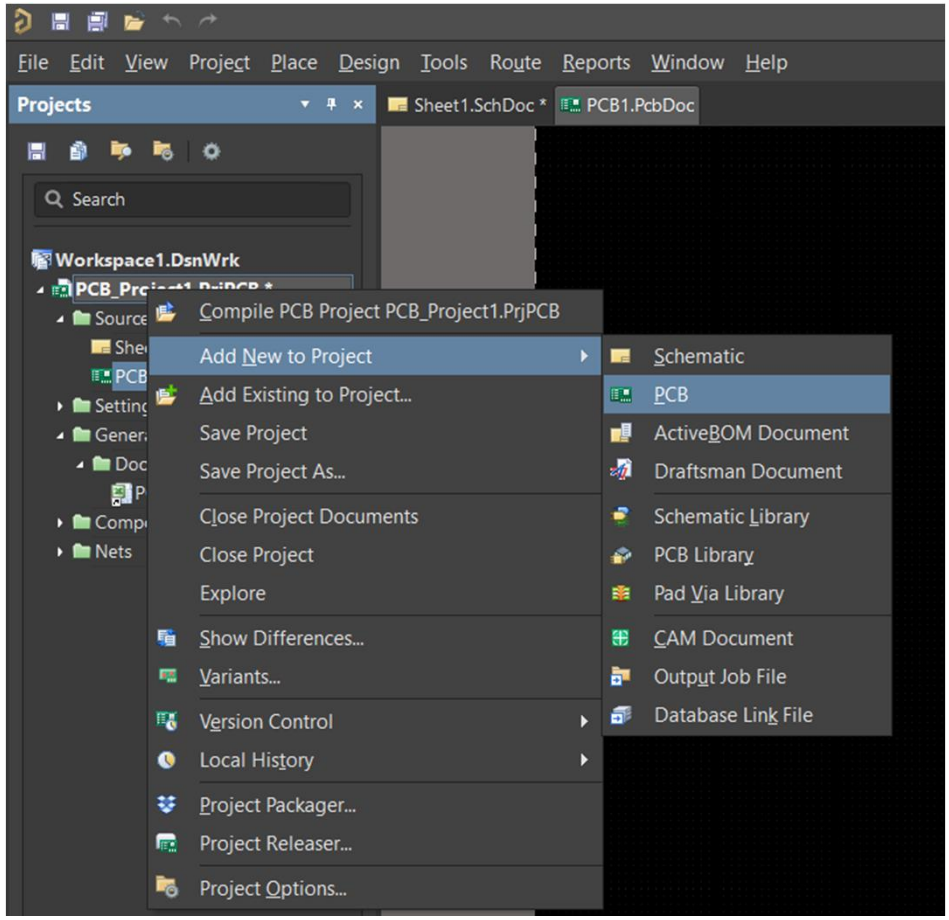
Layout



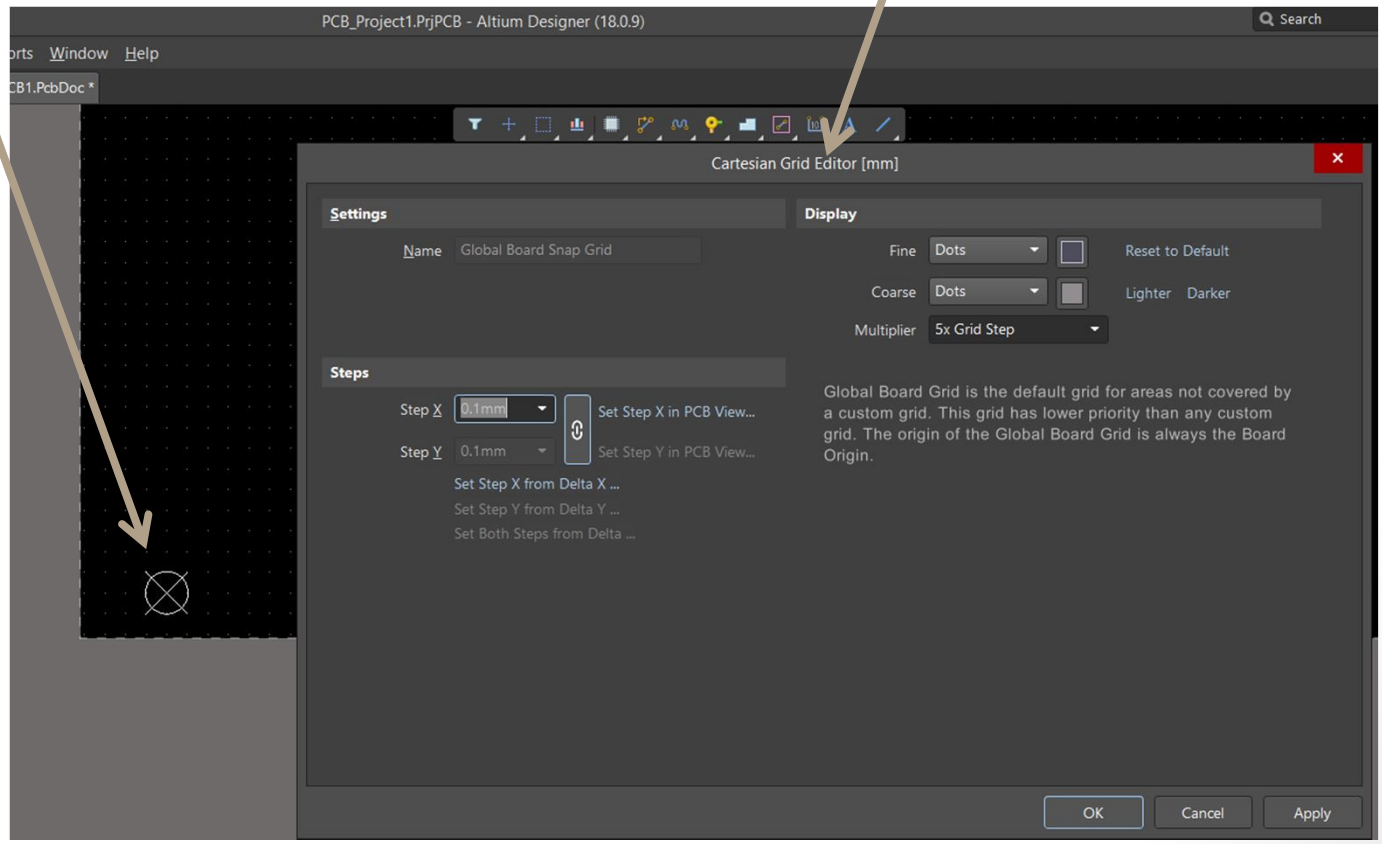
Attenzione all'unità di Misura e alla griglia

Ricordarsi di salvare il Documento pe rpoter fare l'Update da schematico.

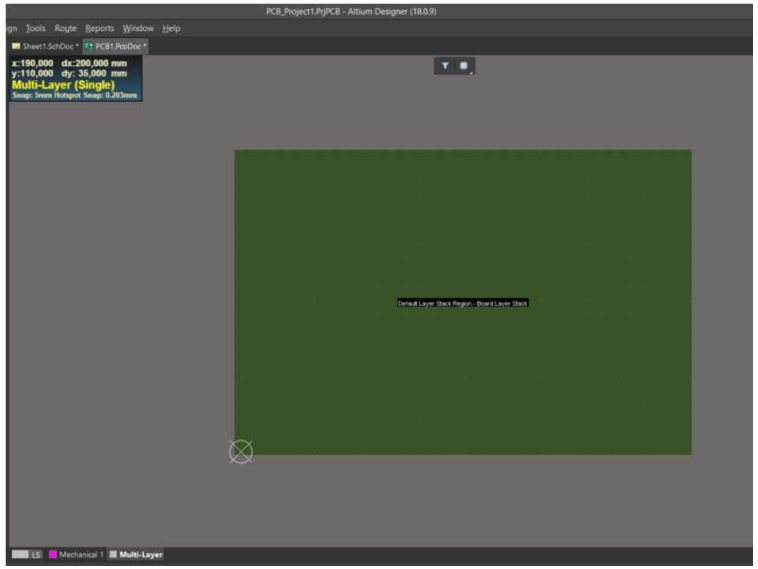
Origine e Griglia

CTRL- G per impostare la griglia
e per fare lo snap VIEW Grids Grids (V G G)

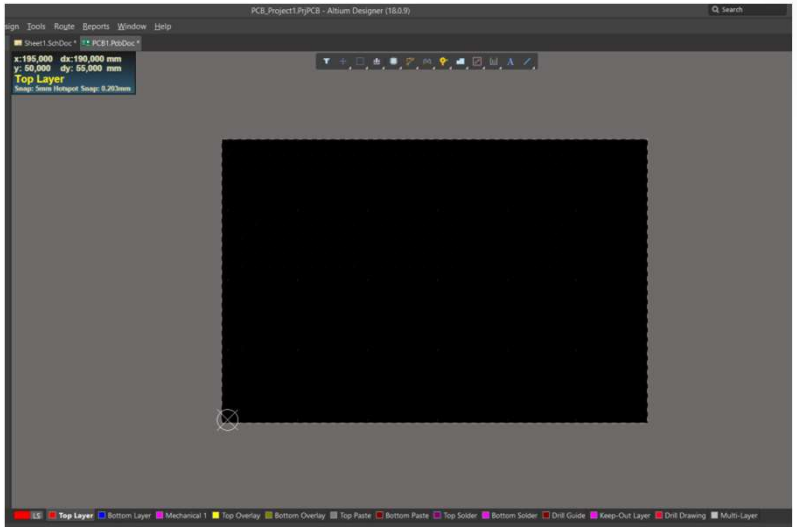
Edit – origin – set
per spostare l'origine



Visualizzazione

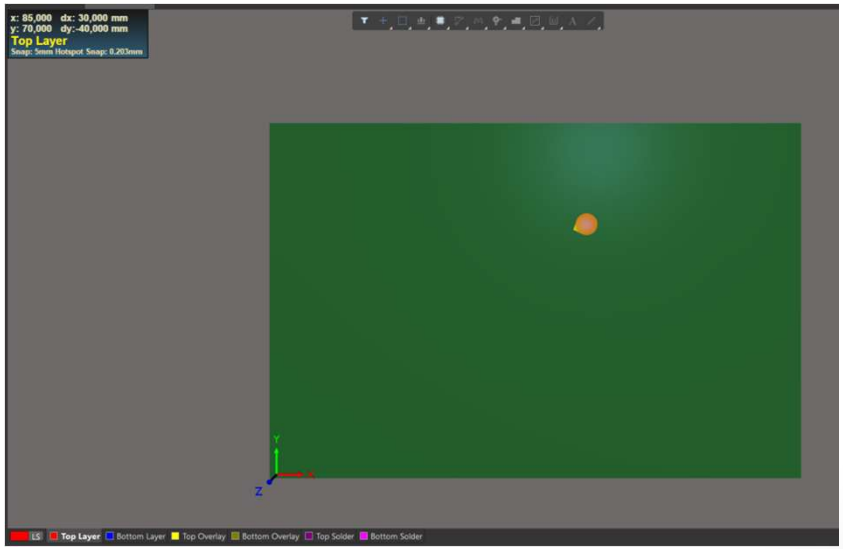


‘’1’ Board Planning Mode

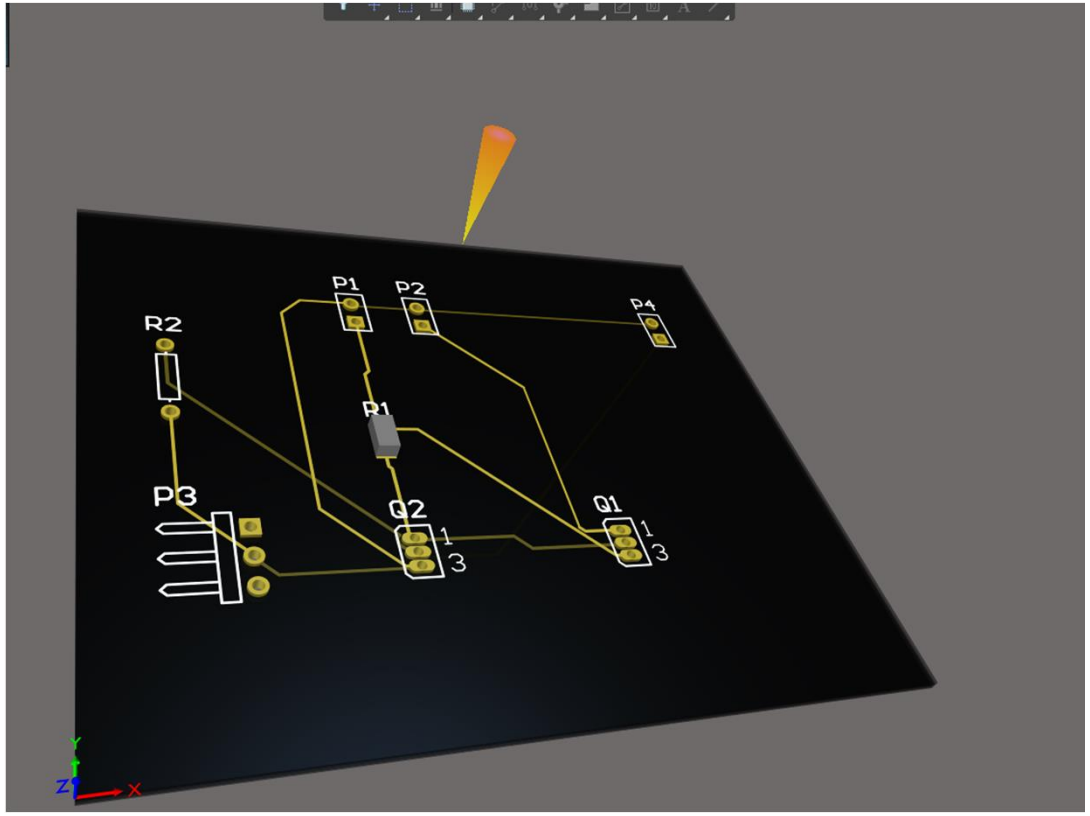


‘2’ 2D Layout Mode

‘3’ 3D Layout Mode



3D Layout Mode

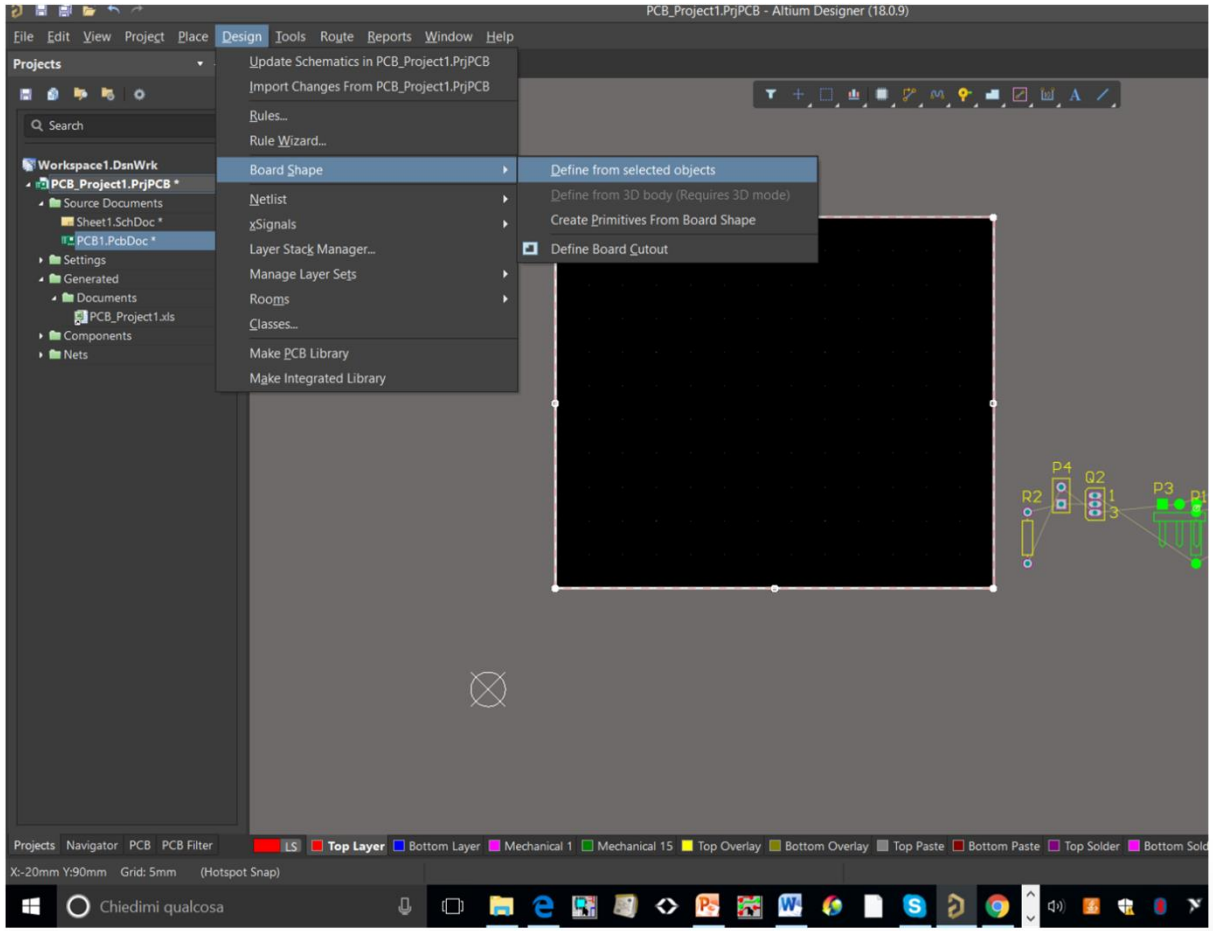


Right-drag Mouse
Per spostare l'immagine

Shift+right-drag mouse
Per ruotare la vista

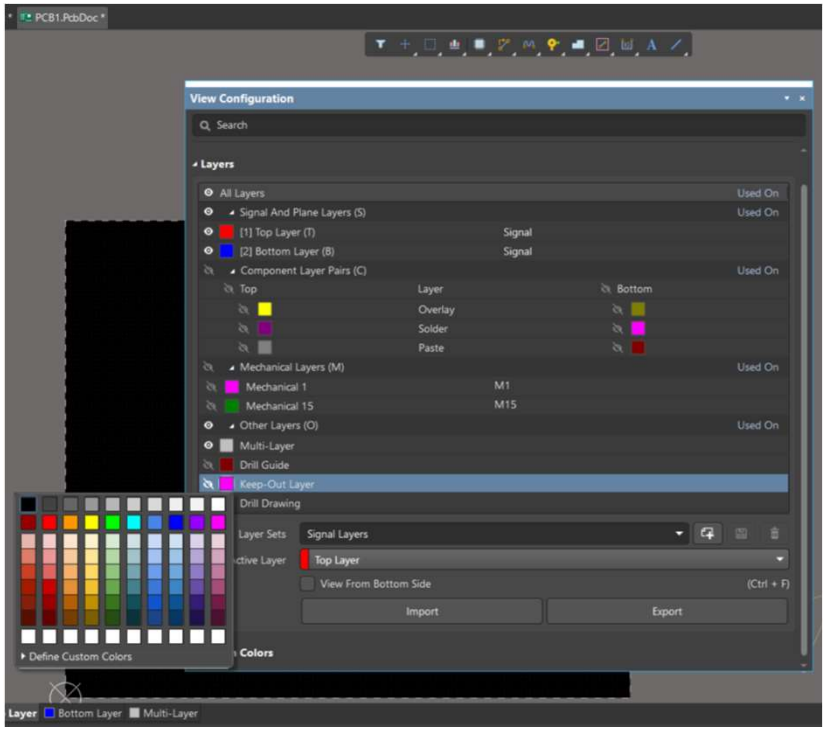
Board OutLine

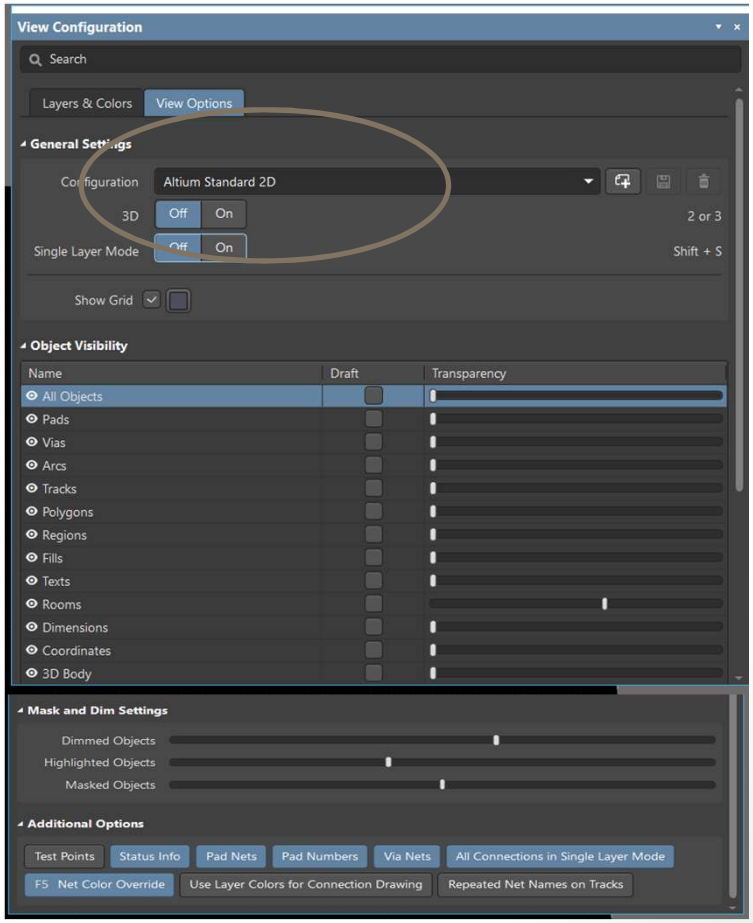
1. Disegnare il contorno graficamente
2. Definirlo con DESIGN- BOARD SHAPE –DEFINE from selected object
3. Se necessario inserire dei CutOut



Layers

Elettrici: 32 + 16 interni di Powerplane
Meccanici: 32
Speciali: silkscreen, solder, paste, drill ……

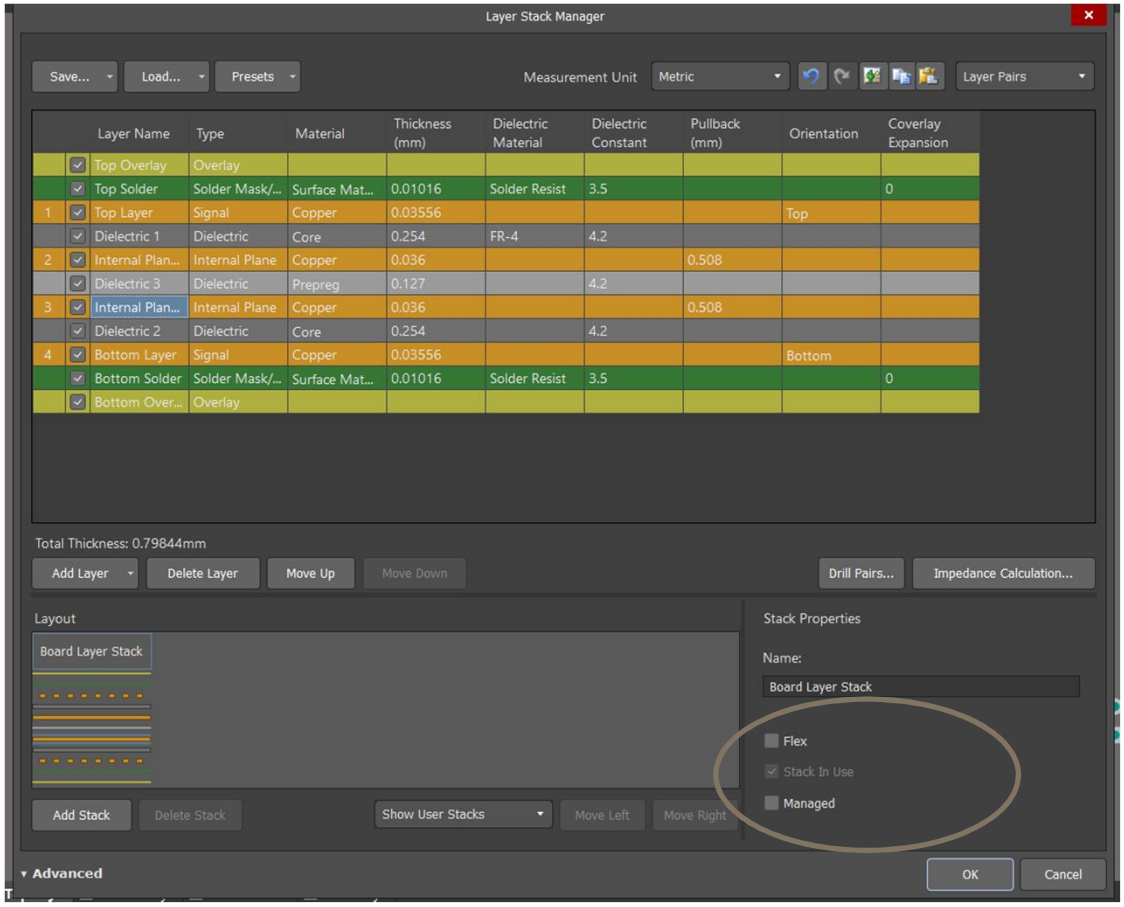




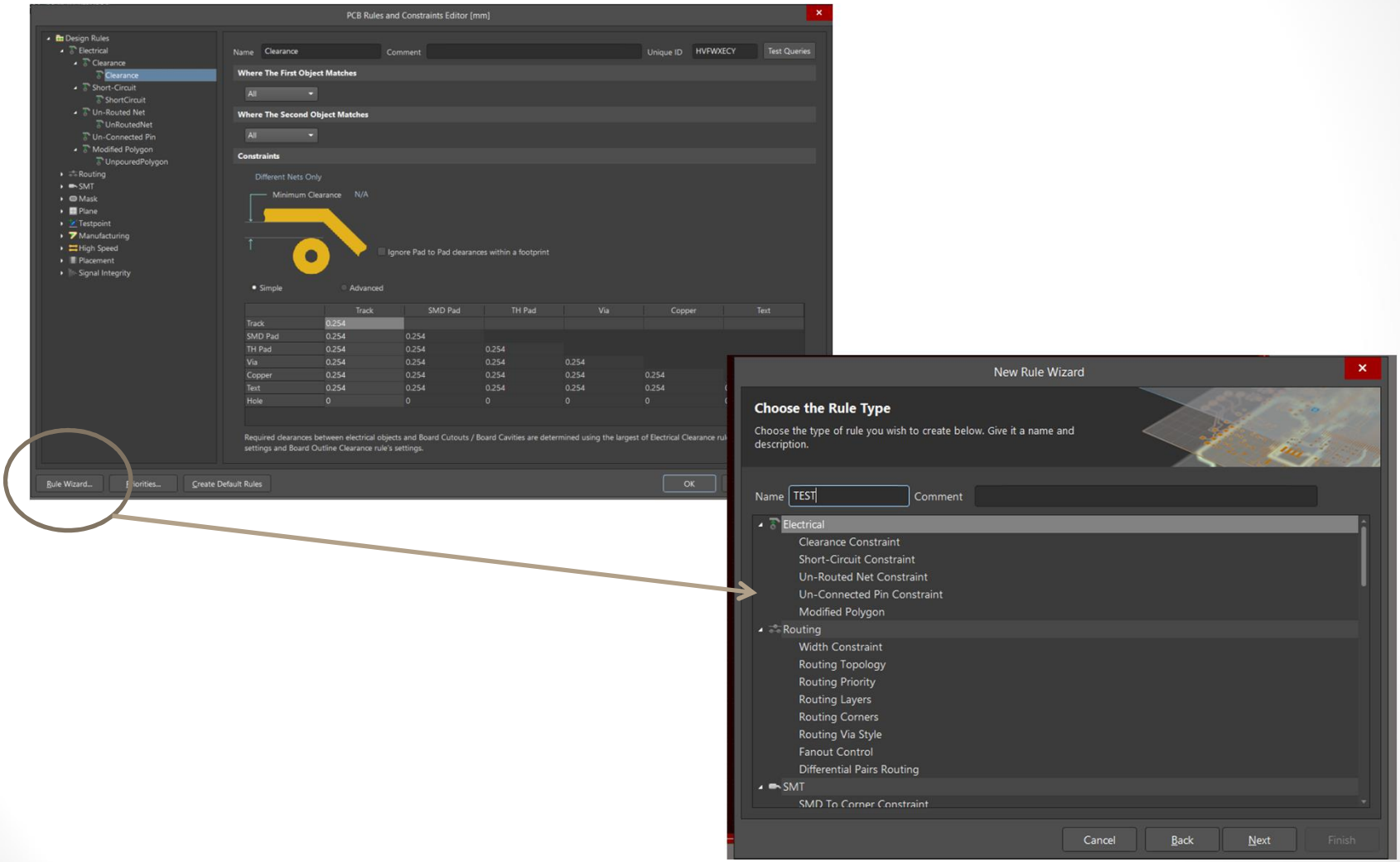
Per vedere la configurazione ‘L’

Layers stack manager

Si accede da Design – Layer Stack Manager



Design Rules



Design Rules

- Rule Types:

  Unary applicato all'oggetto (es Width)

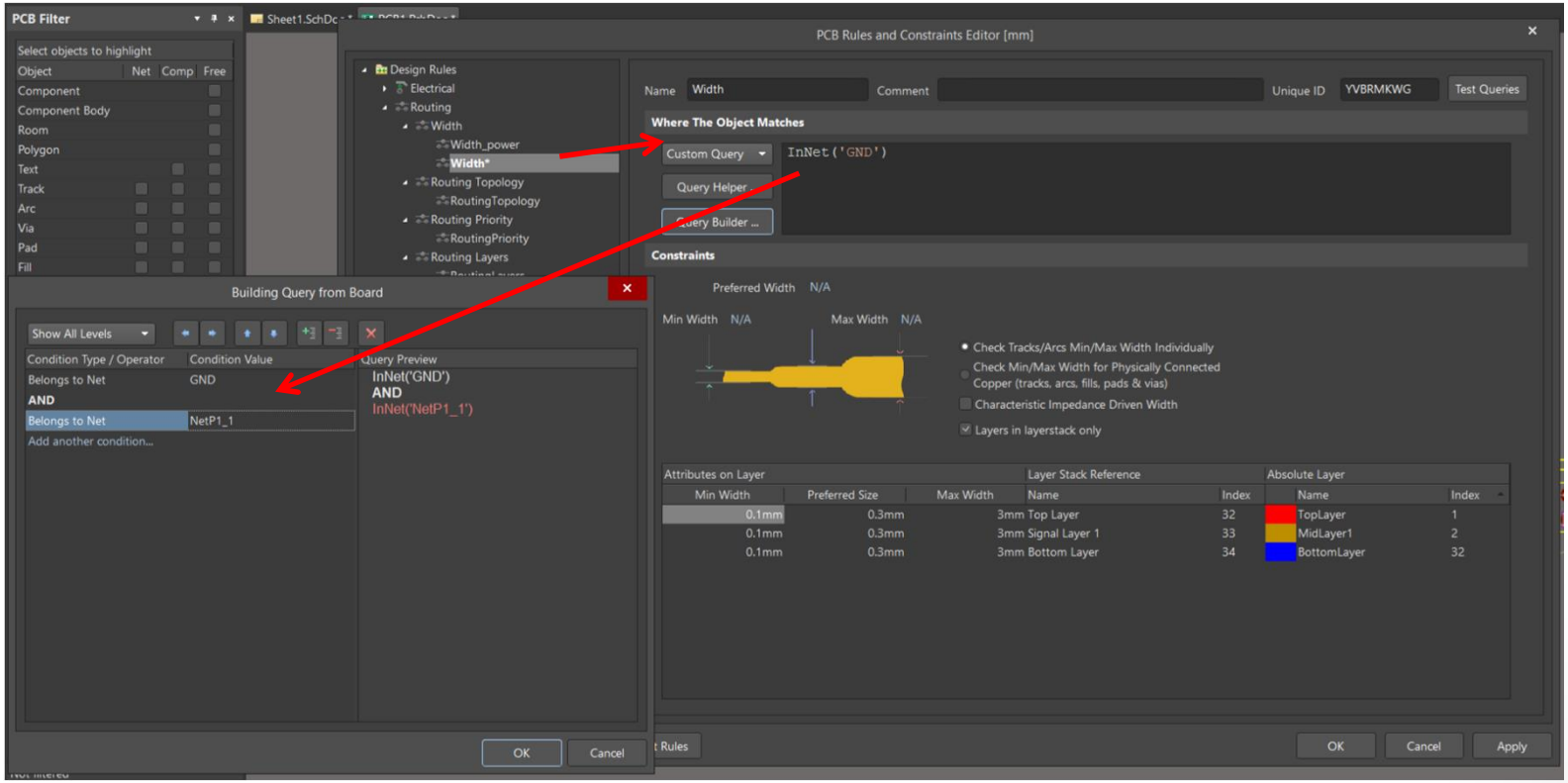
  Binary fra oggetti (esempio isolamento fra due oggetti)

- Rule Priority:

  c'è un ordine di importanza nelle regole di disegno,

  quindi uso generale locale e specifico.

New Rules

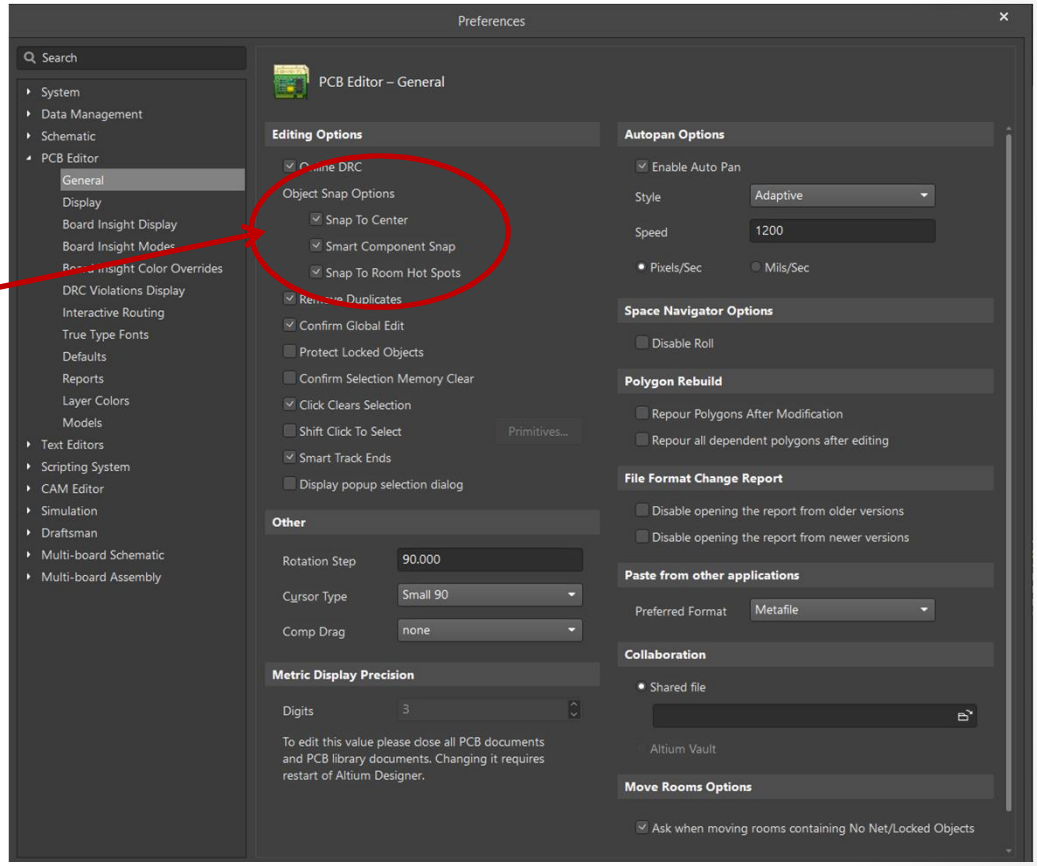

PCB Filter
Select objects to highlight
Object Net Comp Free
Component
Component Body
Room
Polygon
Text
Track
Arc
Via
Pad
Fill
Sheet1.SchDoc
PCB Rules and Constraints Editor [mm]
Design Rules
Electrical
Routing
Width
Width_power
Width*
Routing Topology
RoutingTopology
Routing Priority
RoutingPriority
Routing Layers
Name Width
Comment
Unique ID YVBRMKWG
Test Queries
Where The Object Matches
Custom Query
InNet('GND')
Query Helper ...
Query Builder ...
Constraints
Preferred Width N/A
Min Width N/A
Max Width N/A
Check Tracks/Arcs Min/Max Width Individually
Check Min/Max Width for Physically Connected Copper (tracks, arcs, fills, pads & vias)
Characteristic Impedance Driven Width
Layers in layerstack only
Attributes on Layer
Min Width
Preferred Size
Max Width
Layer Stack Reference
Name
Index
Absolute Layer
Name
Index
0.1mm 0.3mm 3mm Top Layer 32 TopLayer 1
0.1mm 0.3mm 3mm Signal Layer 1 33 MidLayer1 2
0.1mm 0.3mm 3mm Bottom Layer 34 BottomLayer 32
Building Query from Board
Show All Levels
Condition Type / Operator
Condition Value
Query Preview
Belongs to Net GND
AND
Belongs to Net NetP1_1
Add another condition...
InNet('GND')
AND
InNet('NetP1_1')
OK
Cancel
Rules
OK
Cancel
Apply

Posizionamento

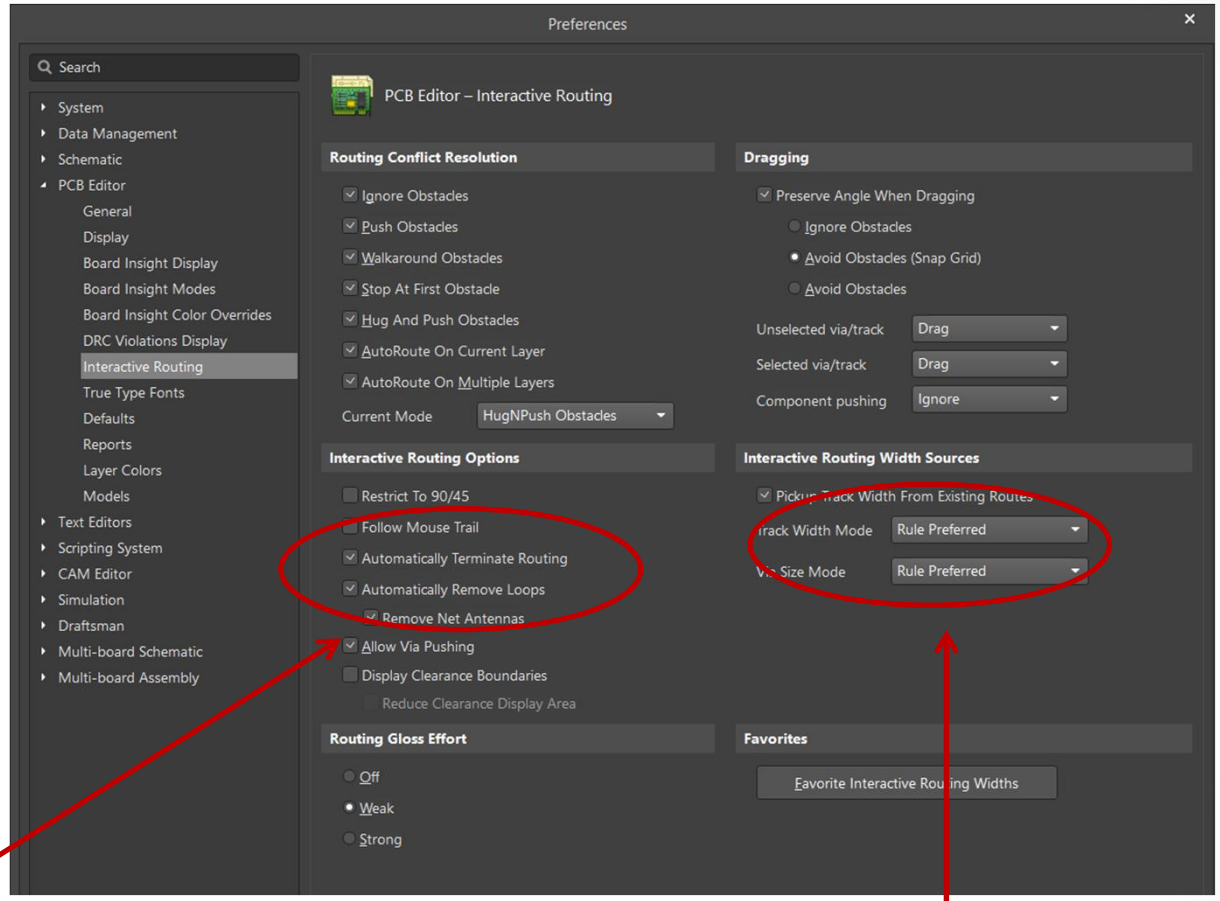
Snap to center mi garantisce che prendo dal centro il componente

Smart Component Snap lo prende dal pad più vicino

Posso disabilitare la visione delle Unconnected Net da View Connections



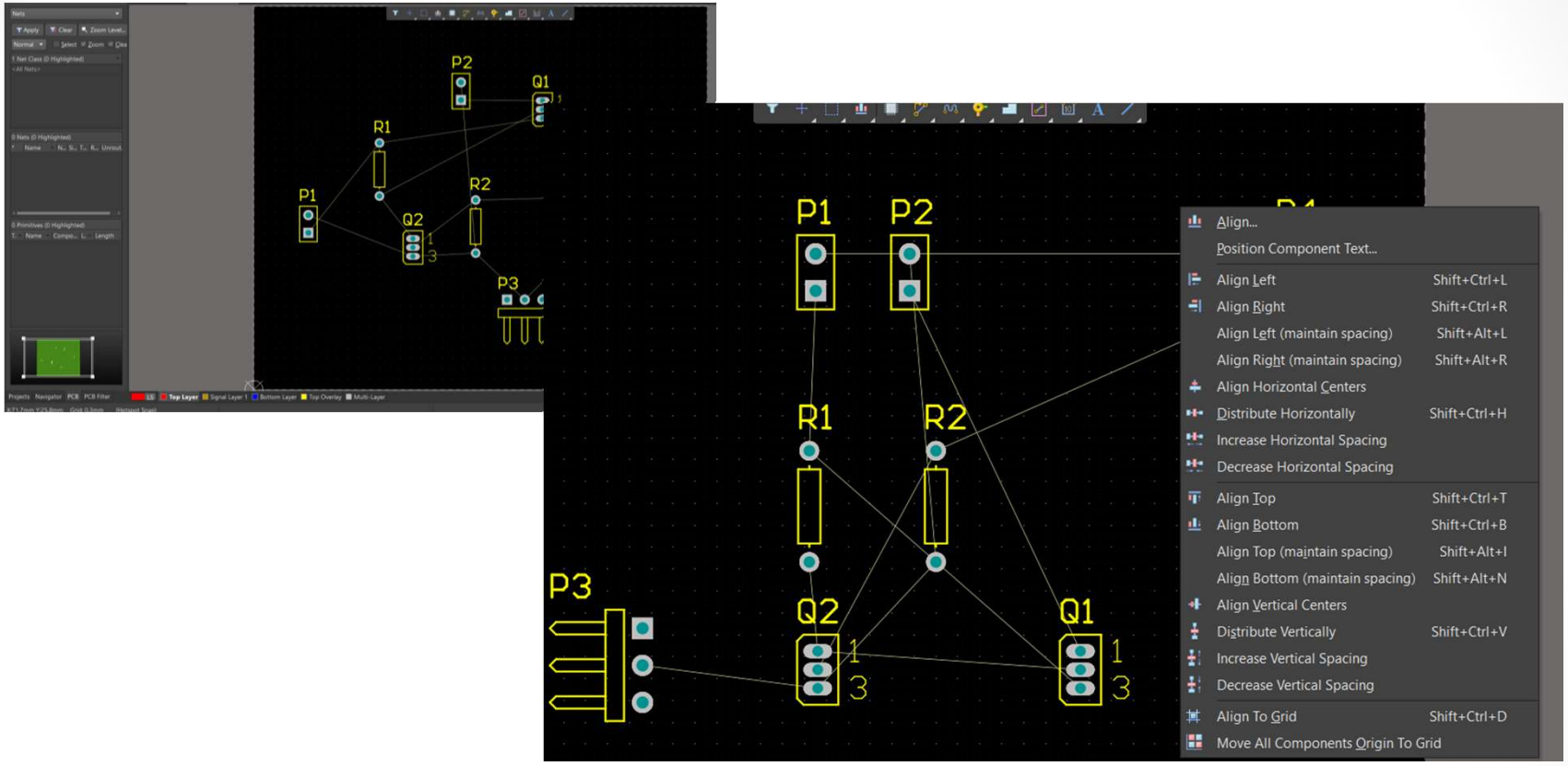
Interactive Routing preferences



Termina in automatico la linea.
EIl loop mi permette di cambiare
la linea eliminando quello che è superfluo

Così usa le regole di preferenza
per vias e wire

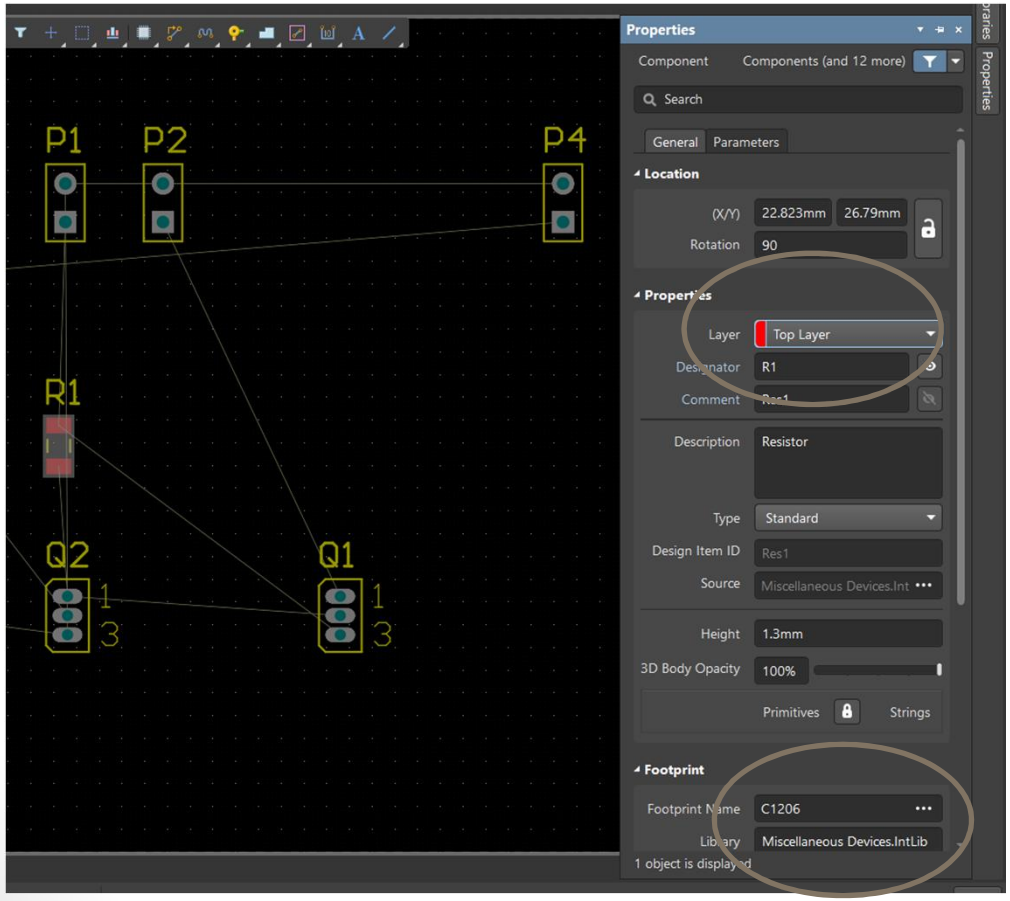
Placement



Spacebar per la rotazione.
Con Shift posso selezionare più parti e fare allineamenti con Align.
Posso muovere i componenti anche con le frecce tenendo CTRL.

Footprint e Layer del componente

Doppio click sul componente e poi posso definire su che layer posizionarlo
e cambiare la footprint -→ facendo Update Schematic aggiorno lo schematico



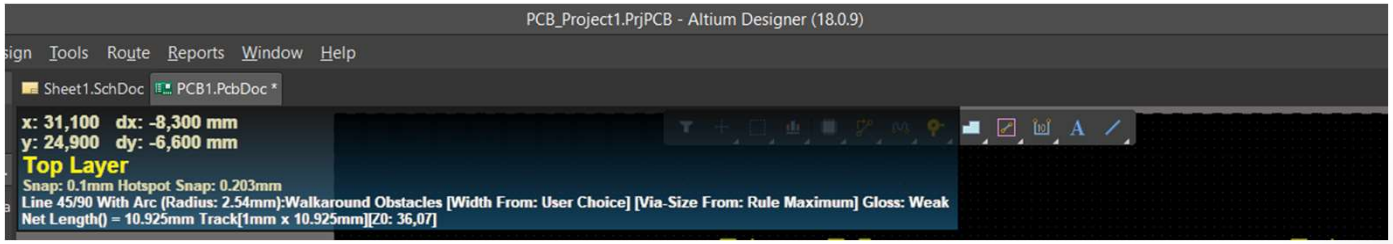
Per dove mettere i nomi (comment) :
In PCB Editor – Default - Preferences comment
o su ogni componente come parametro con doppio click

Routing

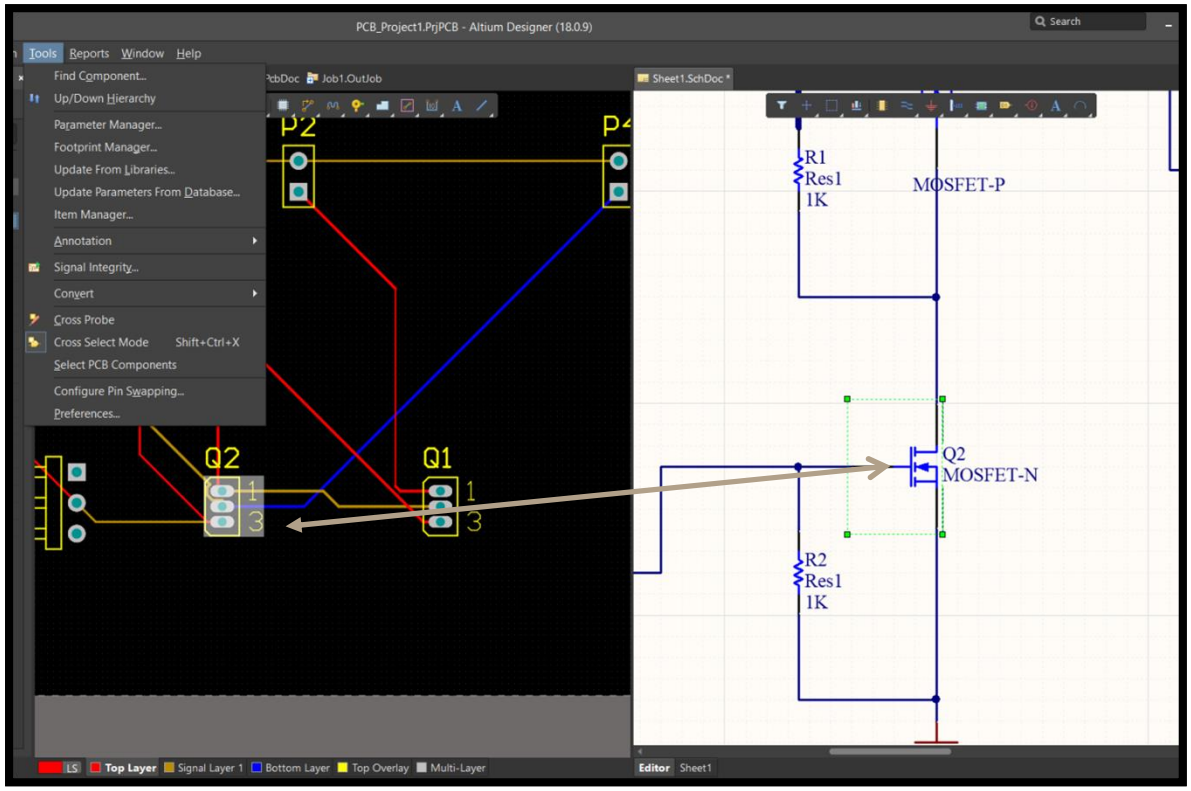
- Shift+F1 per gli short cuts
- Chiusura in automatico di una pista con CTRL +LEFT Click
- U per unroute
- + per cambio layer generico
- * per passare solo fra i layer di segnale
- Oppyure con ctrl+shift + roll
- Shift+W per dimensione pista
- Spacebar per flippare la pista
- Shift + Spacebar per cambiare l'angolo (limitabile da PCB editor - Interactive Routing)
- Shift S per routing in singol layer e posso settare il modo in PCB-Editor Board Insight DIsplay

Routing

- Shift+R per cambiare le modalità
  - Ignore: pista dove voglio
  - Stop at first obstacle
  - WALKAROUND giro intorno
  - Push riposiziona i componenti ch esi possono muovere senza dare errori
  - Hug&Push unione di walkaround e Push
  - Autoroute in current layer
  - Autoroute ion Multipler layer

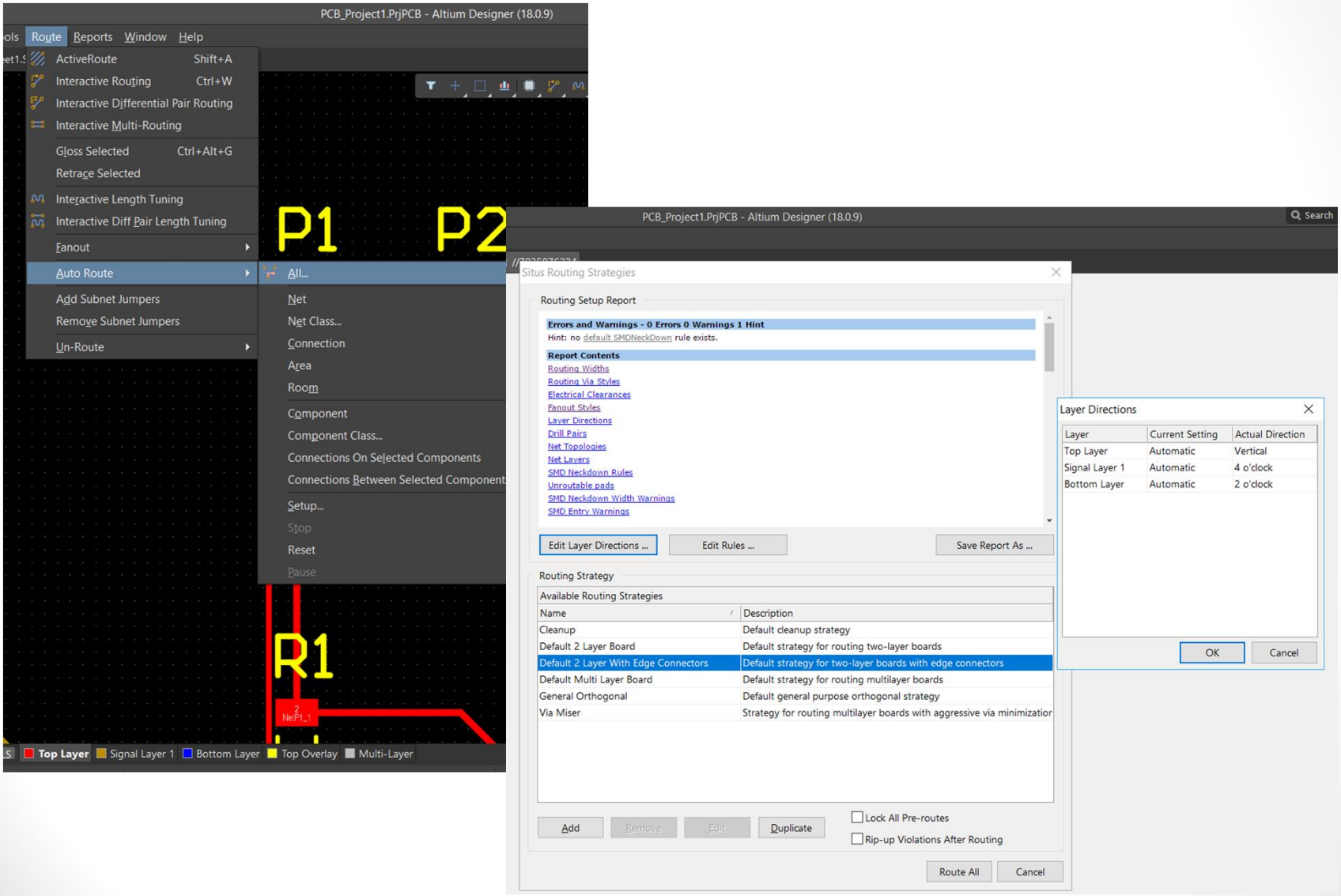


Cross Selector Mode

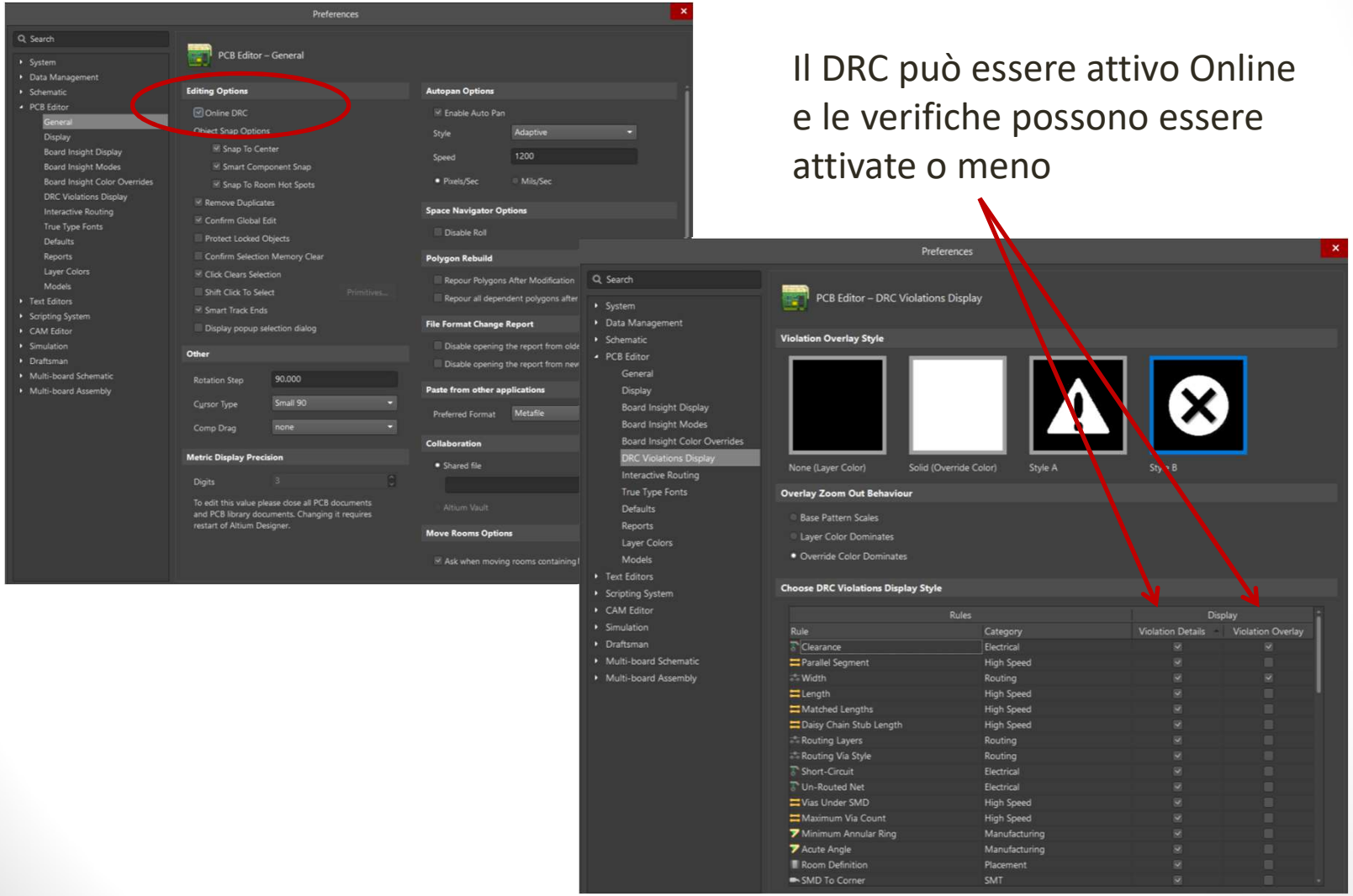


Lo attivo da Tools e deve esserlo sia per layout che per schematico

Autorouting

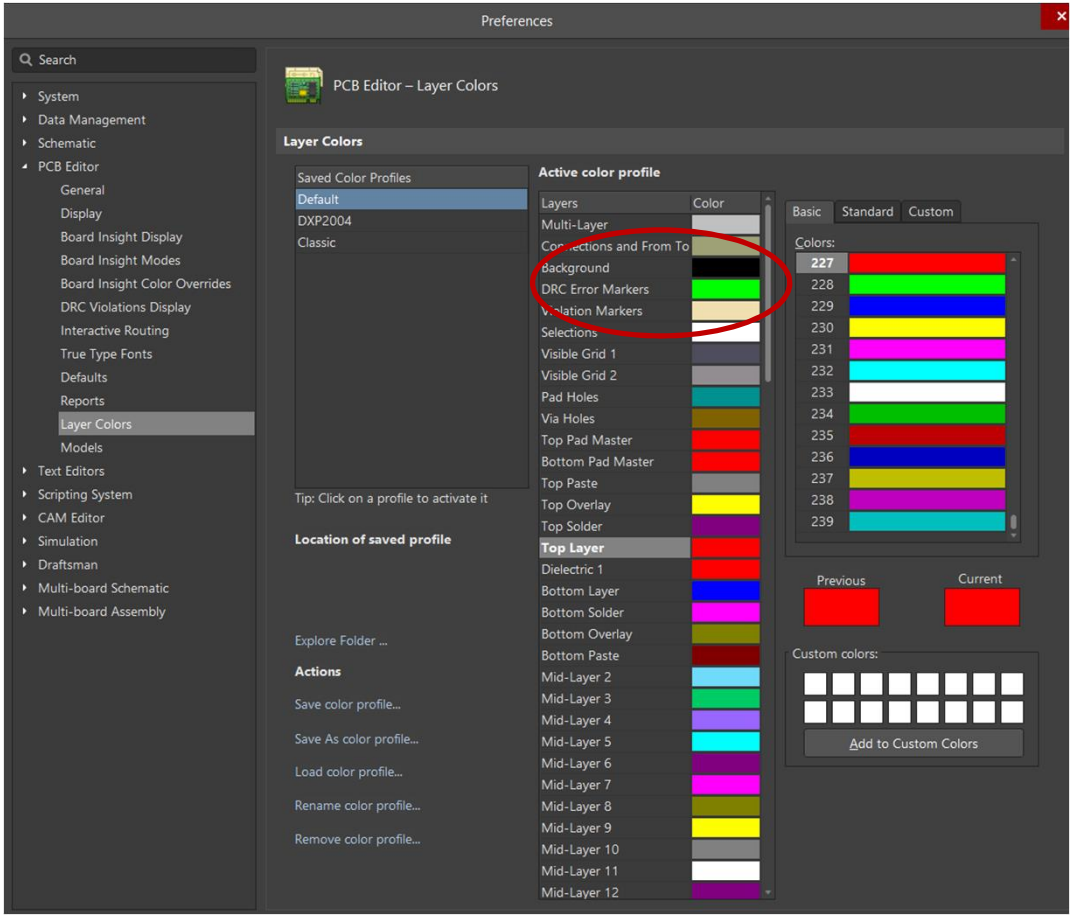


VERIFY

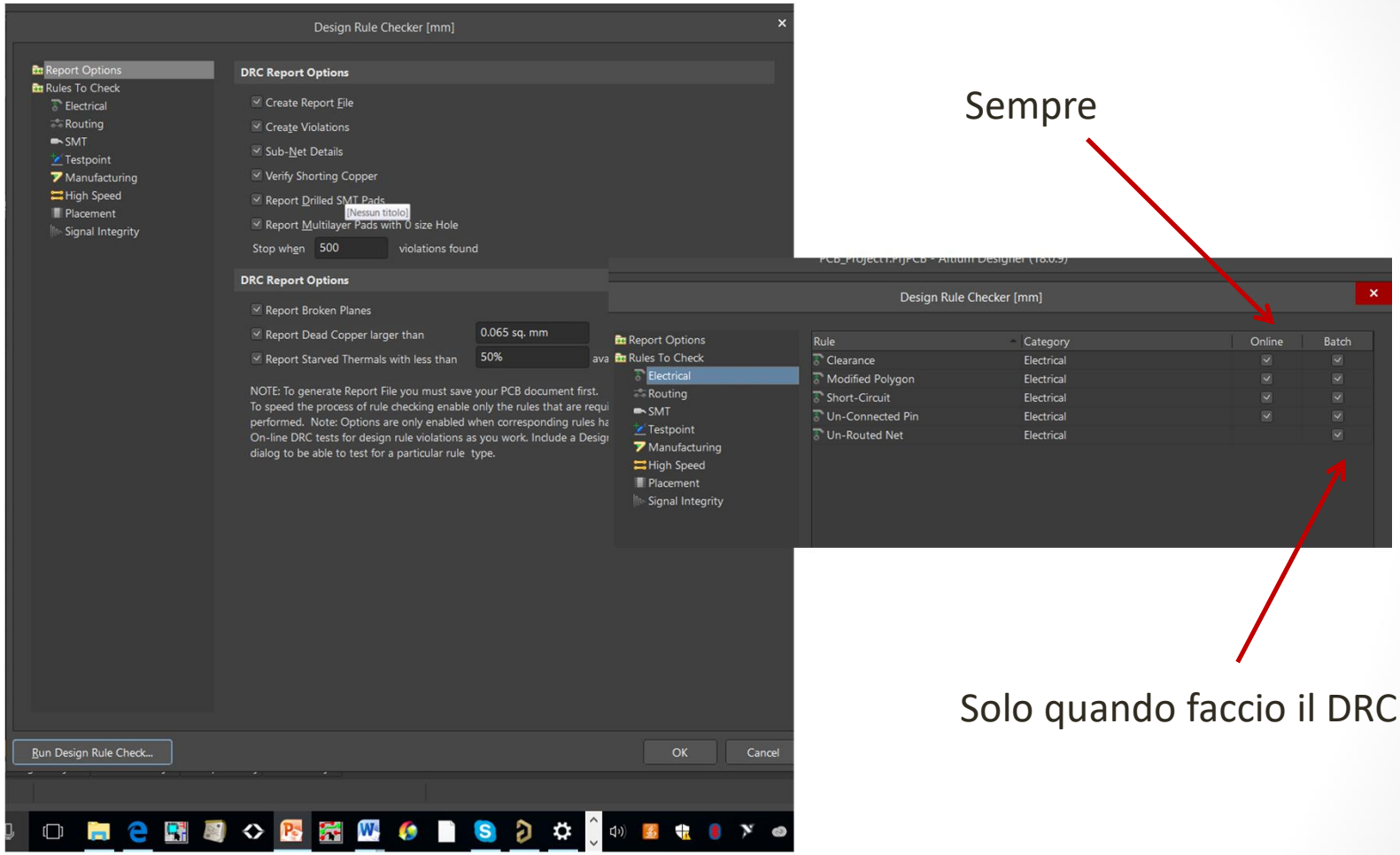


Il DRC può essere attivo Online e le verifiche possono essere attivate o meno

Verify


Preferences
Search
System
Data Management
Schematic
PCB Editor
General
Display
Board Insight Display
Board Insight Modes
Board Insight Color Overrides
DRC Violations Display
Interactive Routing
True Type Fonts
Defaults
Reports
Layer Colors
Models
Text Editors
Scripting System
CAM Editor
Simulation
Draftsman
Multi-board Schematic
Multi-board Assembly
PCB Editor – Layer Colors
Layer Colors
Saved Color Profiles
Default
DXP2004
Classic
Tip: Click on a profile to activate it
Location of saved profile
Explore Folder ...
Actions
Save color profile...
Save As color profile...
Load color profile...
Rename color profile...
Remove color profile...
Active color profile
Layers
Color
Multi-Layer
Background
DRC Error Markers
Selections
Visible Grid 1
Visible Grid 2
Pad Holes
Via Holes
Top Pad Master
Bottom Pad Master
Top Paste
Top Overlay
Top Solder
Top Layer
Dielectric 1
Bottom Layer
Bottom Solder
Bottom Overlay
Bottom Paste
Mid-Layer 2
Mid-Layer 3
Mid-Layer 4
Mid-Layer 5
Mid-Layer 6
Mid-Layer 7
Mid-Layer 8
Mid-Layer 9
Mid-Layer 10
Mid-Layer 11
Mid-Layer 12
Basic
Standard
Custom
Colors:
227
228
229
230
231
232
233
234
235
236
237
238
239
Previous
Current
Custom colors:
Add to Custom Colors

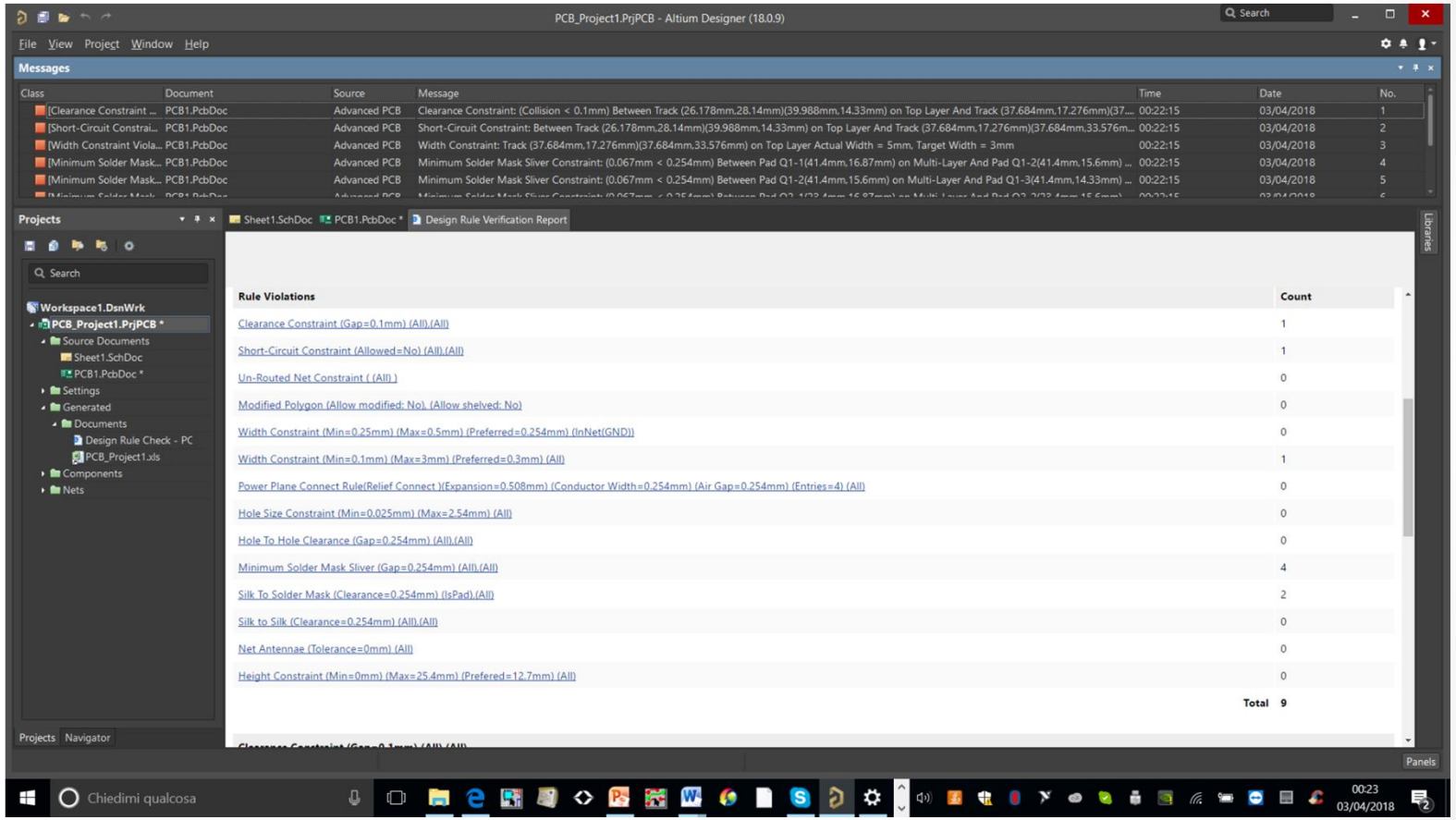
Verify – Tools DRC



Sempre

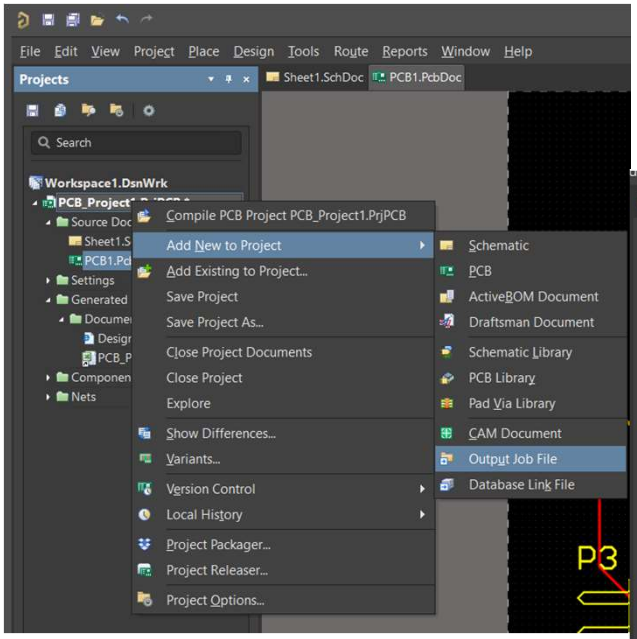
Solo quando faccio il DRC

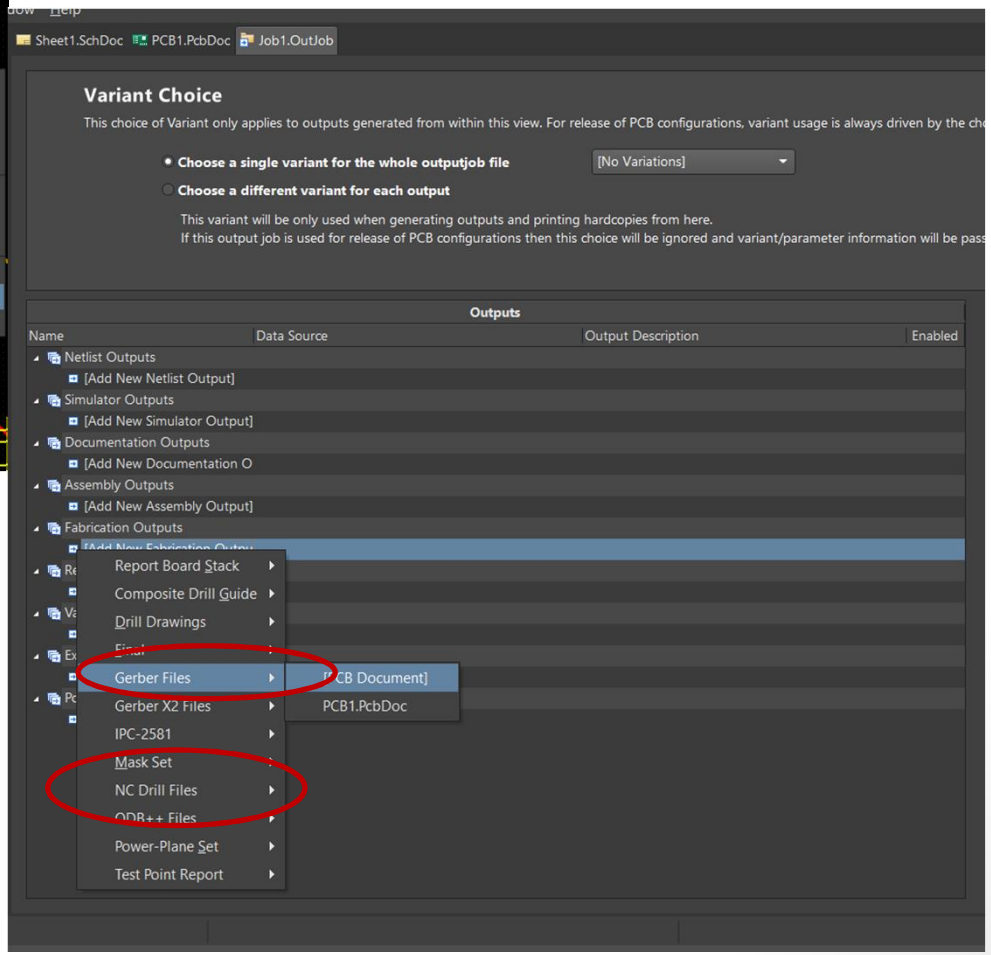
Verify – DRC messages



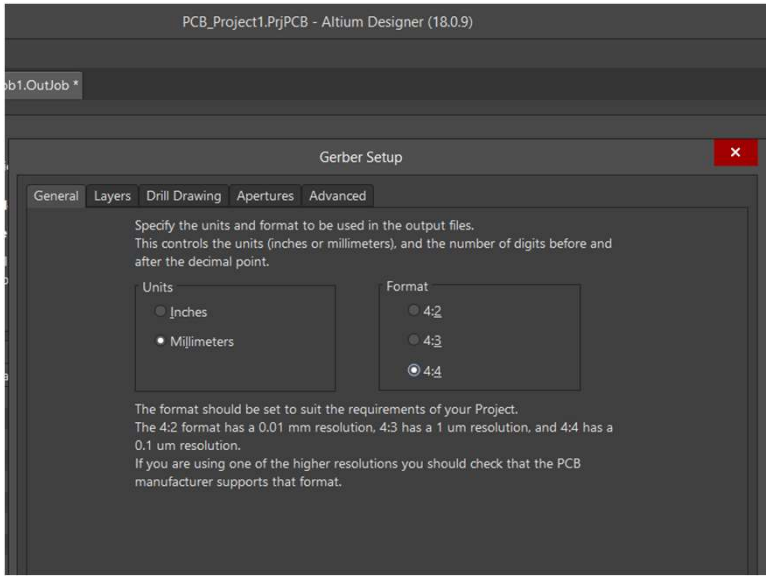
OUTPUT

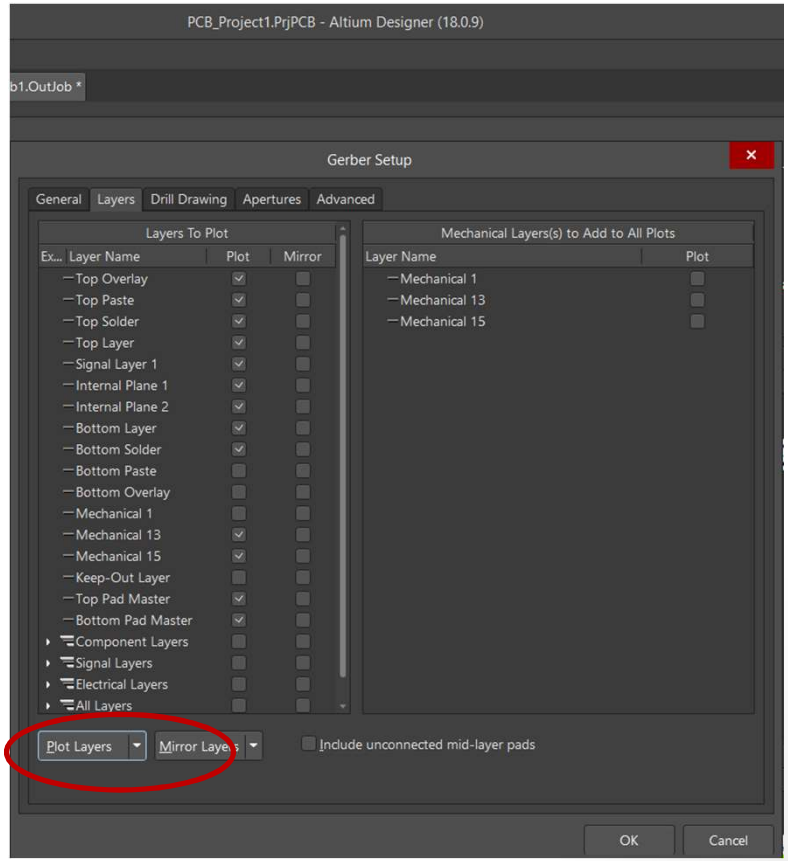
Creare un file output-job e salvarlo



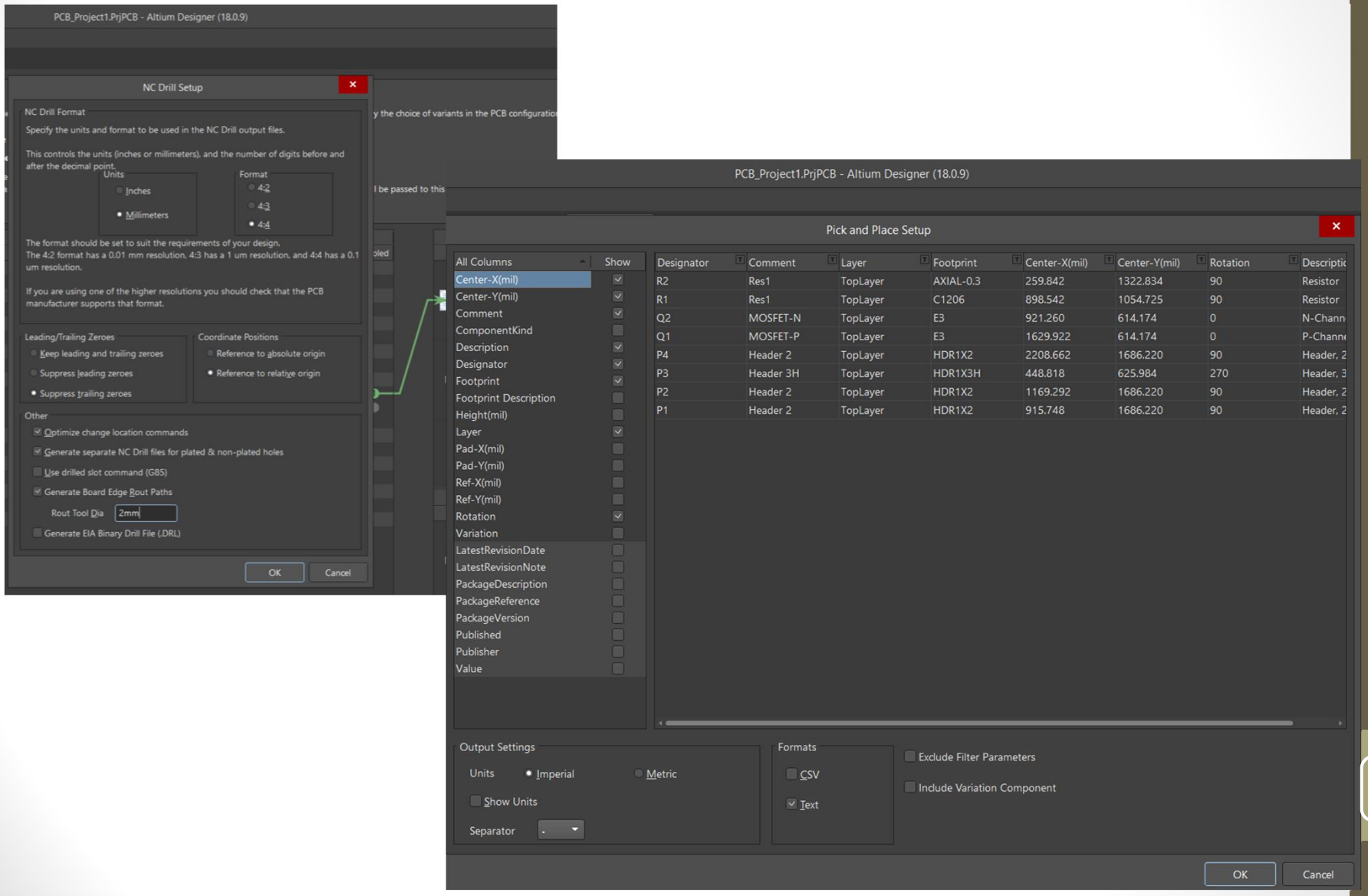


GERBER

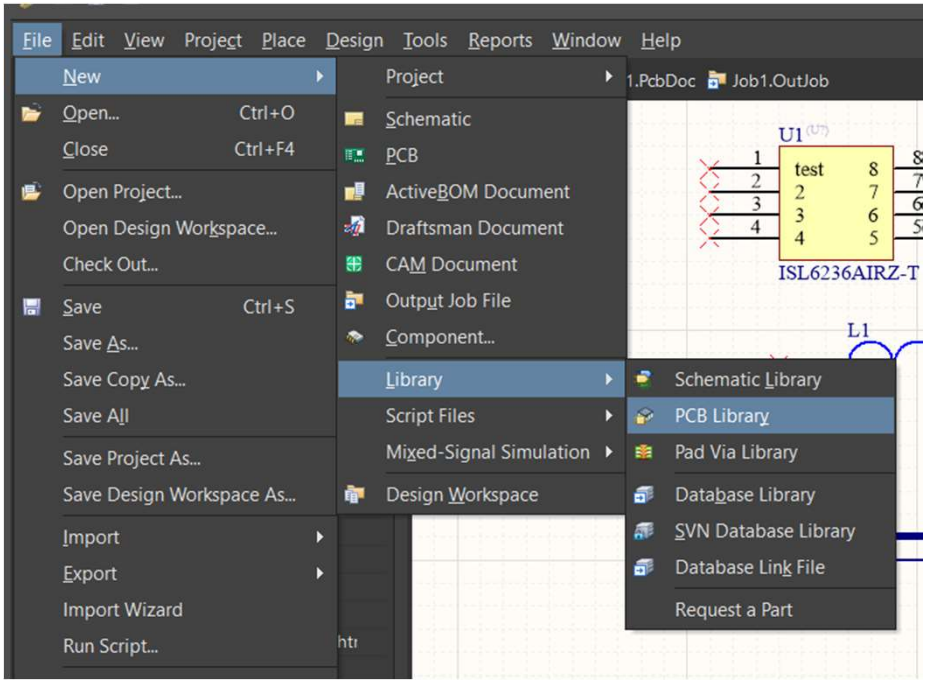

PCB_Project1.PrjPCB - Altium Designer (18.0.9)
Gerber Setup
General
Layers
Drill Drawing
Apertures
Advanced
Specify the units and format to be used in the output files.
This controls the units (inches or millimeters), and the number of digits before and after the decimal point.
Units
Inches
Millimeters
Format
4:2
4:3
4:4
The format should be set to suit the requirements of your Project.
The 4:2 format has a 0.01 mm resolution, 4:3 has a 1 um resolution, and 4:4 has a 0.1 um resolution.
If you are using one of the higher resolutions you should check that the PCB manufacturer supports that format.


PCB_Project1.PrjPCB - Altium Designer (18.0.9)
Gerber Setup
General
Layers
Drill Drawing
Apertures
Advanced
Layers To Plot
Ex...
Layer Name
Plot
Mirror
Top Overlay
Top Paste
Top Solder
Top Layer
Signal Layer 1
Internal Plane 1
Internal Plane 2
Bottom Layer
Bottom Solder
Bottom Paste
Bottom Overlay
Mechanical 1
Mechanical 13
Mechanical 15
Keep-Out Layer
Top Pad Master
Bottom Pad Master
Component Layers
Signal Layers
Electrical Layers
All Layers
Mechanical Layers(s) to Add to All Plots
Layer Name
Plot
Mechanical 1
Mechanical 13
Mechanical 15
Plot Layers
Mirror Layers
Include unconnected mid-layer pads
OK
Cancel

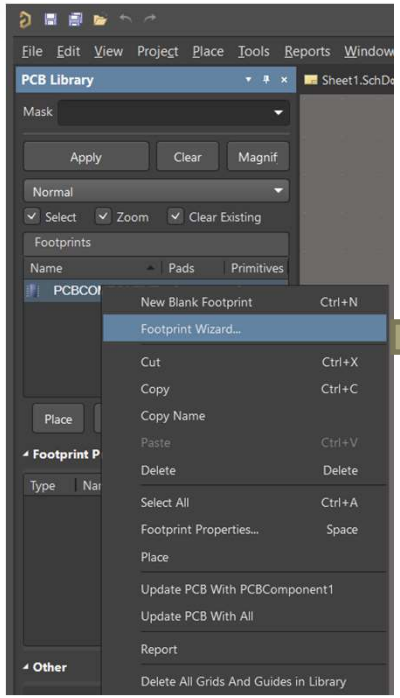
Nc drill e pick and place

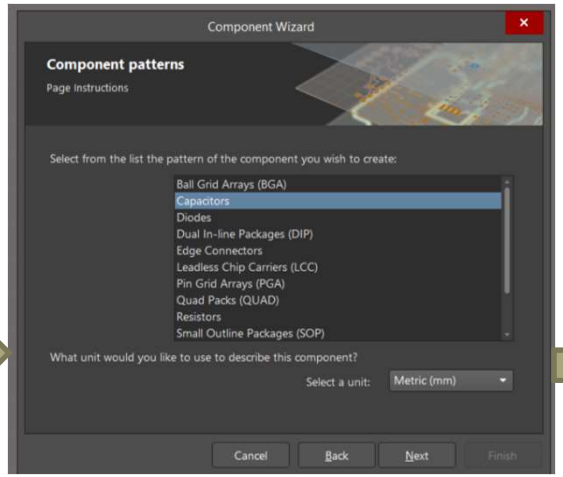


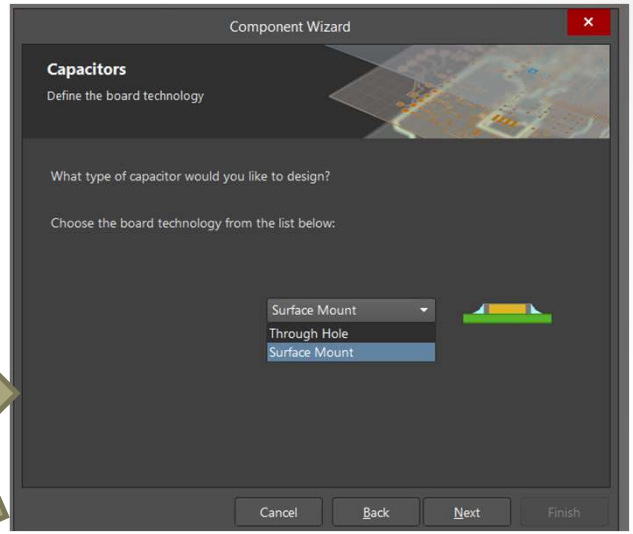
New FootPrint

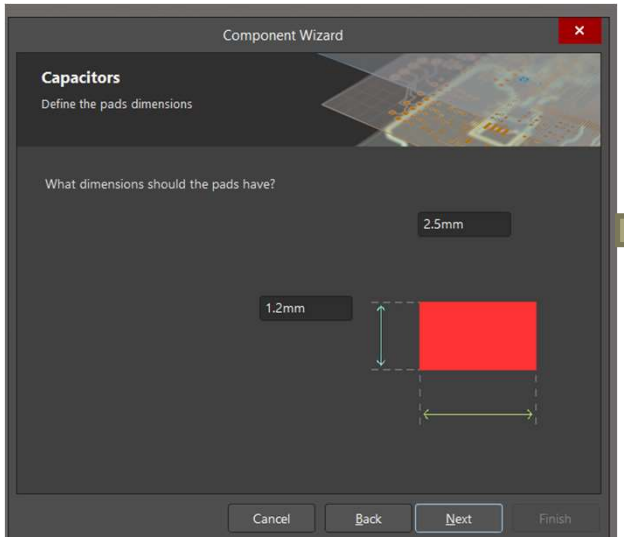


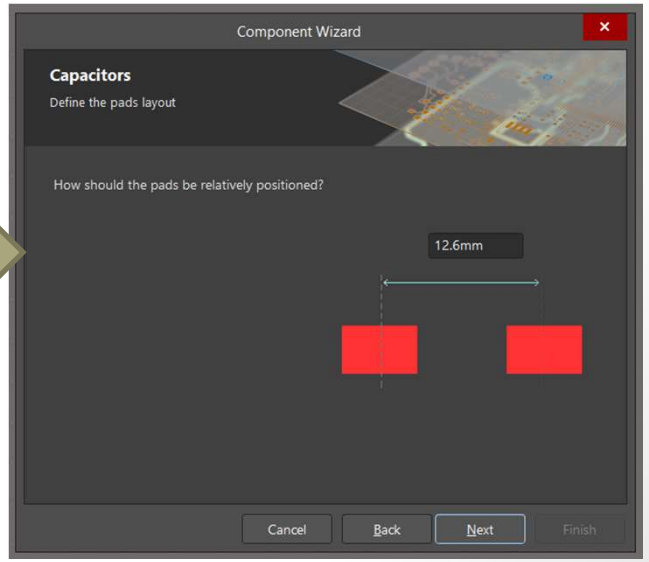
Footprint Wizard





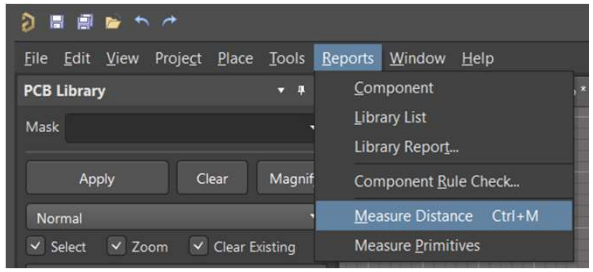


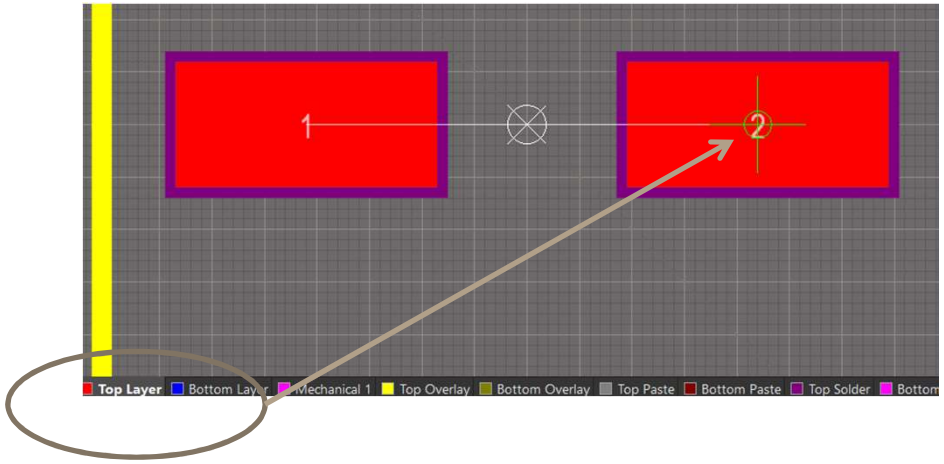




New Footprint

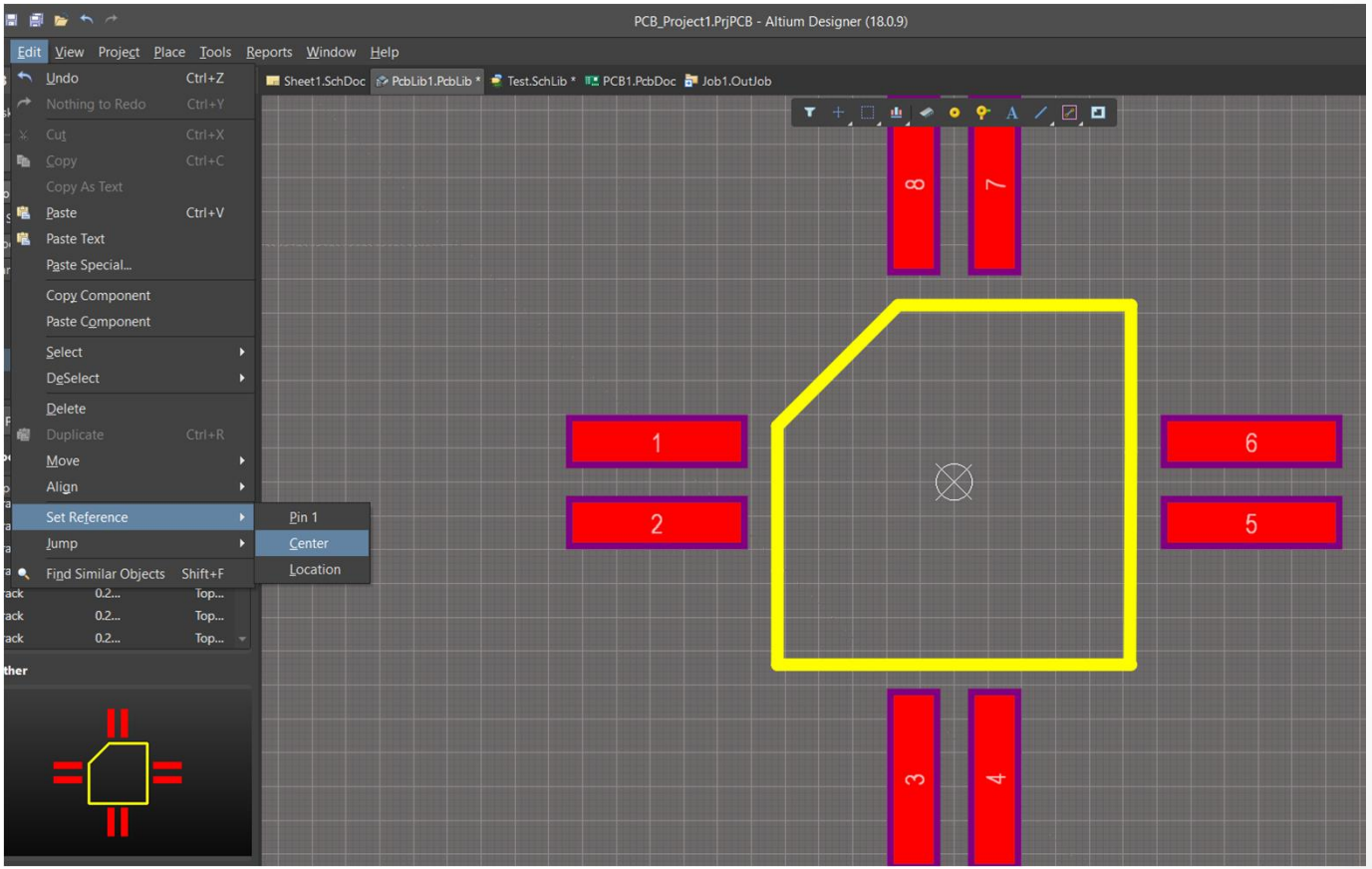
Attenzione alla griglia:anche se ho scelto Metric ho lo schermo in Inch.
Lo cambio con Ctrl +G





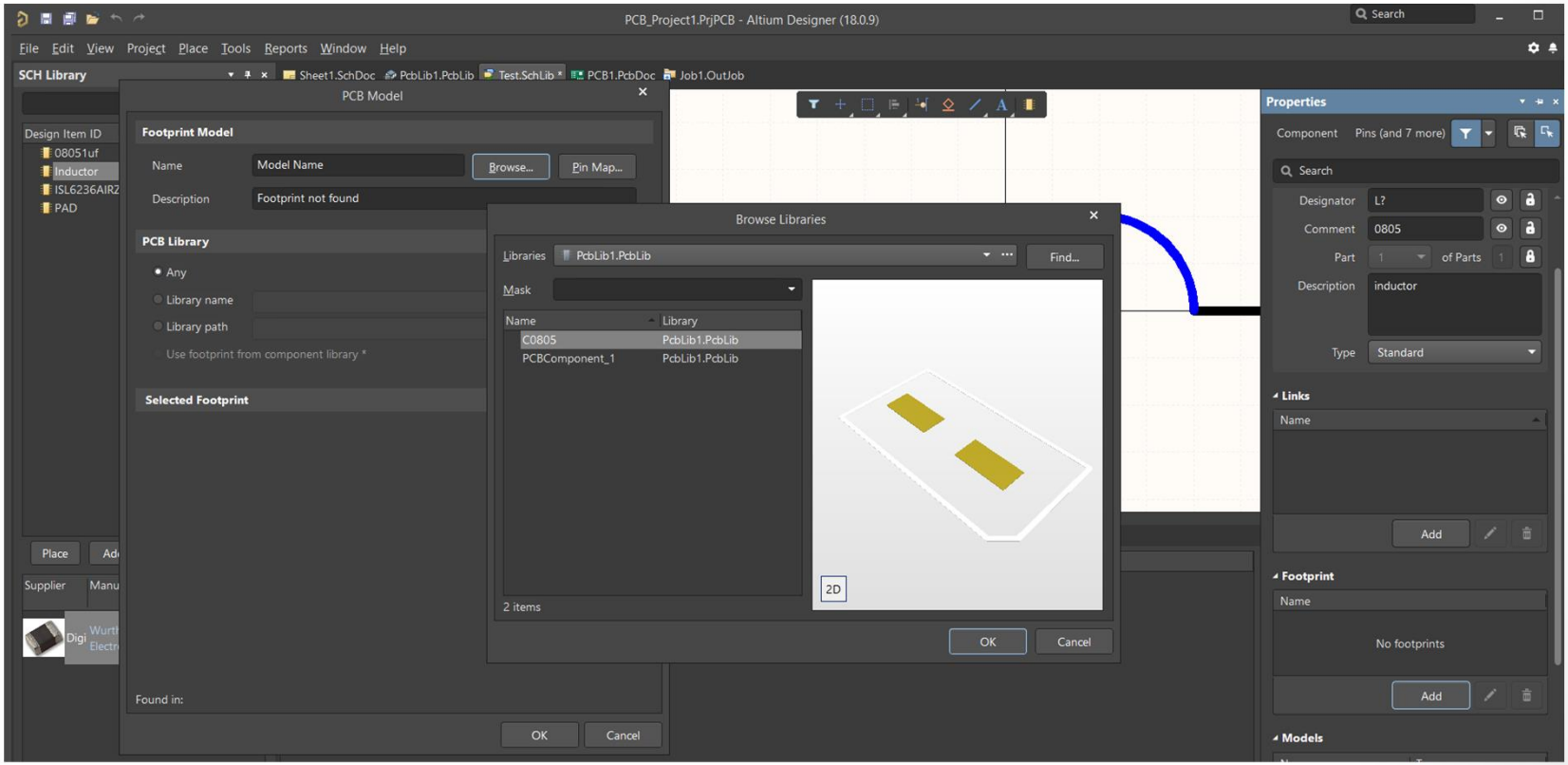
Misurare le distanze : ideale essere sul layer di interesse
Per pulire Shift+C

Reference Point

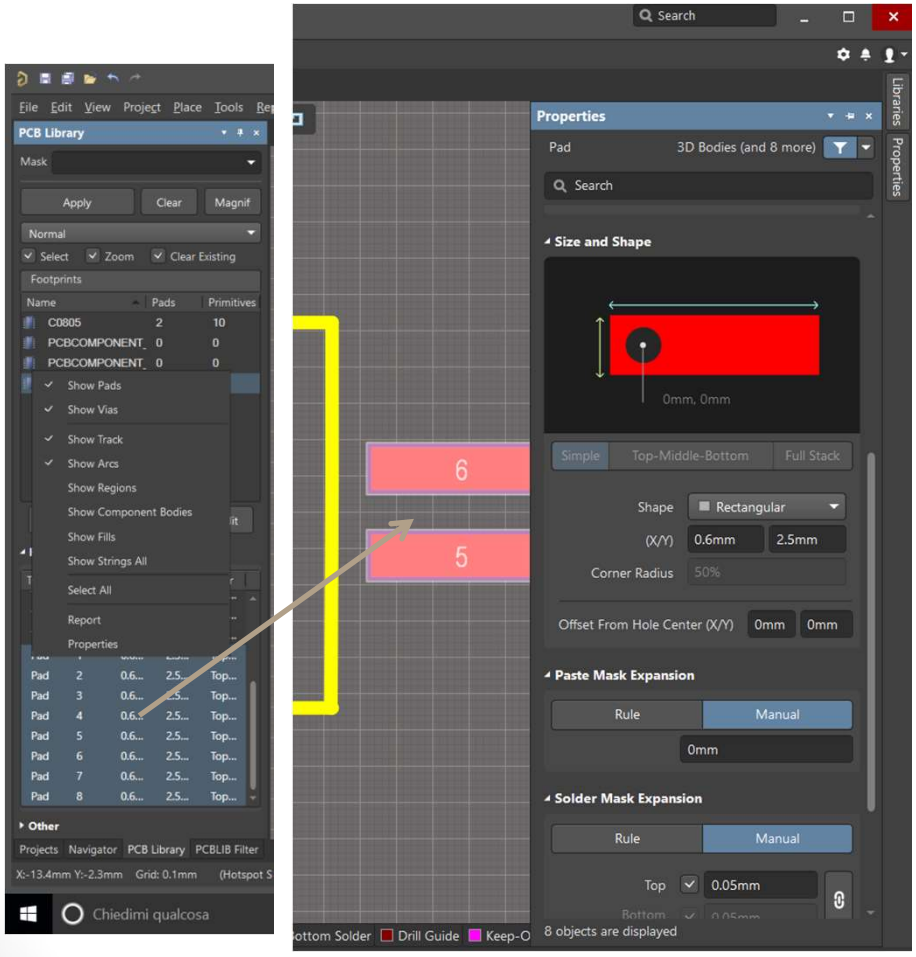


Associazione con Schematico

Da SCH Library ---properties del componente e poi add



Proprietà dei PAD



Nota:attenzione ai fori plated o no

Il nome del PAD deve corrispondere con quello nello chematico

Esempio/ Esercizio

Realizzare il PCB dello schematico di esempio, realizzando una nuova libreria dei componenti.