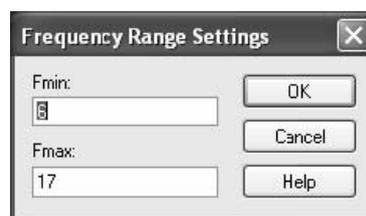
The frequency range for the simulation should be chosen with care. The performance of a transient solver can be degraded if the chosen frequency range is too small.

We recommend using reasonably large bandwidths of 20% to 100% for the transient simulation. In this example, the frequency range is between 6 and 17 GHz. With the center frequency being 11.5 GHz, the bandwidth ( $17 \text{ GHz} - 6 \text{ GHz} = 11 \text{ GHz}$ ) is about 96% of the center frequency, which is sufficiently large. Thus, you can simply choose the frequency range as desired between 6 and 17 GHz.

**Please note:** Assuming that you were interested primarily in a frequency range of e.g. 11.5 to 12.5 GHz (for a narrow band filter), then the bandwidth would only be about 8.3%. In this case, it would make sense to increase the frequency range (without losing accuracy) to a bandwidth of 30% that corresponds to a frequency range of 10.2 – 13.8 GHz. This extension of the frequency range could speed up your simulation by more than a factor of three!

Also the lower frequency can be set to zero without any problems! The calculation time can often be reduced by half if the lower frequency is set to zero rather than e.g. to 0.01 GHz.

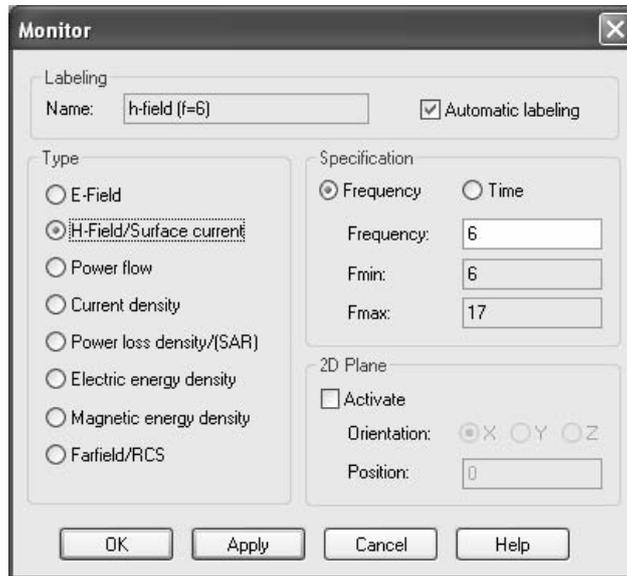
After the proper frequency band for this device has been chosen, you can simply open the frequency range dialog box (*Solve -> Frequency*, ) and enter the range from 6 to 17 (GHz) before pressing the *OK* button (the frequency unit has previously been set to GHz and is displayed in the status bar):



- Define Monitors for the Surface Current

One interesting result for microstrip devices is the current distribution as a function of frequency. The transient solver in CST MICROWAVE STUDIO® is able to obtain the surface current distribution for an arbitrary number of frequency samples from a single calculation run. You can define “field monitors” to specify the frequencies at which the field data shall be stored.

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



In this dialog box you should select the *Type H-Field/Surface current* before you specify the frequency for this monitor in the Frequency field. Afterwards, you should press the *Apply* button to store the monitor’s data. Please define monitors for the following frequencies: 6, 9, 12, 15 (with GHz being the currently active frequency unit). Please make sure that you press the *Apply* button for each monitor. The monitor definition is then added in the *Monitors* folder in the navigation tree. The volume in which the fields are recorded is indicated by a box.

After the monitor definition is complete, you can close this dialog box by pressing the *OK* button.

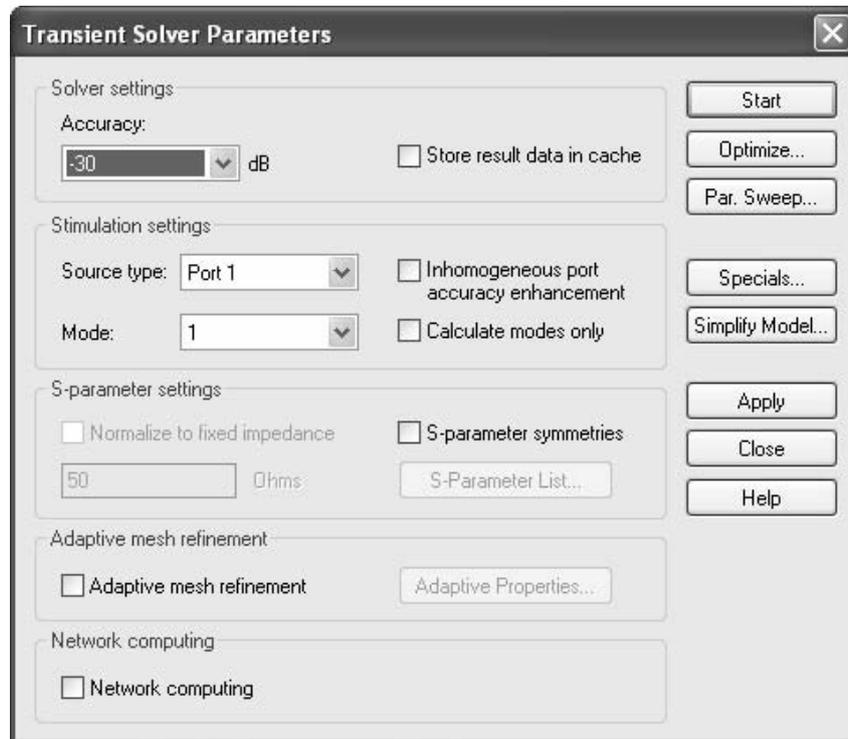
## Field Calculation

A key feature of CST MICROWAVE STUDIO® is the *Method on Demand* approach that allows a simulator or mesh type that is best suited for a particular problem. In this case, we choose the transient simulation with a hexahedral mesh.

## Transient Solver

- Transient Solver Settings

The transient solver parameters are specified in the solver control dialog box that can be opened by selecting *Solve -> Transient Solver* from the main menu or by pressing the “Transient solver” icon (🔧) in the toolbar.



Because the structure is fully symmetric, so change the Source type to Port 1.

Finally, press the *Start* button to start the calculation. A progress bar and abort button appear in the status bar, displaying some information about the solver stages.

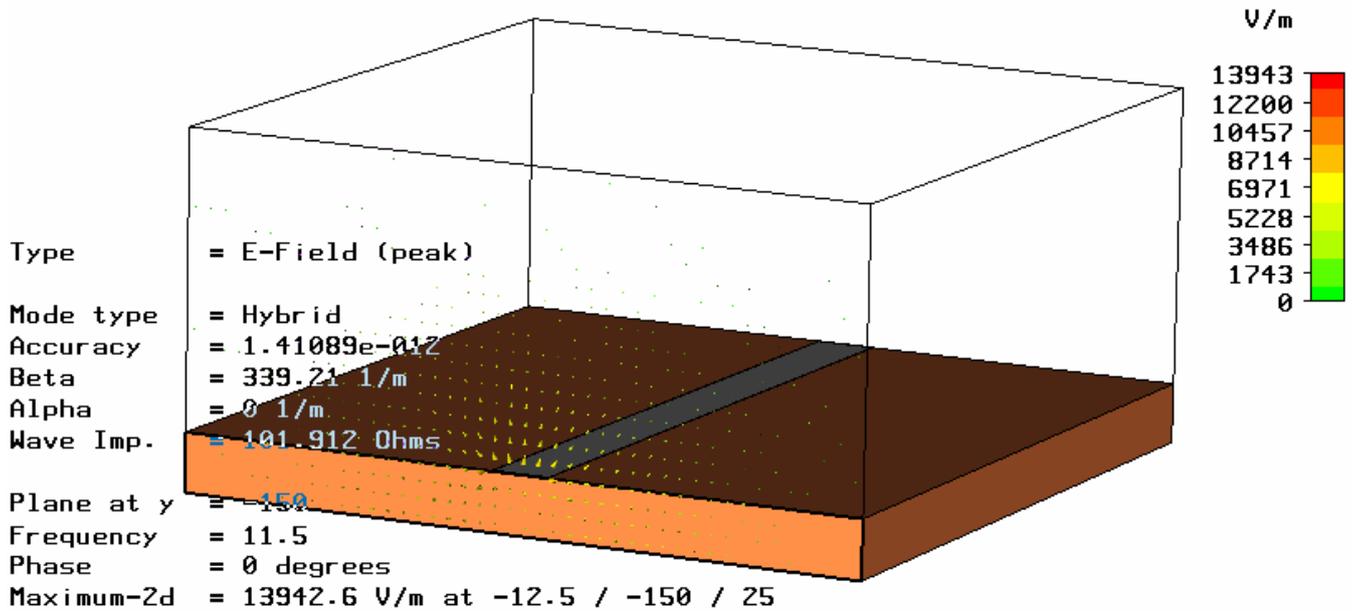


During the simulation, the “Message Window” will show some details about the performed simulation.

Congratulations, you have simulated the microstrip line using the transient solver! Let’s review the results.

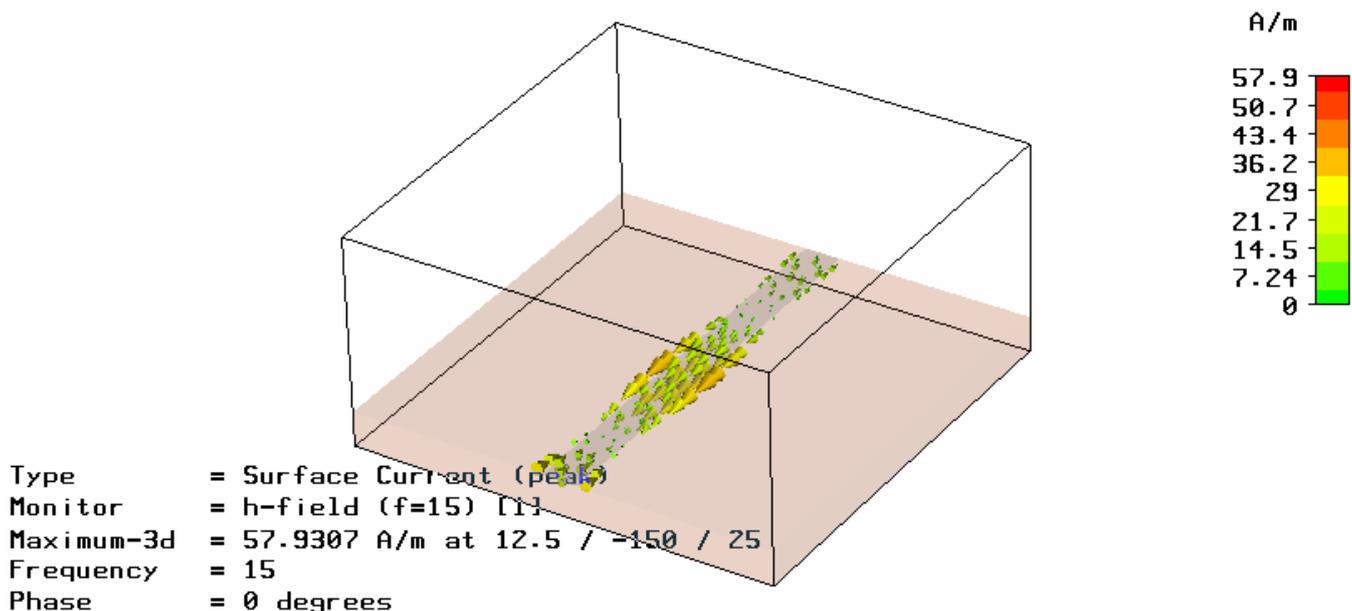
- 2D and 3D Results (Port Modes and Field Monitors)

Finally, you can observe the 2D and 3D field results. You should first inspect the port modes that can be easily displayed by opening the *2D/3D Results -> Port Modes -> Port1* folder of the navigation tree. To visualize the electric field of the fundamental port mode you should click on the *e1* folder. After properly rotating the view and tuning some settings in the plot properties dialog box, you should obtain a plot similar to the following picture.



The plot also shows some important properties of the mode such as mode type, propagation constant and line impedance. The port mode at the second port can be visualized in the same manner.

The three-dimensional surface current distribution on the conductors can be shown by selecting one of the entries in the *2D/3D Results* -> *Surface Current* folder in the navigation tree. The surface current at a frequency of 15 GHz can thus be visualized by clicking at the *2D/3D Results* -> *Surface Current* -> *h-field (f=15) [1]* entry:



You can toggle an animation of the currents on and off by selecting *Results* -> *Animate Fields*. The surface currents for the other frequencies can be visualized in the same manner.