

NCTU IEE 5046

高頻電路設計與實驗

PCB design with Pad Designer

Lecturer: Professor Yu-Jiu Wang

TA: 李道一 michael@rfvlsi.ee.nctu.edu.tw

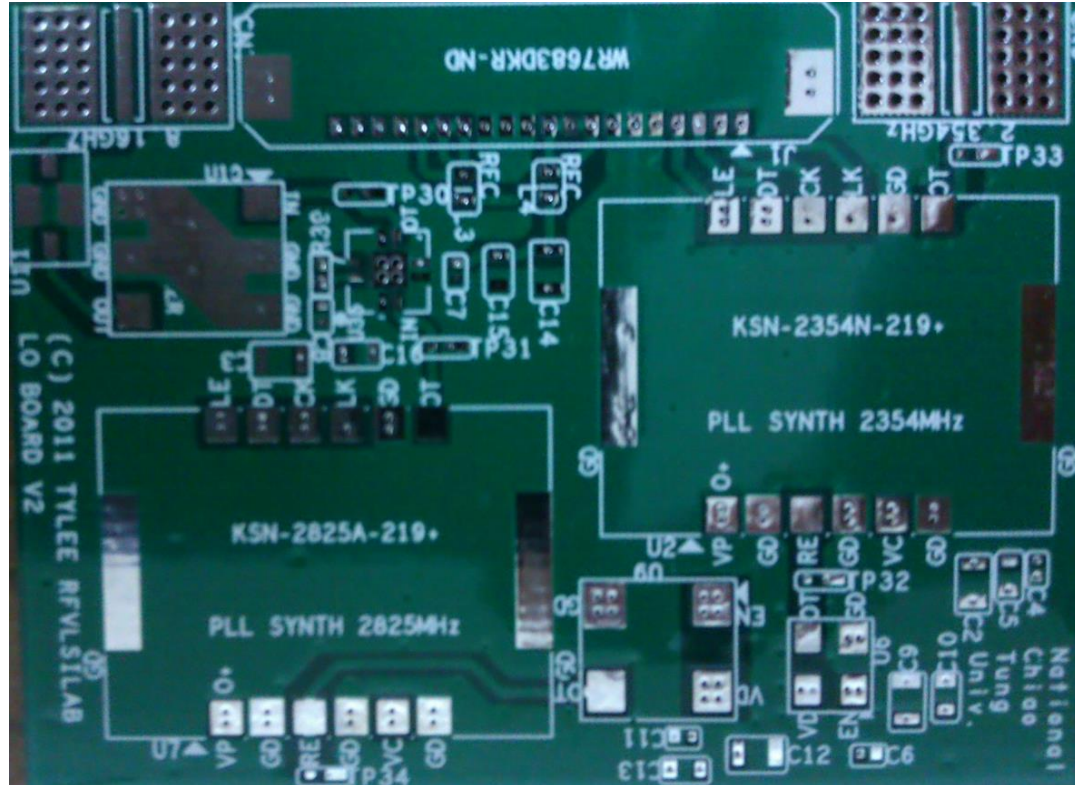
Document coauthor: Jon-Jin Chen

Related EDA tools

- Pad Designer  Pad Designer

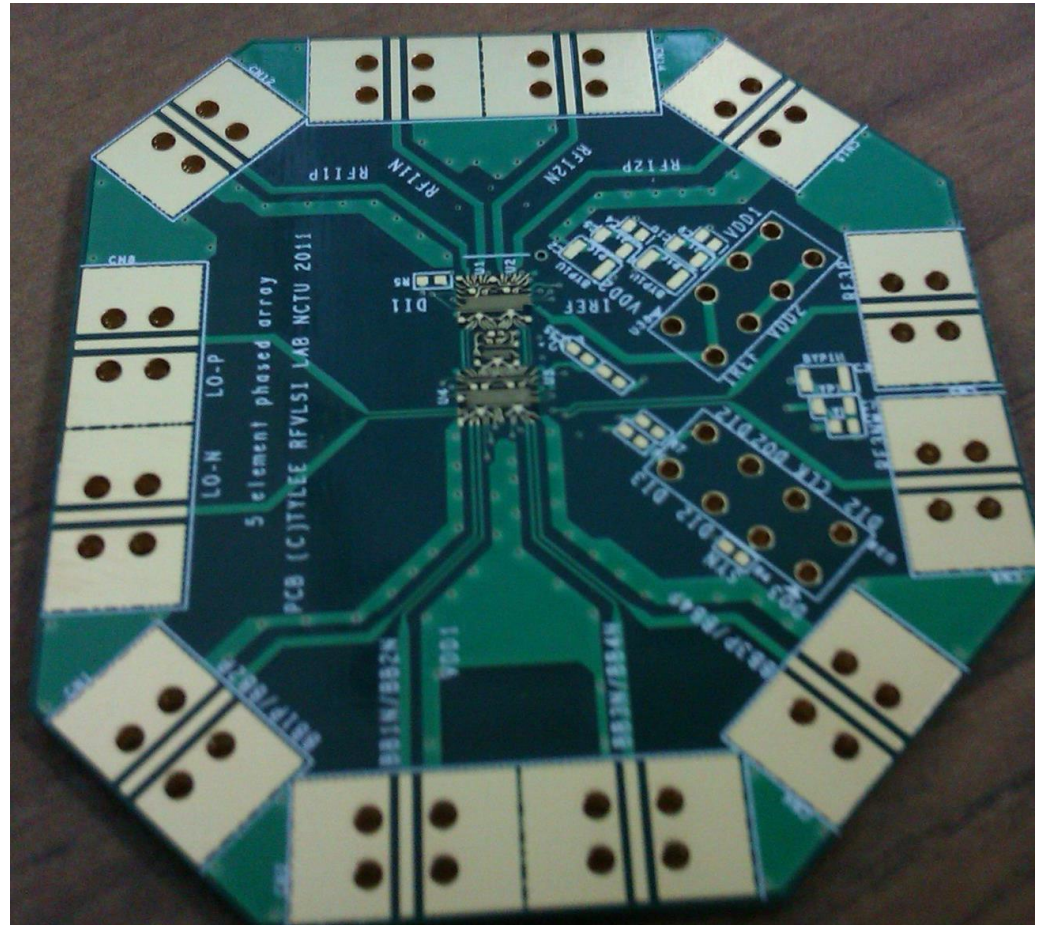
PCB basics

- Solder mask
 - Dark Green
- Etch layer
 - i.e. Conductors
- Drill holes
 - Look closer to the pads
- Silk Screen
 - White characters and lines



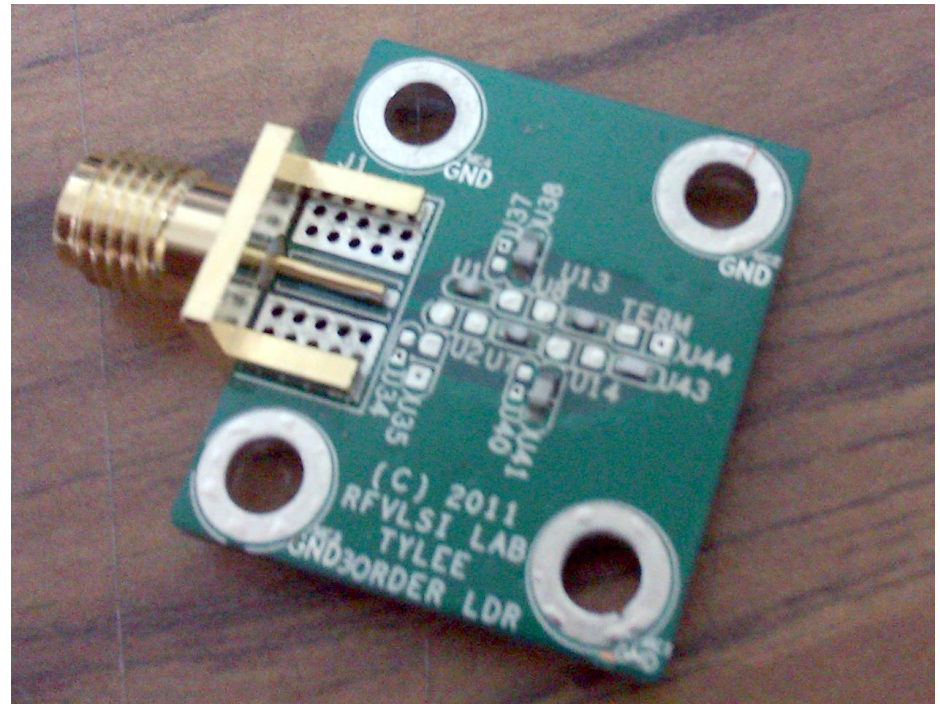
High Frequency PCB

- Impedance control
- GND plane
- Cable to board transition
- EMI/EMC

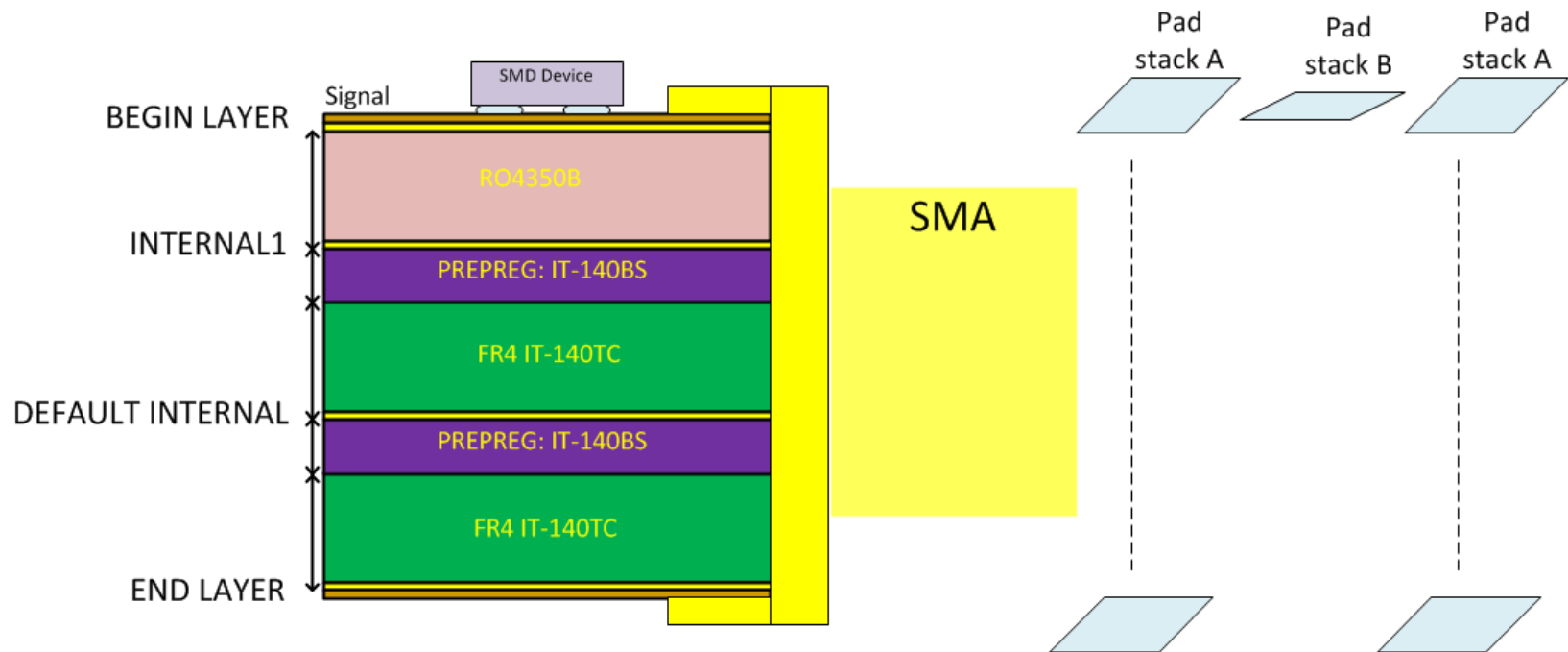


SMT Soldering

- Solder
- Solder paste
- Solder iron

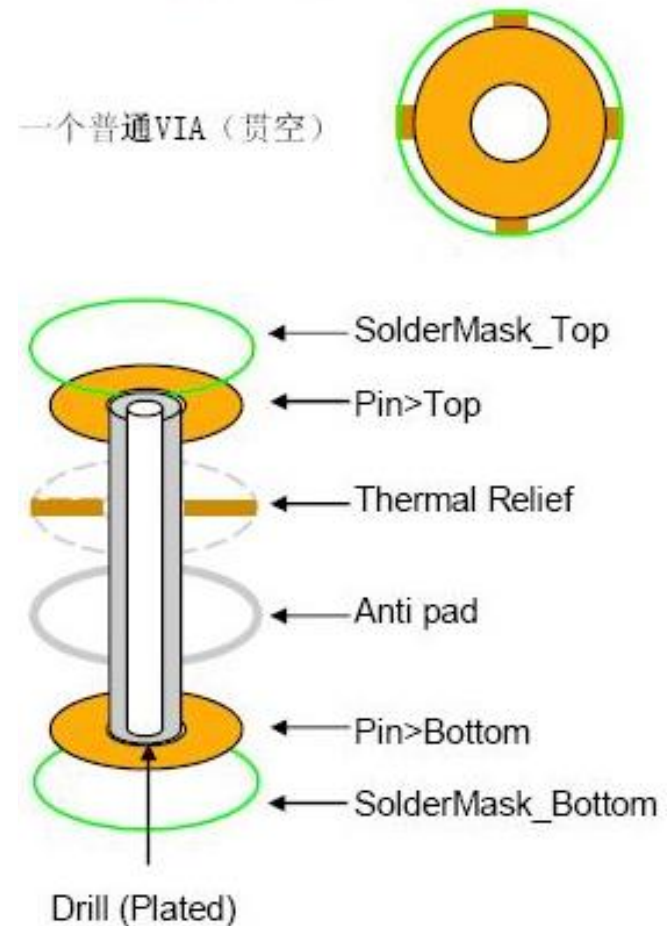


Pad-Stack (4 Layer Stack Up)



Definitions

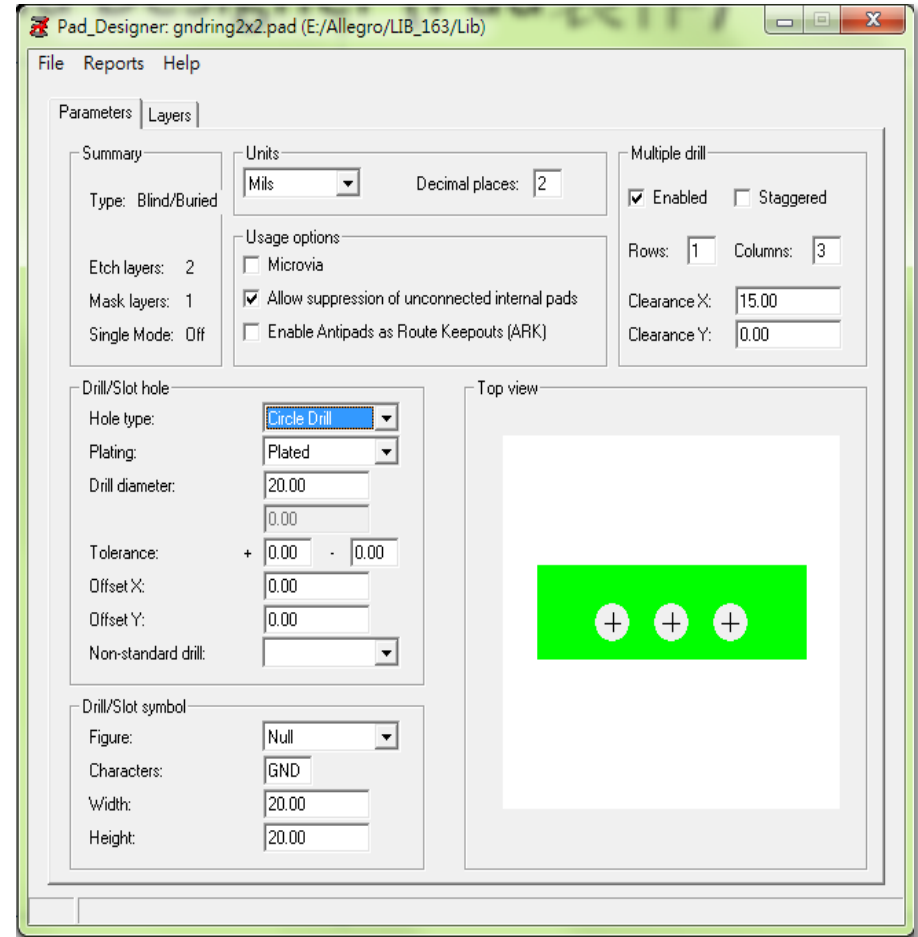
- Copper pad
- ***Antipad***
- Thermal Relief
- Soldermask
- Drill
 - Plated: Conductive
 - Non-Plated: Isolated



Create a Pad in *Pad Designer*

“Parameters” Tab

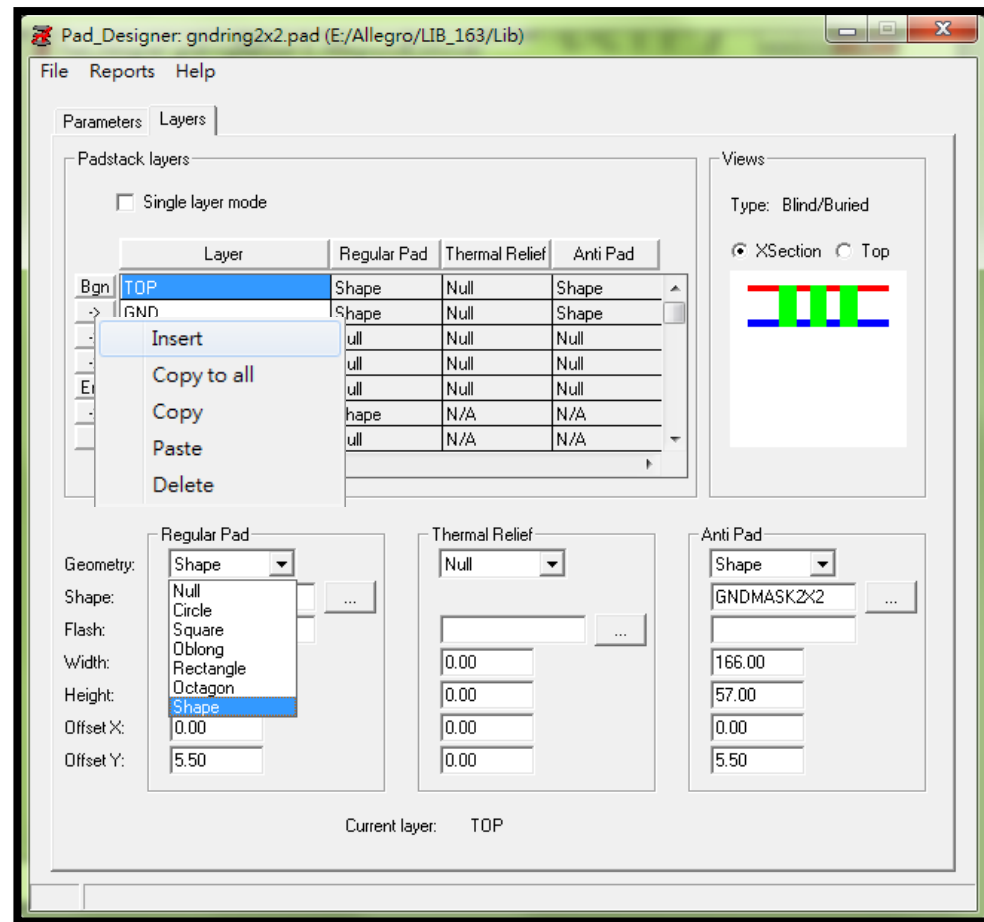
- Start Pad Designer
- Switch to “Parameters” tab
- Set units to “mils” with decimal places 2
- Decide whether to place drill holes on pad, if no drills needed, set drill diameter to 0.
- Do not use rectangle shaped drill holes in your project
- Drill/Slot symbol: optional
- Define multiple drill on the upper right pane, check “enabled” to enable; Specify clearances if needed.



Create a Pad in Pad Designer

“Layers” Tab

- For SMD: Check “Single layer mode”
- Right click on “Bgn” to insert layers
- Select regular pad shape and specify size
- **Antipads**
 - for PCB with internal conducting layers, define antipads
 - should be larger than regular pad , 6 mils around
- **Solder-masks** should also be larger than regular pad
- Save your *.pad to the specified “PADPATH” in Allegro



Create *antipad* / *soldermask* for a padstack

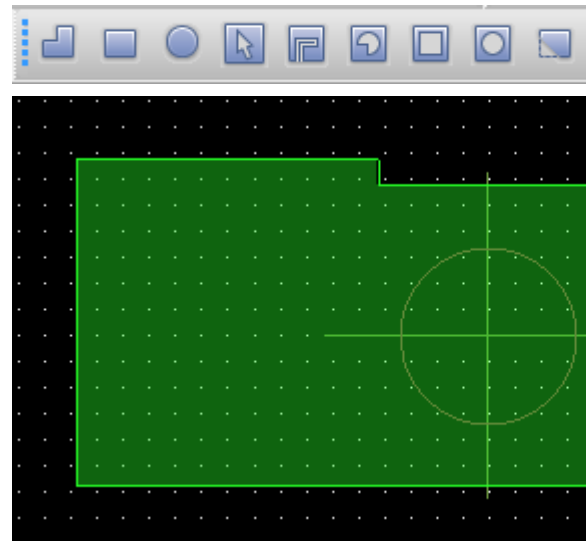
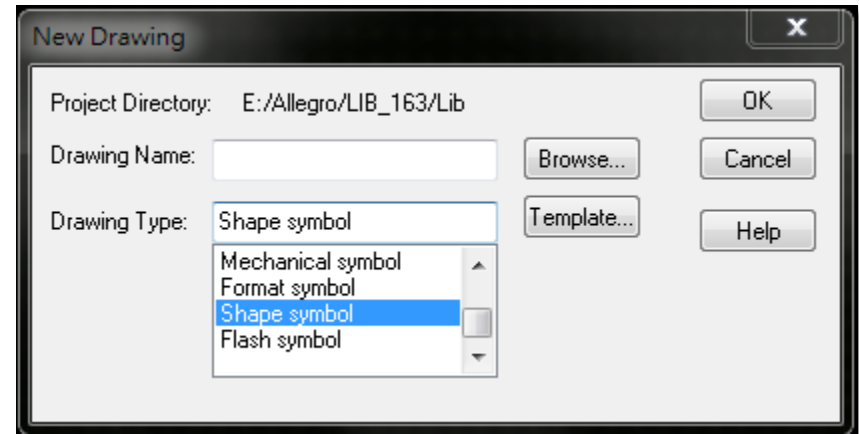
- Generally, antipad and soldermask encloses regular pad by 3 mil (Depends on the min. etching gap of PCB process)

	Circle	Rectangle	Shape
Regular Pad	R mil	W mil \times L mil	Manually assign
Antipad	R+6 mil	W+6mil \times L+6 mil	Manually assign (Align Origin)
Solder Mask Top	R+6 mil	W+6mil \times L+6 mil	Manually assign (Align Origin)
Thermal Relief	R+6 mil	W+6mil \times L+6 mil	Manually assign (Align Origin)

Special Shaped Pads

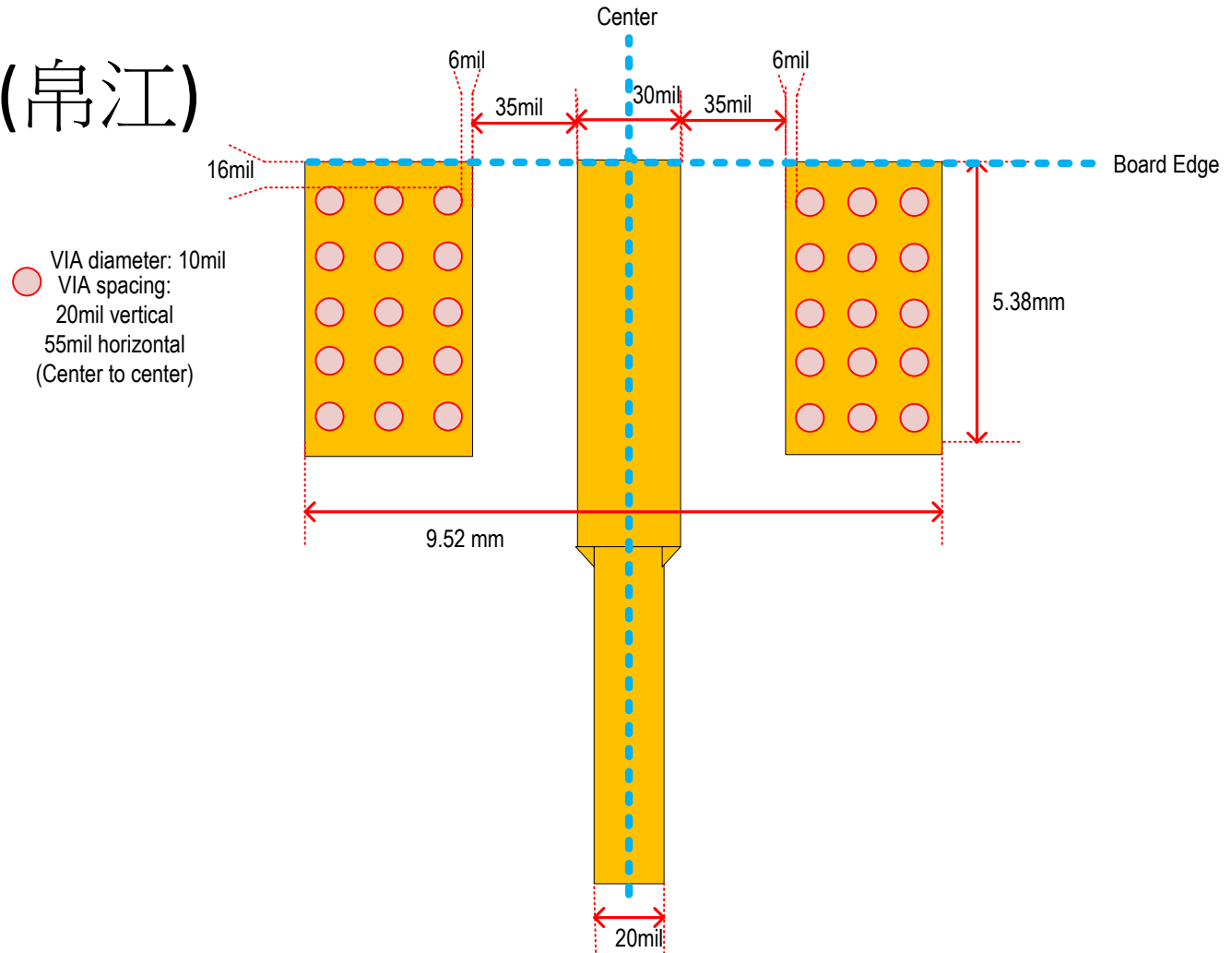
New “Shape symbol” in PCB Editor

- File -> New
 - Create a *shape symbol*
- 此時只有Etch可以用
- 用上列工具列畫出想要的圖形後
- Shape -> Merge Shapes可以把圖合再一起，才不會有DRC Error
- 畫完後直接在外面加一層3mil的框用來做Pad的Soldermask (要另存)這樣他們就會對齊
- Default drill hole location is the origin



SMA Connector Padstack

- Bo-Jiang(帛江)
- 7867LS



Surface Current Density Plot @ Radio Frequency

