13

This chapter shows the basics of using Layout for creating physical designs, for generating layouts from schematics, and simulating from layout or schematic. It also demonstrates Boolean operations, DRC, and the GCC.

Lab 13: Layout Basics

About this lab exercise:

This lab is only for those who are interested in ADS Layout. It is optional because there is a separate specific course for layout (Physical Design) and because this course is primarily focused on circuit design and simulation using schematics. However, this lab will get you started using layout and many of its features.

OBJECTIVES

Learn basic layout features, including the dual placement, ground plane and clearance creation, and the new graphical cell compiler.

PROCEDURE: Part 1 – Microstrip Layout and Tuning

1. **Create a new project named: layout.**

2. **Create a new schematic design and name it: ms_filter**

3. **Open the layout and set the preferences**

    a. In the schematic window, click:**Window > Layout**. This will open the corresponding layout window with the same name. Every .dsn file has a schematic and a layout.

    b. In the layout window, click: **Options > Preferences**. Next, go to the **Grid/Snap** tab and set the following for X and Y: Snap distance: 1, Minor Grid: 20, Major Grid: 200 as shown here. Then click **Apply**.

    In mils, the major grid points are now 0.2 inches apart (1 x 200 = 200 mils or 0.2 inches). Of course, you could set the grid to any desired values.

    c. Notice the top tool bar of the layout window. It has most of the same

NOTE: You can also set the snap to vertex or edge or use the icons.
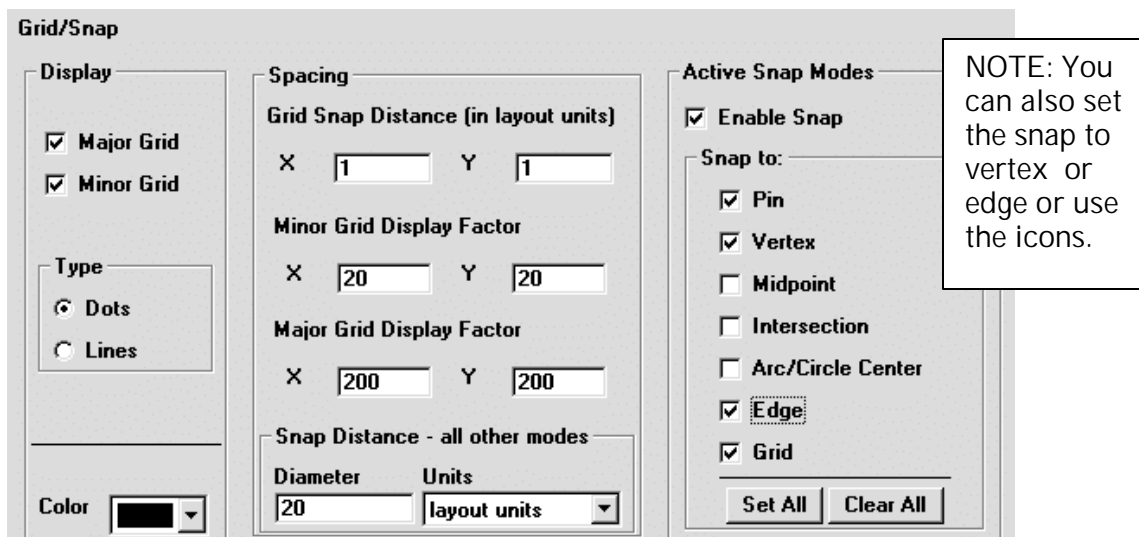
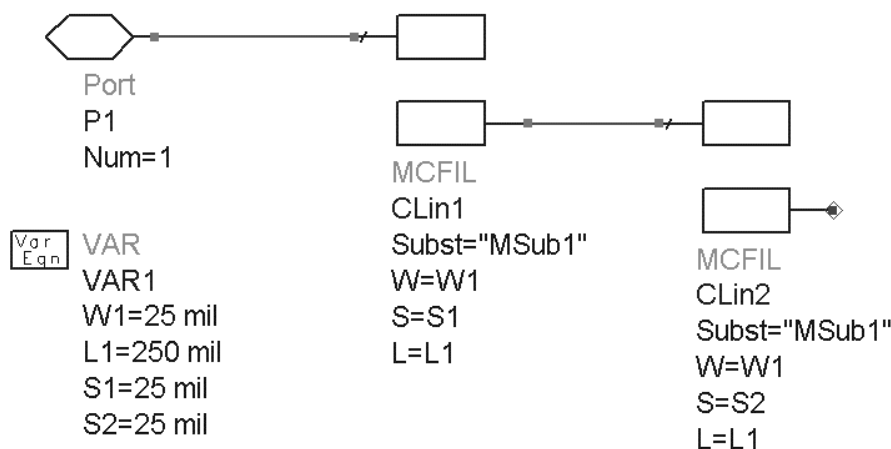icons as these snapping icons which are very useful when drawing or placing objects in layout:

SNAP ICONS: Enable, Pin, Intersection, Midpoint, Circle, Edge and Grid

d. In the Preferences dialog box, go through the various Tabs and examine the available settings: Select, Trace, Placement, etc. However, no other settings are required at this time in the lab. Click **OK** to dismiss the box.

4. **In the schematic window, build a simple coupled line filter**

Use the following steps to build the partial circuit shown here:

a. Insert a **VAR** and assign the values as show: **W1=25 mil, L1=250 mil, S1=25 mil** and **S2=25mil**.

b. From the **TLines-Microstrip** palette, insert two **MCFIL** microstrip coupled lines, and change the values to the VAR values.   Disregard the Msub1 for now.

c. Insert a port and connect the circuit as shown.

d. Generate the Layout: click: **Layout > Generate/Update Layout**.

e. In the schematic menu bar, click **Layout > Generate/Update Layout**.

Port
P1
Num=1

```
Var   VAR
Eqn
      VAR1
      W1=25 mil
      L1=250 mil
      S1=25 mil
      S2=25 mil
```

MCFIL
CLin1
Subst="MSub1"
W=W1
S=S1
L=L1

MCFIL
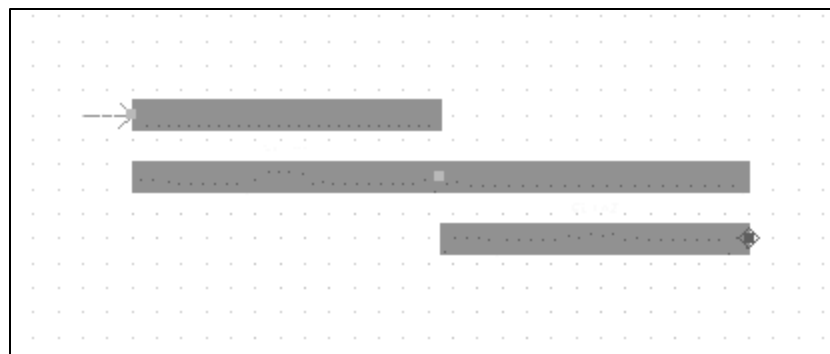CLin2
Subst="MSub1"
W=W1
S=S2
L=L1

A dialog box will appear. This dialog shows the starting point for creating the layout - click **OK** and another dialog will appear.
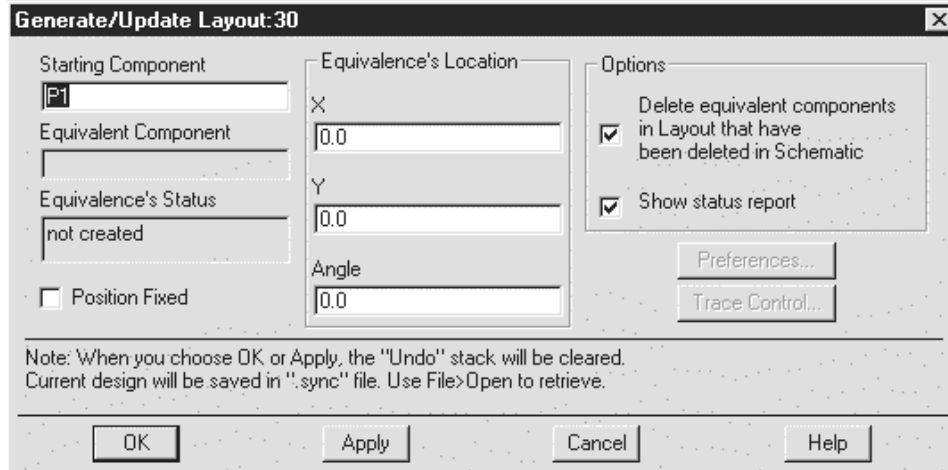
This next dialog shows the results…disregard any other messages or dialogs.
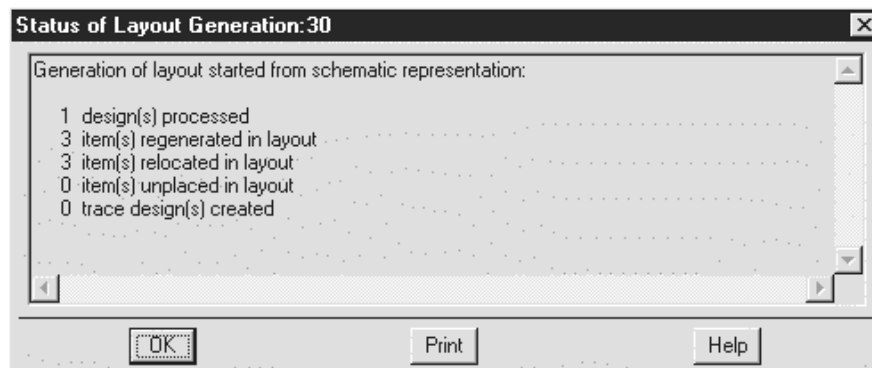

The layout window will appear with the microstrip, pins and an arrow showing the input port.
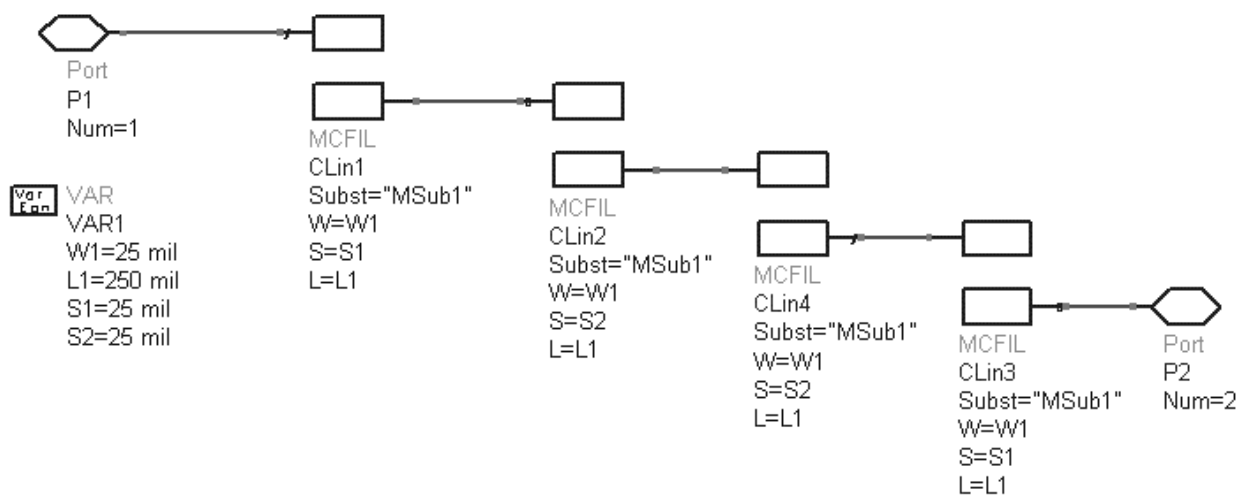
5. Complete the filter in Layout and update the Schematic

Follow these steps to copy the layout, rotate it 180 degrees and connect the copy as a symmetrical half of the final design.

a. In the layout window, click **Select > Select All**.

b. Use the copy command to copy the selected items and then rotate them and connect as shown where the port 2 arrow is pointing inward.
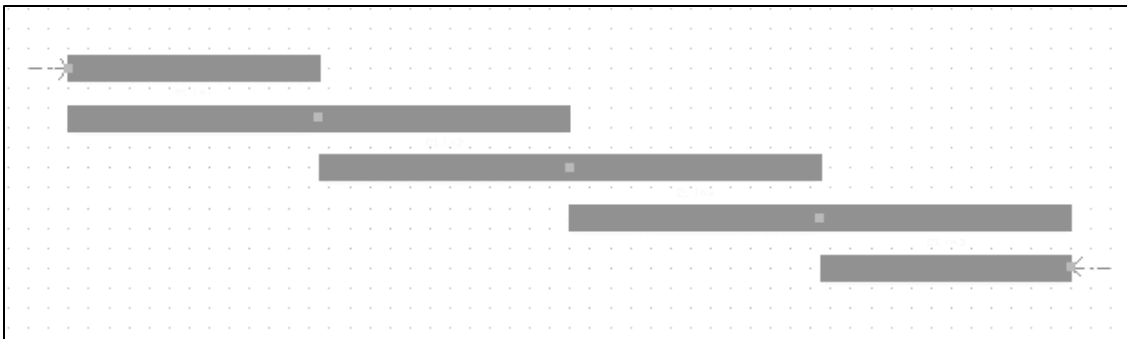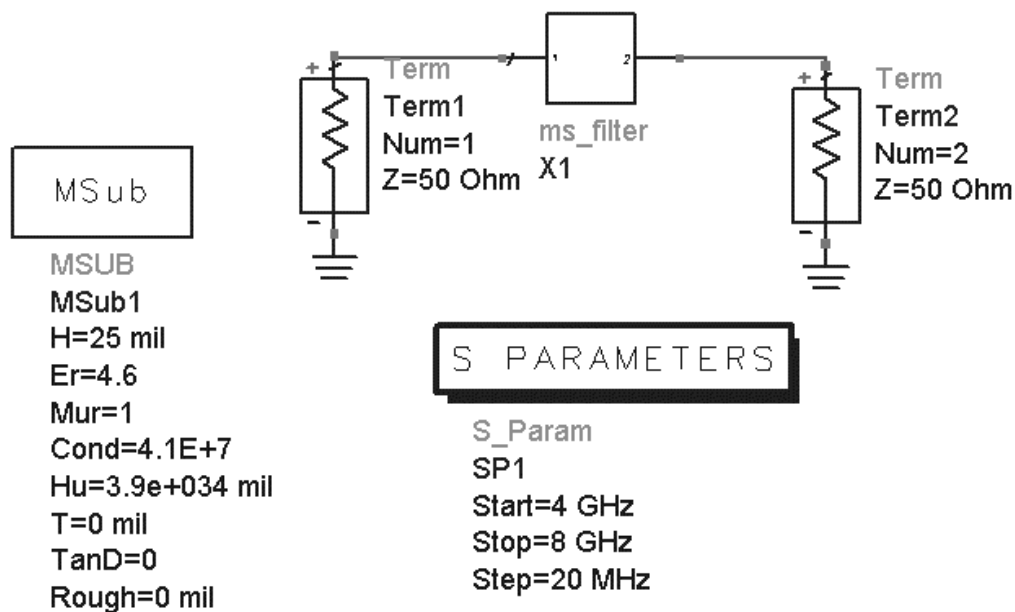

c. Save the layout and then click the menu command**: Schematic > Generate/Update Schematic** and watch the schematic updated from the layout, including the port.

d. Change the port 2 **num = 2**.

Port
P1
Num=1

VAR
VAR1
W1=25 mil
L1=250 mil
S1=25 mil
S2=25 mil

MCFIL
CLin1
Subst="MSub1"
W=W1
S=S1
L=L1

MCFIL
CLin2
Subst="MSub1"
W=W1
S=S2
L=L1

MCFIL
CLin4
Subst="MSub1"
W=W1
S=S2
L=L1

MCFIL
CLin3
Subst="MSub1"
W=W1
S=S1
L=L1

Port
P2
Num=2

6. Save the design and open anew schematic window

a. If you are familiar with the File >Deisgn Parameters feature, give the sub-circuit a different component instance name (if you do not want it to be "X1". Save the design (ms_filter) and open a new schematic window. Save the new schematic as bp_filter.

b. Use the library icon and insert the **ms_filter** from the browser.

c. Complete the schematic as shown here, including an **MSUB** from the Microstrip palette, terminations and an S-parameter simulation controller from the S-parameter palette.

The Msub is the substrate definition and has an Er of 4.6 (FR4) and copper conductivity with a 25 mil height.

```
        ┌──────┐
  Term  │ 1  2 │         Term
  Term1 └──────┘         Term2
  Num=1   ms_filter      Num=2
  Z=50 Ohm  X1           Z=50 Ohm

  MSub
  MSUB
  MSub1
  H=25 mil
  Er=4.6        S PARAMETERS
  Mur=1
  Cond=4.1E+7   S_Param
  Hu=3.9e+034 mil  SP1
  T=0 mil       Start=4 GHz
  TanD=0        Stop=8 GHz
  Rough=0 mil   Step=20 MHz
```

d. Wire the circuit together.

e. Push into the ms_filter and note that your first design is there.
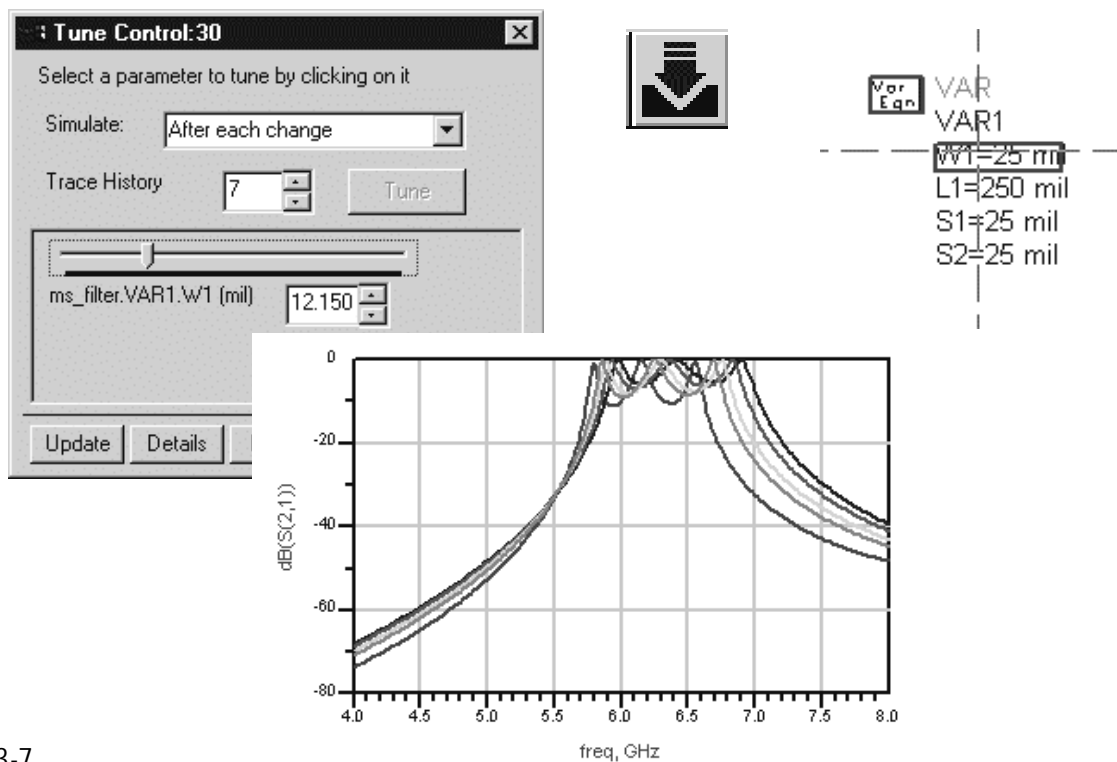
7. Simulate and plot the results

You should see the band pass as shown.
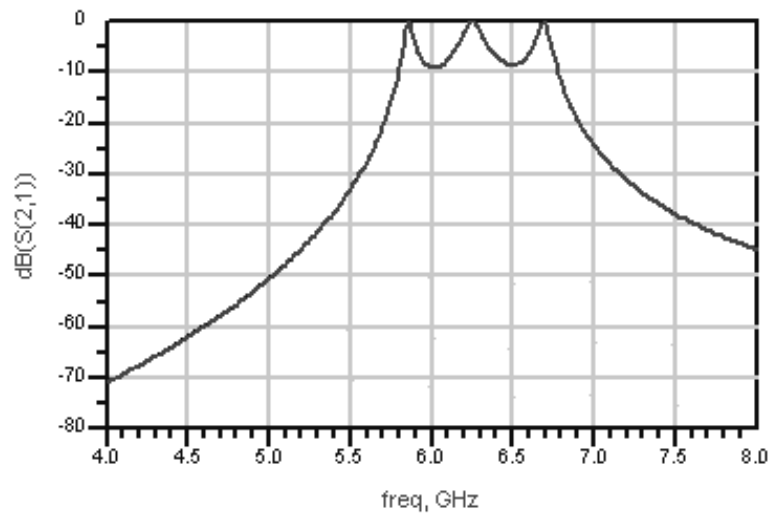
8. Use Tune Mode to modify the performance

a. Start tuning in the upper level circuit. Then push into the lower level schematic and try tuning w1:

b. Try tuning and other parameters. When finished, reset the variables to their original values.

9. Place schematic components directly into layout

This step will show you how to create a layout by placing the components from the schematic.

a. Inside the layout and delete the lower symmetrical half of the ms_filter, including the pin.

b. Go to the lower level schematic (ms_filter) and click the menu command: **Layout > Place Components from Schem to Layout**. Immediately, the schematic components will be highlighted. Position the schematic on top of the layout so you can see both windows.

c. Select the first highlighted MCFIL in schematic and move the cursor to the layout window. The cursor will now have the layout item attached. Connect it to the pin as shown here:
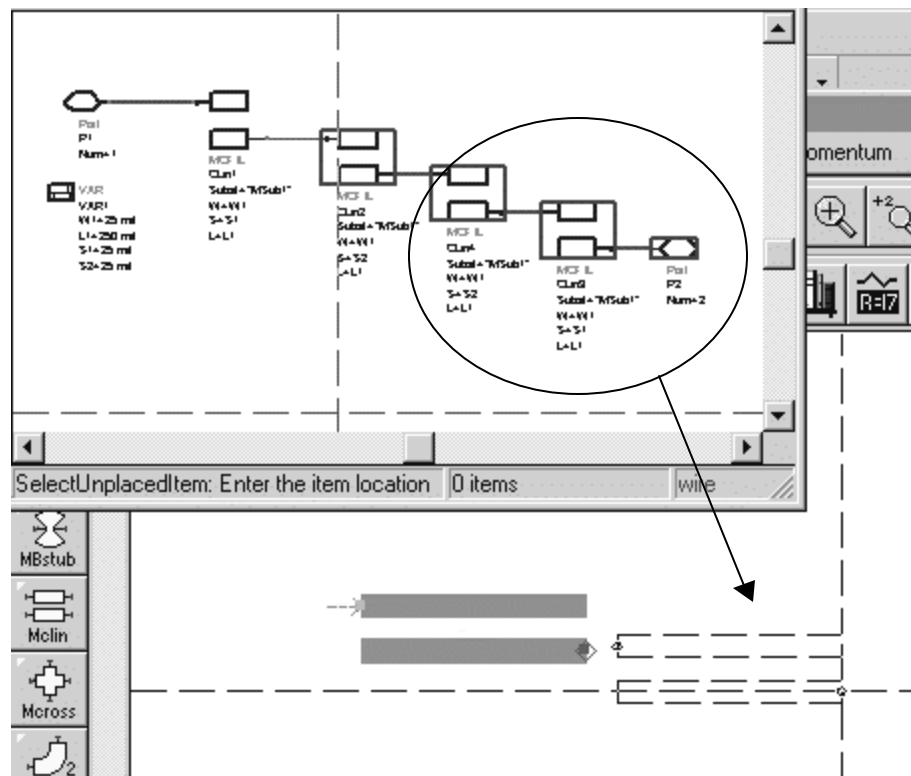

This method works best when seperately or in addition to the automatic layout generation, especially when you are concerned about the placement of the components. You can experiment with this feature. You will have to rotate some components with ports.

10. Checking the distance of a line

Put the cursor on a pin and click. Then move the cursor to the end of the line and notice the distance in the lower right corner of the layout window. This is how you check distance.

Check this distance

11. Building the ms_filter in dual placement mode: Schematic & Layout

a. Delete both the schematic and layout**: Edit > Delete All**.

b. In the schematic window, go to the **Options > Preferences** and select the **Placement** tab. Click the box for **Dual Representation**. Then click **Apply** and **OK**. Now, you have set the system to place components in both schematic and layout at the same time.

c. Dual Placement: Place both the Schematic window and the Layout window near each other. Now, in the Schematic window, build the ms_ filter again, starting with the first component in schematic. Next, move the cursor into the Layout window and insert it there. Go back to the Schematic window and select the next component and continue. Build the same circuit as you did before (refer to the picture) until complete.

12. Create a path / trace

In the layout, try using the Create a Path icon to draw traces – experiment as desired.

## PROCEDURE: Part 2 – ADS Example - RF board & ground plane

Save and close all the windows from the filter designs in the previous steps.

13. **Go to the** Examples **directory and open** RF_Board **directory and the** MixerPager_prj **example. This is similar to the pager mixer you created in the course.**

   a. Open the **MixerLayout** design and open its layout window. As you will see, the layout has a ground plane, consisting of 3 areas, that is at least 10 mils away from all the components.
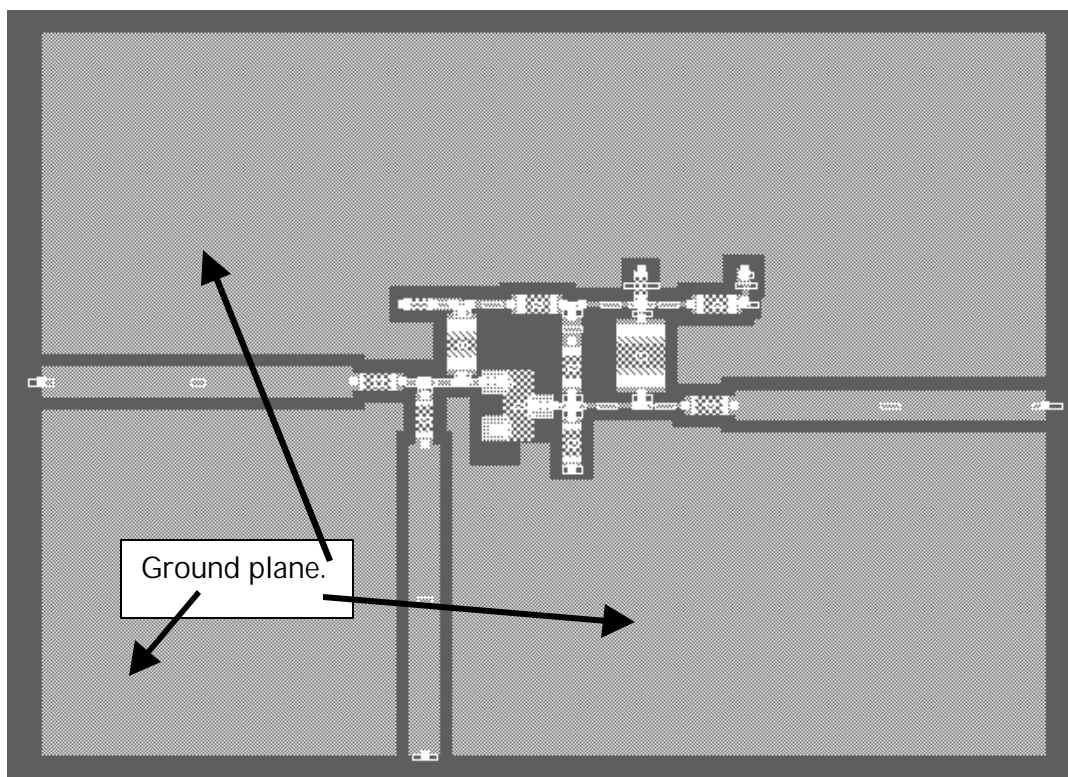
Ground plane.

   b. Now, delete the existing ground plane (3 separate blocks) by selecting each one and delete it. What remains is only the layout without any ground as shown here.
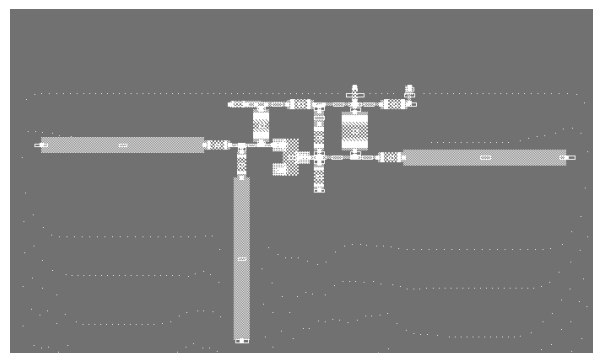
c. To create a new Ground Plane, select the **rectangle** from the tool bar and draw a large rectangle around the existing circuit layout. This will make the layout very difficult to see because the rectangle is drawn on the same layer (cond) as the microstrip as shown here.
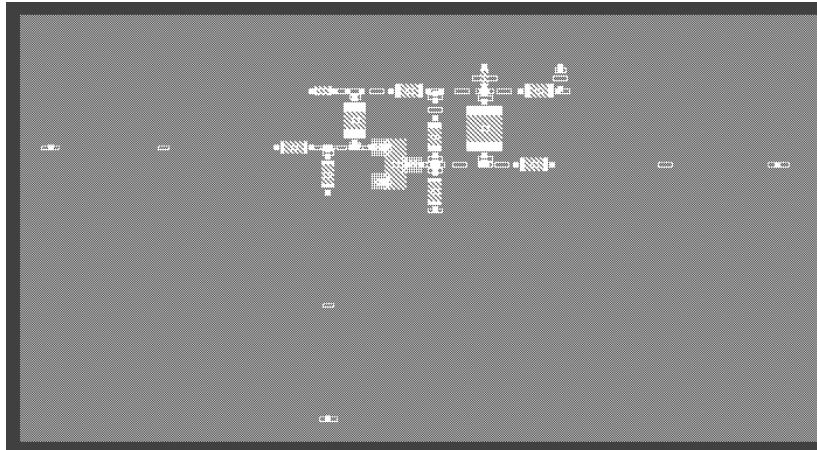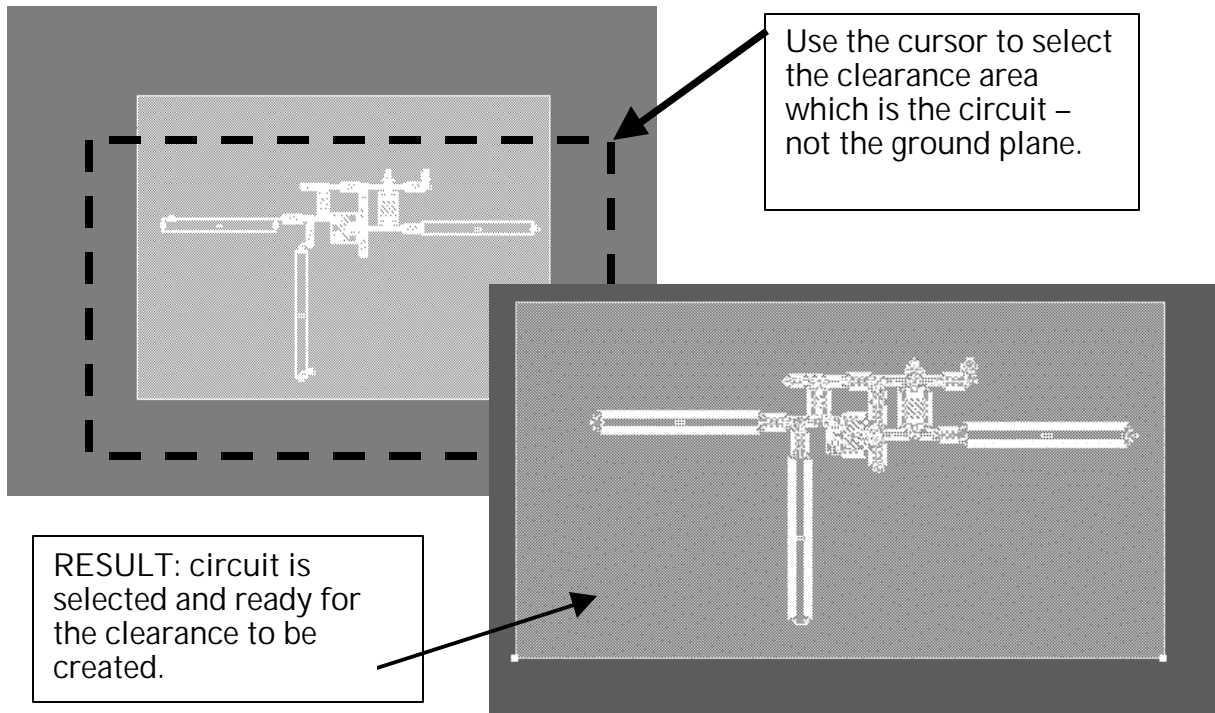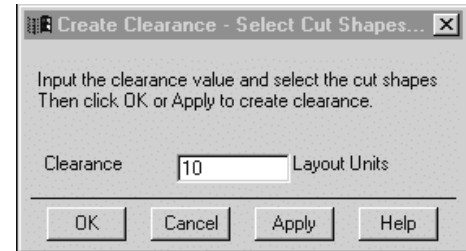

d. To begin creating the clearance, click: **Edit > Create Clearance**. A dialog will appear instructing you to select the ground plane you just drew on cond.   First, select the ground plane and then click OK in the dialog box.
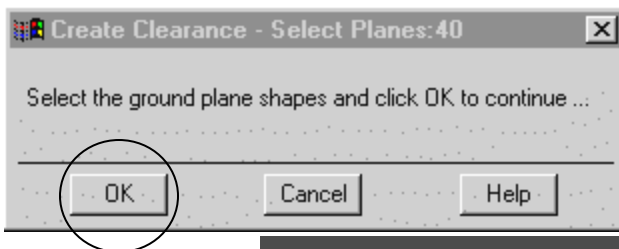
First, select the ground plane and then click OK
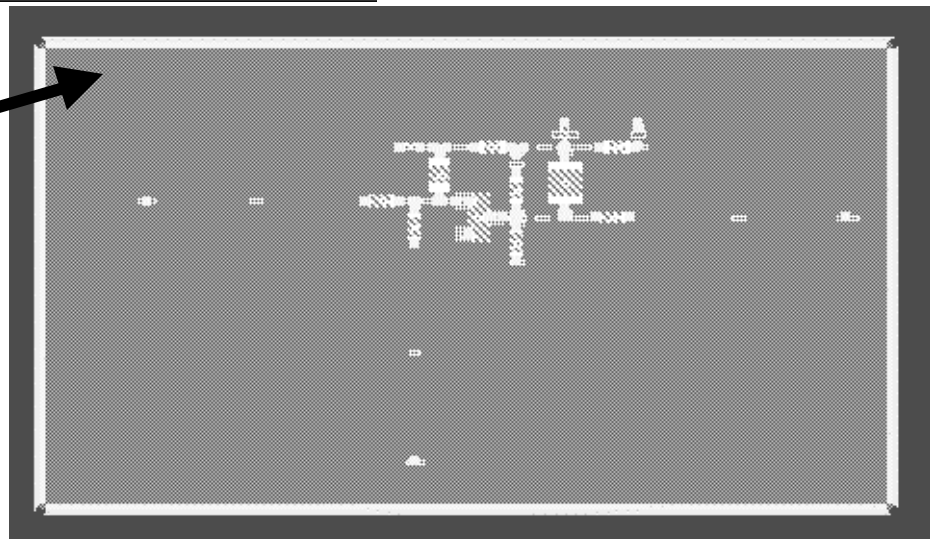
e. When the next, enter the clearance value:
   10, but do **NOT click OK** or Apply yet.

f. Now, select the clearance area by rubber
   banding (selecting) the circuit - which is
   everything except the ground plane.

Use the cursor to select
the clearance area
which is the circuit –
not the ground plane.

RESULT: circuit is
selected and ready for
the clearance to be
created.

g. With the mixer pads and other items on all the layers selected, click the
   OK button and then cancel the dialog
   box.   Zoom in to see the result.

14. Cleaning up the Ground Plane
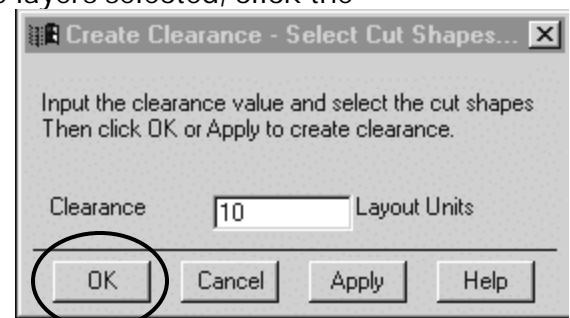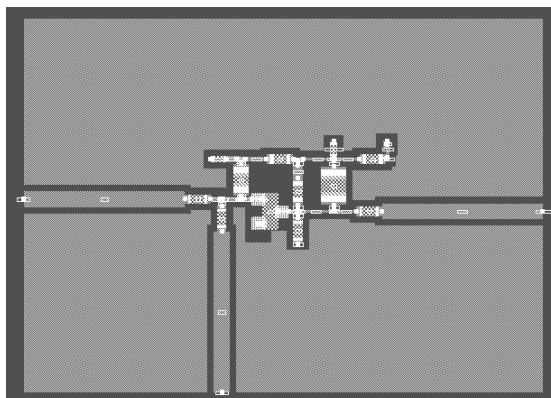
a. With the layout components selected, change the insert layer to: **pc1** in the top of the layout window.

b. Turn on edge snap and vertex snap modes only. Turn all other snapping modes off.

c. Zoom in on an area and draw rectangles on pc1 where you want to remove the cond material as shown:

Area to remove using boolean logic.
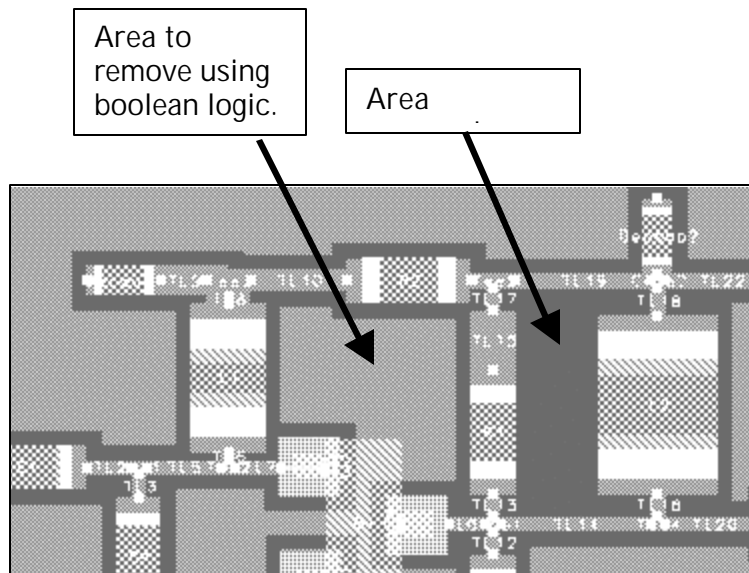
Area

d. **Click Edit > Boolean Logical**. When the dialog appears, set it as shown here:
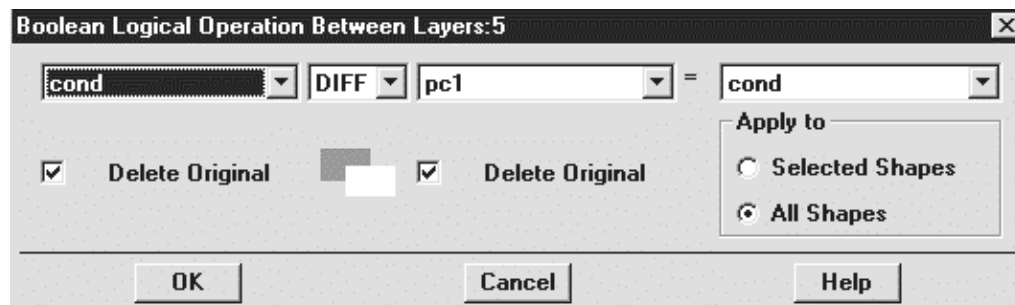
e. Click the OK button and the pc1 area will be subtracted from the cond layer leaving the desired clearance. This is how the clean up is done.

15. OPTIONAL - Graphical Cell Compiler

For this final step, use the on-line manuals and the Step-by-Step procedure from the GCC Getting Started manual.

    a. In the Main Window, click; HELP > Manuals.

    b. Go to the Contents and select the GCC. Size the window so you can read the text and use the tutorial steps.

This concludes the layout lab exercise.