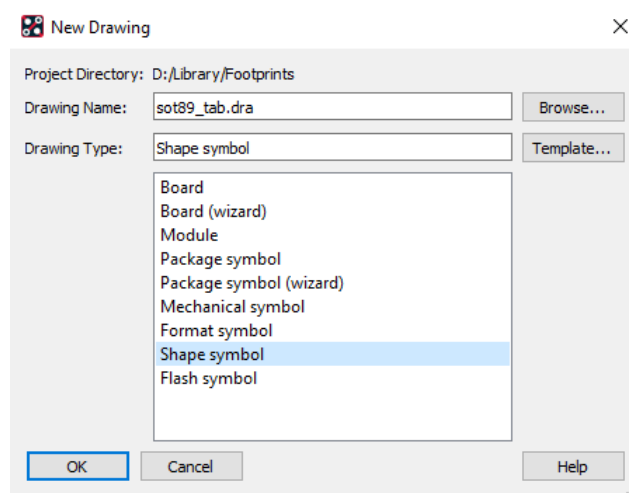## Introduction

The Cadence® OrCAD® and Allegro® Padstack Editor allows for the creation of the following pad shapes: - circle, square, oblong, rectangle, rounded rectangle, chamfered rectangle, octagon, donut and n-sided polygons, however there are times when something more complex is required, in this situation a shape needs to be created and used. This application note will show you how to create and use a shape for use as a SOT89 tab pin.

How to create an alternative shaped pad or pin

Launch PCB Editor, and create a new shape symbol: File > New… will open the New Drawing Dialogue, then change the 'Drawing Type' to Shape Symbol, enter a 'Drawing Name' and click OK.

Once in shape symbol editor draw your shape using Etch\Top to create the shape of the tab pin. The 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.
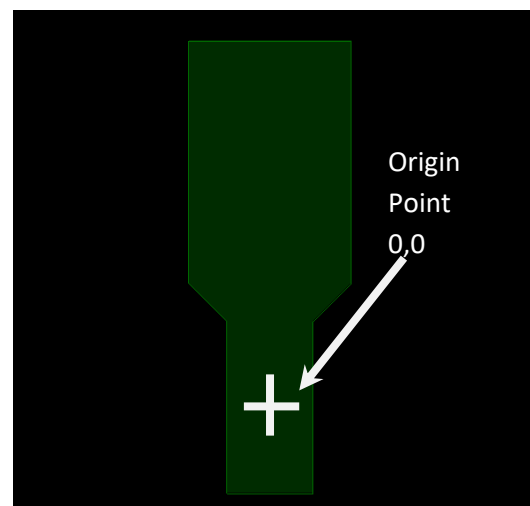

For the SOT89 tab pad enter the following at the command line after invoking the add shape command.

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

An alternative method is to add this complete line at the command line and then the shape is drawn in one go: - x -18 -28; ix 36; iy 72; ix 16 16; iy 102; ix -68; iy -102; ix 16 -16; iy -72. By using the semi-colon means you can add a sequence of commands to draw a completed shape.
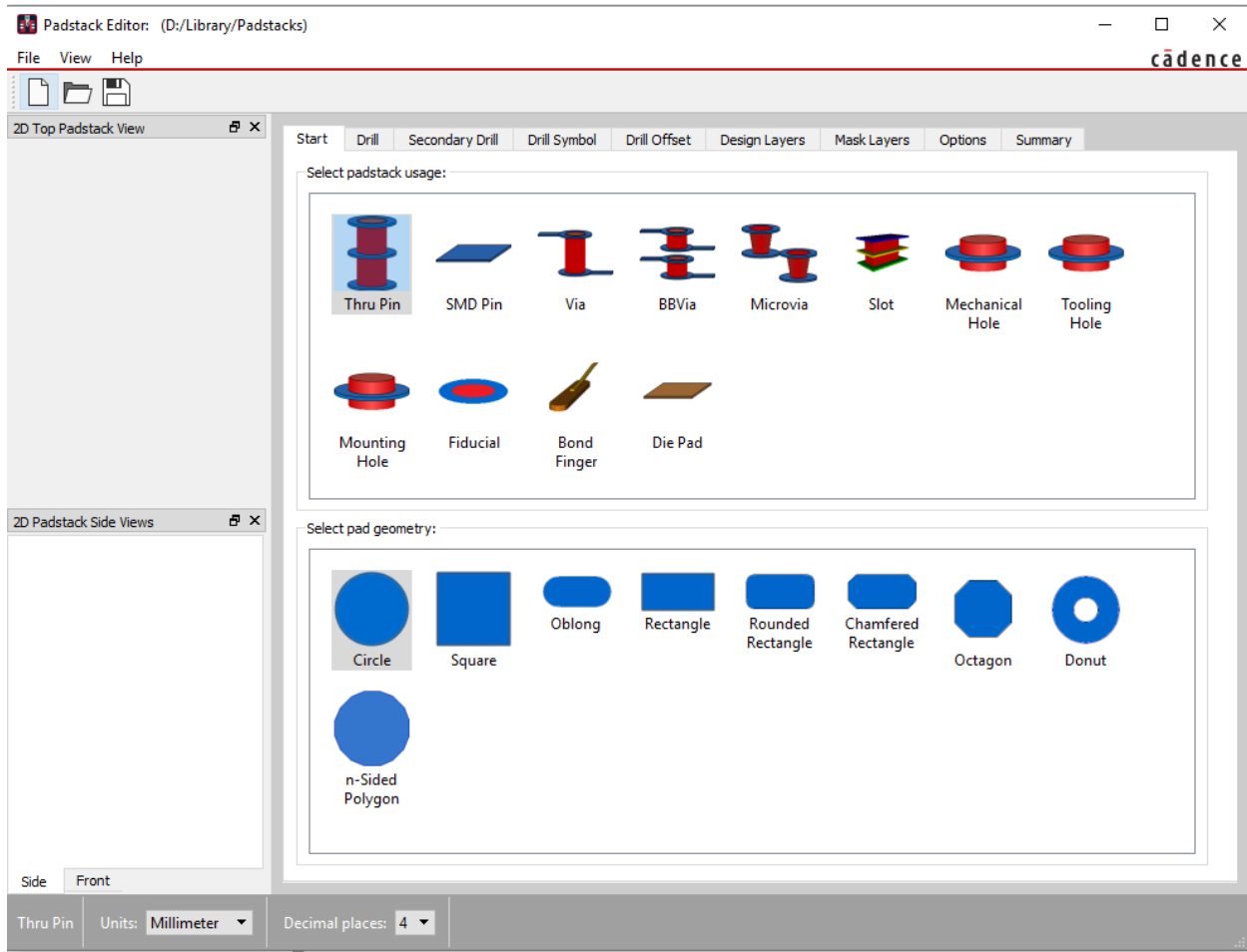
This will draw the shape shown above guaranteeing that 0,0 is at the centre of the pad shown.

Save the shape file as sot89_tab.ssm in the psmpath location.

Open Pad Designer (Start > All Programs > Cadence PCB Editor Utilities XX-XXXX > Padstack Editor XX.X)


Select File > New, specify a name for the padstack ensuring the save directory is set to the padpath location. Set the Padstack usage to SMD Pin. Set the units and decimal places as required. On the Design Layers tab change the Geometry dropdown for Regular Pad on Layer BEGIN LAYER from None to Shape Symbol.
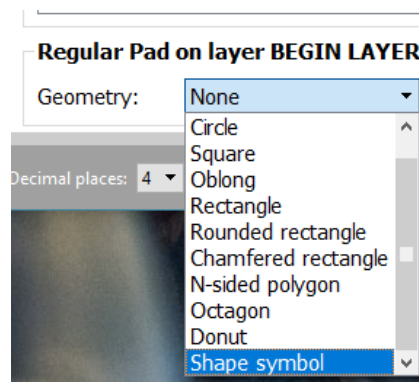
Select the browse button next to Shape symbol and choose the shape symbol you generated earlier. Another method if you haven't already created the shape symbol is to select the "Create New Shape Symbol" option which will launch PCB Editor and start a new shape symbol based on the name you specify. Once you have chosen the shape symbol and selected OK the shape name is applied to the Regular Pad for BEGIN LAYER.
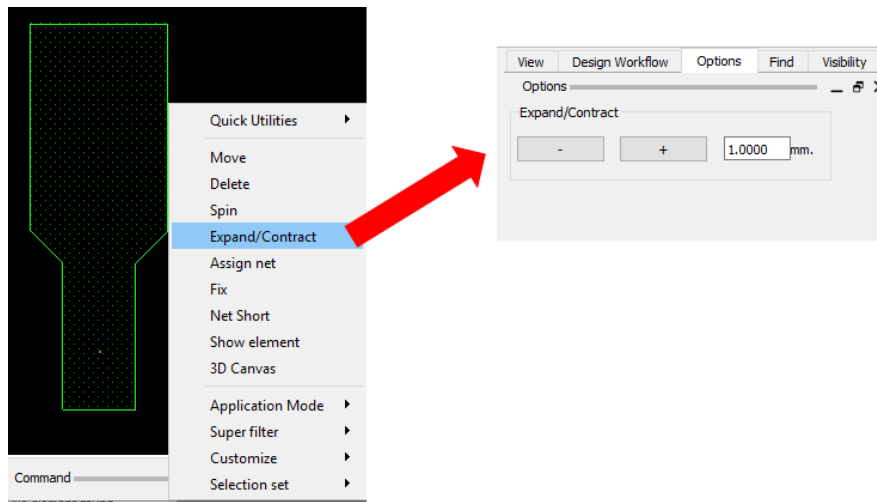
]
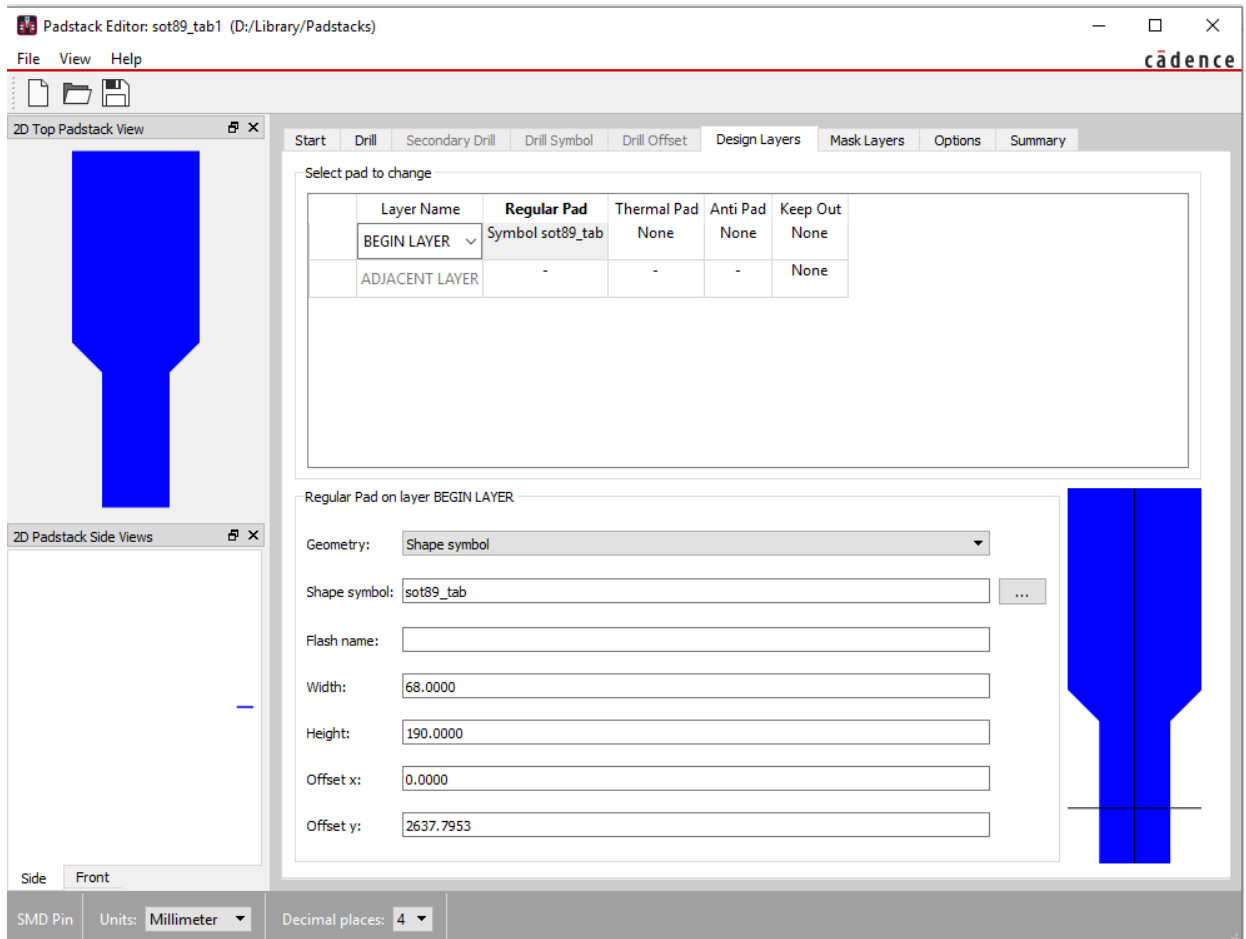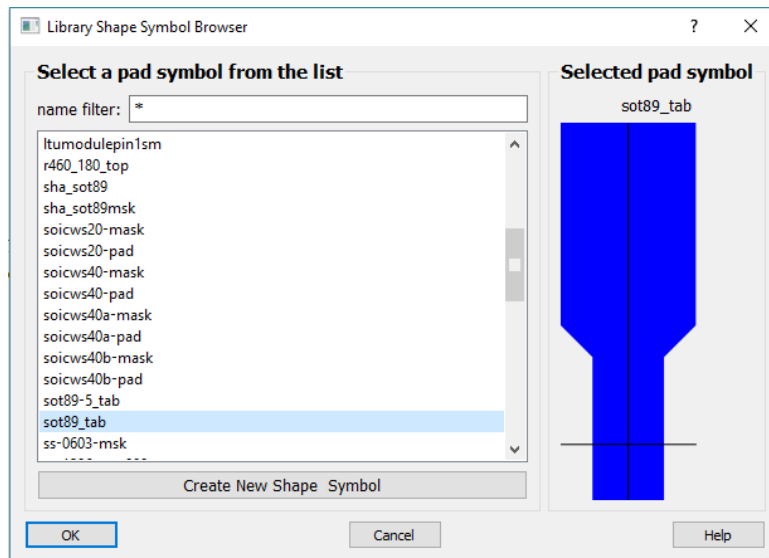
Repeat these steps for the soldermask and pastemask definitions via the Mask Layers tab. Remember if you under or oversize the pads for these layers you will need to generate different shape.ssm file as required. Once complete save the pad in the padpath location.

There is a useful option if you need to oversize/undersize the shape for the mask layers. Open the shape symbol then hover over the shape boundary until it highlights (ensure you are in General Edit App mode first) then right click > Expand/Contract and the set the values via the Options Pane. Then use File > Save As to save as a soldermask or pastemask version.

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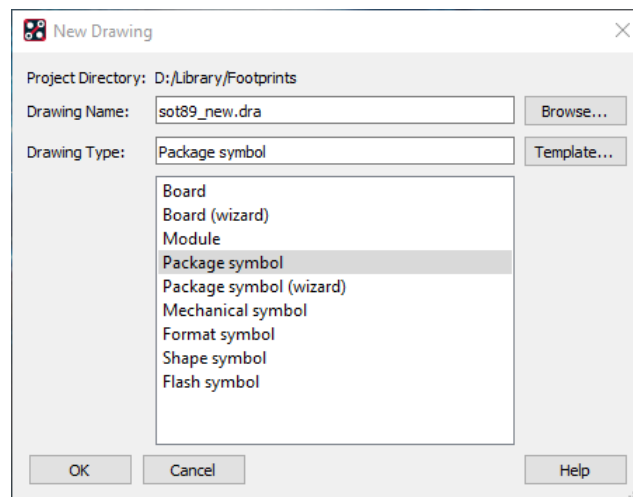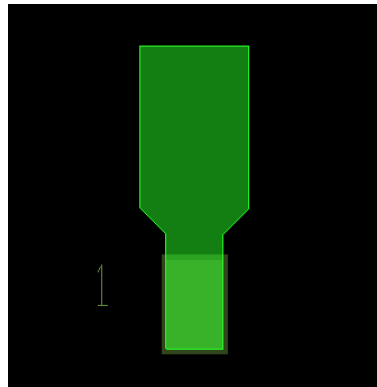

To add the pins use Layout > Pins. Browse to the pad name shown in the options menu
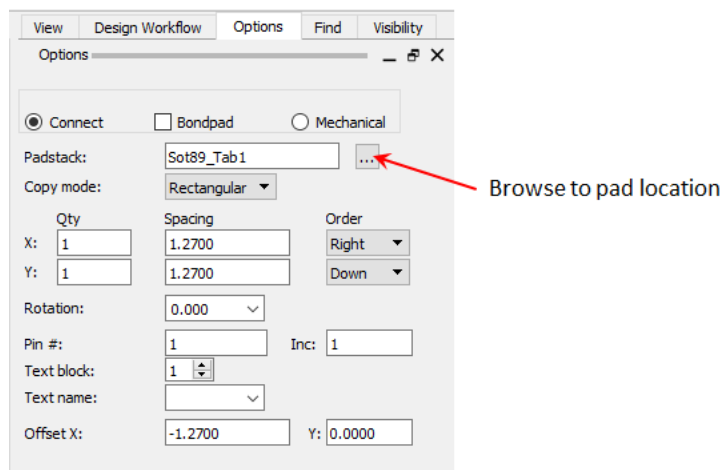
Add the newly created pad as you would normally.

Creating padstacks for through hole pins using shapes

The same principle can be used for creating padstacks for through hole pins. In this case the same Shape is assigned to all of the padstack layers, Begin Layer, Default Inner and End Layer. If the Soldermask and Pastemask layers are going to be the same, the same shape can also be assigned to them. If different shapes are required for Soldermask and Pastemask layers, these can be designed as shape symbols in PCB Editor and then used by the Pad Designer. Use the Expand / Contract option for the shape as described above.

Note that, even though shape symbols in PCB Editor can only be defined on the Etch / Top and Etch Bottom layers, the Pad Designer and the Package Symbol Editor in PCB Editor will correctly process the shape to the required board design layer.
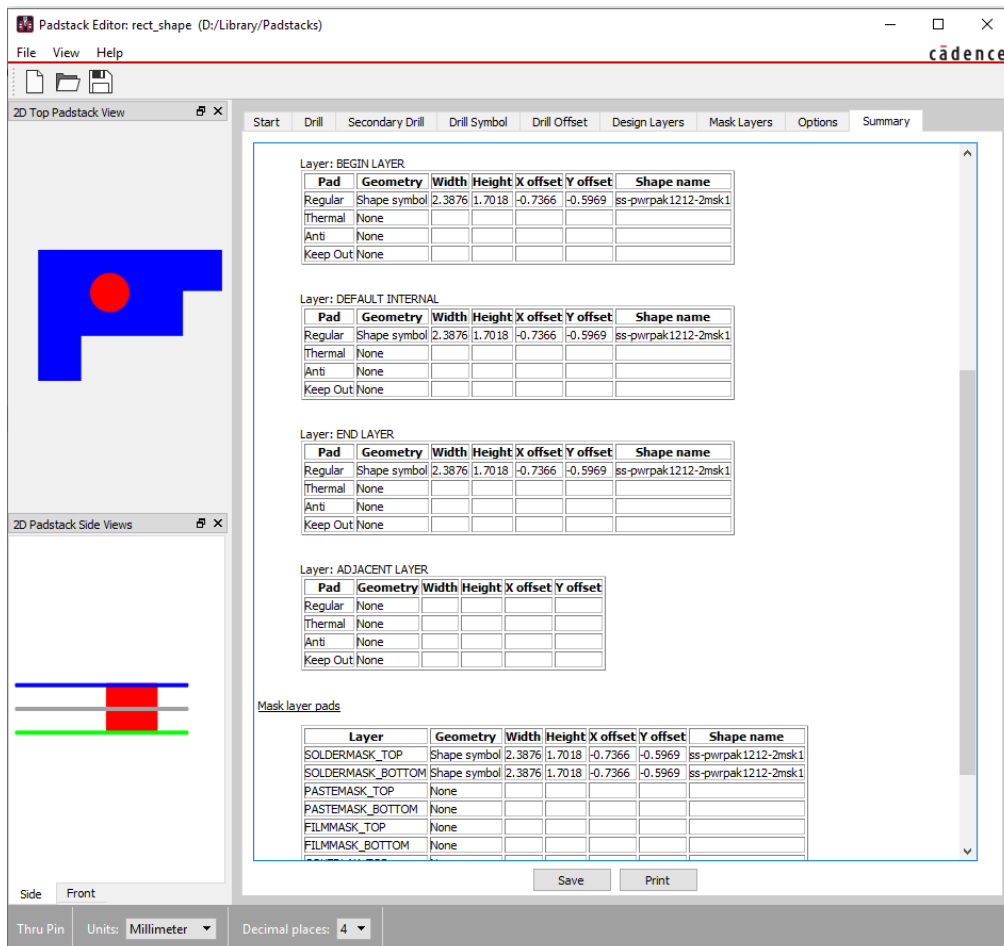
In this case, the same shape has been assigned to all layers, the shape was added to the Begin Layer and then Copied to all the other layers using CTRL+C and CTRL+V including the Soldermask and Pastemask layers on the Mask Layers tab.

Define the drill as normal on the Drill tab:
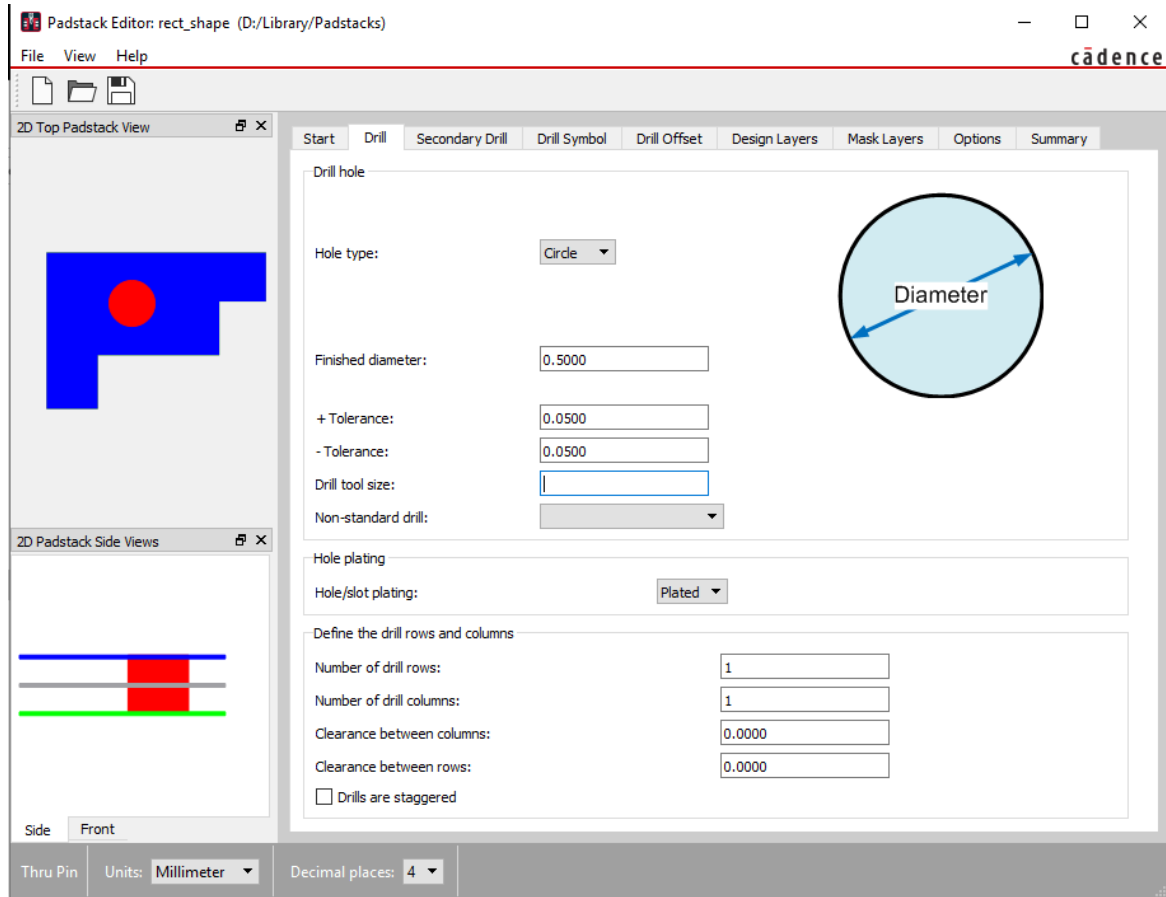
Don't forget to define a Drill Symbol on the Drill Symbol tab or accept the warning.

Save the padstack definition to a "padpath" location so that PCB Editor can find it. Then File > New, Package Symbol and design a symbol using the new padstack.