Wireless Circuits and Systems Laboratory

Procedure #4 Using the Layout Program in ADS

Follow these instructions very closely! If you don't, the TA will not be able to machine your circuit and you will not be able to complete the lab assignment.

In this course you will be designing some of your own microwave circuits, which will be produced using the computer-controlled milling machine. The Teaching Assistant will be responsible to running the milling machine, but you must provide the circuit drawing.

The drawing package you will use is the Layout Program in ADS; once you have started ADS, the drawing window is accessed by clicking on the Layout icon which is located in the ADS Main Program window (the layout icon is the 7th button from the left). ADS treats your drawing as a circuit design, that is specified by drawing the geometry as opposed to creating a circuit schematic. Thus the file will be saved with the .dsn extension by default. This means that you should choose a unique name for drawing files, so as not to conflict with any associated circuit schematics you may generate as part of a lab experiment.

Pay special attention to the following directions:

- **When you begin your drawing ensure that the unit of length is set to mm. Also, set the Snap option to vertex only (not grid or pin). Refer to steps 3 and 4 below. Do your layout in mm to be consistent with the other procedures in the lab.**
- **You do not need to use more than 2 significant digits with your coordinate values (the milling machine does not have accuracy in the micron range!).**
- **The name of the layout file (.dsn) that you give to the TA (on a 3.5" disk) must follow the naming procedure outlined in step 2 (below).**
- **You must also provide the TA with the .dxf file that uses same naming convention from step 2.**

1. Creating Circuit Artwork using the Layout Program

1. From the ADS main program window, click on the Layout icon (the 7th button from the left). This will cause the Layout window to pop up.

2. It's good practice to give your design a name right as you begin. To do this, File/Save As and then enter the name of your design (you do not have to add the suffix). This file will be saved in the Networks subdirectory of your project directory. For the filenames use the following syntax: GnSmLx, where n=your group number, m=the laboratory section you are in (use only the last digit) and x=the experiment.

3. Examine the small boxes at the bottom of the layout window. In one of these boxes you should see 'mm' which indicates the drawing units that are being used. (You should have chosen mm as the default unit of length when you initially created the directory that is being used.) We will use mm to do the CAD drawing, since this is consistent with the other work you will do in the lab.

4. From the Options pull-down menu, make sure that the following are **selected**: Snap Enabled and Vertex Snap. Make sure that the following are **disabled**: Grid Snap and Pin Snap.

Now all you have to do is draw your circuit. We will draw all of our shapes using **Polylines**. These are objects which have an unlimited number of vertices, which we will enter using the keyboard. **WORD OF CAUTION**: the layout package is not easily coerced into modifying incorrectly specified vertices, so be careful when you enter each

point. A good approach is to draw your design on paper first and calculate the positions (x and y) of all the vertices first (x runs left to right, y runs bottom to top).

5. Go to the Draw pull-down menu and select Polyline. Now return to the Draw menu and select Coordinate entry... This will allow you to enter all the vertices of your polyline objects exactly, using the keyboard.

6. Enter the x and y positions of the first point and click on the Apply button. Edit the x and y positions for the second points and click on the Apply button. Repeat for all the vertices until you have returned to the starting point, and have "closed" the polyline. On the last point, click on the Apply button twice to finish the polyline. You can now repeat the procedure for all subsequent polylines. Note: if you are working in mm, do NOT use more than two decimal places when you enter the coordinates - the third decimal place is at the micron level, and this type of resolution cannot be achieved using the milling machine!

7. Two things that are convenient to use as you proceed are: 1) View > Redraw View (this will refresh the screen), and 2) View > Zoom Area (this allows you to point in one corner and draw a box around your drawing so that you can enlarge the view).

8. It is recommended that you save your design periodically, as this will allow you to recover from mistakes more easily.

9. If you do happen to enter a vertex incorrectly when drawing a polyline you have two options. Option 1 is to click on Draw -> Undo Vertex; this will erase only the last point that you entered. If you have really messed things up, and are still in the coordinate entry box: click twice on twice, then Cancel and Edit>Undo. This will remove the last (incorrect) polyline. **Do NOT hit the Undo button repeatedly, as it will continue to delete polylines in the order in which they were entered**.

10. If, after finishing a polyline and clicking twice on Apply, the cursor does not release (i.e., the dotted cross-hairs still appear in the window if you move the mouse around): click on Cancel in the coordinate entry pop-up window, and then click on the large white arrow button along the top menu row. You have probably not properly returned to the starting point of your polyline (first and last points entered are not the same), and so you should delete the last polyline and start over.

When you are finished with the drawing:

1) Click on **Options>Preferences** and select the **Units/Scale** tab.

2) Check the Scale Factor for Length in **mm**, click on **Apply** and then **OK**.

3) Click on **File>Export** and select the **Mask file (.msk)** file type, then click on **OK**.

4) Click on **File>Export** and select the **DXF** file type, then click on **OK**.

   a) At this point a pop-up window will appear that requires you to enter the name of a mask file. Click on **Browse** and select the file you created in the previous step. Then click on the **Translate** button.

   b) When the MTOOLS Log window pops up, you can simply **close** this (do not click on Save As in this file window).

   c) Finally, click on **Exit** to close the MTOOLS translator.

5) At this point, the .dxf version of the layout file should appear in the main project directory.

You will need to provide two files to the TA. The first is the normal layout file, which will have a .dsn extension (by default, this is written to the Networks subdirectory of your project directory). The second is the version with the .dxf extension.

Make sure to indicate on the disk the exact files that you want the TA to use (if there are more than the two mentioned here). Either mark on the label, or tape a piece of paper to the disk.