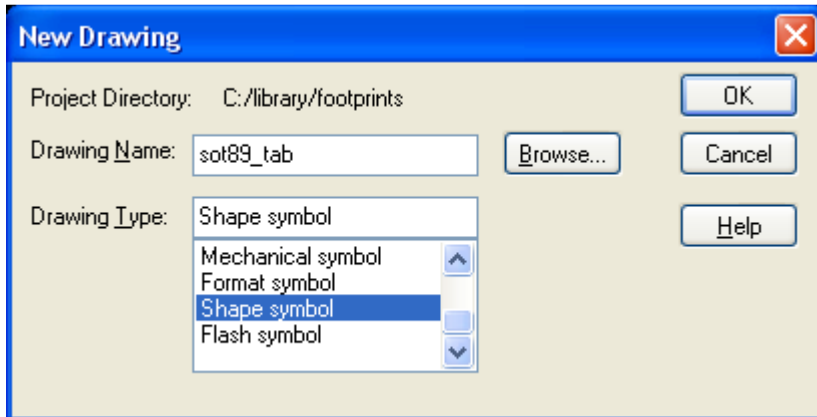


How to Create an alternative shaped pad/pin (Like a SOT89 Tab pin).

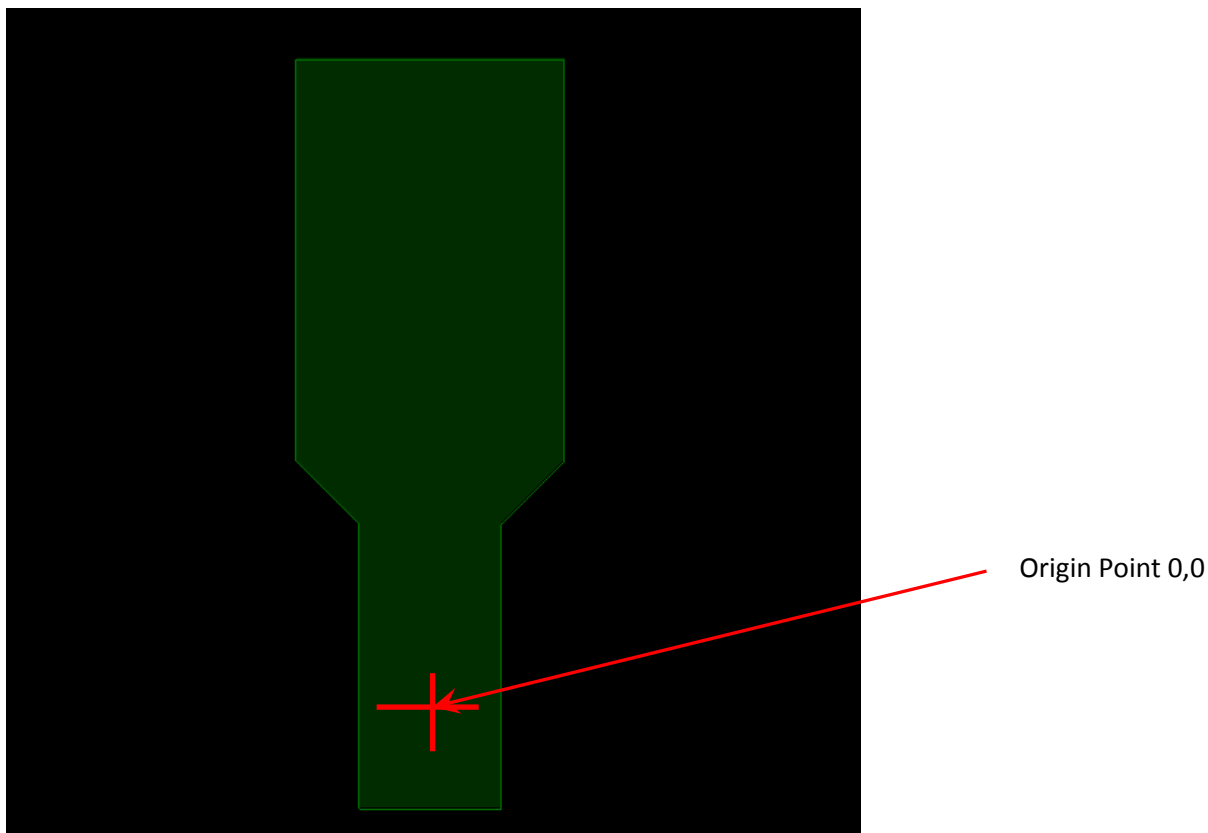
Start PCB Editor File → New → Shape Symbol and enter a name



Once in shape symbol editor draw your shape using Etch – Top to create the shape of the tab pin. Most Important part of this step is to ensure that the origin point 0,0 is at the location inside the shape where you want the pin connection point to be. You can use the command line to enter specific co-ordinates to create this shape so that 0,0 is at the correct location. For the SOT89 tab pad enter the following at the command line after invoking the add shape command.

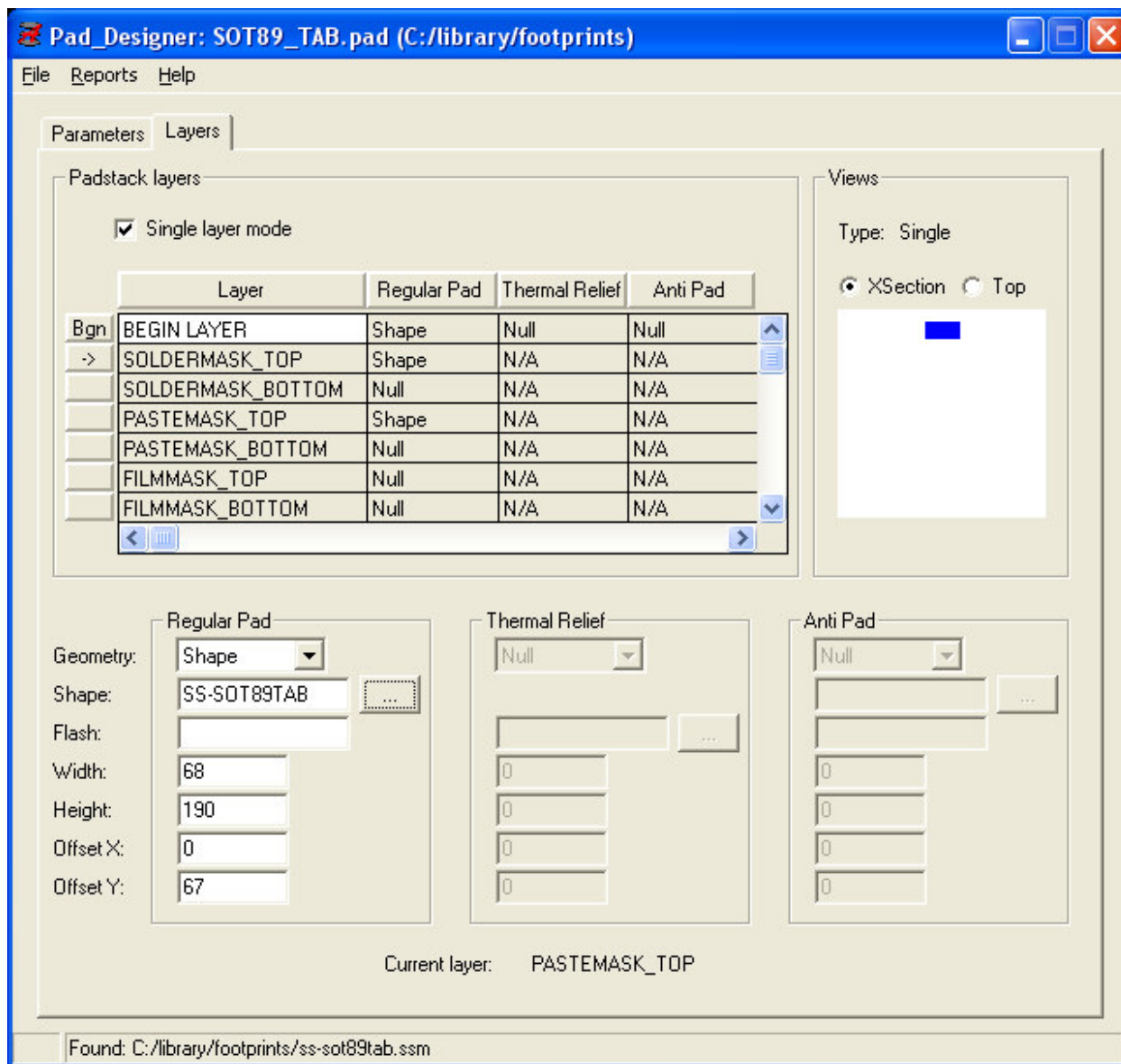
x -18 -28, ix 36, iy 72, ix 16 16, iy 102, ix -68, iy -102, ix 16 -16, iy -72.

This will draw the shape shown below guaranteeing that 0,0 is at the centre of the pad shown.



Save the shape file as sot89_tab.ssm in the pad/psm path location.

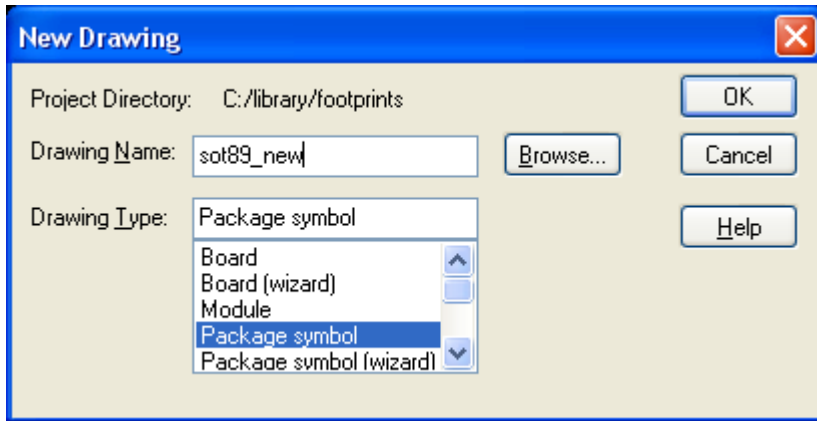
Open Pad Designer (Start → All Programs → Orcad16.x → Orcad PCB Editor Utilities → Pad Designer)



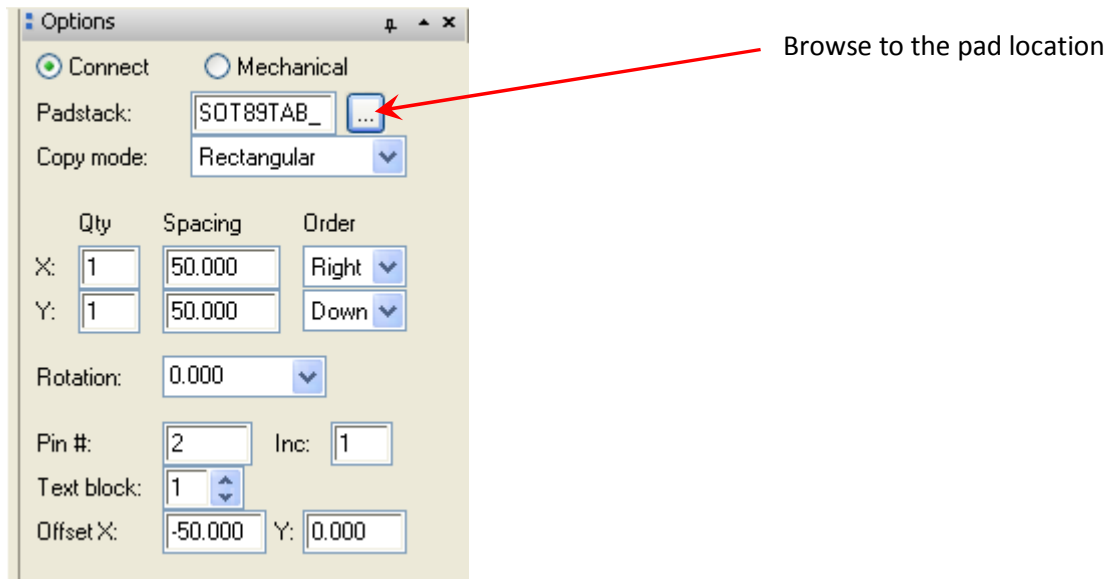
Because this is a surface mount pad I have selected the single layer mode type. Under the layers tab instead of defining geometry (circle, square, oblong etc) check the button next to Shape and browse to the ssm file saved earlier. Repeat this step for Soldermask and Pastemask. Remember if you under or oversize the pads for these layers you will need to generate different shape.ssm files as required. Once complete save the pad in the pad/psm path location.

You can also use this method for Plated through holes, just remember that the 0,0 location on your shape file will be where the drill hole is centred.

To use the pad when you create your symbol (dra) file go to PCB Editor File → New → Package Symbol



To add pads use Layout→Pins. Browse to the pad name shown in the options menu



Add the newly created pad as you would normally.

