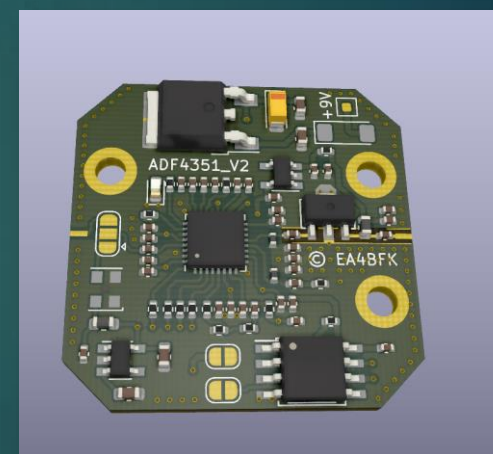
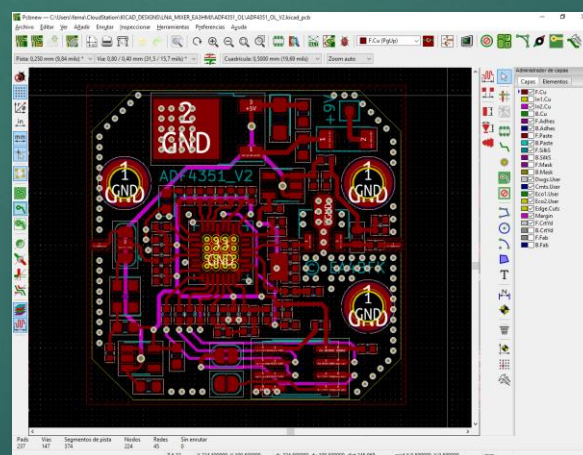
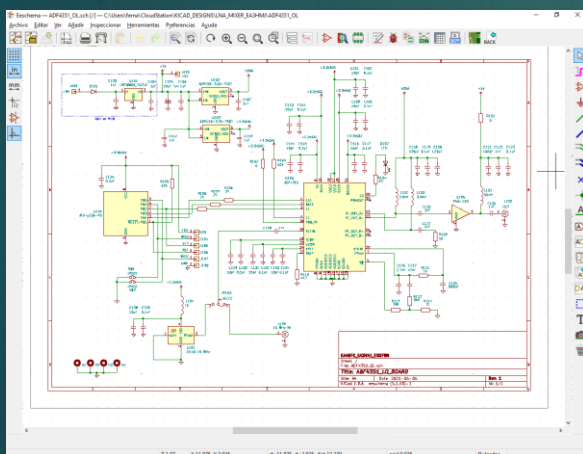




# INTRODUCCIÓN A



# Agenda



## Parte I ( 8 de Octubre)

- ▶ **Kicad EDA** Descripción
- ▶ Flujo de Diseño
- ▶ **EEschema**
  - ▶ Visión General
  - ▶ Atajos de teclado y el Botón izquierdo
  - ▶ Librerías de Símbolos
  - ▶ Propiedades de los Símbolos
  - ▶ Creación de esquema sencillo
    - ▶ Trazado
    - ▶ Anotación
    - ▶ DRC
    - ▶ Asociación de Huellas
    - ▶ Netlist
  - ▶ Esquemas Jerárquicos o Anidados

# Agenda



## Parte II ( 29 de Octubre)

### ▶ **Diseño de PCBs**

- ▶ Tipos de PCB y elementos que contiene (Capas, Materiales, etc)

### ▶ **PcbNew**

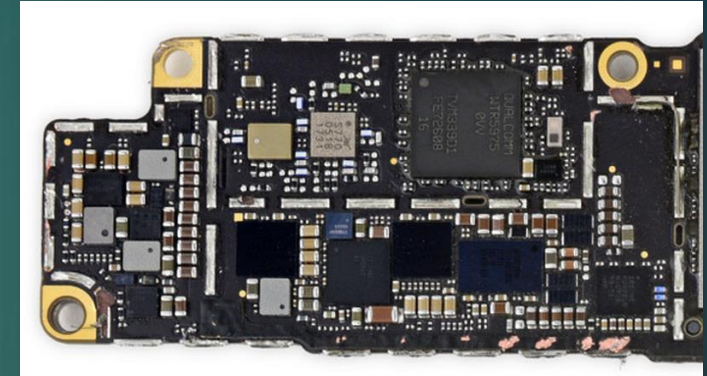
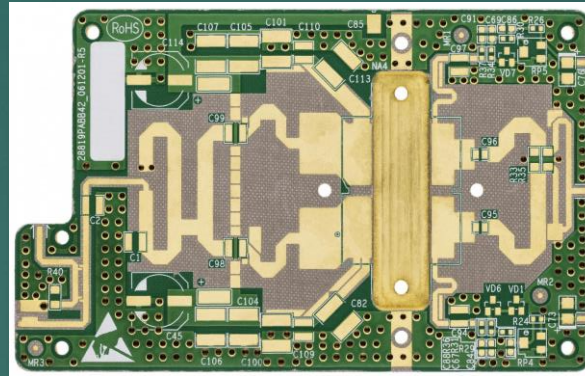
- ▶ Visión General
- ▶ Atajos de teclado y Ratón
- ▶ Librerías de Huellas y Objetos 3D
- ▶ Propiedades de la huellas
- ▶ Reglas de Diseño
- ▶ Importación de la Rejilla
- ▶ Creación de un PCB.
  - ▶ Algunas Ideas
  - ▶ Demostración

# Diseño de PCB's



## ► ¿Qué es un Circuito Impreso o PCB? (Printed Circuit Board)

- Es el soporte de los componentes electrónicos, que permite la interconexión entre ellos para permitir su funcionamiento.



## ► Estructura de un PCB (2 Capas)

TOP SOLDER MASK  
TOP COPPER LAYER

SUBSTRATE

BOTTOM COPPER  
BOTTOM SOLDER MASK



Resina protectora

Lámina de cobre. 1 oz / 35 um

Sustrato: Fibra de Vidrio (FR4) / PTFE / Cerámica

Lámina de cobre. 1 oz / 35 um

Resina protectora



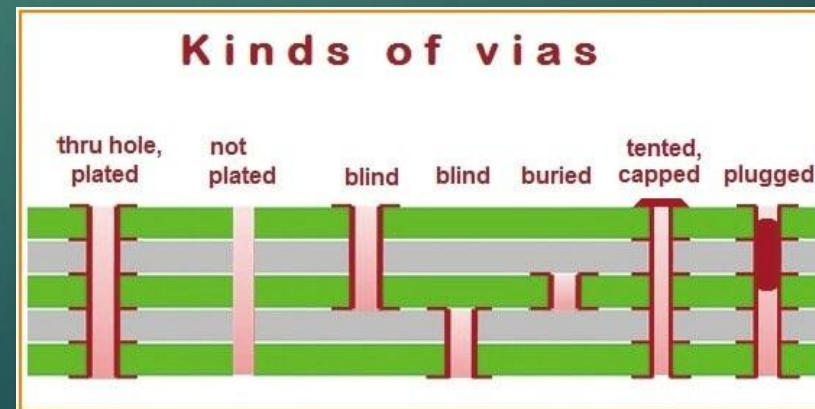
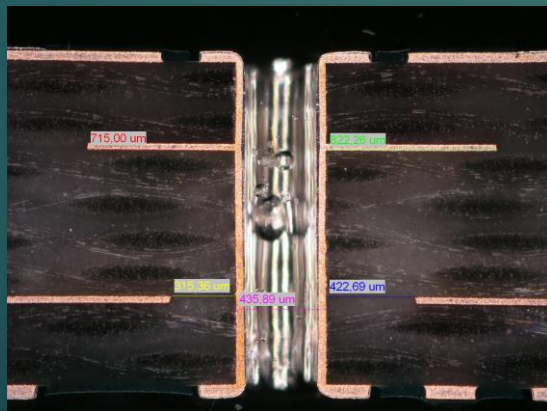
# Diseño de PCB's



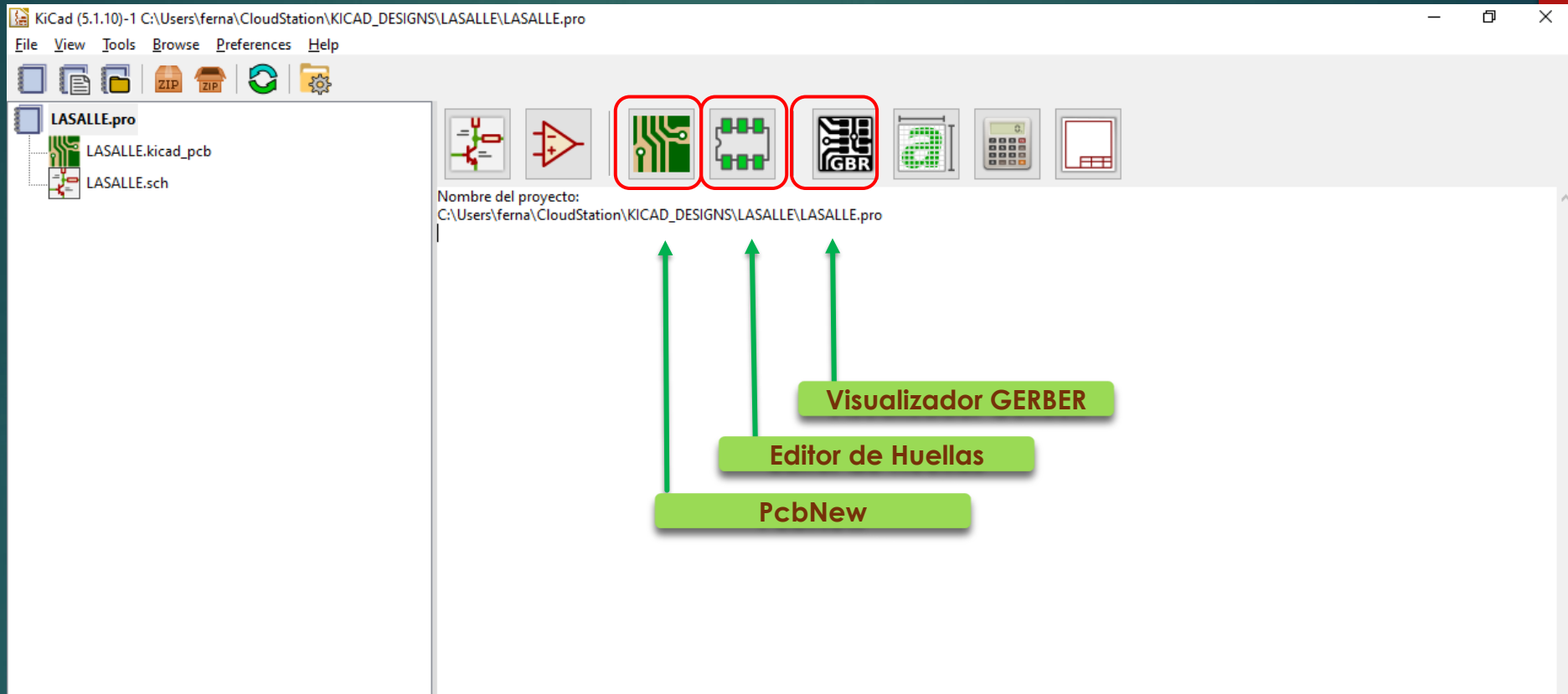
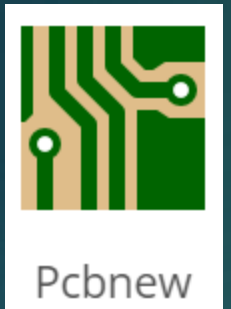
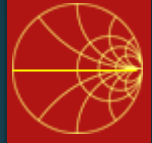
## ► Estructura de un PCB 4 capas



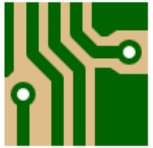
## ► Vías, conexiones entre capas



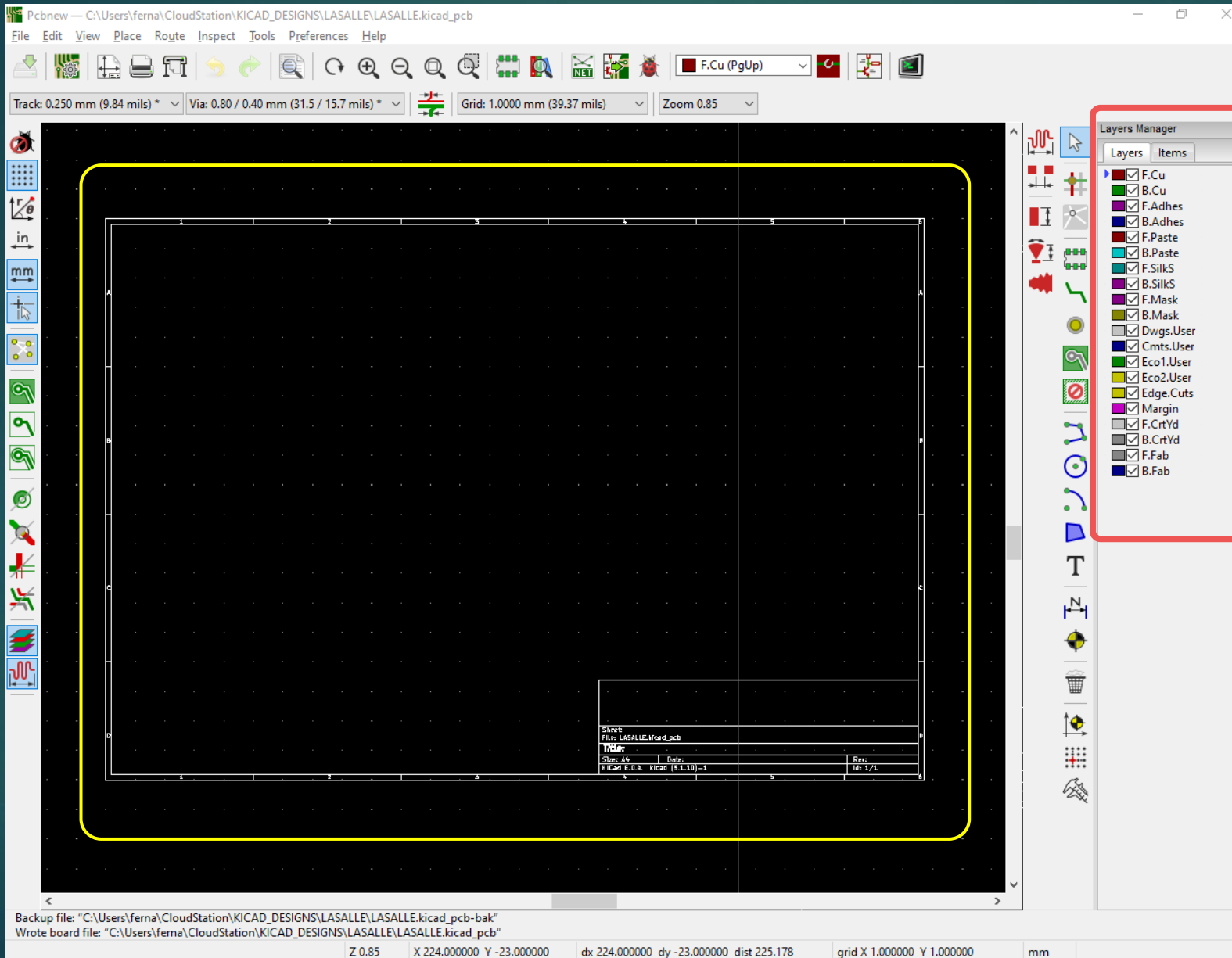
# Kicad EDA - PCBnew



# PCBnew- Descripción



Pcbnew



## Selector de Visualización

- Capas
- Objetos
- Colores



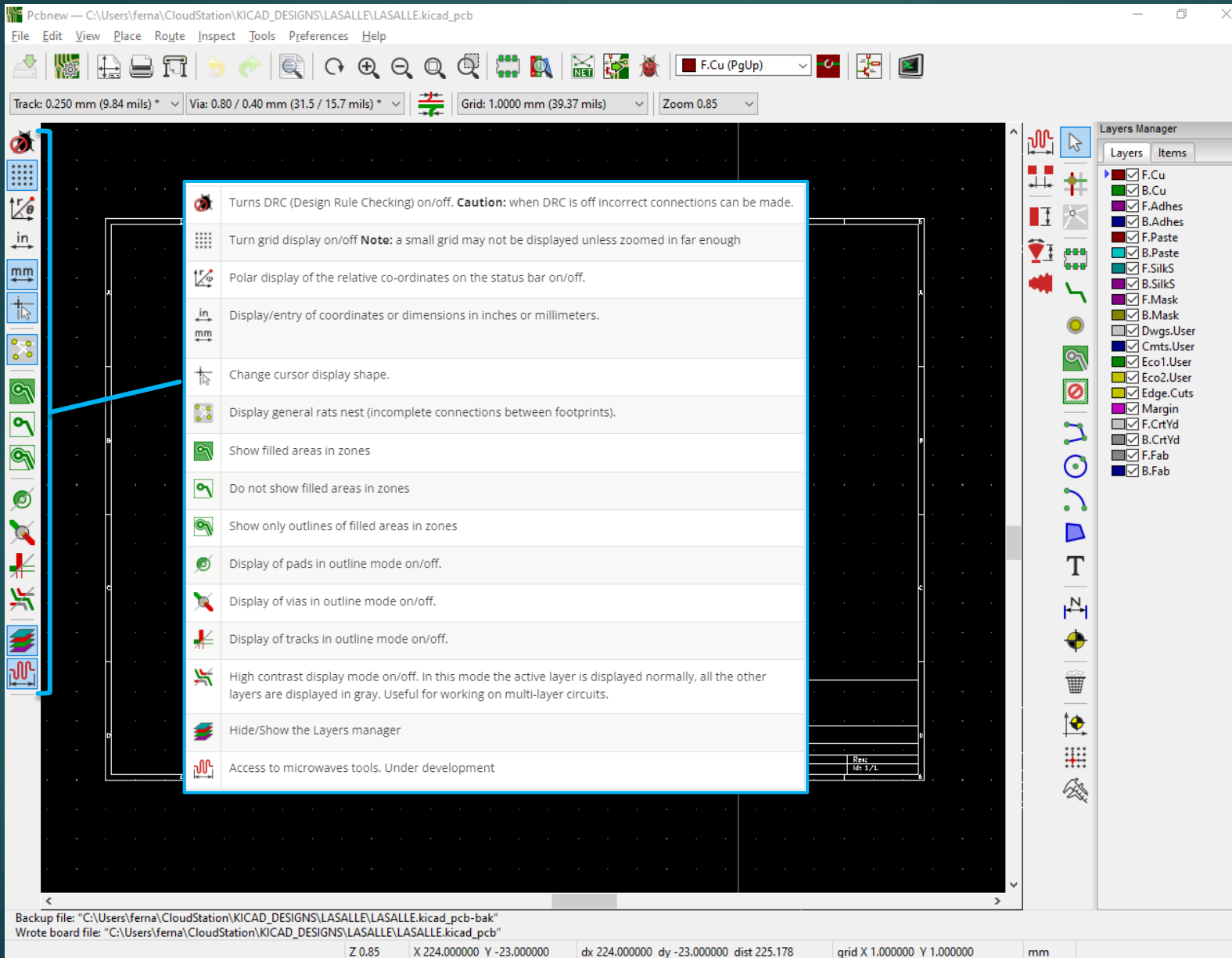
Visibilidad

Primer Plano

# PCBnew- Descripción



Pcbnew





# PCBnew- Descripción



Pcbnew



PCBnew — C:\Users\ferna\CloudStation\KICAD\_DESIGN\LASALLE\LASALLE.kicad\_pcb

File Edit View Place Route Inspect Tools Preferences Help

Track: 0.250

F.Cu (PgUp)

	Save printed circuit.
	Selection of the page size and modification of the file properties.
	Opens Footprint Editor to edit library or pcb footprint.
	Opens Footprint Viewer to display library or pcb footprint.
	Undo/Redo last commands (10 levels)
	Display print menu.
	Display plot menu.
	Zoom in and Zoom out (relative to the center of screen).
	Redraw the screen
	Fit to page
	Find footprint or text.
	Netlist operations (selection, reading, testing and compiling).
	DRC (Design Rule Check): Automatic check of the tracks.
	Selection of the working layer.
	Selection of layer pair (for vias)
	Show / Hide the Python scripting console

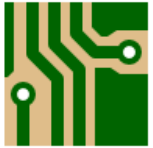
Layers Manager

Layers	Items
<input checked="" type="checkbox"/>	F.Cu
<input checked="" type="checkbox"/>	B.Cu
<input checked="" type="checkbox"/>	F.Adhes
<input checked="" type="checkbox"/>	B.Adhes
<input checked="" type="checkbox"/>	F.Paste
<input checked="" type="checkbox"/>	B.Paste
<input checked="" type="checkbox"/>	F.SilkS
<input checked="" type="checkbox"/>	B.SilkS
<input checked="" type="checkbox"/>	F.Mask
<input checked="" type="checkbox"/>	B.Mask
<input checked="" type="checkbox"/>	Dwgs.User
<input checked="" type="checkbox"/>	Cmts.User
<input checked="" type="checkbox"/>	Eco1.User
<input checked="" type="checkbox"/>	Eco2.User
<input checked="" type="checkbox"/>	Edge.Cuts
<input checked="" type="checkbox"/>	Margin
<input checked="" type="checkbox"/>	F.CrtYd
<input checked="" type="checkbox"/>	B.CrtYd
<input checked="" type="checkbox"/>	F.Fab
<input checked="" type="checkbox"/>	B.Fab

Backup file: "C:\Users\ferna\CloudStation\KICAD\_DESIGN\LASALLE\LASALLE.kicad\_pcb-bak"  
Wrote board file: "C:\Users\ferna\CloudStation\KICAD\_DESIGN\LASALLE\LASALLE.kicad\_pcb"

Z 0.85 X 224.000000 Y -23.000000 dx 224.000000 dy -23.000000 dist 225.178 grid X 1.000000 Y 1.000000 mm

# PCBnew- Descripción



Pcbnew



Pcbnew — C:\Users\ferna\CloudStation\KICAD\_DESIGN\LASALLE\LASALLE.kicad\_pcb

File Edit View Place Route Inspect Tools Preferences Help

Track: 0.250 mm (9.84 mils) \* Via: 0.80 / 0.40 mm (31.5 / 15.7 mils) \* Grid: 1.0000 mm (39.37 mils) Zoom 0.85

F.Cu (PgUp)

Track	Via	Grid	Zoom
17.0	65.0	50.0	128

Selection of thickness of track already in use.

Selection of a dimension of via already in use.

Automatic track width: if enabled when creating a new track, when starting on an existing track, the width of the new track is set to the width of the existing track.

Selection of the grid size.

Selection of the zoom.

Layers Manager

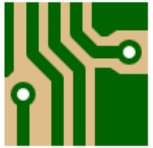
Layers	Items
F.Cu	<input checked="" type="checkbox"/>
B.Cu	<input checked="" type="checkbox"/>
F.Adhes	<input checked="" type="checkbox"/>
B.Adhes	<input checked="" type="checkbox"/>
F.Paste	<input checked="" type="checkbox"/>
B.Paste	<input checked="" type="checkbox"/>
F.SilkS	<input checked="" type="checkbox"/>
B.SilkS	<input checked="" type="checkbox"/>
F.Mask	<input checked="" type="checkbox"/>
B.Mask	<input checked="" type="checkbox"/>
Dwgs.User	<input checked="" type="checkbox"/>
Cmts.User	<input checked="" type="checkbox"/>
Eco1.User	<input checked="" type="checkbox"/>
Eco2.User	<input checked="" type="checkbox"/>
Edge.Cuts	<input checked="" type="checkbox"/>
Margin	<input checked="" type="checkbox"/>
F.CrtYd	<input checked="" type="checkbox"/>
B.CrtYd	<input checked="" type="checkbox"/>
F.Fab	<input checked="" type="checkbox"/>
B.Fab	<input checked="" type="checkbox"/>

Backup file: "C:\Users\ferna\CloudStation\KICAD\_DESIGN\LASALLE\LASALLE.kicad\_pcb-bak"

Wrote board file: "C:\Users\ferna\CloudStation\KICAD\_DESIGN\LASALLE\LASALLE.kicad\_pcb"

Z 0.85 X 224.000000 Y -23.000000 dx 224.000000 dy -23.000000 dist 225.178 grid X 1.000000 Y 1.000000 mm

# PCBnew- Descripción



Pcbnew



PCBnew — C:\Users\ferna\CloudStation\KICAD\_DESIGN\LASALLE\LASALLE.kicad\_pcb

File Edit View Place Route Inspect Tools Preferences Help

Track: 0.250 mm (9.84 mils) \* Via: 0.80 / 0.40 mm (31.5 / 15.7 mils) \* Grid: 1.0000 mm (39.37 mils) Zoom 0.85

Layers Manager

Layers	Items
<input checked="" type="checkbox"/> F.Cu	
<input checked="" type="checkbox"/> B.Cu	
<input checked="" type="checkbox"/> F.Adhes	
<input checked="" type="checkbox"/> B.Adhes	
<input checked="" type="checkbox"/> F.Paste	
<input checked="" type="checkbox"/> B.Paste	
<input checked="" type="checkbox"/> F.SilkS	
<input checked="" type="checkbox"/> B.SilkS	
<input checked="" type="checkbox"/> F.Mask	
<input checked="" type="checkbox"/> B.Mask	
<input checked="" type="checkbox"/> Dwgs.User	
<input checked="" type="checkbox"/> Cmts.User	
<input checked="" type="checkbox"/> Eco1.User	
<input checked="" type="checkbox"/> Eco2.User	
<input checked="" type="checkbox"/> Edge.Cuts	
<input checked="" type="checkbox"/> Margin	
<input checked="" type="checkbox"/> F.CrtYd	
<input checked="" type="checkbox"/> B.CrtYd	
<input checked="" type="checkbox"/> F.Fab	
<input checked="" type="checkbox"/> B.Fab	

Select the standard mouse mode.

Highlight net selected by clicking on a track or pad.

Display local ratsnest (Pad or Footprint).

Add a footprint from a library.

Placement of tracks and vias.

Placement of zones (copper planes).

Placement of keepout areas ( on copper layers ).

Draw Lines on technical layers (i.e. not a copper layer).

Draw Circles on technical layers (i.e. not a copper layer).

Draw Arcs on technical layers (i.e. not a copper layer).

Placement of text.

Draw Dimensions on technical layers (i.e. not the copper layer).

Draw Alignment Marks (appearing on all layers).

Delete element pointed to by the cursor  
**Note:** When Deleting, if several superimposed elements are pointed to, priority is given to the smallest (in the decreasing set of priorities tracks, text, footprint). The function "Undelete" of the upper toolbar allows the cancellation of the last item deleted.

Offset adjust for drilling and place files.

Grid origin. (grid offset). Useful mainly for editing and placement of footprints. Can also be set in Dimensions/Grid menu.

Backup file: "C:\Users\ferna\CloudStation\KICAD\_DESIGN\LASALLE\LASALLE.kicad\_pcb-bak"  
Wrote board file: "C:\Users\ferna\CloudStation\KICAD\_DESIGN\LASALLE\LASALLE.kicad\_pcb"

Z 0.85 X 224.000000 Y -23.000000 dx 224.000000 dy -23.000000 dist 225.178 grid X 1.000000 Y 1.000000 mm

# PCBnew - Atajos de Teclado 1/2



- Existen múltiples atajos de teclado, que facilitan acciones de una forma muy sencilla e intuitiva. (Windows: Ctrl+F1 / Mac: Cmd+F1)

GENERALES	
Shortcut	Action
Ctrl+N	New
Ctrl+O	Open
Ctrl+S	Save
Ctrl+Shift+S	Save As
Ctrl+P	Print
Ctrl+Z	Undo
Ctrl+Y	Redo
Ctrl+X	Cut
Ctrl+C	Copy
Ctrl+V	Paste
Ctrl+F1	List Hotkeys
Ctrl+,	Preferences
F1	Zoom In
F2	Zoom Out
F3	Zoom Redraw
F4	Zoom Center
Home	Zoom Auto
Ctrl+F5	Zoom to Selection
Alt+3	3D Viewer
Ctrl+U	Switch Units
Space	Reset Local Coordinates
S	Set Grid Origin
Z	Reset Grid Origin
Return	Mouse Left Click
End	Mouse Left Double Click
}	Increment Layer Transparency (Modern Toolset only)
{	Decrement Layer Transparency (Modern Toolset only)
Ctrl+Shift+X	Toggle Cursor Display (Modern Toolset only)
Ctrl+Shift+M	Measure Distance (Modern Toolset only)

PCBNEW	
Shortcut	Action
P	Place Item
O	Add Footprint
Ctrl+Shift+V	Add Vias
Ctrl+Shift+Z	Add Filled Zone
Ctrl+Shift+K	Add Keepout Area
Shift+C	Add a Zone Cutout
Ctrl+>	Add a Similar Zone
Ctrl+Shift+L	Draw Line
Ctrl+Shift+C	Draw Circle
Ctrl+Shift+A	Draw Arc
Ctrl+Shift+P	Draw Graphic Polygon
Ctrl+Shift+T	Add Text
Ctrl+Shift+H	Add Dimension
Ctrl+Shift+F	Place DXF
X	Add New Track
/	Switch Track Posture
D	Drag Track Keep Slope
V	Add Through Via
Alt+Shift+V	Add Blind/Buried Via
Alt+V	Add MicroVia
6	Route Differential Pair (Modern Toolset only)
7	Tune Single Track (Modern Toolset only)
8	Tune Differential Pair Length (Modern Toolset only)
9	Tune Differential Pair Skew (Modern Toolset only)
Ctrl+<	Routing Options
Ctrl+L	Length Tuning Settings (Modern Toolset only)
1	Increase meander spacing by one step.
2	Decrease meander spacing by one step.
3	Increase meander amplitude by one step.
4	Decrease meander amplitude by one step.
Ctrl+Shift+R	Differential Pair Dimensions
B	Fill or Refill All Zones
Ctrl+B	Remove Filled Areas in All Zones
Ins	Insert Corner (Modern Toolset only)
U	Select Single Track
I	Select Connected Tracks
`	Toggle Highlight of Selected Net (Modern Toolset only)
K	Track Display Mode
Q	Custom Track/Via Size
W	Switch Track Width To Next
Shift+W	Switch Track Width To Previous

PCBNEW	
Shortcut	Action
'	Increase Via Size
\	Decrease Via Size
Del	Delete Full Track
Back	Delete Track Segment
G	Drag Item
C	Copy Item
M	Move Item
T	Get and Move Footprint
Ctrl+M	Move Item Exactly
Ctrl+R	Position Item Relative
F	Flip Item
R	Rotate Item
Shift+R	Rotate Item Clockwise (Modern Toolset only)
<	Select Layer and Add Through Via
Alt+<	Select Layer and Add Blind/Buried Via
Ctrl+D	Duplicate Item
Ctrl+Shift+D	Duplicate Item and Increment
Ctrl+T	Create Array
L	Lock/Unlock Footprint
Ctrl+F	Find Item
E	Edit Item
Ctrl+E	Edit with Footprint Editor
PgUp	Switch to Component (F.Cu) layer
PgDn	Switch to Copper (B.Cu) layer
F5	Switch to Inner layer 1
F6	Switch to Inner layer 2
F7	Switch to Inner layer 3
F8	Switch to Inner layer 4
Shift+F5	Switch to Inner layer 5
Shift+F6	Switch to Inner layer 6
+	Switch to Next Layer
-	Switch to Previous Layer
Alt+1	Switch Grid To Fast Grid1
Alt+2	Switch Grid To Fast Grid2
N	Switch Grid To Next
Shift+N	Switch Grid To Previous
Ctrl+H	Toggle High Contrast Mode

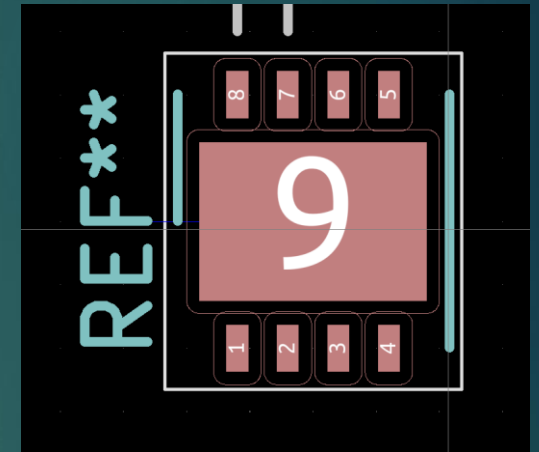
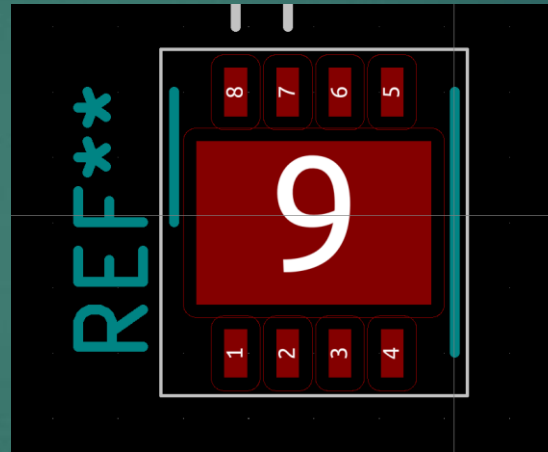
# PCBnew - Atajos de Teclado 2/2 y Ratón



FOOTPRINT EDITOR	
Shortcut	Action
Ctrl+Shift+L	Draw Line
Ctrl+Shift+C	Draw Circle
Ctrl+Shift+A	Draw Arc
Ctrl+Shift+P	Draw Graphic Polygon
Ctrl+Shift+T	Add Text
Ctrl+Shift+N	Place the Footprint Anchor
Del	Delete Full Track
M	Move Item
Ctrl+M	Move Item Exactly
R	Rotate Item
/	Switch Track Posture
E	Edit Item
Ctrl+D	Duplicate Item
Ctrl+Shift+D	Duplicate Item and Increment
Ctrl+T	Create Array
Alt+1	Switch Grid To Fast Grid1
Alt+2	Switch Grid To Fast Grid2
N	Switch Grid To Next
Shift+N	Switch Grid To Previous
PgUp	Switch to Component (F.Cu) layer
PgDn	Switch to Copper (B.Cu) layer
Ctrl+H	Toggle High Contrast Mode
F9	Switch to Legacy Toolset (not all features will be available)
F12	Switch to Modern Toolset with software graphics (fall-back)
F11	Switch to Modern Toolset with hardware-accelerated graphics (recommended)

## ► Uso del ratón

- En PCBnew el objeto seleccionado si se destaca



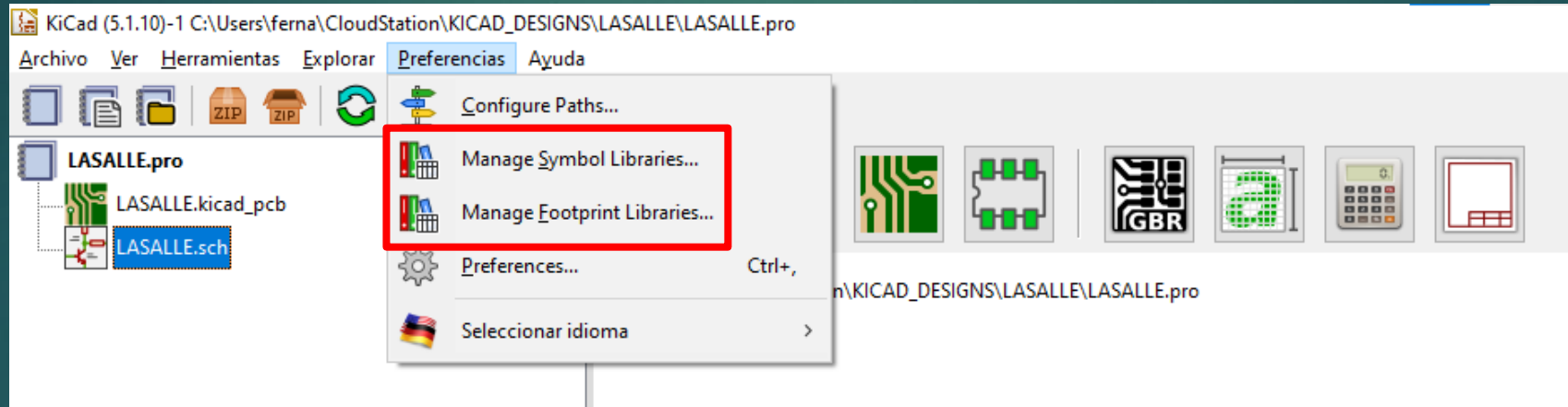
- Rueda: Zoom In, Zoom Out



# Librerías de Huellas y objetos 3D

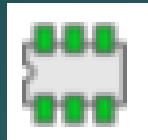


- ▶ La Instalación incluye todas las librerías de símbolos, huellas y modelos 3D que ofrece Kicad.org
- ▶ El Path a los ficheros de Librerías se puede modificar en Preferencias del menú Proyecto de Kicad

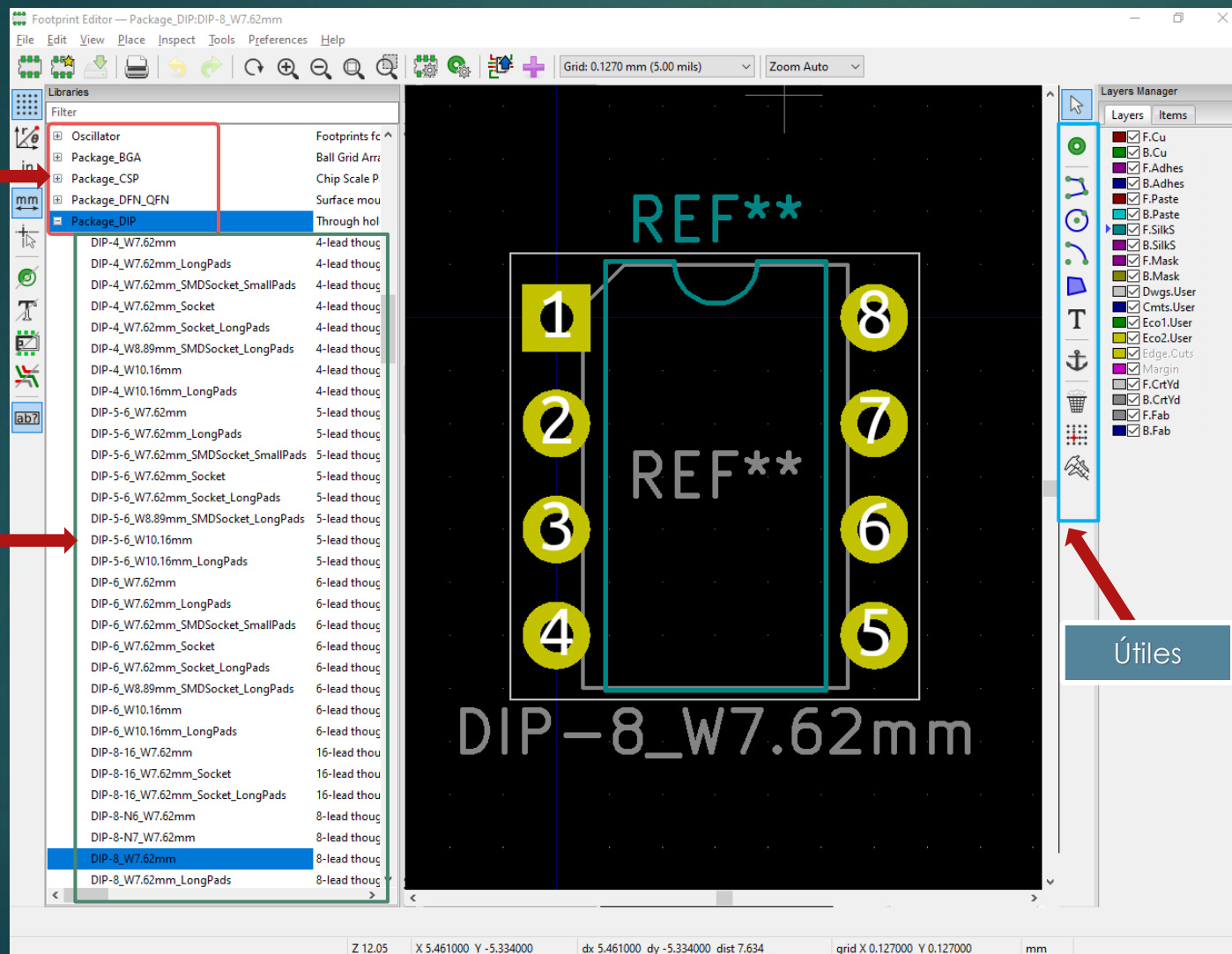


- ▶ Otras fuentes de librerías de símbolos y huellas:
  - ▶  <https://www.snapeda.com/>
  - ▶ Webs de Distribuidores (Mouser, Digi-Key, Farnell,...)
  - ▶ Crearlo con el Editor de Huellas 

# Editor de Huellas



- Módulo que permite la Edición o creación de nuevas huellas para PCBnew

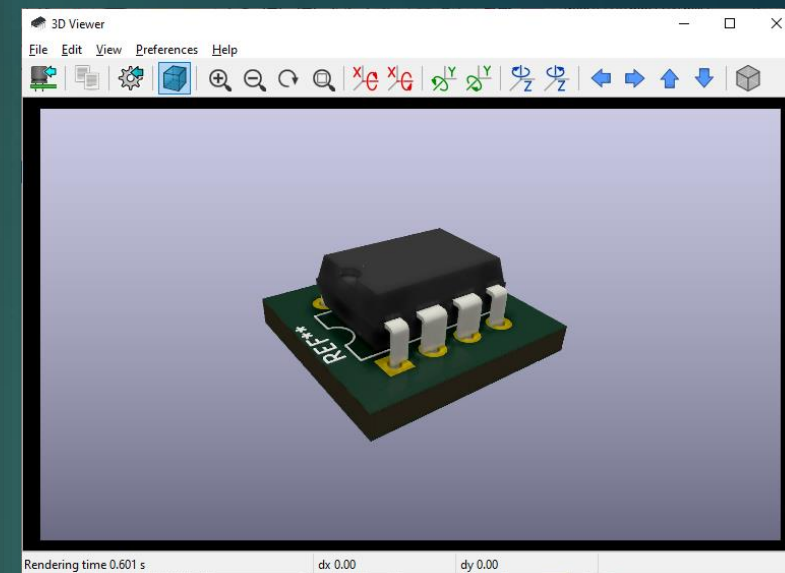


Librerías

Huellas

Útiles

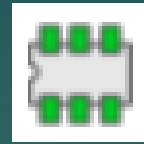
[Alt +3]



Librerías de huellas  
<https://www.snapeda.com/>

Librerías 3D  
<https://www.3dcontentcentral.es/>

# Propiedades de las huellas



- Seleccionar y [e] para visualizar y editar las propiedades

Footprint Properties

General Local Clearance and Settings 3D Settings

	Text Items	Show	Width	Height	Thickness	Italic	Layer
Reference	REF**	<input checked="" type="checkbox"/>	1 mm	1 mm	0.15 mm	<input type="checkbox"/>	F.SilkS
Value	DFN-8-1EP_4x4mm_P0.8mm_EP2.5x3.6mm	<input checked="" type="checkbox"/>	1 mm	1 mm	0.15 mm	<input type="checkbox"/>	F.Fab
	%R	<input checked="" type="checkbox"/>	1 mm	1 mm	0.15 mm	<input type="checkbox"/>	F.Fab

Position X: 146.63 mm  
Position Y: 65.615 mm

Orientation  
☐ 0.0  
☒ 90.0  
☐ -90.0  
☐ 180.0  
☐ Other: 90.0

Board side: Front

Move and Place  
☐ Free  
☒ Lock pads  
☐ Lock footprint

Auto-placement Rules  
Allow 90 degree rotated placement: 0 10  
Allow 180 degree rotated placement: 0 10

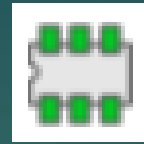
Fabrication Attributes  
☐ Through hole  
☒ Surface mount  
☐ Virtual

Update Footprint from Library...  
Change Footprint...  
Edit Footprint...  
Edit Library Footprint...

Library reference: Package\_DFN\_QFN:DFN-8-1EP\_4x4mm\_P0.8mm\_EP2.5x3.6mm

OK Cancel

# Propiedades de las huellas



- Seleccionar y [e] para visualizar y editar las propiedades

The image shows a PCB design software interface with a footprint editor. On the left, a footprint is shown with a large white '9' in the center and eight small red rectangles labeled 1 through 8 around it. The footprint is labeled 'REF\*\*' in large blue letters. On the right, two 'Footprint Properties' dialog boxes are open. The first dialog box has the '3D Settings' tab selected, showing a table with the following data:

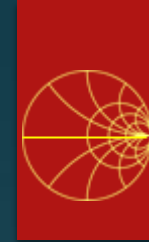
Reference	Value
REF**	DFN-8-1EP_4x4mm_P0.8mm_EP2.5x3.6mm
	%R

Below the table, the 'Position X' is 146.63 mm and 'Position Y' is 65.615 mm. The 'Orientation' is set to 90.0 degrees. The 'Board side' is 'Front'. The 'Library reference' is 'Package\_DFN\_QFN:DFN-8-1EP\_4x4mm\_P0.8mm'. The second dialog box has the '3D Model(s)' tab selected, showing a table with the following data:

3D Model(s)	Preview
{KISYS3DMOD}/Package_DFN_QFN.3dshapes/DFN-8-1EP_4x4mm_P0.8mm_EP2.5x3.6mm.wrl	<input checked="" type="checkbox"/>

Below the table, the 'Scale' is set to X: 1.0000, Y: 1.0000, Z: 1.0000. The 'Rotation' is set to X: 0.00 deg, Y: 0.00 deg, Z: 0.00 deg. The 'Offset' is set to X: 0.0000 mm, Y: 0.0000 mm, Z: 0.0000 mm. The 'Library reference' is 'Package\_DFN\_QFN:DFN-8-1EP\_4x4mm\_P0.8mm\_EP2.5x3.6mm'. A 3D preview of the footprint is shown on the right, labeled 'REF\*\*'.

# PCBnew – Reglas de Diseño



- ▶ Reglas definidas por los fabricantes de PCB y que se pueden incorporar al diseño para evitar errores de fabricación
- ▶ Información disponible en las webs de los Fabricantes

<https://jlcpcb.com/capabilities/Capabilities>

## PCB Capabilities

Know JLCPCB's Capabilities & Get your PCBs Built Fast

PCB Specifications

+

Drill/Hole Size

+

Minimum Annular Ring

+

Minimum clearance

+

Minimum trace width and spacing

+

BGA

+

Solder Mask

+

Legend

+

Board Outlines

+

Panelization

+

Features	Capability	Notes	Patterns
Layer count	1,2,4,6 layers	The number of copper layers in the board.	
Controlled Impedance	4/6 layer, default layer stack-up	<a href="#">Controlled Impedance PCB Layer Stackup</a> <a href="#">JLCPCB Impedance Calculator</a>	
Material	FR-4	FR-4 Standard Tg 130-140/ Tg 155	
Dielectric constant	4.5(double-sided PCB)	7028 Prepreg 4.6 2313 Prepreg 4.05 2116 Prepreg 4.25	
Max. Dimension	400x500mm	The maximum dimension JLCPCB can accept	
Dimension Tolerance	±0.2mm	±0.2mm for CNC routing, and ±0.4mm for V-scoring	
Board Thickness	0.4/0.6/0.8/1.0/1.2/1.6/2.0mm	The thickness of finished board.	
Thickness Tolerance (Thickness≥1.0mm)	± 10%	e.g. For the 1.6mm board thickness, the finished board thickness ranges from 1.44mm(T-1.6×10%) to 1.76mm(T+1.6×10%)	
Thickness Tolerance (Thickness<1.0mm)	± 0.1mm	e.g. For the 0.8mm board thickness, the finished board thickness ranges from 0.7mm(T-0.1) to 0.9mm(T+0.1).	
Finished Outer Layer Copper	1 oz/2 oz (35um/70um)	Finished copper weight of outer layer is 1oz or 2oz.	
Finished Inner Layer Copper	0.5 oz (17.5um)	Finished copper weight of inner layer is 0.5oz by default.	



# PCBnew – Reglas de Diseño



- ▶ Reglas definidas por los fabricantes de PCB y que se pueden incorporar al diseño para evitar errores de fabricación
- ▶ Información disponible en las webs de los Fabricantes

<https://jlcpcb.com/capabilities/Capabilities>

**PCB Capabilities**  
Know JLCPCB's Capabilities & Get your PCBs Built Fast

- PCB Specifications
- Drill/Hole Size
- Minimum Annular Ring
- Minimum clearance
- Minimum trace width and spacing
- BGA
- Solder Mask
- Legend
- Board Outlines
- Panelization

Drill/Hole Size			
Features	Capability	Notes	Patterns
Drill Hole Size	0.20mm- 0.30mm	Min. drill size is 0.20mm. Max. drill size is 0.30mm.	
Drill Hole Size Tolerance	+0.13/-0.08mm	e.g. for the 0.6mm hole size, the finished hole size between 0.52mm to 0.73mm is acceptable.	
Blind/Buried Vias	Don't support	Currently we don't support Blind/Buried Vias, only make through holes.	
Min. Via hole size	0.2mm	For Single&Double Layer PCB, the minimum via hole size is 0.3mm; For Multi Layer PCB, the minimum via hole size is 0.2mm	
Min. Via diameter	0.4mm	For Single&Double Layer PCB, the minimum Via diameter is 0.6mm; For Multi Layer PCB, the minimum via diameter is 0.4mm.	
PTH hole Size	0.20mm - 0.35mm	The annular ring size will be enlarged to 0.15mm in production.	
Pad Size	0.70mm- 0.35mm	The pad hole size will be enlarged 0.15mm in production.	

# PCBnew – Reglas de Diseño




- ▶ Reglas definidas por los fabricantes de PCB y que se pueden incorporar al diseño para evitar errores de fabricación
- ▶ Información disponible en las webs de los Fabricantes

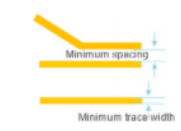
<https://jlcpcb.com/capabilities/Capabilities>





### PCB Capabilities

Know JLCPCB's Capabilities & Get your PCBs Built Fast



- PCB Specifications +
- Drill/Hole Size +
- Minimum Annular Ring +
- Minimum clearance +
- Minimum trace width and spacing +
- BGA +
- Solder Mask +
- Legend +
- Board Outlines +
- Panelization +


Minimum trace width and spacing			
Copper weight	Min. Trace width	Min. Spacing	Patterns
H/HOZ (Inner layer)	5mil (0.127mm)	5mil (0.127mm)	
1oz (Outer layer)	1/2 layers: 5mil (0.127mm) 4/6 layers: 3.5mil(0.09mm)	1/2 layers: 5mil (0.127mm) 4/6 layers: 3.5mil(0.09mm)	
2oz (Outer layer)	8mil (0.2mm)	8mil (0.2mm)	

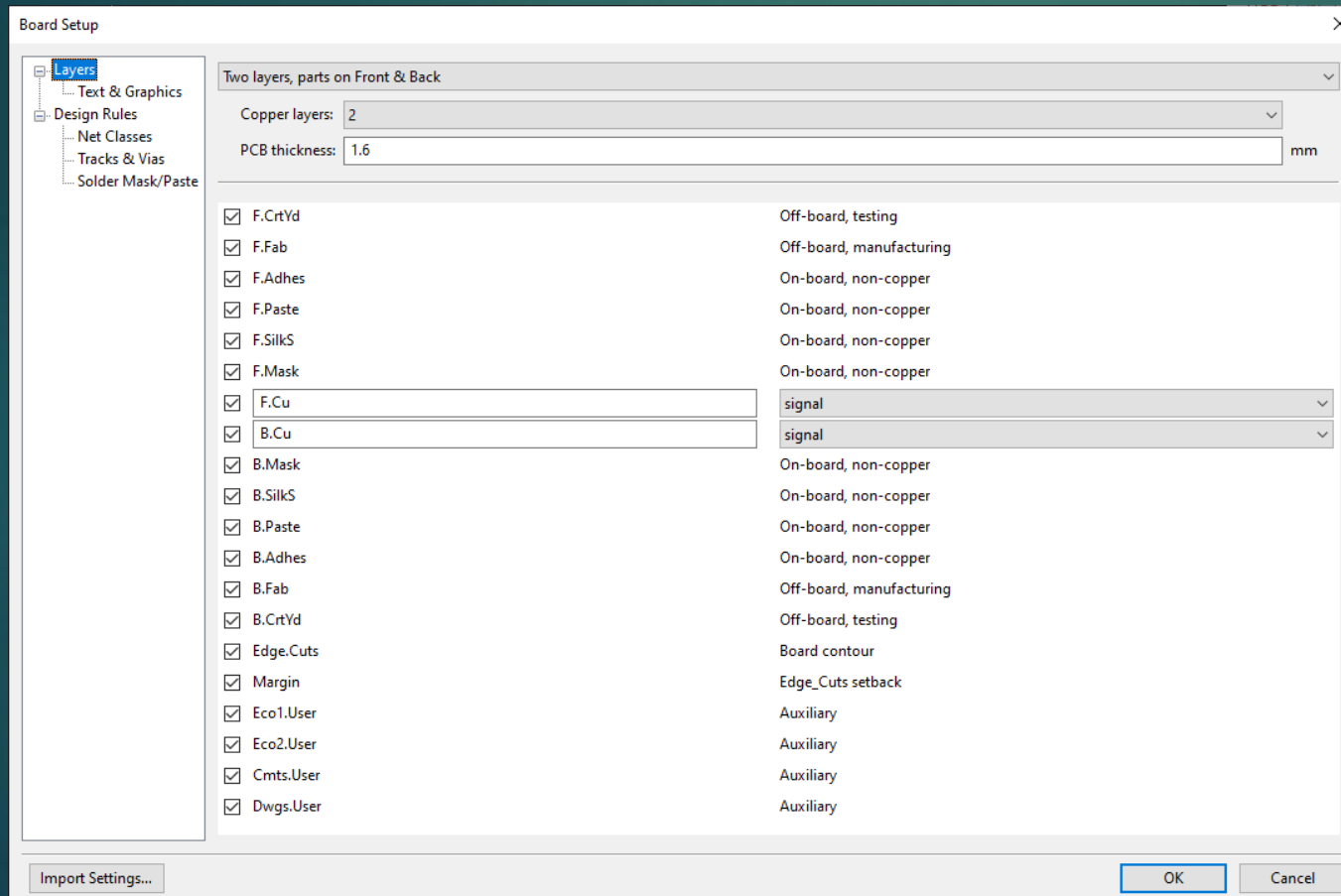
Legend			
Features	Capabilities	Notes	Patterns
Minimum Line Width	6 mil (0.153mm)	Characters width less than 6mil(0.153mm) will be unidentifiable.	
Minimum text height	32 mil (0.8mm)	Characters height less than 32mil(0.8mm) will be unidentifiable.	
Character width to height ratio	1:6	The preferred ratio of width to height is 1:6.	
Pad To Silkscreen	0.15mm	The Minimum Distance Between Pad and Silkscreen is 0.15mm.	

# PCBnew – Reglas de Diseño




## ► Inclusión de la Reglas de Diseño en PCBnew

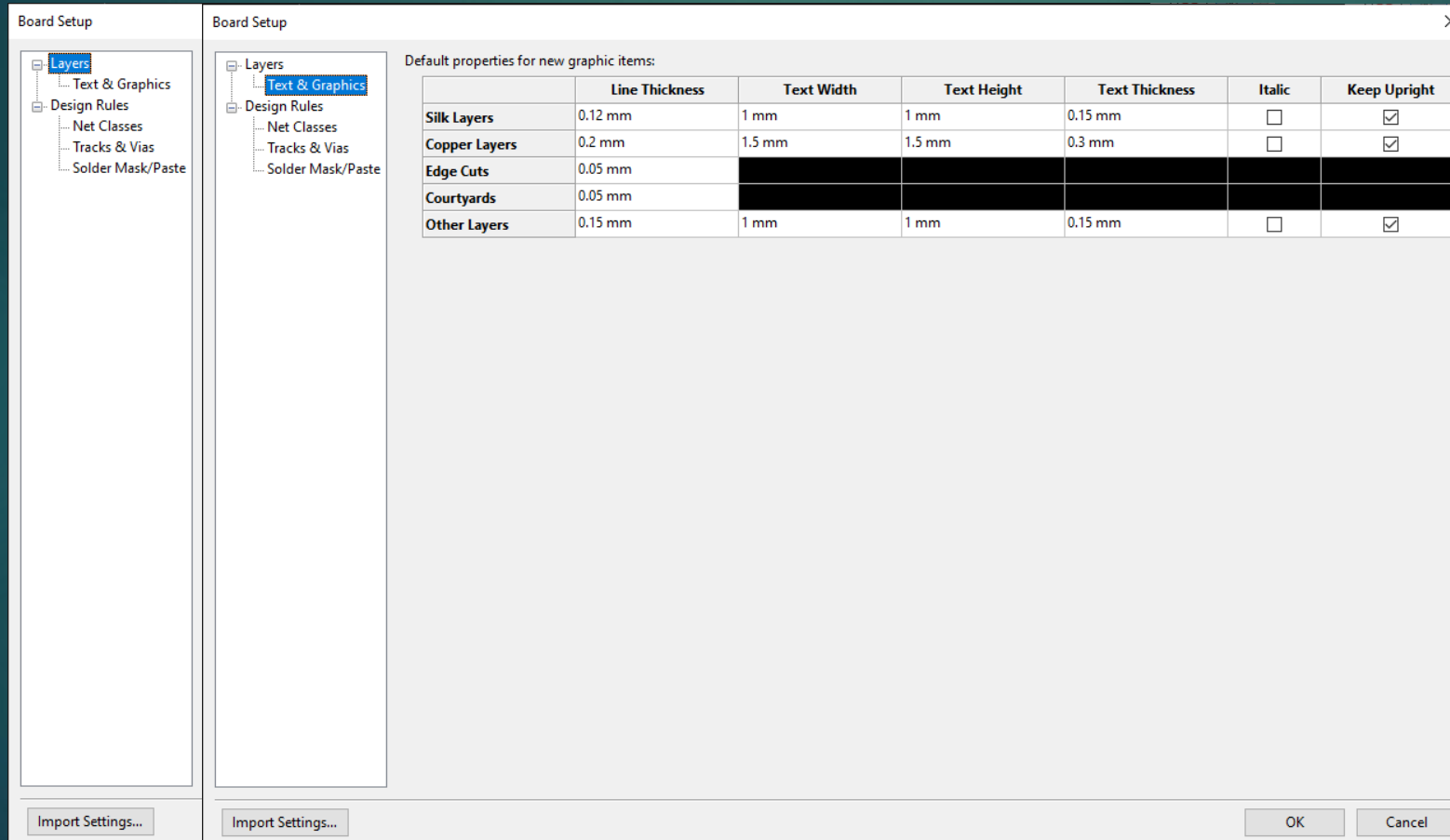
- Mediante el icono  (Board Setup) ó > File > Board Setup accedemos a las ventanas de las reglas de diseño



# PCBnew – Reglas de Diseño




- Inclusión de la Reglas de Diseño en PCBnew
  - Mediante el icono  (Board Setup) ó > File > Board Setup accedemos a las ventanas de las reglas de diseño

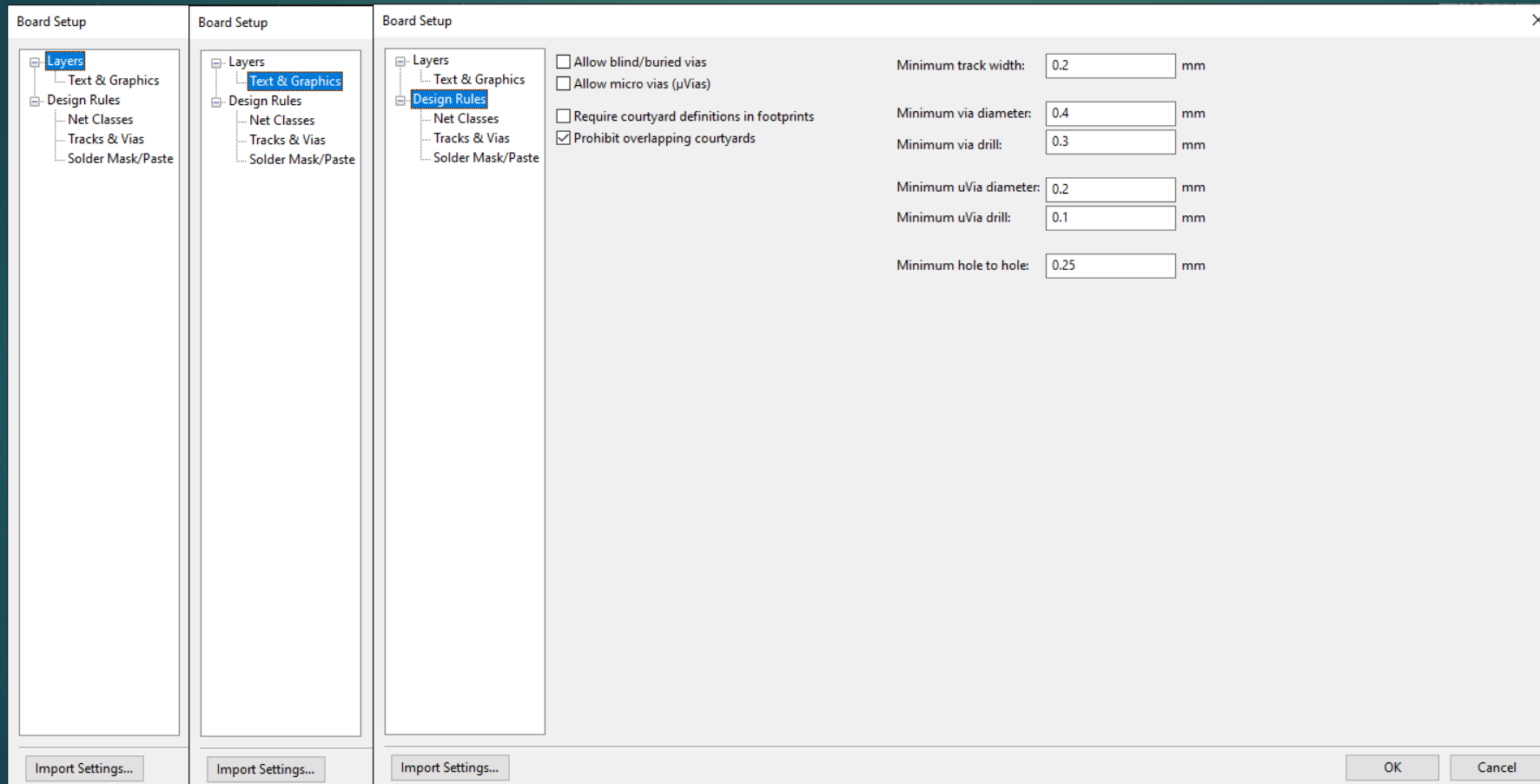


# PCBnew – Reglas de Diseño



## ► Inclusión de la Reglas de Diseño en PCBnew

- Mediante el icono  (Board Setup) ó > File > Board Setup accedemos a las ventanas de las reglas de diseño




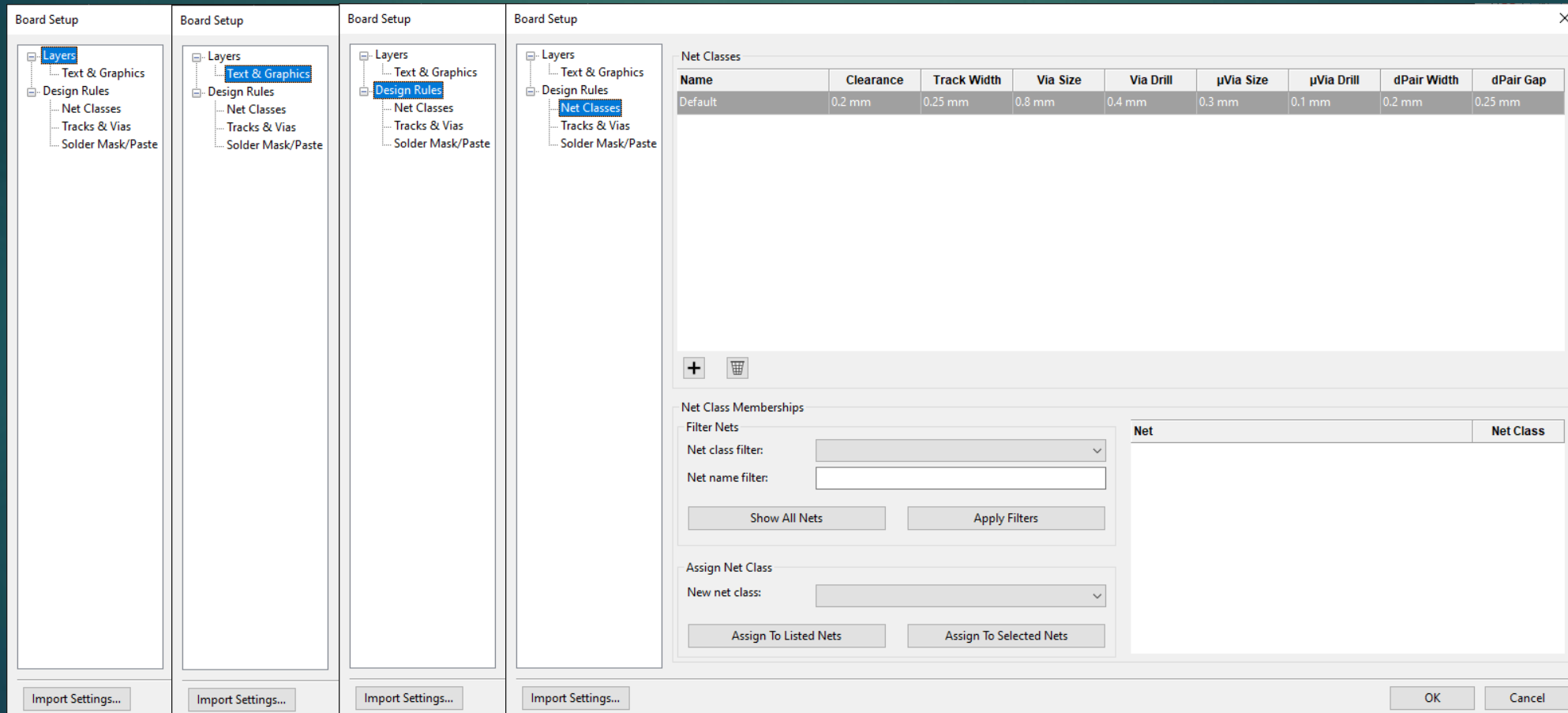


# PCBnew – Reglas de Diseño



## ► Inclusión de la Reglas de Diseño en PCBnew


- Mediante el icono  (Board Setup) ó > File > Board Setup accedemos a las ventanas de las reglas de diseño

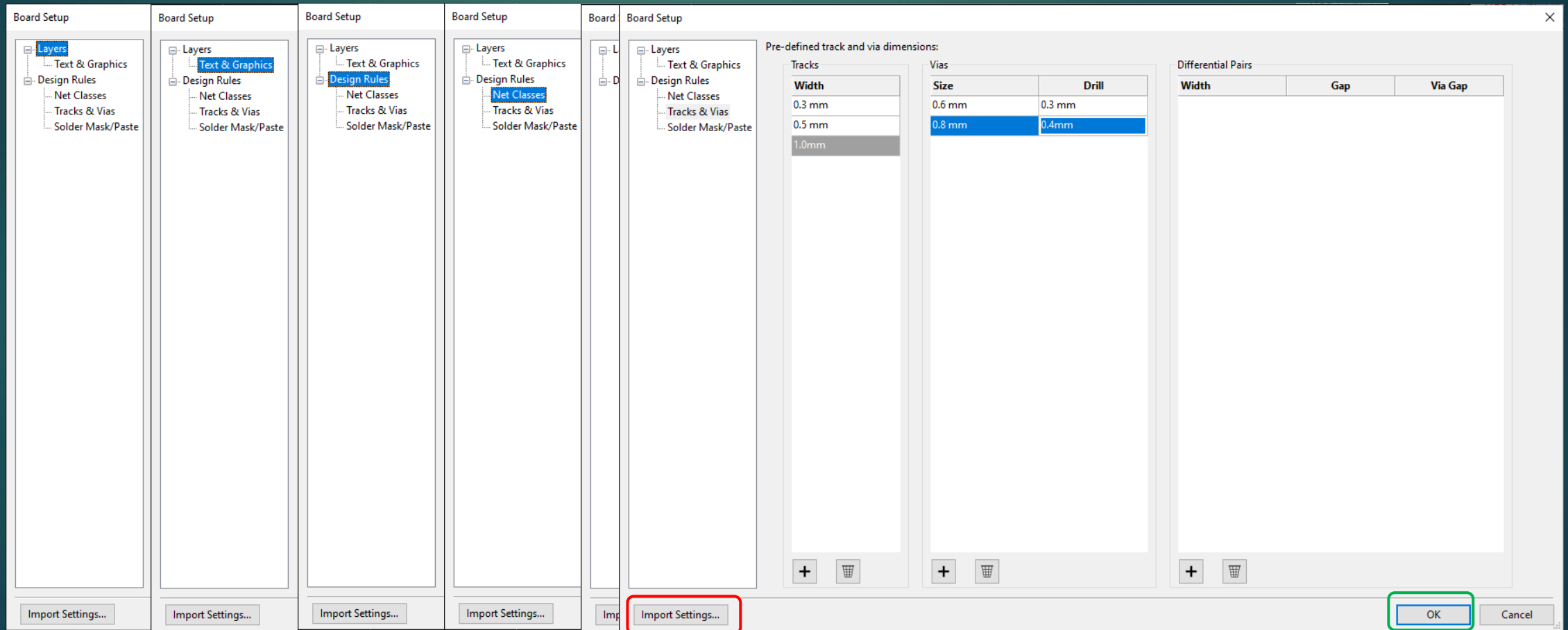


# PCBnew – Reglas de Diseño



## ► Inclusión de la Reglas de Diseño en PCBnew

- Mediante el icono  (Board Setup) ó > File > Board Setup accedemos a las ventanas de las reglas de diseño

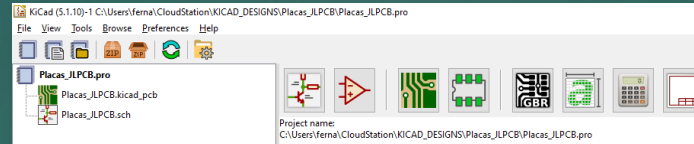


# PCBnew – Reglas de Diseño

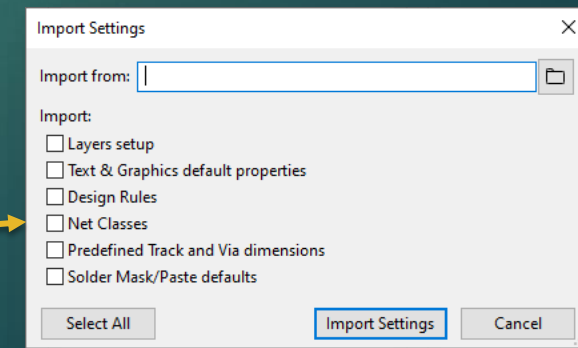
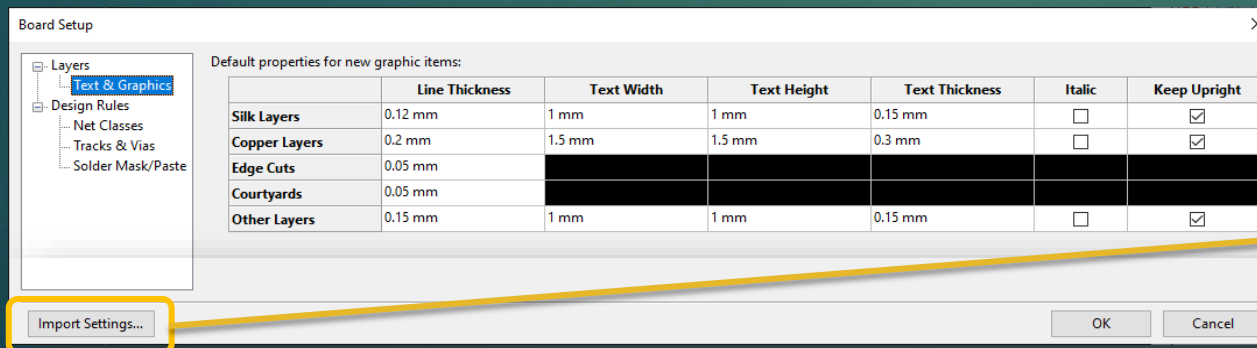


- ▶ Las reglas de diseño se guardan en el fichero del Proyecto (\*.pro) no en el fichero de PCBnew (\*.kicad\_pcb)

- ▶ Crear un proyecto genérico mediante el gestor de proyectos de Kicad, por ejemplo Placas\_JLPCB.pro



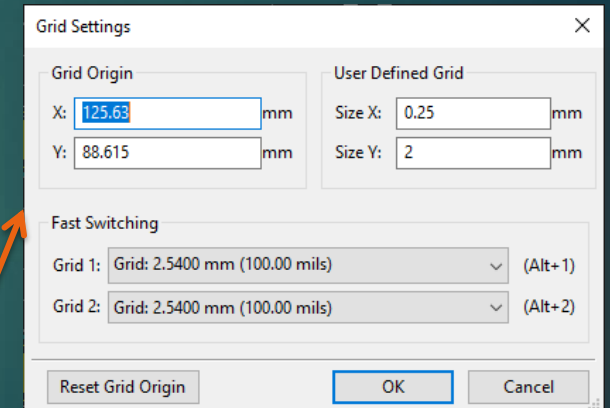
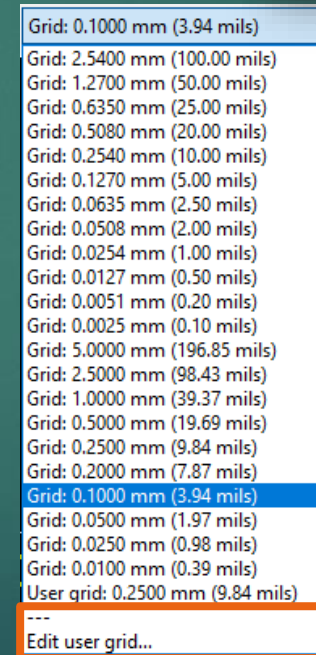
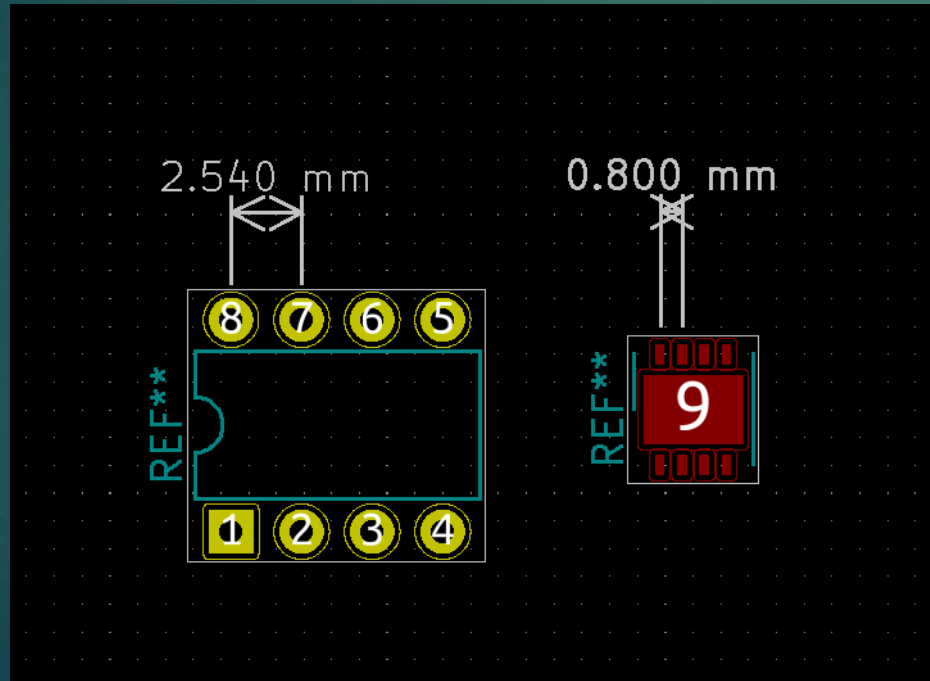
- ▶ Abrir con PCBnew el fichero del PCB que se crea por defecto e incluir todas las reglas de diseño del fabricante, así como diversos anchos de pista y tamaños de vías.
- ▶ Guardar el proyecto
- ▶ En cualquier otro proyecto, en lugar de volver a entrar las Reglas de Diseño, podemos importarlas desde este proyecto genérico con el botón **Import Settings** de la ventana Board Setup y seleccionando que ajustes de reglas queremos importar



# PCBnew- Uso de la rejilla



- ▶ Posiciones en la hoja de diseño donde se anclan componentes, pistas, textos
  - ▶ Standard habitual con medidas en “mils” (1 mil = 1 milésimo de Pulgada = 0,0254 mm)
  - ▶ Nuevos componentes con medidas en mm
- ▶ Escoger la rejilla (Grid) igual a la mitad de la separación de las patillas de los componentes con mas pines. (Circuitos Integrados por ejemplo)
- ▶ Modificación del Origen (0,0) de la rejilla, mediante el icono

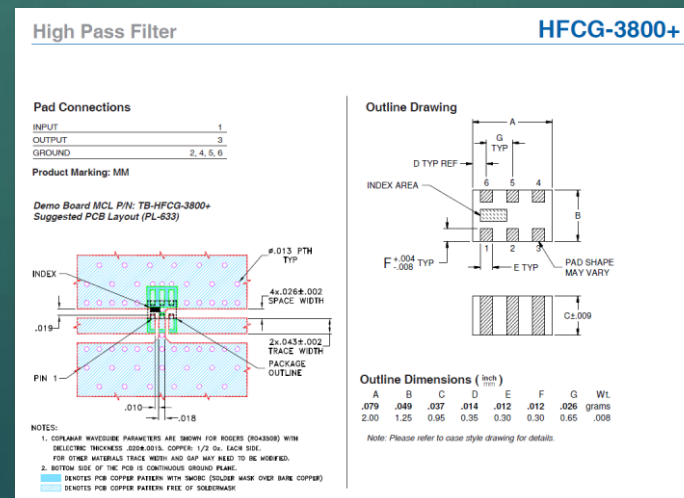


# Creación de un PCB. Algunas ideas

- ▶ Decidir la tecnología a emplear (PTH o SMD)
- ▶ Definir las medidas de la caja metálica que vamos a emplear para adecuar el PCB
  - ▶ <http://www.schubert-gehaeuse.de/weissblechgehaeuse.html>



- ▶ Revisar los data sheets de los componentes a emplear, especialmente los semiconductores, circuitos integrados, componentes especiales y verificar que las huellas a emplear son adecuadas y las medidas son correctas
  - ▶ Tamaño de los Pads, recomendaciones de trazado de pistas





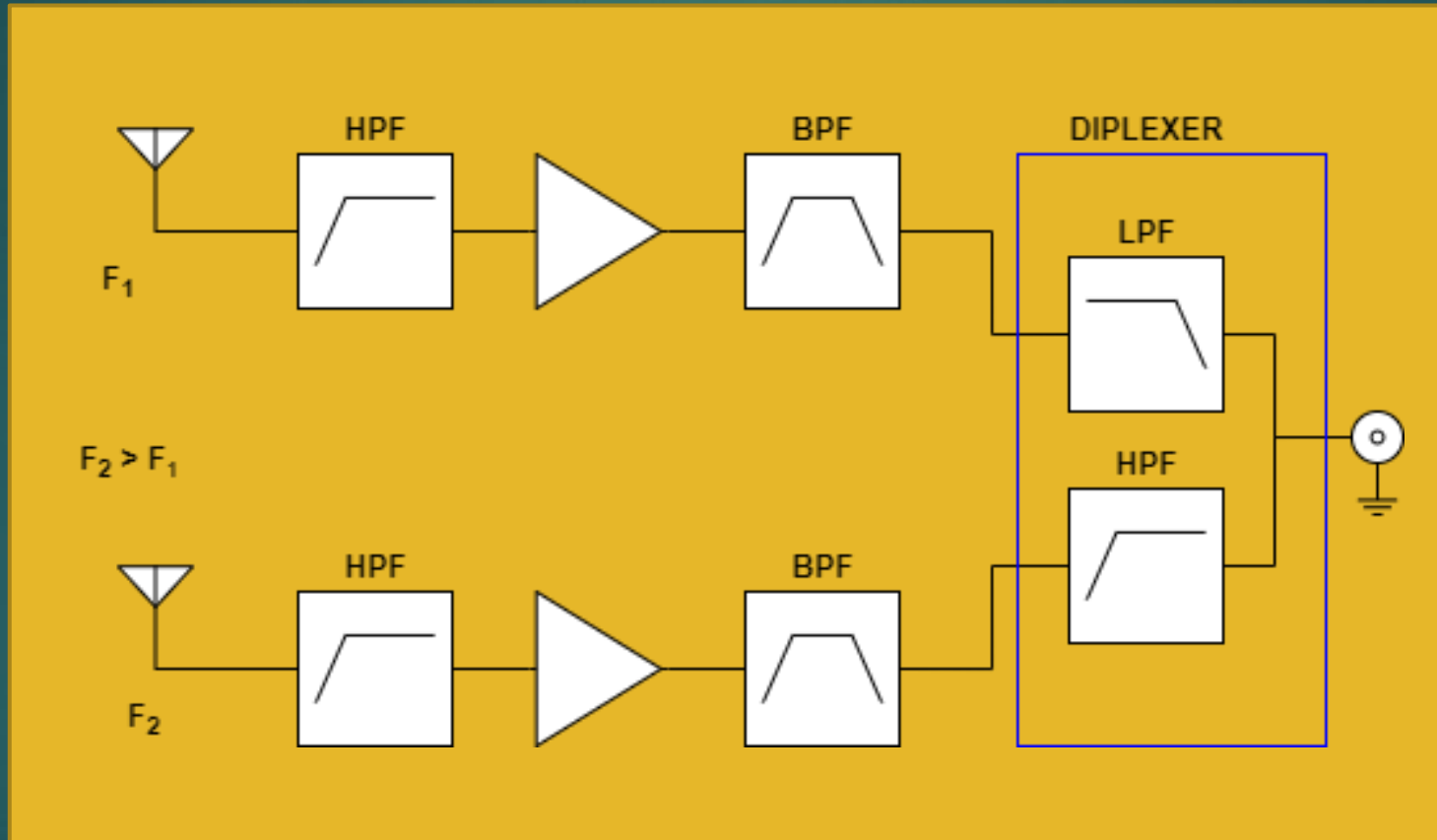
# Creación de un PCB. Proceso



- ▶ Revisar nuevamente el Esquema y verificar la correcta asociación de huellas a cada componente.
- ▶ Abrir PCBnew
- ▶ Importar Reglas de Diseño
- ▶ Establecer la rejilla inicial ( por ejemplo 1mm)
- ▶ Importar el Netlist del Esquema
- ▶ Dibujar el contorno del PCB en función del tamaño de la caja a emplear
- ▶ Mover y colocar los taladros en las esquinas del PCB
- ▶ Mover y colocar las huellas de los componentes / Mirar Esquema periódicamente
  - ▶ Separación entre componentes
- ▶ Trazar las Pistas de Conexión
  - ▶ Conexiones cortas. Evitar ángulos de 90°
- ▶ Crear Planos de masa si son necesarios
- ▶ Verificar con DRC

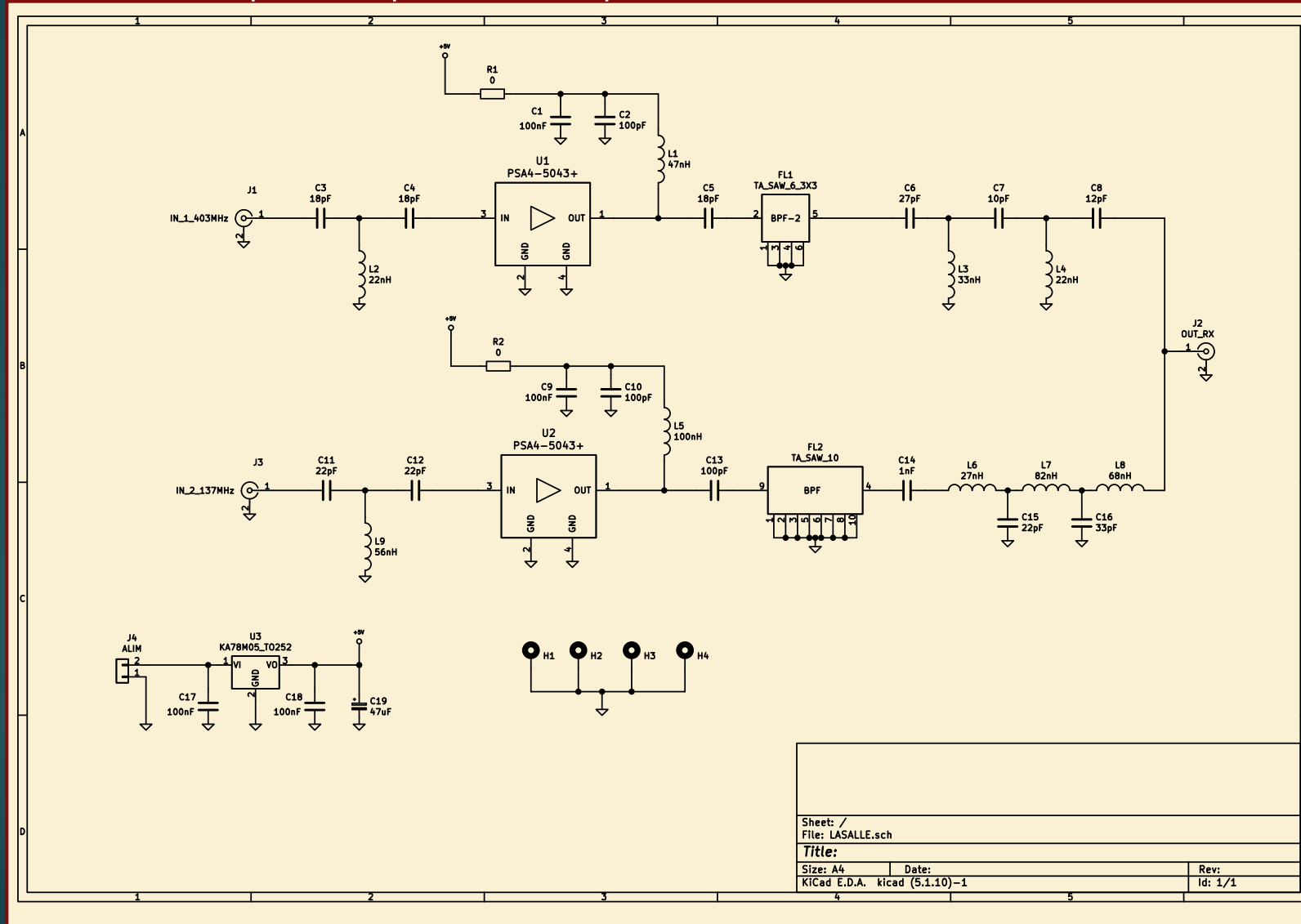
# Creación de un Esquema Sencillo

- LNA Doble con Diplexor para recepción con SDR



# Creación de un Esquema Sencillo

## ► LNA Doble con Diplexor para recepción con SDR



# Creación de un Esquema Sencillo

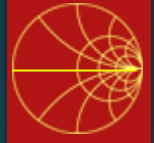


DEMO



# Preguntas

# Enlaces interesantes



Kicad <https://www.kicad.org/>

Draw.io <https://www.diagrams.net/>

Librerías de Símbolos y huellas <https://www.snapeda.com/>

Tutoriales de Kicad <https://www.youtube.com/user/contextualelectronic/playlists>

Librerías 3D <https://www.3dcontentcentral.es/>

Cajas metálicas <http://www.schubert-gehaeuse.de/weissblechgehaeuse.html>



# Agenda



## Parte III ( 5 de Noviembre)

- ▶ **Diseño PCB para RF.**
- ▶ Proceso de Diseño. Objetivos / Especificaciones del dispositivo
  - ▶ Esquema de Bloques
  - ▶ Simulación con QUCS Studio
  - ▶ Esquema con Kicad
  - ▶ PCB con Kicad
    - ▶ Diseño de PCBs para RF.
    - ▶ Tipos de pistas de RF (Microstrip, CPWG, Stripline, etc)
    - ▶ Cálculo de CPWG
    - ▶ Diseñando pistas de RF con KICAD.
  - ▶ Los errores mas habituales y como evitarlos
  - ▶ Ejemplos