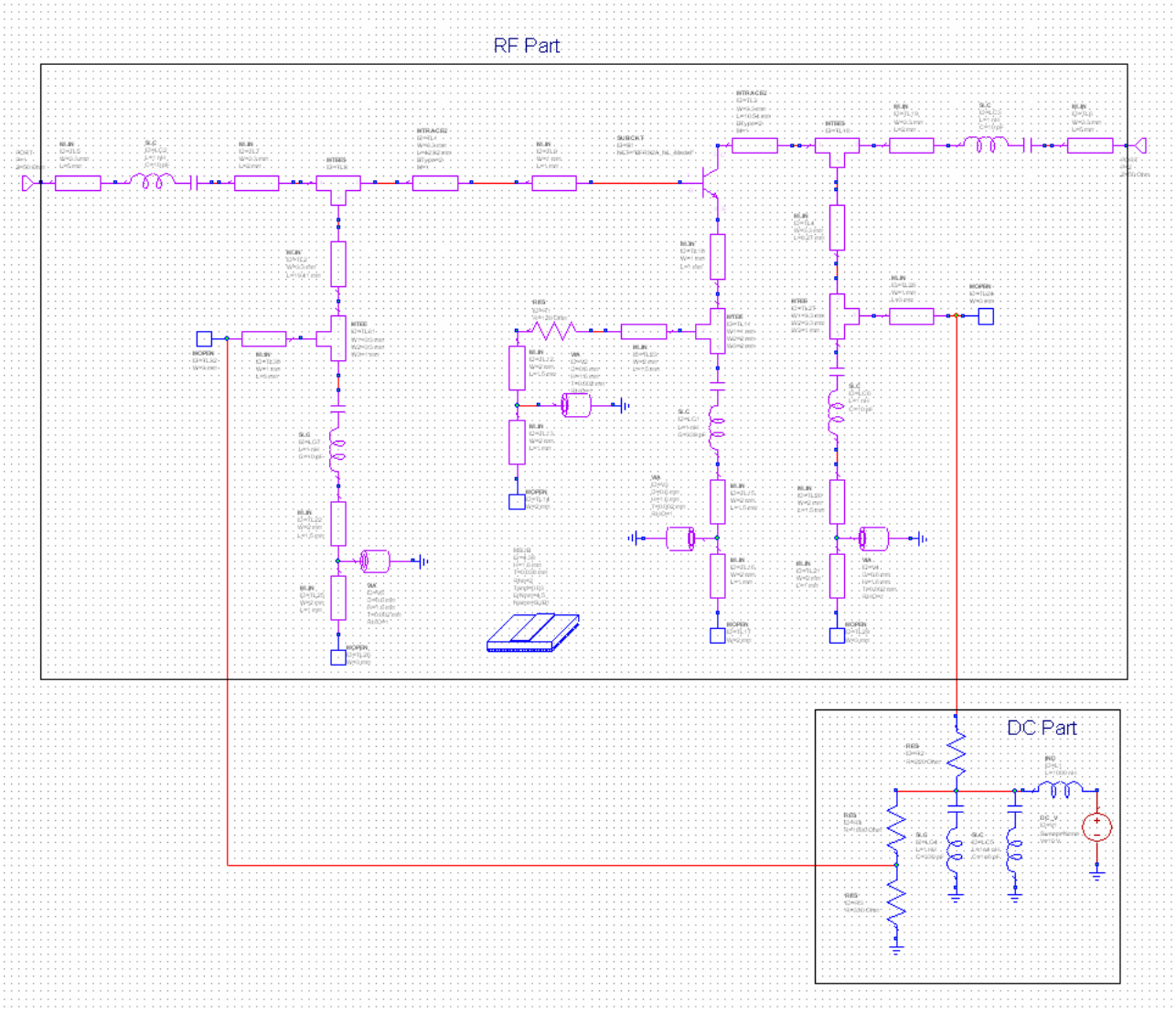
Microwave Office 2006 - Layout

Once you have designed the circuit and happy with its response you must perform the layout. It is recommended you copy the schematic into a new Schematic so that you can easily retrace your steps. Use the schematic copy to generate the layout.

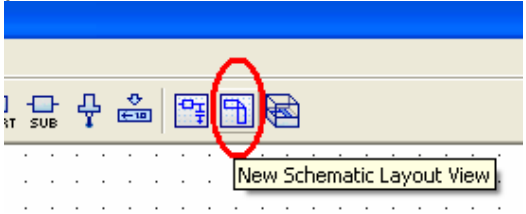
1. Schematic

Having created a copy of your schematic of the circuit, you will need to include interconnecting microstriplines and most importantly microstrip discontinuities (step-width, Tee-junction, etc). If you simulate this new schematic you will find that its response differs from the original schematic. You will probably need to adjust stub lengths and positions.



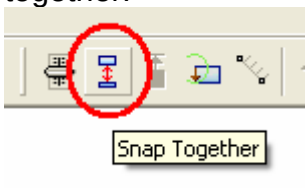
2. Initialise the Layout

Whilst in the schematic window, click on the "New Schematic Layout View" icon. This will open a new window which shows the layout. But the elements will not be in the correct positions.



3. Layout

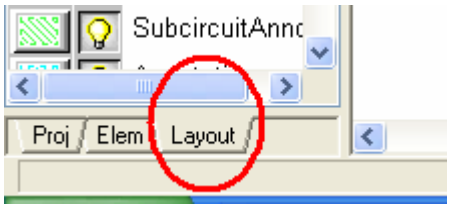
In the layout window you can move, rotate and flip microstrip elements and snap them together.



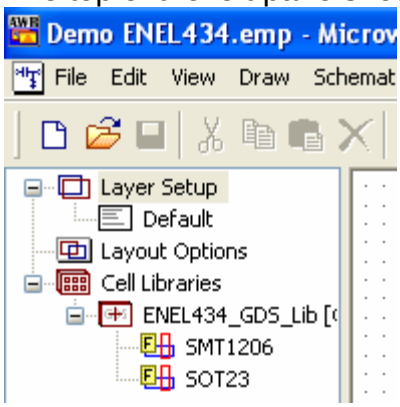
4. Footprints

What you will need to manufacture the pcb is the layout of the copper tracks. Microstriplines automatically have copper tracks associated with them in the layout. Although not essential, it is useful to have footprints of transistors and surface mount components. These aid alignment of copper tracks.

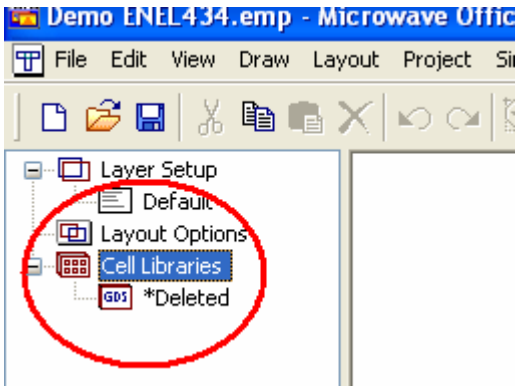
Click on the layout tab at the lower left-hand corner of the screen:



The top of the left pane should look like this:



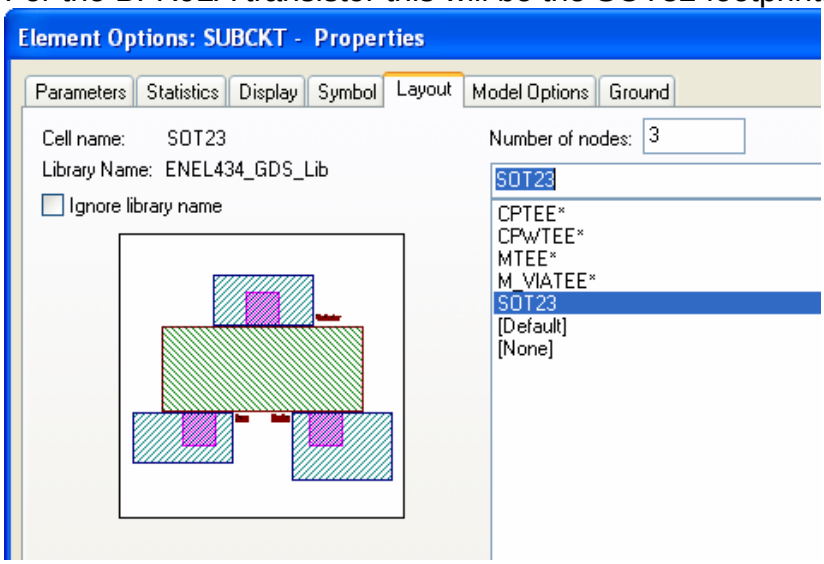
If this is not the case then you need to download a file named “ENEL434_GDS_Lib.gds from the W drive. Then you have to load it into the MWO Project file. To do this right-click on Cell Libraries:



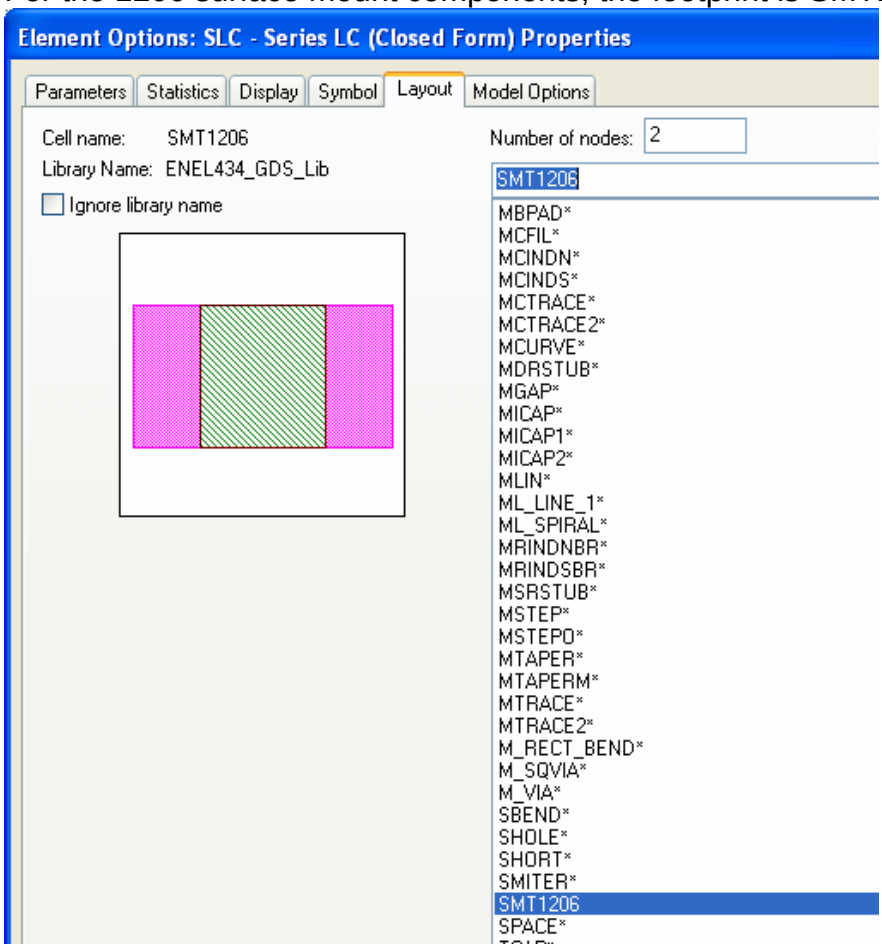
Then select “Read GDSII Library … “.

To associate footprints with the components, return to the schematic and double click on a component. Then click the Layout tab of the dialogue box and then select the footprint.

For the BFR92A transistor this will be the SOT32 footprint:

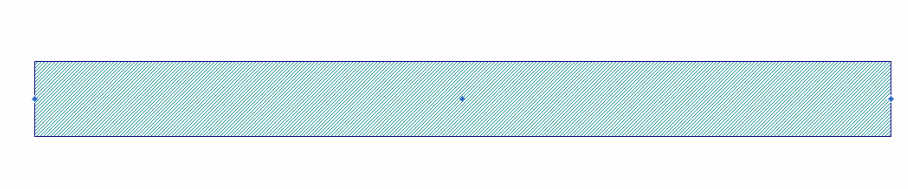


For the 1206 surface mount components, the footprint is SMT1206:

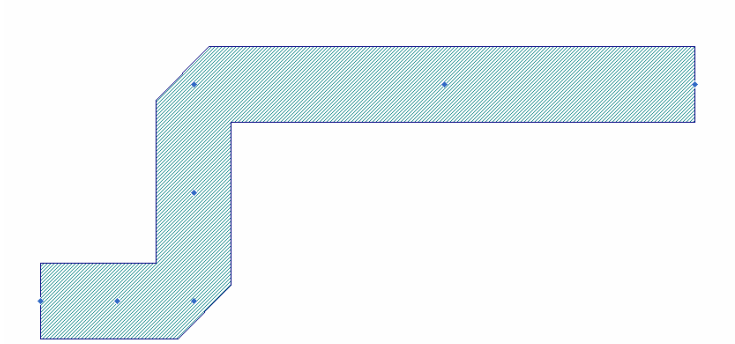


5. Meandering Microstriplines

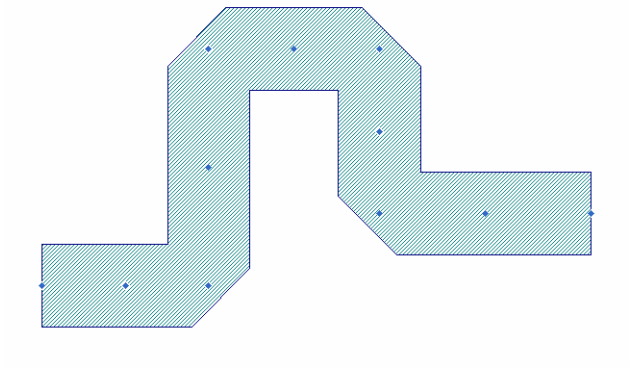
Microstriplines can be lengthy but may be compressed into a smaller space by meandering thereby conserving board area. In the schematic in place of the MLIN element choose the MTRACE2 element. In the layout double click on the MTRACE2 element this will reveal small blue diamonds on the element:



Drag and drop the small blue to shape the meandering:



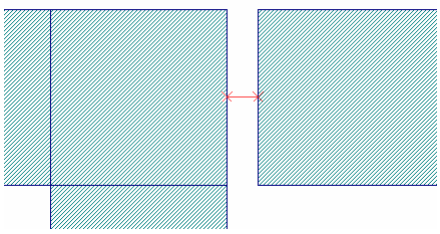
To add extra bends, hold the Ctrl key down while dragging a diamond:



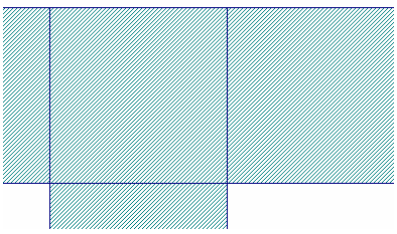
6. Check

You should ensure that all elements are joined together and arranged in the correct position. Zoom as necessary to ensure that all elements are properly joined.

Improperly joined:



Properly joined:



7. Completed Layout

Check the response of your layout schematic to see that it performs as desired. Adjust stub lengths and positions etc if necessary.

Add your name and cutting lines etc to complete the layout. The icons shown are used for these embellishments.

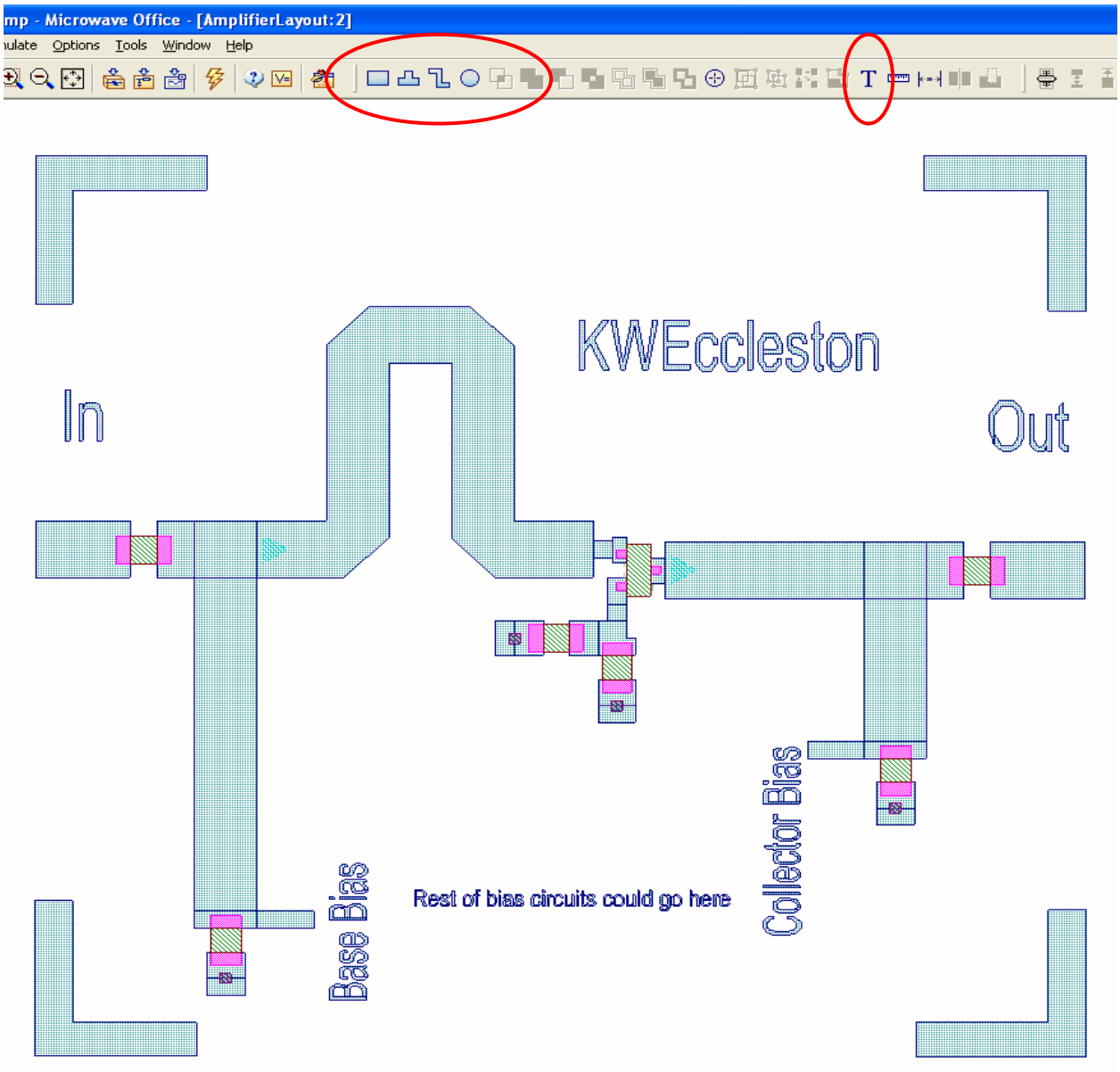
Make sure that these added objects are “thick Metal”. They should appear the same colour as the microstriplines. If not, right-click on them and select “Shape Properties”, etc.

Make sure that the input and output ports are flush with their respective cutting edges.

Make sure that there is a gap of at least 3 substrate heights between the side of a microstripline and another element or object including text, cutting lines, and edge of pcb.

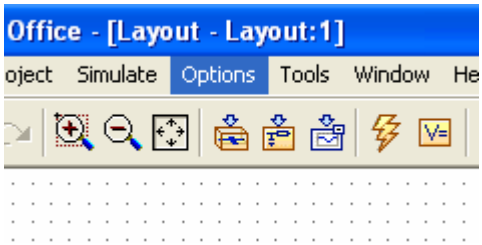
Make sure that there is room for a solder joint.

A completed layout looks like …



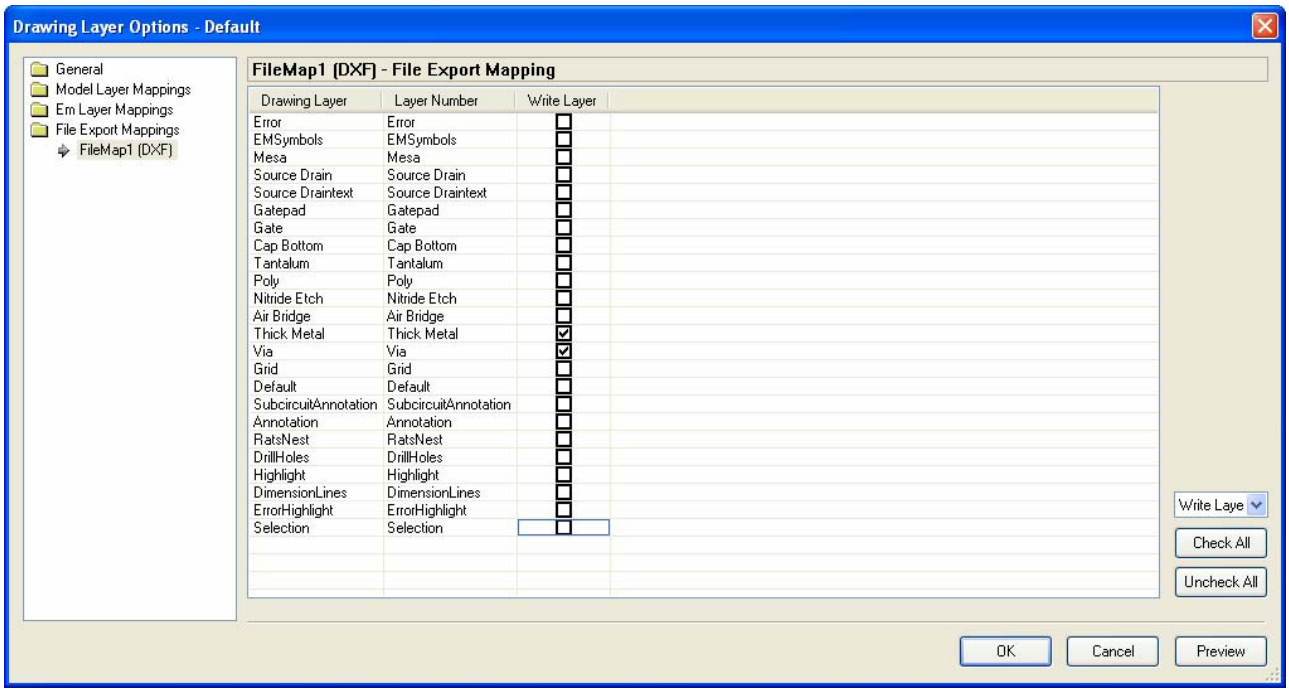
8. Export Instructions

You need to inform the method in which the layout is to be exported to a DXF file. Click on the Options menu:



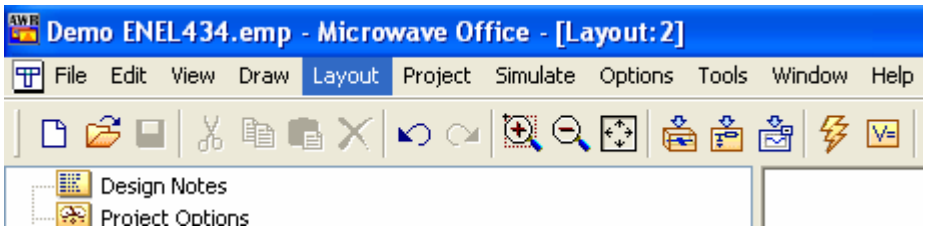
and select “Drawing Layers …”.

Right click on the “File Export Mappings” folder tab and select “New DXF File Export Mapping”. In the “Write Layer” column tick only the layers that you will need exported to the DXF file. This should only be “Thick Metal” for the copper tracks and “Via”:

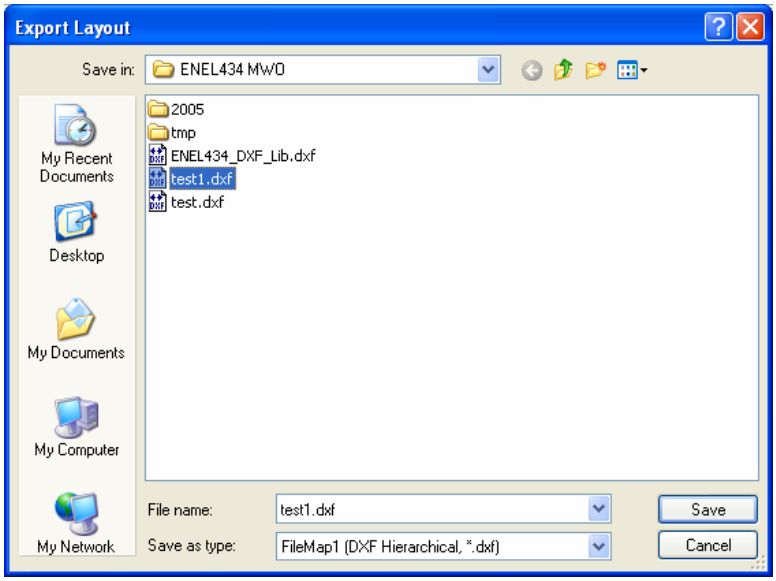


8. Export

Once you are happy with the layout, export the layout to a DXF file. To do this (within layout view): Click “Layout” menu.

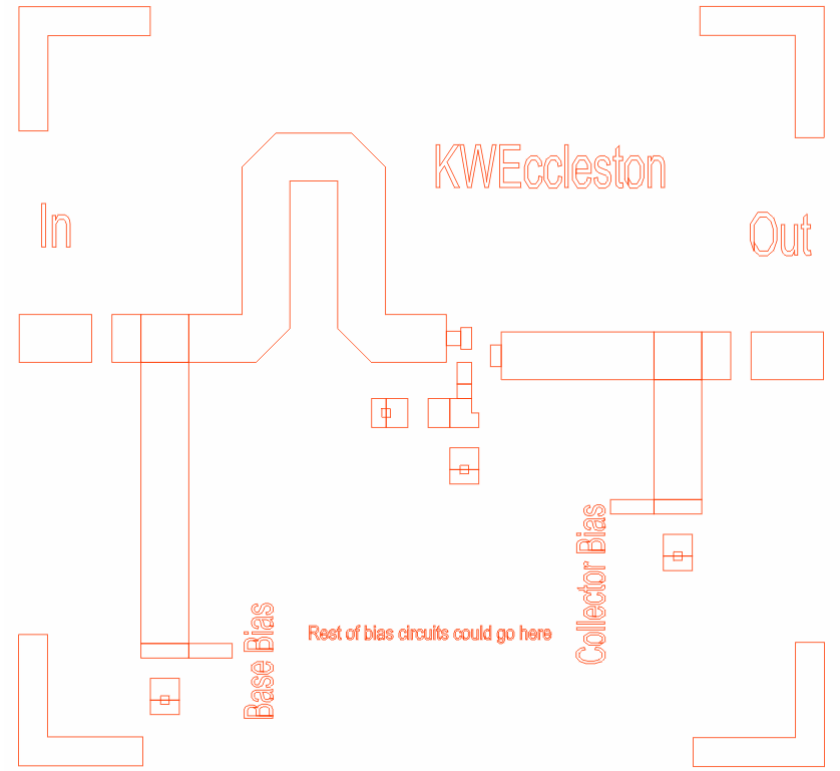


Then select “Export”, and then save to a FileMap (DXF Hierarchial, *.dxf) file.

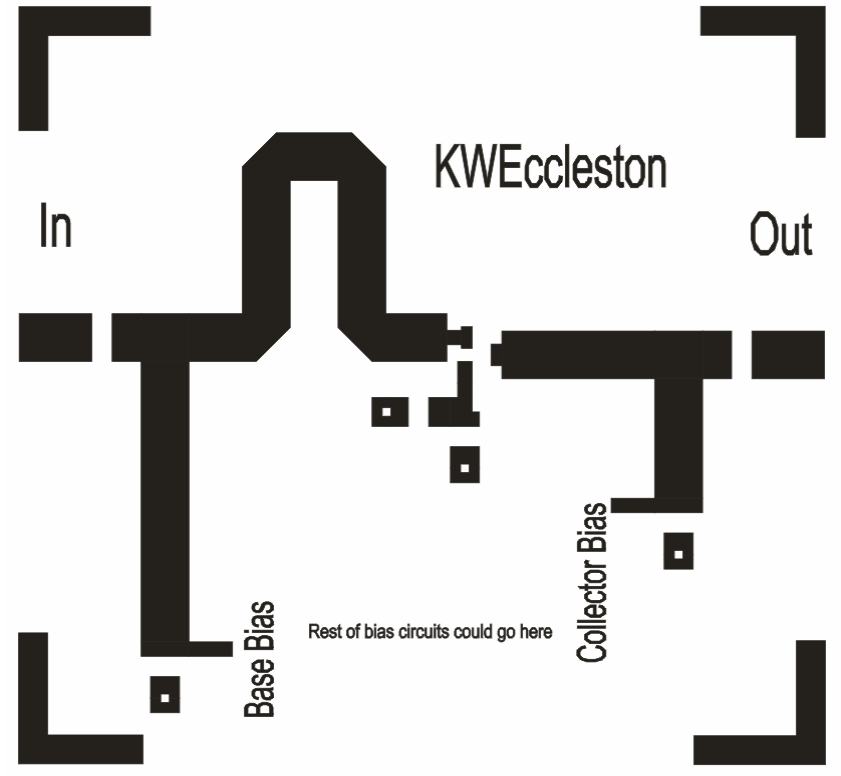


9. Editing the layout in CorelDraw

Import the DXF file into CorelDraw. You should see the outlines of the copper tracks:



You then need to fill the polygons black and make the outlines black (hairline):



Before submission check that the artwork is what you intend. Make sure that via drill holes appear.

Make sure that the file name coincides with the protocol given in the assignment instruction sheet. Failure to do this could result in your layout been misplaced.