Introduction

Slots are a common occurrence in PCB Designs today. They allow "float" for fixing hole locations or may be there for other reasons. This App note describes how to create the slot using the Cadence Padstack Editor. It also covers the steps required to output the data for fabrication. It is important to note that this App note only covers the settings for slots in Padstack Editor. For the full explanation of the Padstack Editor settings please refer to the Cadence documentation.

How to create a Slot

Start by designing the slot using the Padstack Editor from Start – All Programs – Cadence – Release XX.X - PCB Editor Utilities – Padstack Editor.
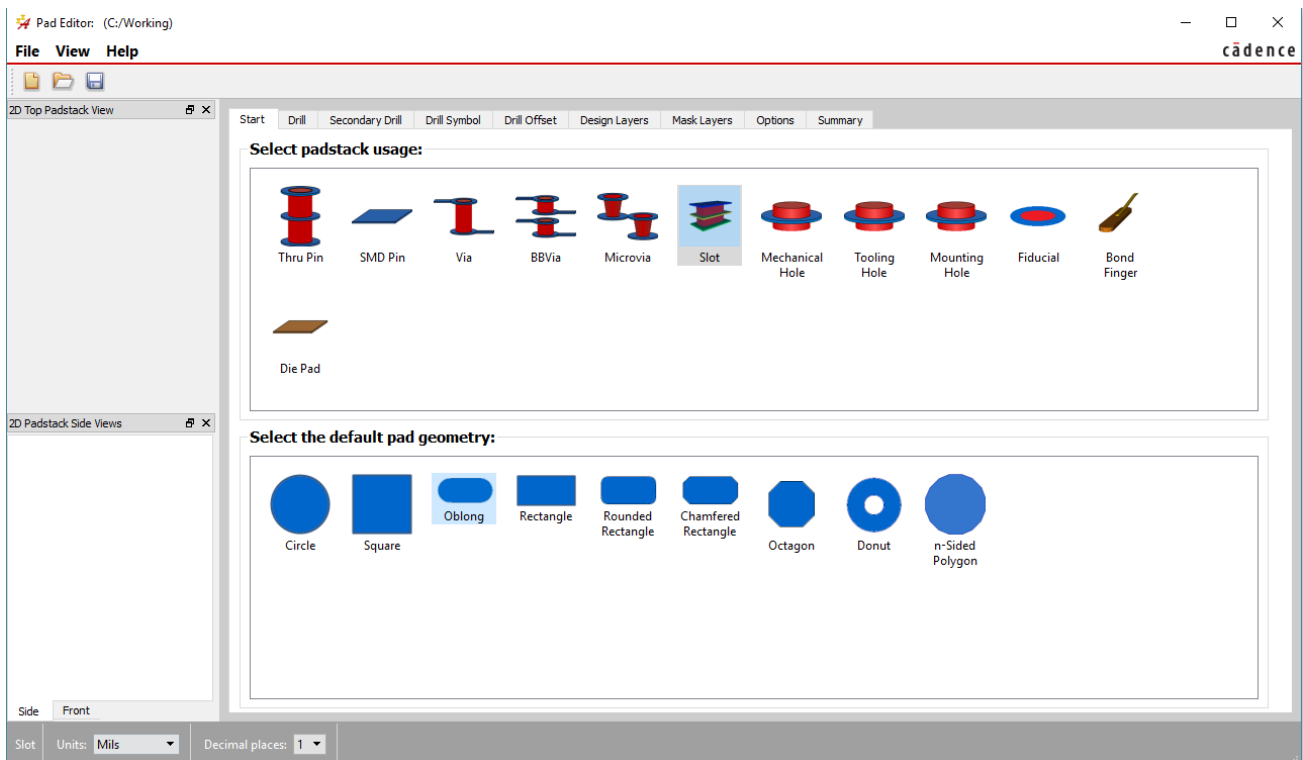

Define a name using File – New then specify units, decimal places, set the padstack usage to be slot and the choose a default pad geometry. For this example, we are going to design a 1mm x 2mm Oval slot with a drill tolerance of ±0.3mm. Now select the Drill tab and choose oval slot as the slot type. The Slot Y size is the diameter of the slot and the Slot X size is the length of the slot based on overall length. Set also the X and Y tolerances to match ±0.3mm. You also have to choice and to whether the slot is Plated or Non-Plated. Once the slot sizes are entered you can see a preview of the slot drill size.
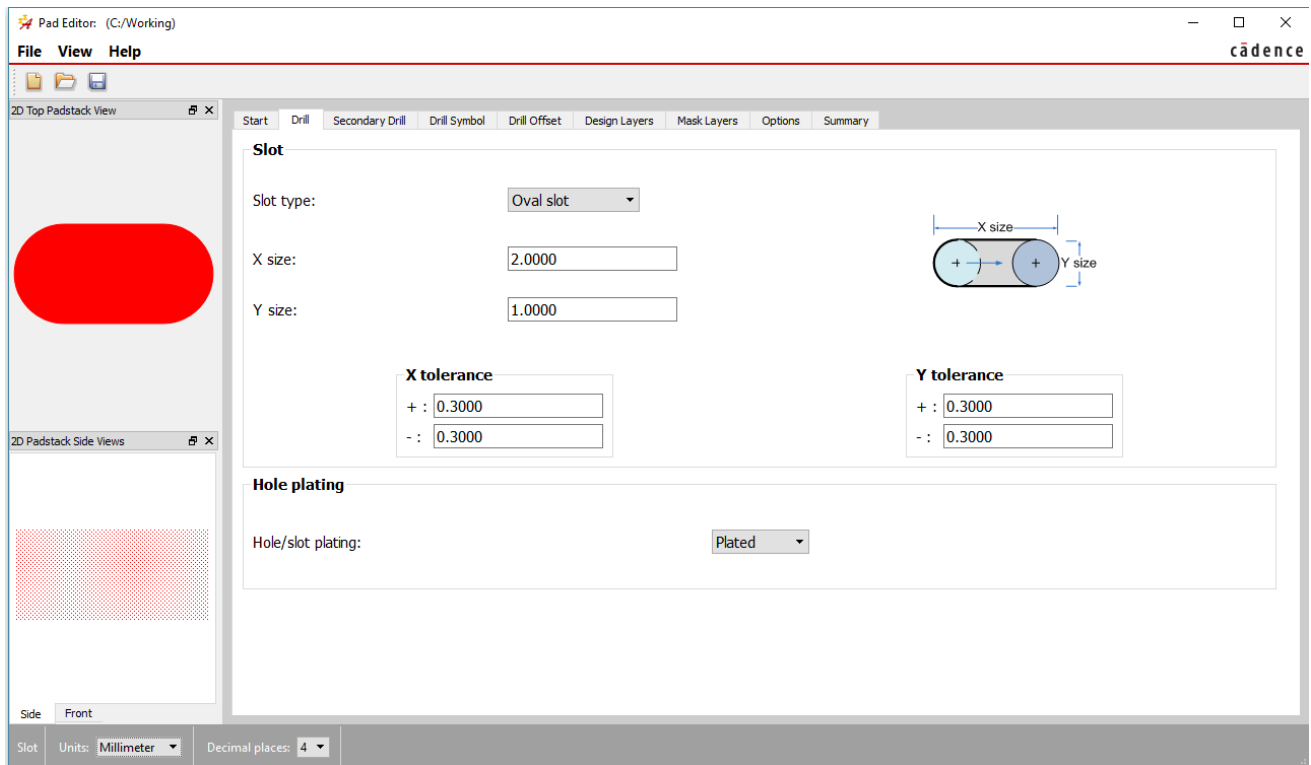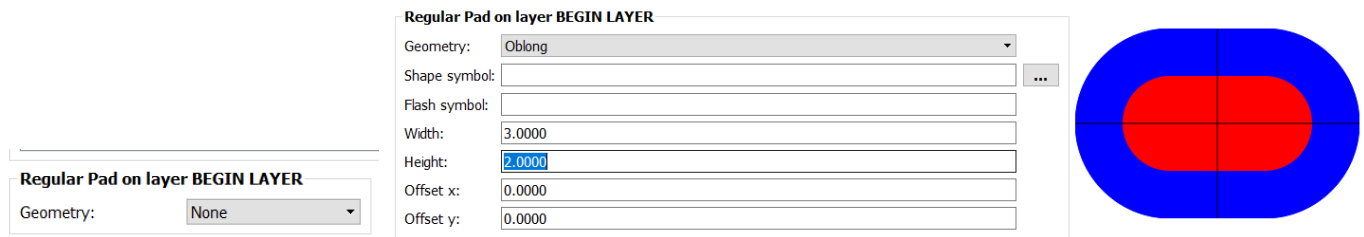
Now choose The Drill Symbol tab to define the slot symbol Figure. The size of the figure is defined by default, based on the slot x and y sizes. You can add a Character if required. These are used for the NC Drill Legend in PCB Editor.
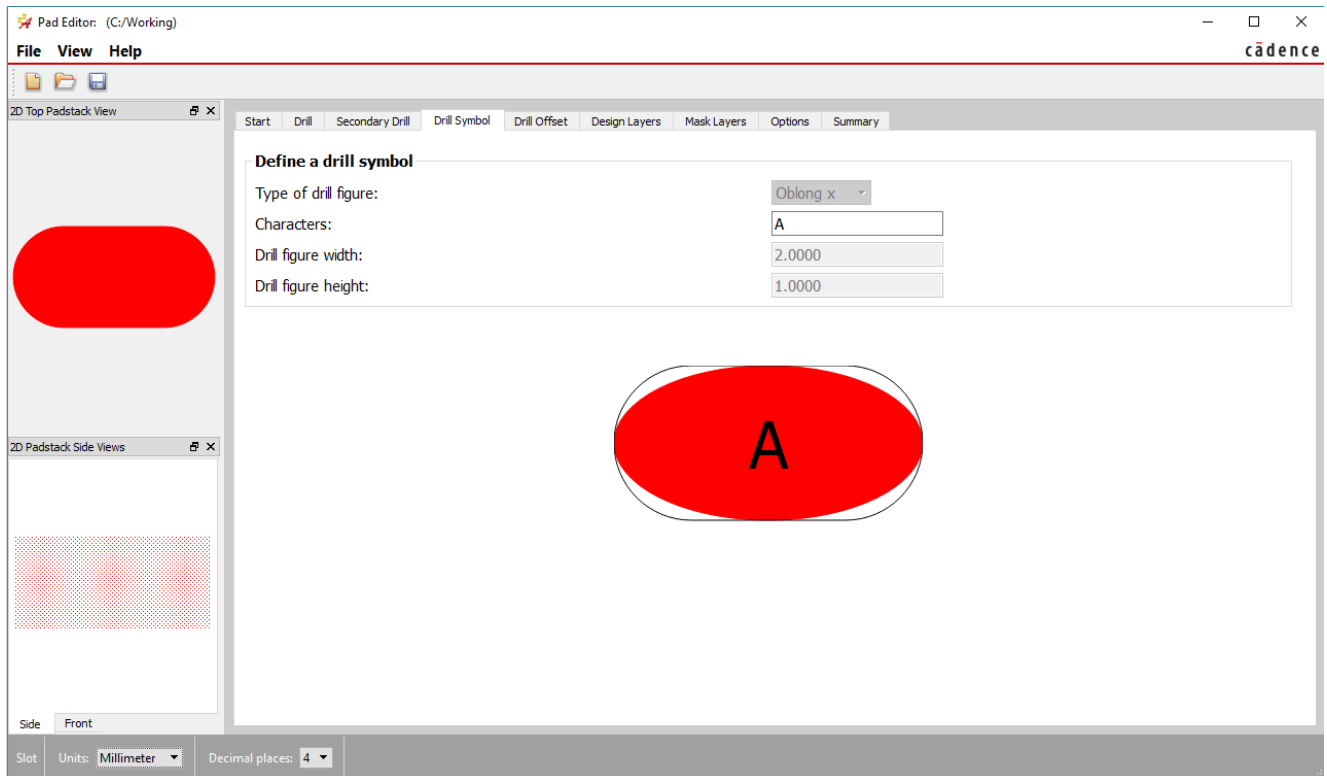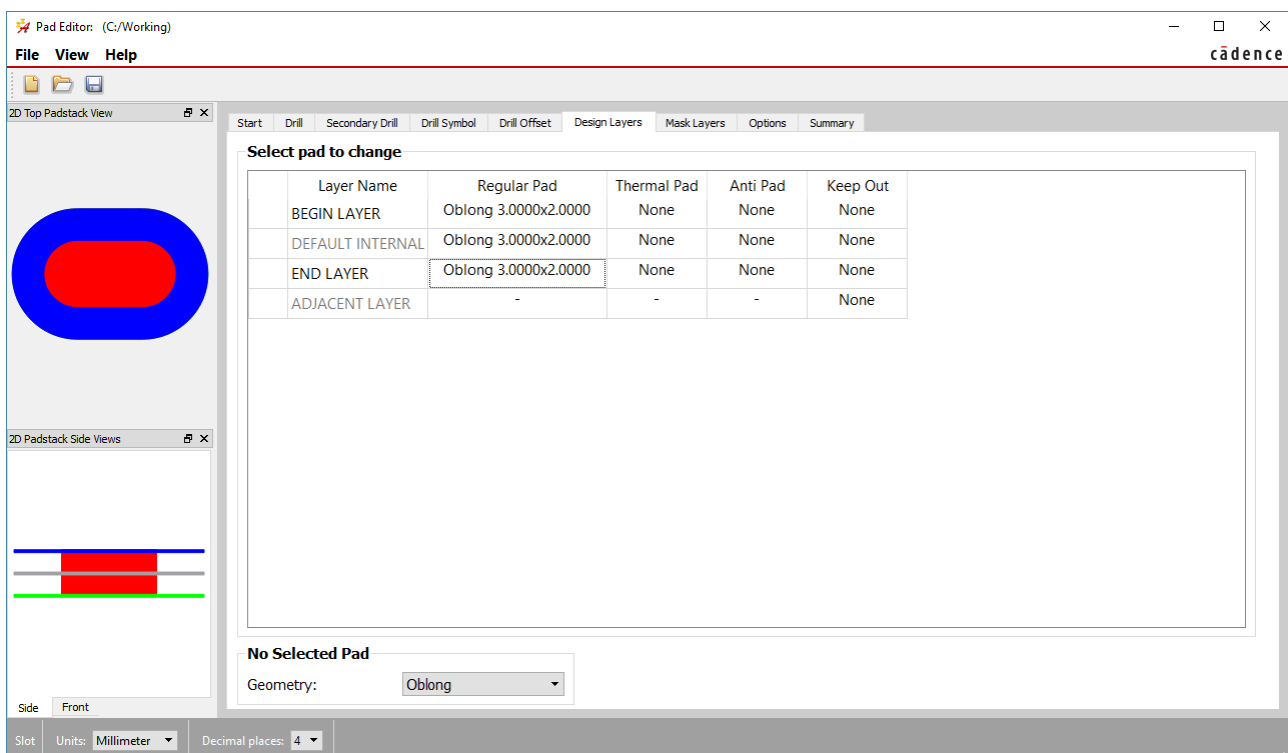

We now need to define the Design Layers (actual pad sizes required for our Plated slot). Choose the Design Layers tab.

Depending on if the slot is a mechanical based slot then you may only need to specify Solder Mask layers using the Mask Layers tab. For this example, we are going to design a plated slot so need the Pads Definition to cover the Begin, Default Internal and End layers.

Change the Geometry for the BEGIN Layer dropdown from None to Oblong in this example and then enter values for the width and height sizes of the slot. You can also enter offsets for the slot centre (if required) or even use a shape symbol for the pad geometry. In this example I have increased the pad size to be 1mm larger than the actual slot drill size giving an overall increase of 0.5mm . You may need to confirm this with the IPC 7251 standard or the manufacturers datasheet to match your requirements.

Regular Pad on layer BEGIN LAYER

Geometry:	Oblong
Shape symbol:	
Flash symbol:	
Width:	3.0000
Height:	2.0000
Offset x:	0.0000
Offset y:	0.0000

Regular Pad on layer BEGIN LAYER

Geometry:	None

Once the BEGIN layer has been defined we can use CTRL+C (Copy) and CTRL+V (Paste) to define the Default Internal and END layer definitions. In this example we are not defining thermal or anti-pads. Most modern designs use positive dynamic shapes and PCB Editor rules can be used to define clearances / thermal reliefs to shapes using Constraint Manager.

Pad Editor: (C:/Working)

File   View   Help

Select pad to change

Layer Name	Regular Pad	Thermal Pad	Anti Pad	Keep Out
BEGIN LAYER	Oblong 3.0000x2.0000	None	None	None
DEFAULT INTERNAL	Oblong 3.0000x2.0000	None	None	None
END LAYER	Oblong 3.0000x2.0000	None	None	None
ADJACENT LAYER	-	-	-	None

No Selected Pad

Geometry: Oblong

Slot   Units: Millimeter   Decimal places: 4

Once defined, you will be able to see a Top and side view of the defined pad stack in the Views window. You can also define Soldermask's and Pastemask's accordingly to complete your pad design by choosing the Mask Layers tab and repeating the steps above. The Pad is then saved to the relevant padpath library for use.
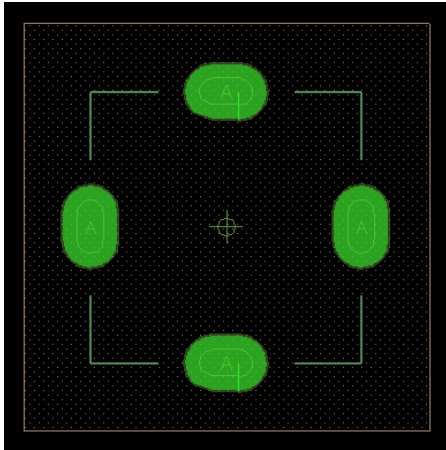
The Pad (slot) can now be used in a symbol (Package or Mechanical). The slot is added exactly the same way you add any other pin. Use the Layout – Pins command, then from the options menu browse to select and use the Pad you designed. This can be a connect pin or a mechanical pin for a Package Symbol and a mechanical pin only for a Mechanical Symbol. The example below is a Mechanical Symbol. The slots have been added, along with a Via and Route Keepout area and some Assembly lines to represent the mechanical body outline. For further details on symbol creation please refer to the Cadence documentation.

Using the symbol slots in a Board File

Mechanical symbols can be used in a board file using the Place – Mechanical Symbols (OrCAD) or Place – Manually (Allegro) command, change the Components by refdes to Mechanical Symbols (you may also need to select Library from the Advanced settings tab) to see the list of available symbols. Pick the part you require and place on the board.

The NC Drill legend does reference the slots (as shown below) but the NC Drill does not reference the slots. You will see a WARNING in the NC Drill file that the slots cannot be drilled and must be routed:-
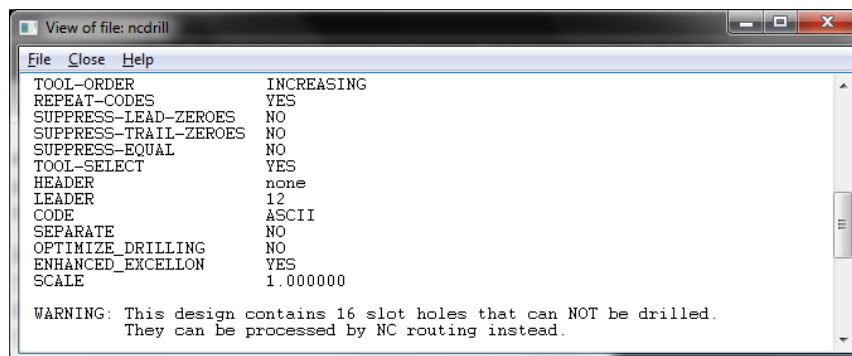
If you are using version 16.5 or earlier before you create a NC Route file you will need to create an ncroutebits.txt file. Since version 16.6 this file is created automatically. The ncroutebits.txt file specifies the tool size assigned to a specific tool number. The ncroutebits.txt file should include a tool for each minor dimension of all the slotted holes and any additional routing information. In our example the slot is 1mm x 2mm so the minor width of 1.0 is

defined and assigned to T01. The line width on the BOARD GEOMTERY/NCROUTE_PATH is 2.54mm so 2.54 is defined and assigned to T02. The ncroutebits.txt file can be stored in the same location as the board file.

```
1.00 T01
2.54 T02
```

For routing of the PCB profile you can if you choose define a start and finish point. The direction in which a route path is cut can be controlled by adding the character '1' at (or near) the vertex that you want to be the start point of the cut. This character (referred to as a 'start point indicator') is added by using the ADD TEXT command and adding text on the layer BOARD GEOMTERY/NCROUTE_PATH. In the case of a complete loop, direction can be specified by adding the character '1' at the first point on the path and the character '2' at the second point along the path.

To run the route tools, use Manufacture – NC – NC Route (Allegro) or Export – NC Route (OrCAD). Define the NC Parameters if required, and then click on Route. The Route file filename.rou is stored in the job directory. Below is a partial output of NC Route.

```
M48
METRIC
T01C1.
T02C2.54
;EXTENTS: -16.002 -13.462 37.338  29.718
;LEADER: 12
;HEADER:
;CODE  : ASCII
;FILE  : filename.rou for board filename.brd
%
G90
F1
M16
T02
M16
G00X-01270Y-01270
M15
G01X-01270Y52070
G01X52070Y52070
G01X52070Y-01270
G01X-01270Y-01270
M16
```