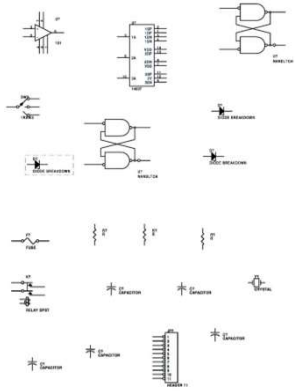
PROGETTAZIONE PCB

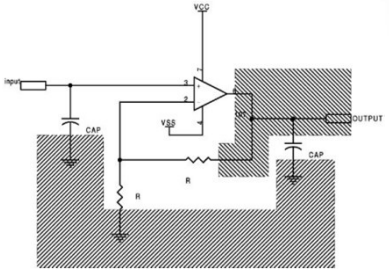
- Ciclo di Progettazione
  - Funzionalità del sistema
  - Schema Elettrico
  - Vincoli meccanici
  - Layout
  - Produzione
- Altium Designer
  - Schematic Project
  - Layout Project
  - Production Files

Ciclo di Progettazione

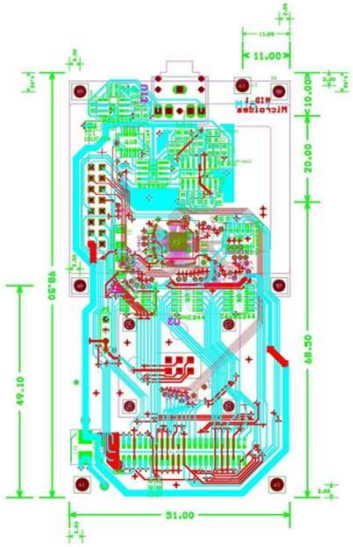


Simboli

Lista collegamenti

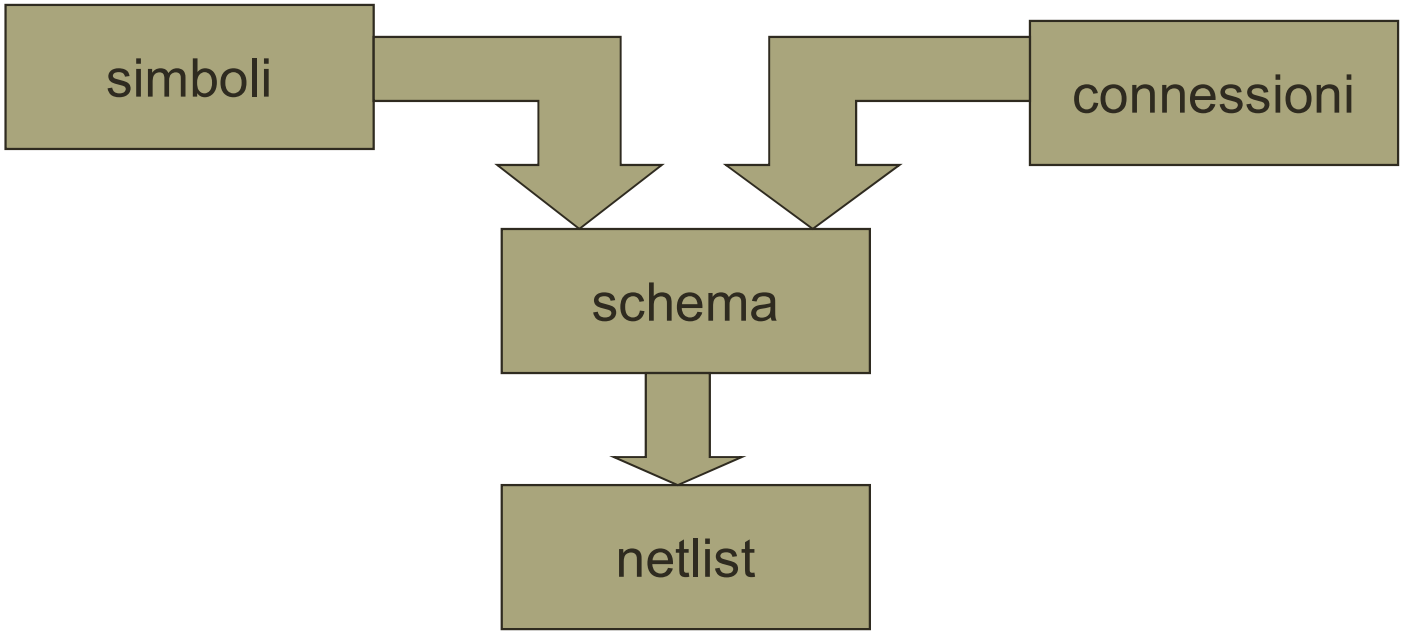


Schematico

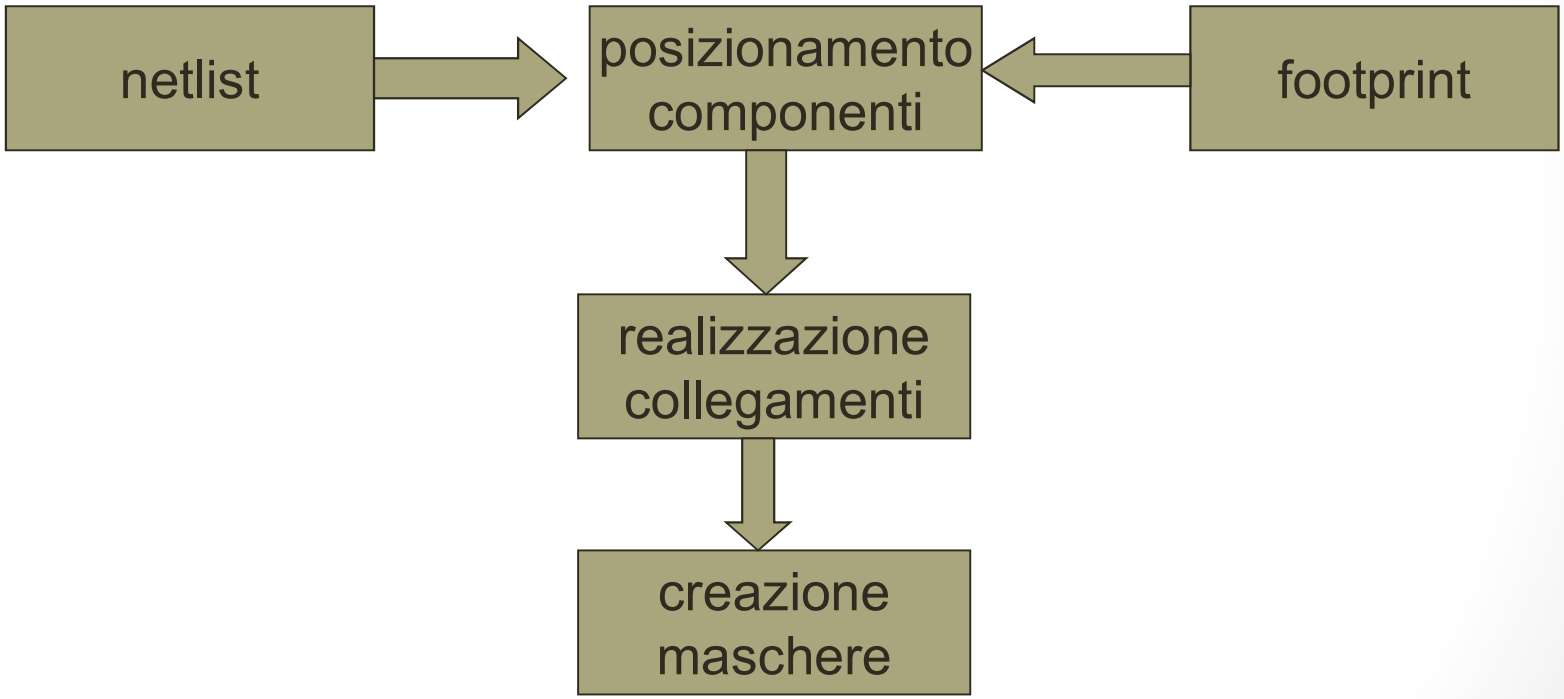


PCB

Ciclo di Progettazione - Schematico

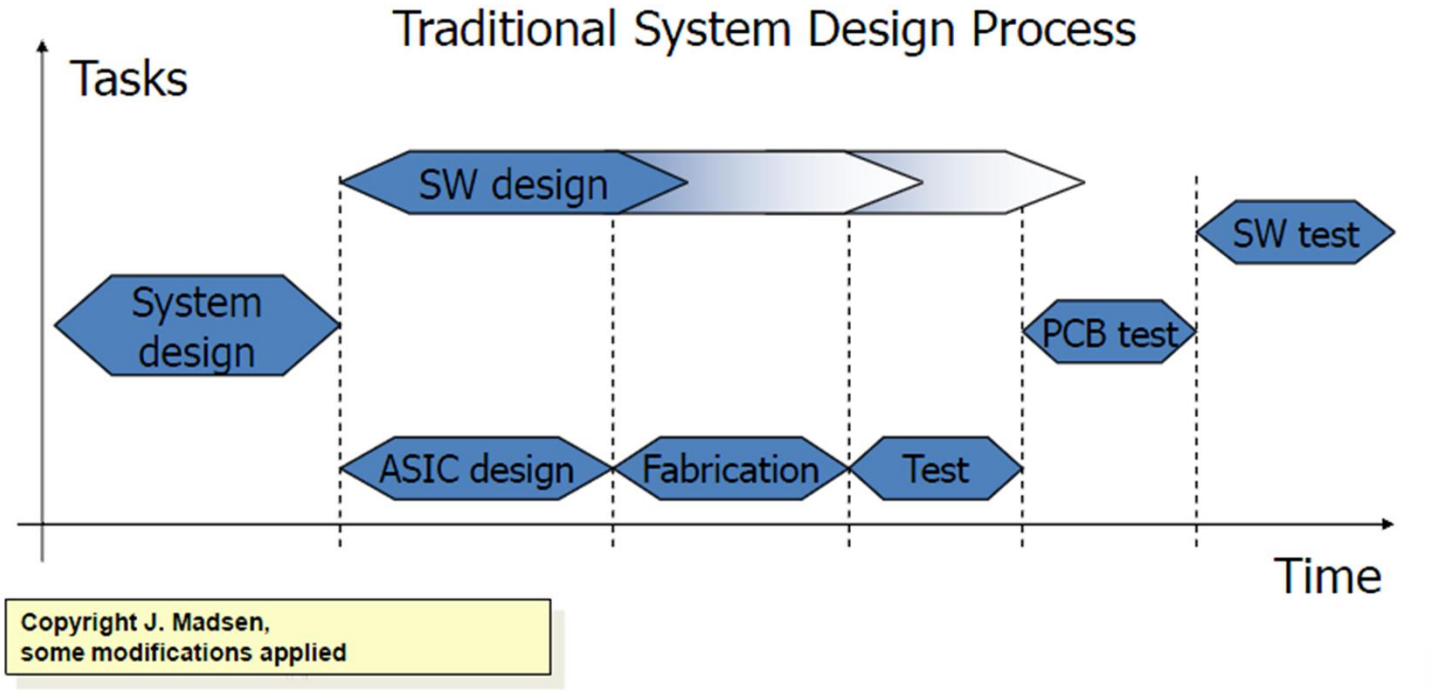


Ciclo di Progettazione -LAYOUT

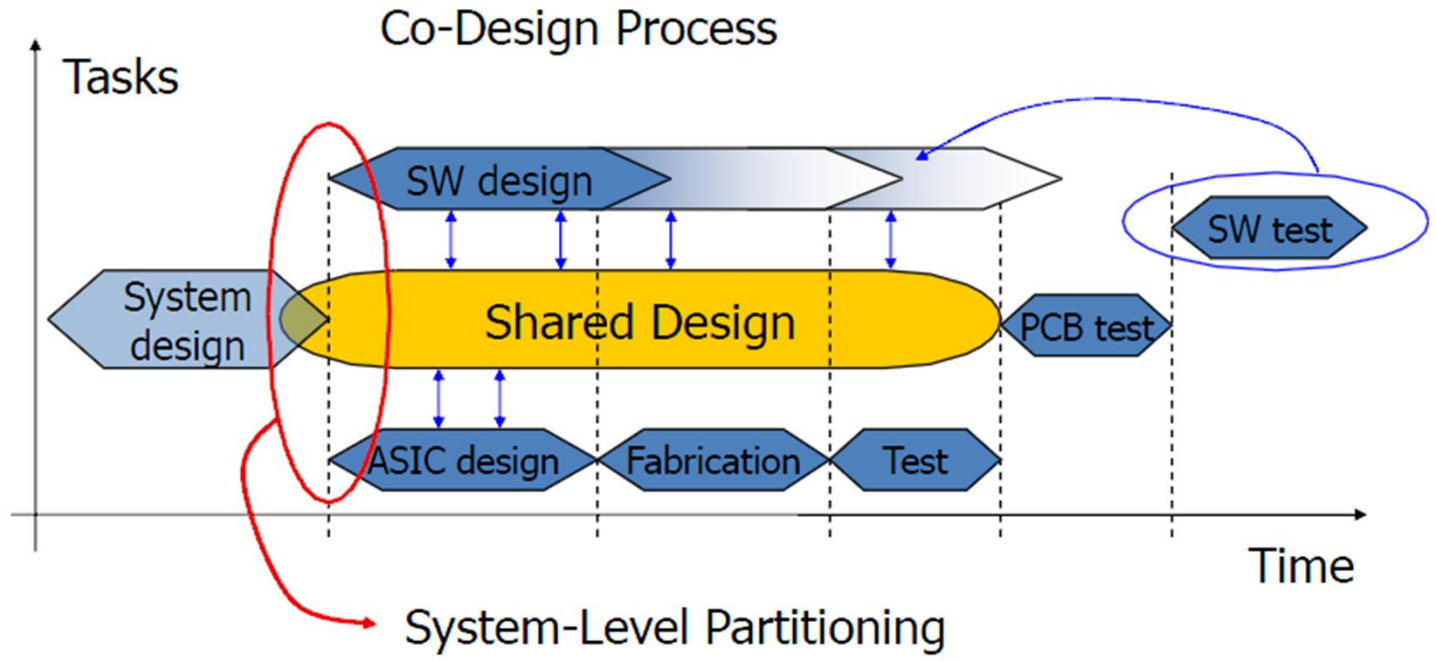


Ciclo di Progettazione

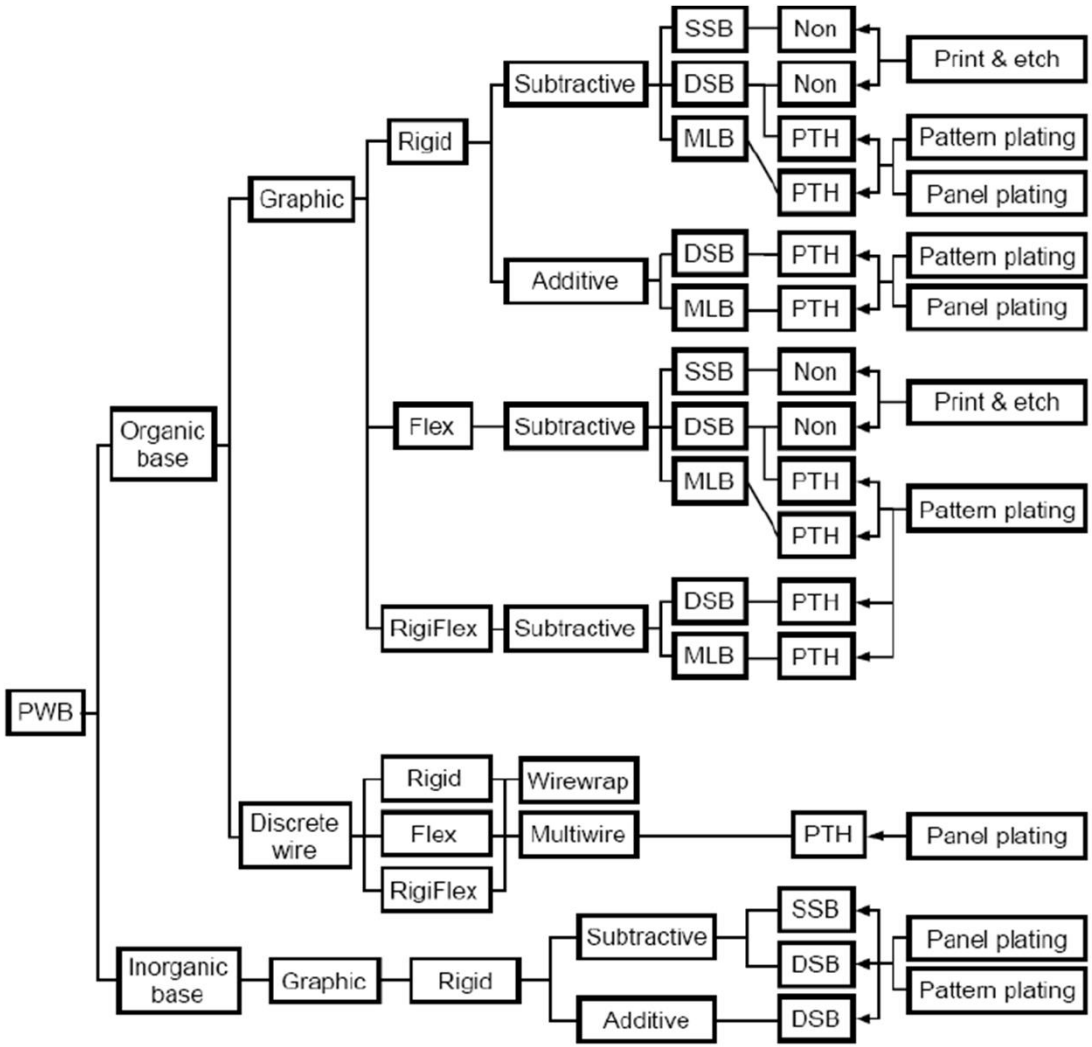
Ciclo di Progettazione



Ciclo di Progettazione

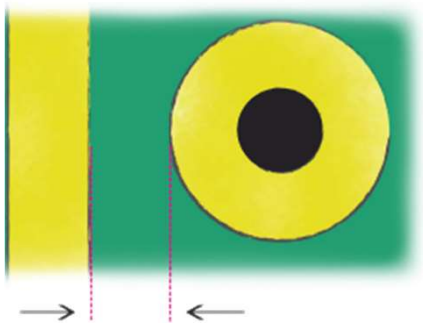


Ciclo di Progettazione Produzione PCB

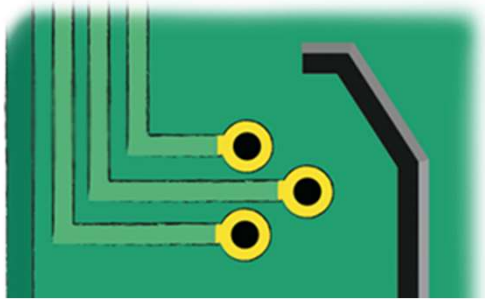


Ciclo di Progettazione Produzione PCB

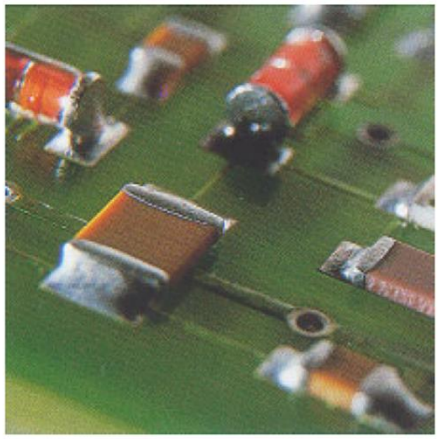
Vincoli Elettrici



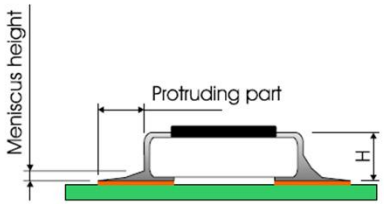
Vincoli Meccanici



Ciclo di Progettazione Assemblaggio

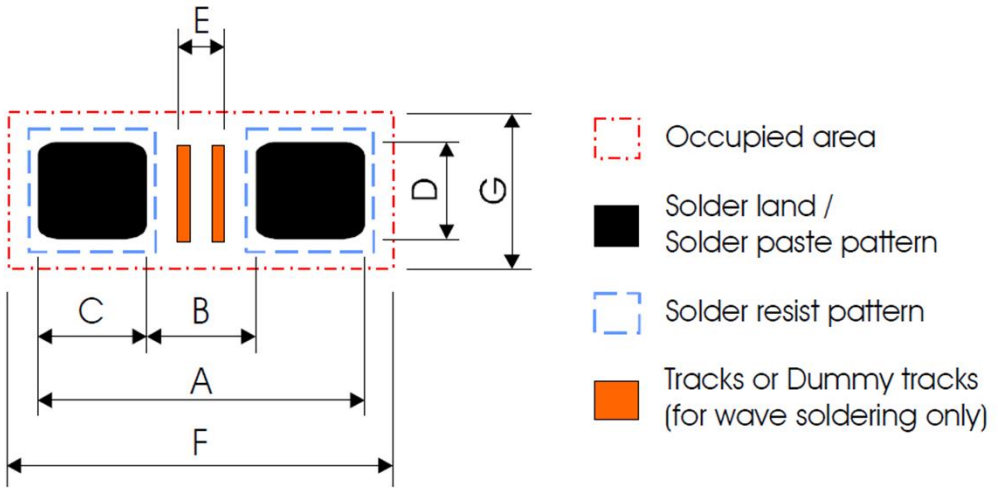


Acceptable

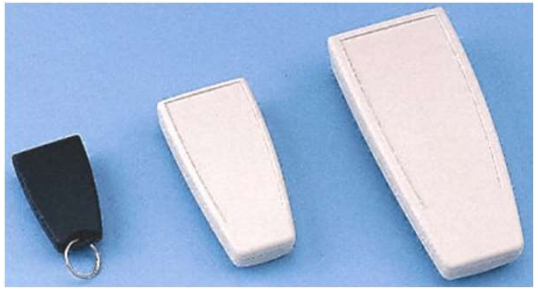


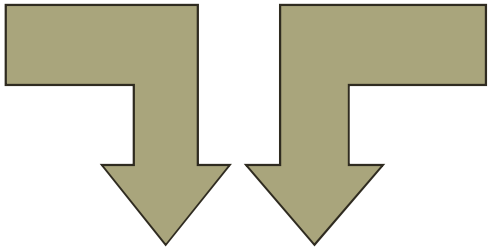
For Components with a termination height H < 1.2mm : The height of the meniscus must be at least 1/3 of the height H

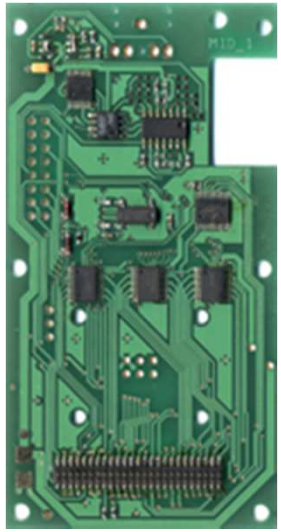
For components with a termination height H ≥ 1.2mm : The height of the meniscus must be at least 0.4mm.



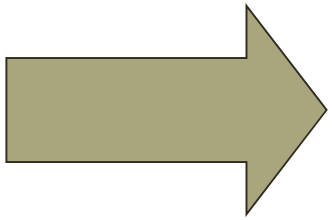
Ciclo di Progettazione Meccanici - Designer

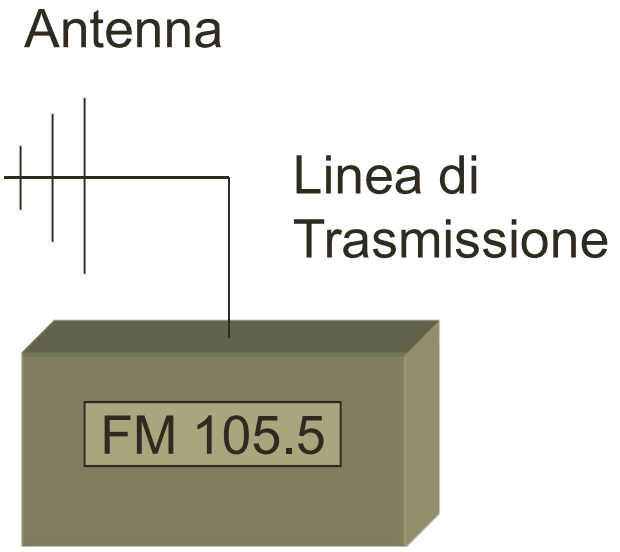






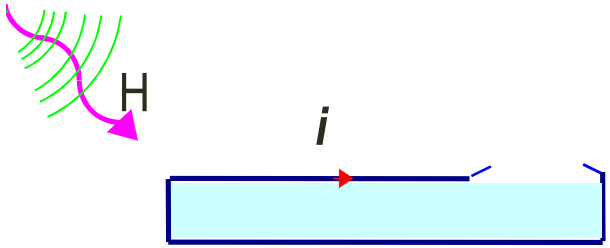
Ciclo di Progettazione EMC





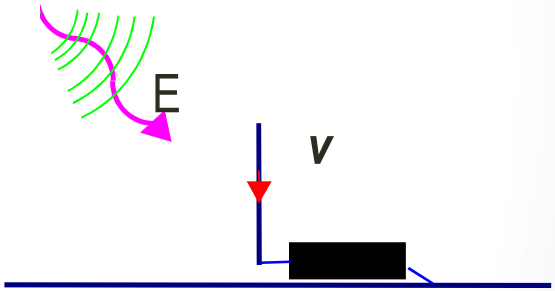
Ciclo di Progettazione EMC

Accoppiamento Magnetico



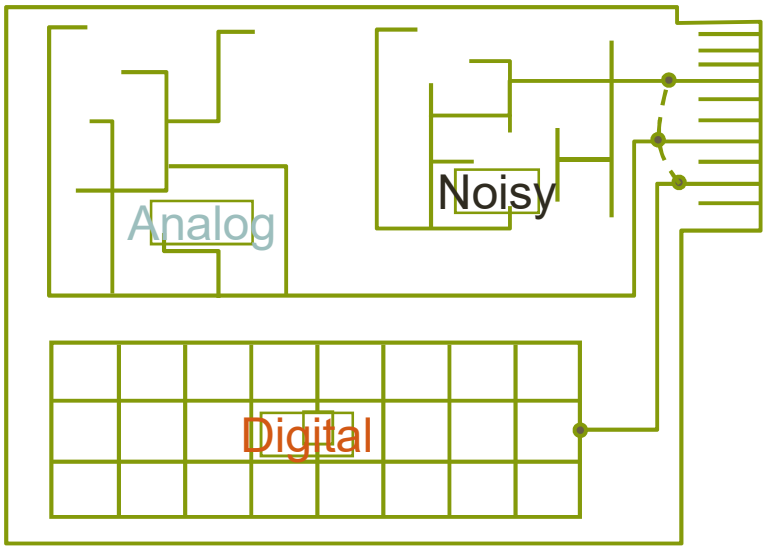
Generazione di corrente I per campo magnetico H (A/m)

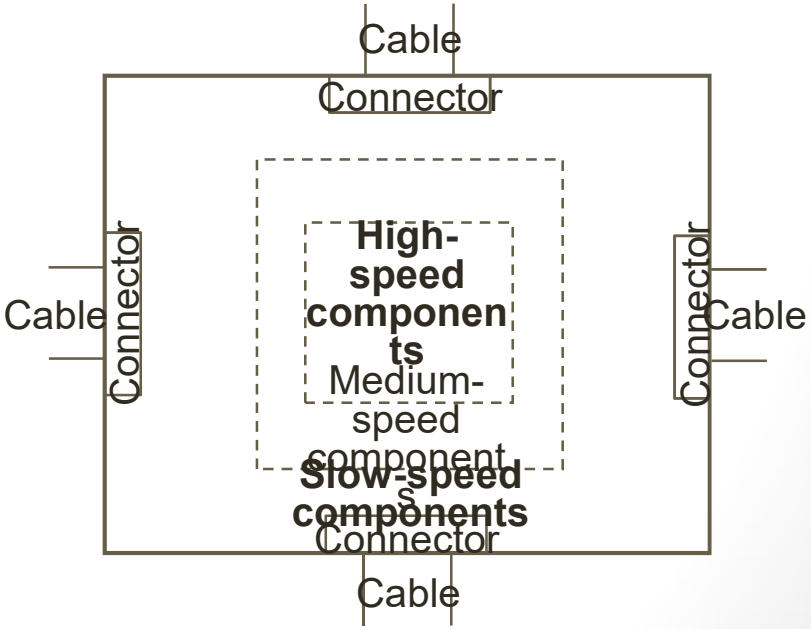
Accoppiamento Elettrico



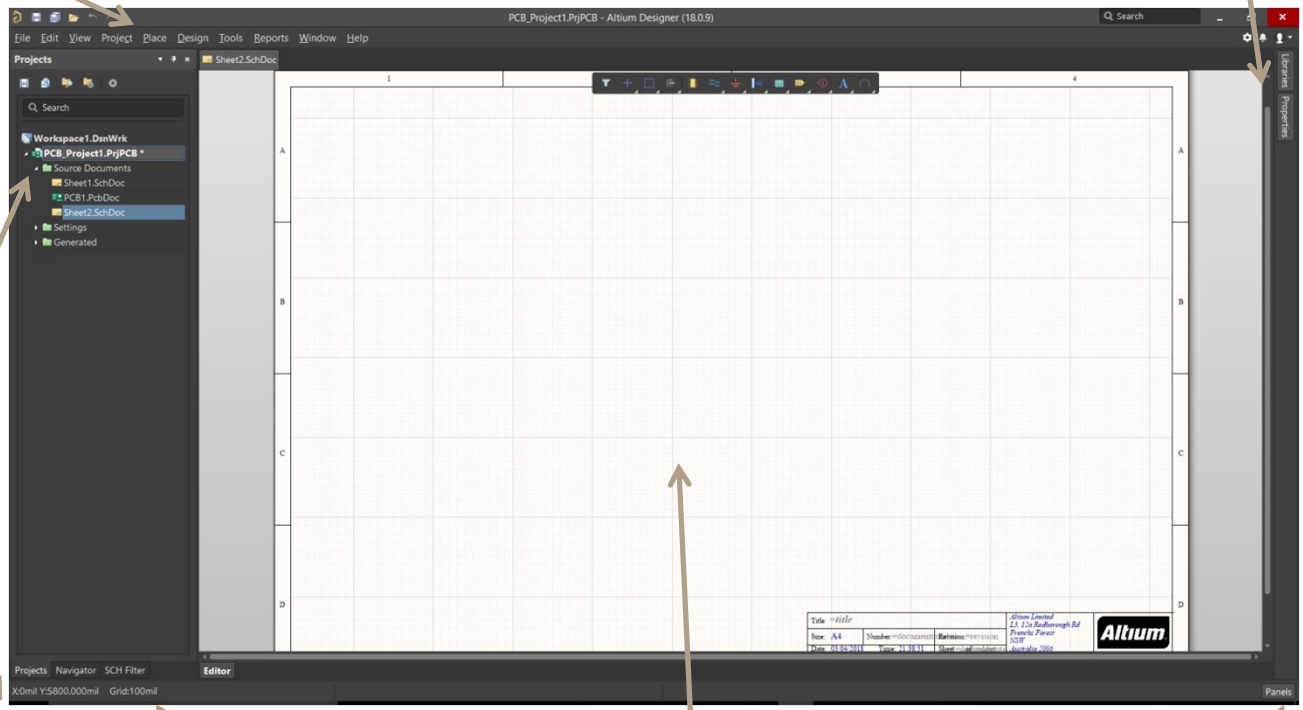
Tensione V generata per campo elettrico E (V/m)

Ciclo di Progettazione EMC

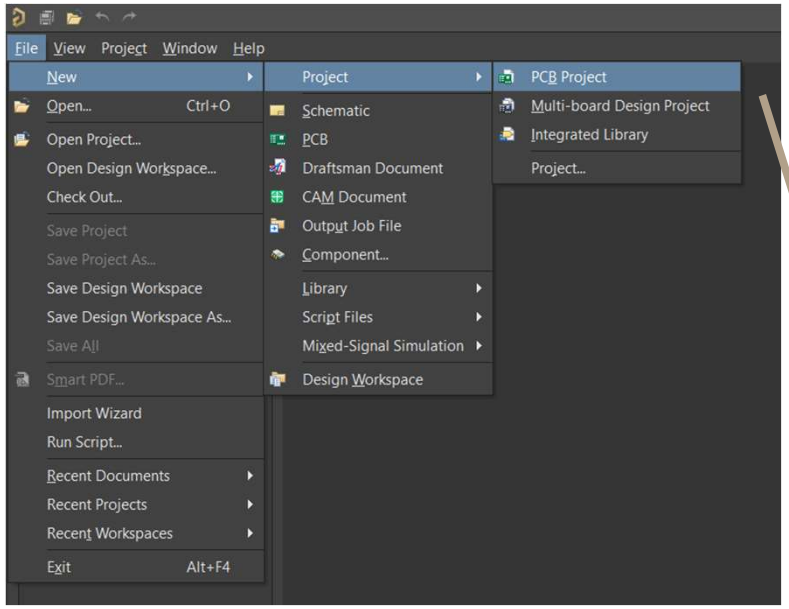


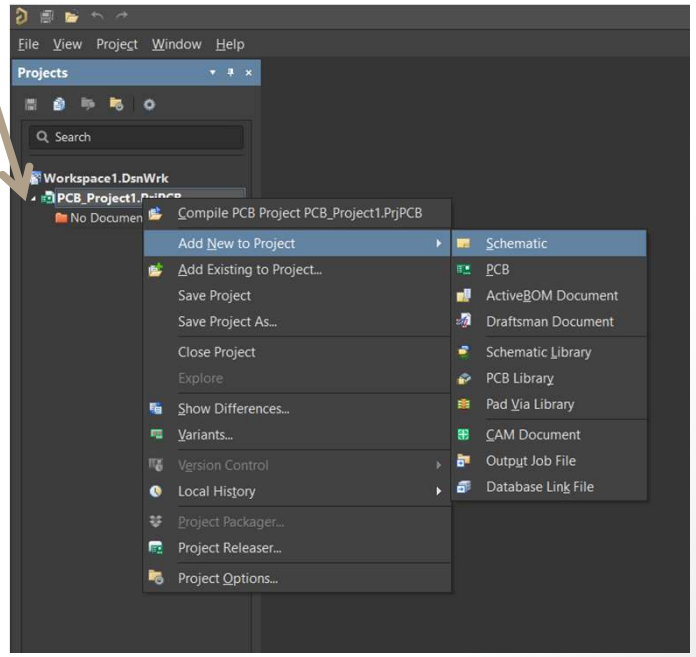


Altium Designer - Schematics



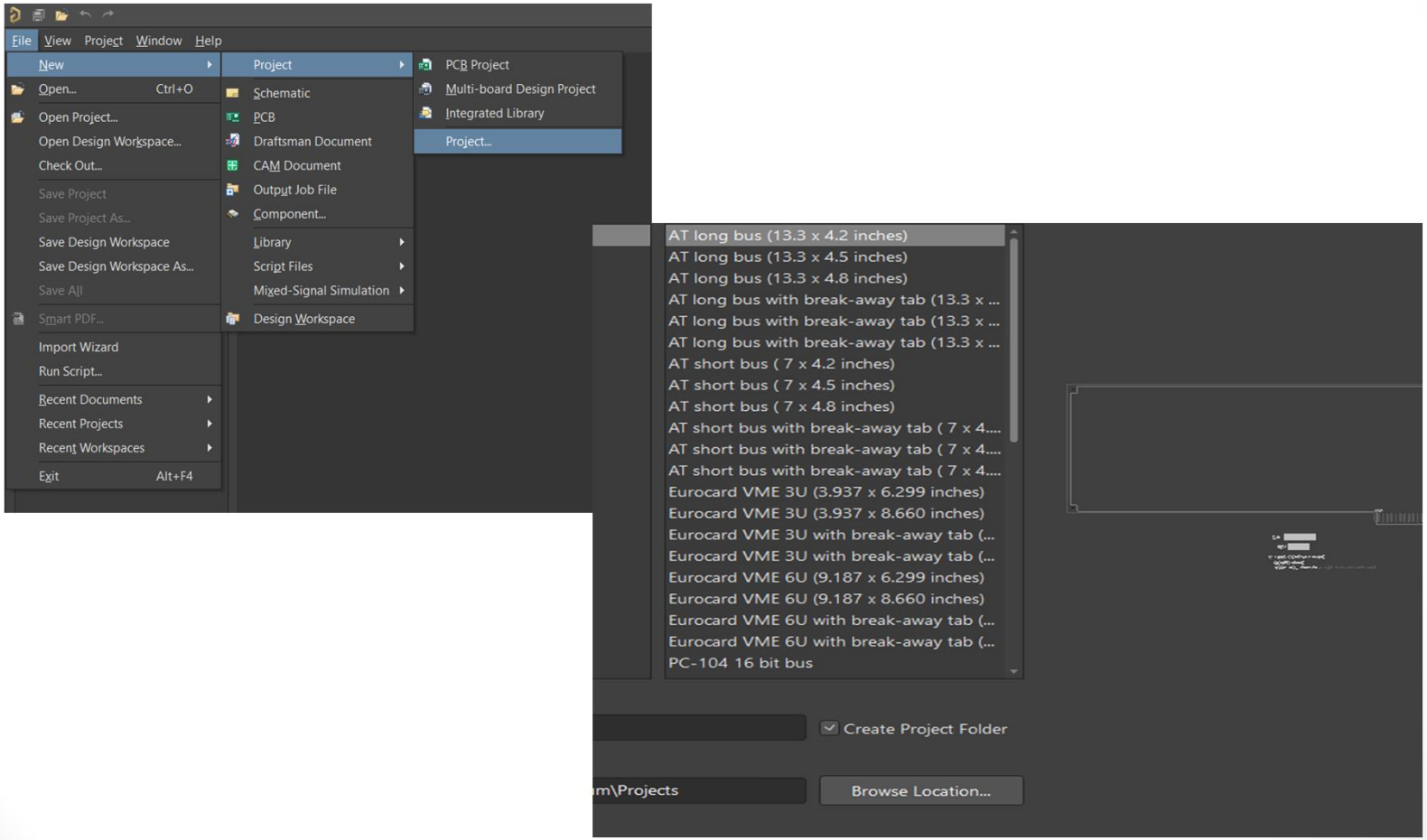
Schematics - New Project



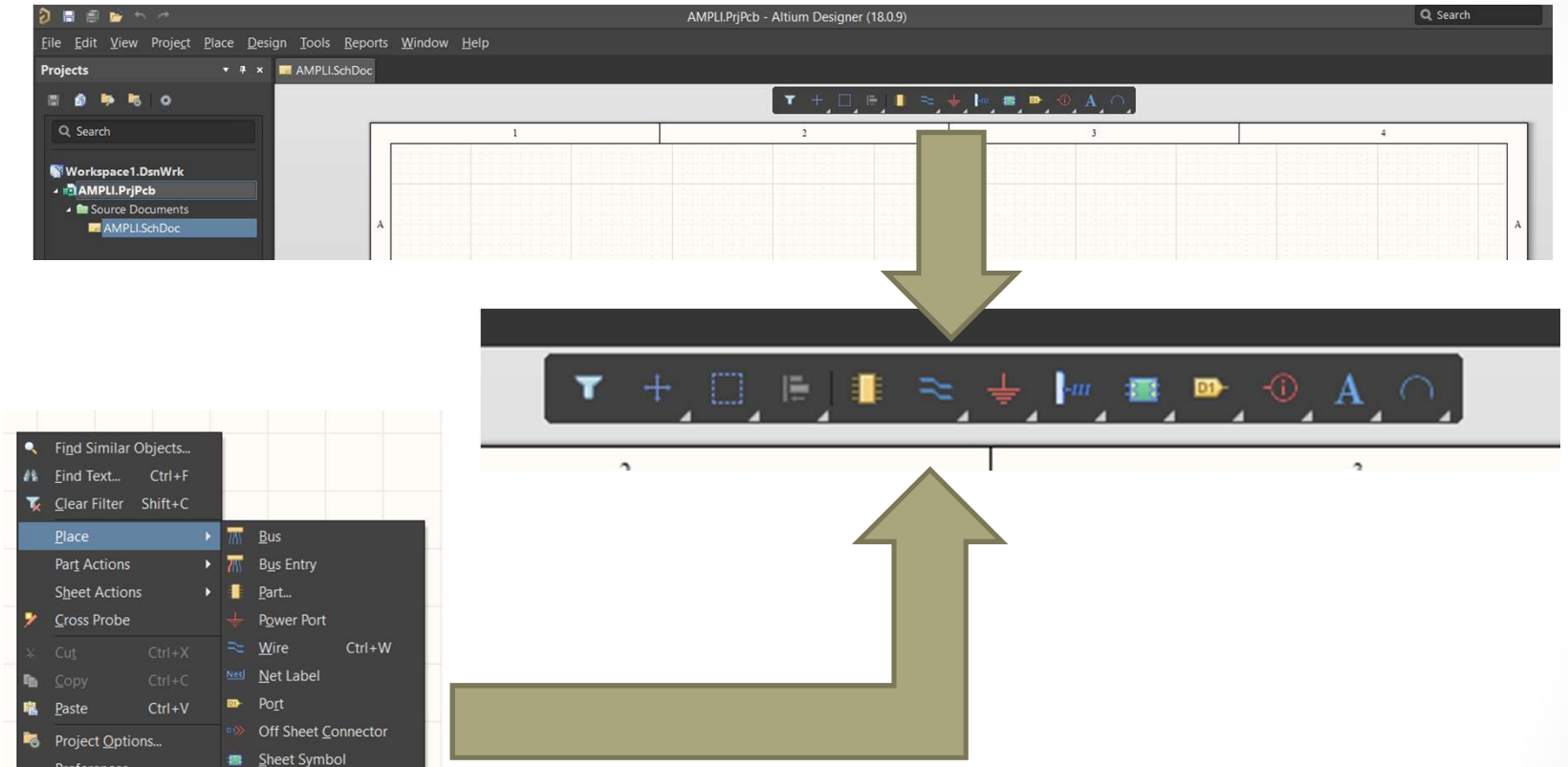


Nota:
Un nuovo documento è utilizzabile solo dopo essere stato salvato.

Schematics
New Project Source

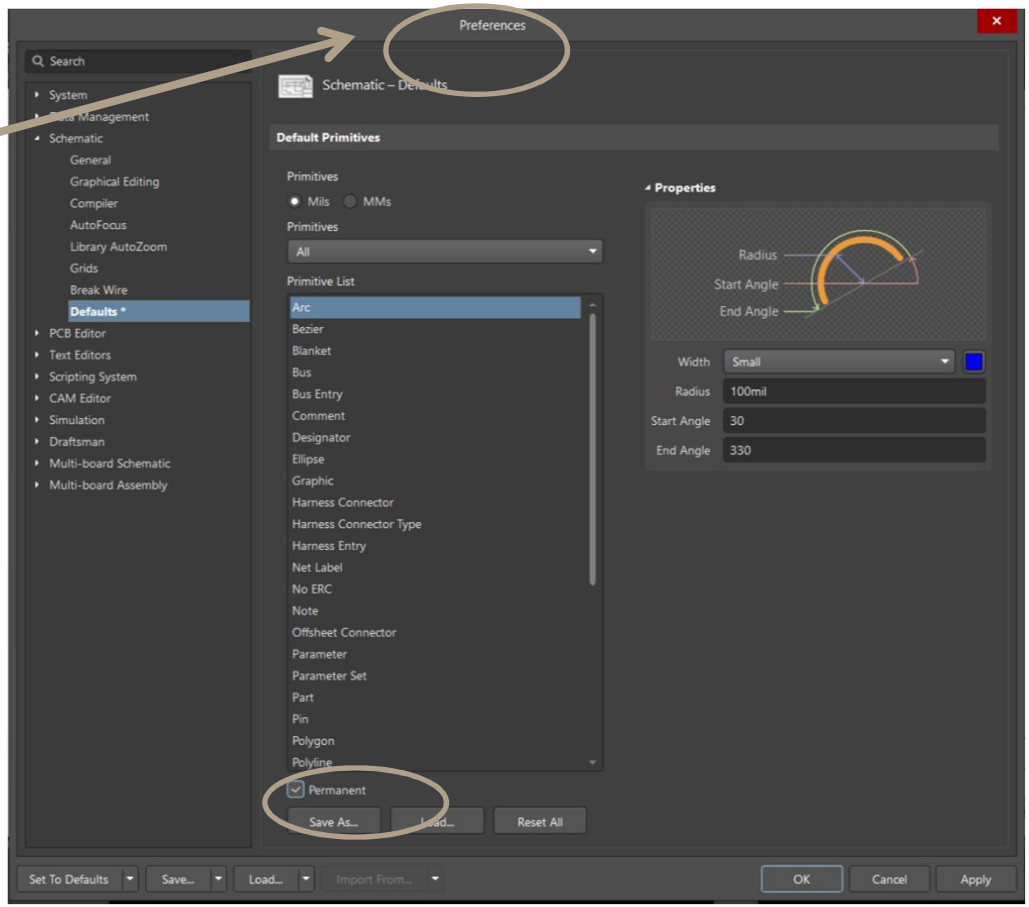


Schematics Place



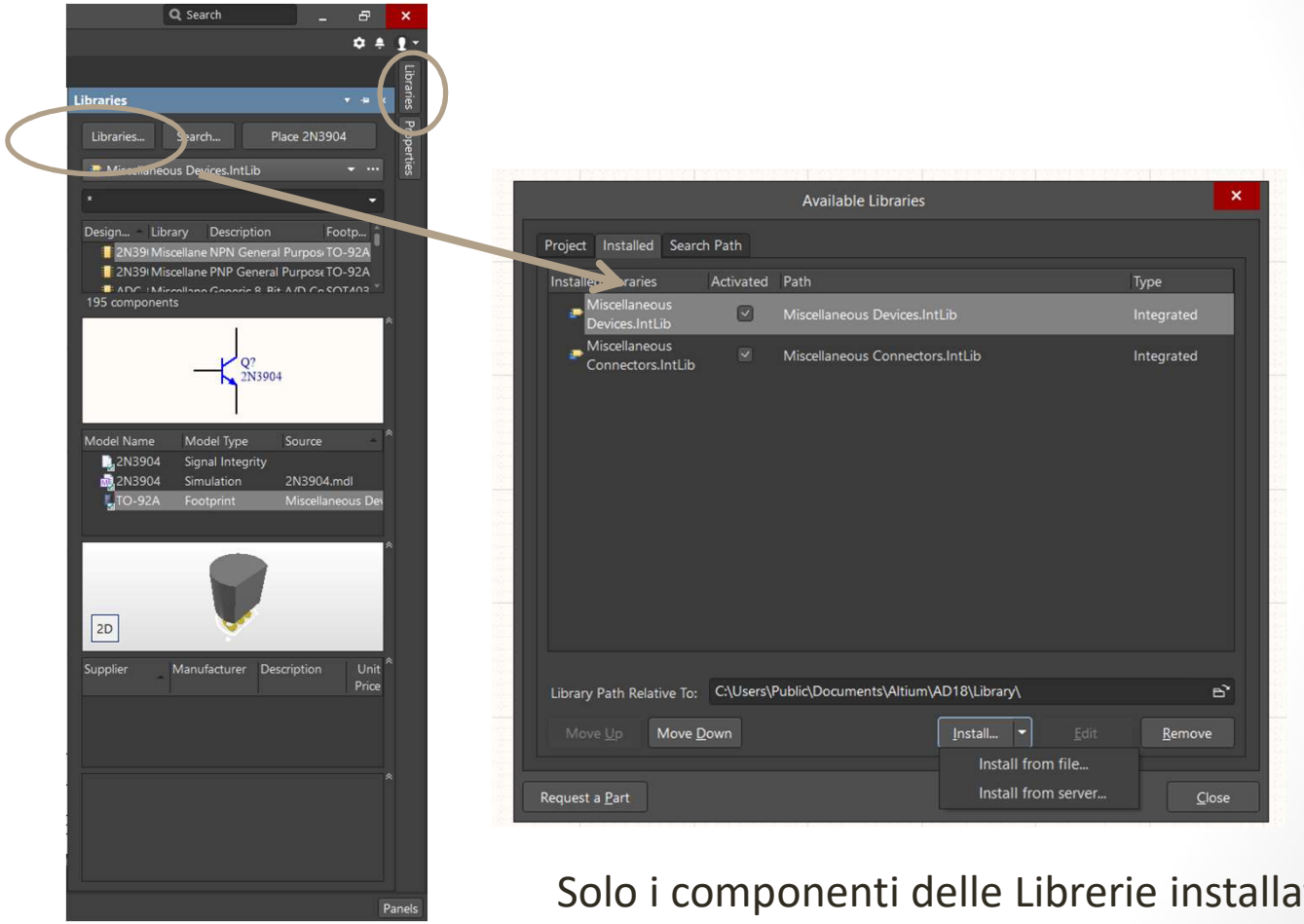
Schematics Preferences

From TOOLS



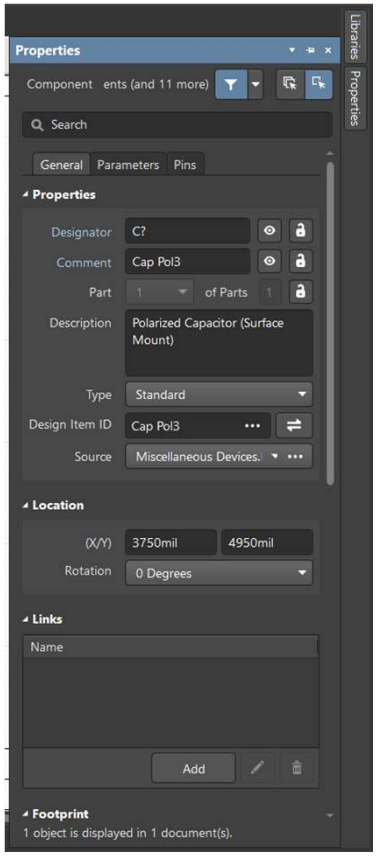
While attributes can be modified during placement (**Tab** to bring up associated properties dialog), bear in mind that these will become the default settings for further placement unless the **Permanent** option on the **Schematic – Default Primitives**page of the *Preferences* dialog is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

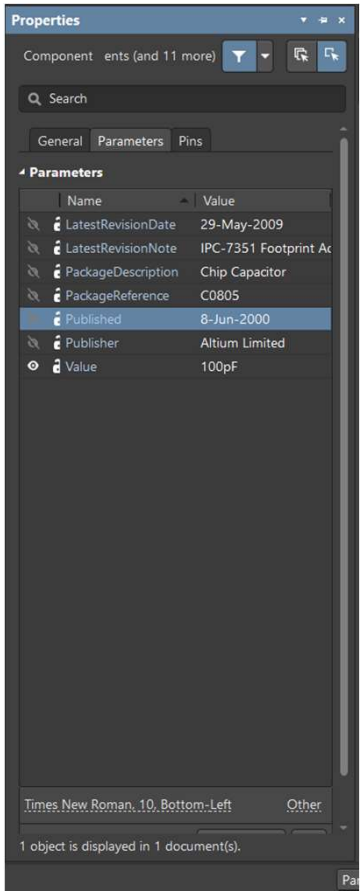
Schematics Libraries

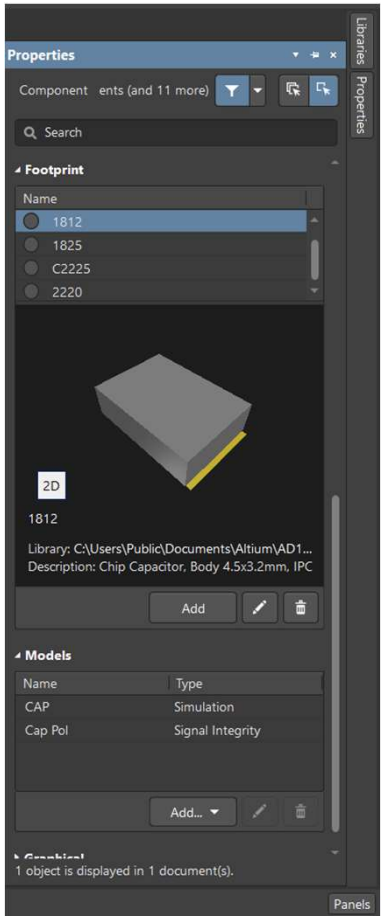


Solo i componenti delle Librerie installate possono essere utilizzati

Schematics Libraries
Components Properties







NOTA: Per la simulazione è necessario inserire tutti i parametri richiesti.
Es: per Resistenza serve il «Value»

Schematics Navigation

- Spacebar → ruota in senso orario il componente
- Shift + Spacebar → ruota in senso antiorario il componente

- Mentre è Floating Y → mirror verticale
  X → mirror orizzontale

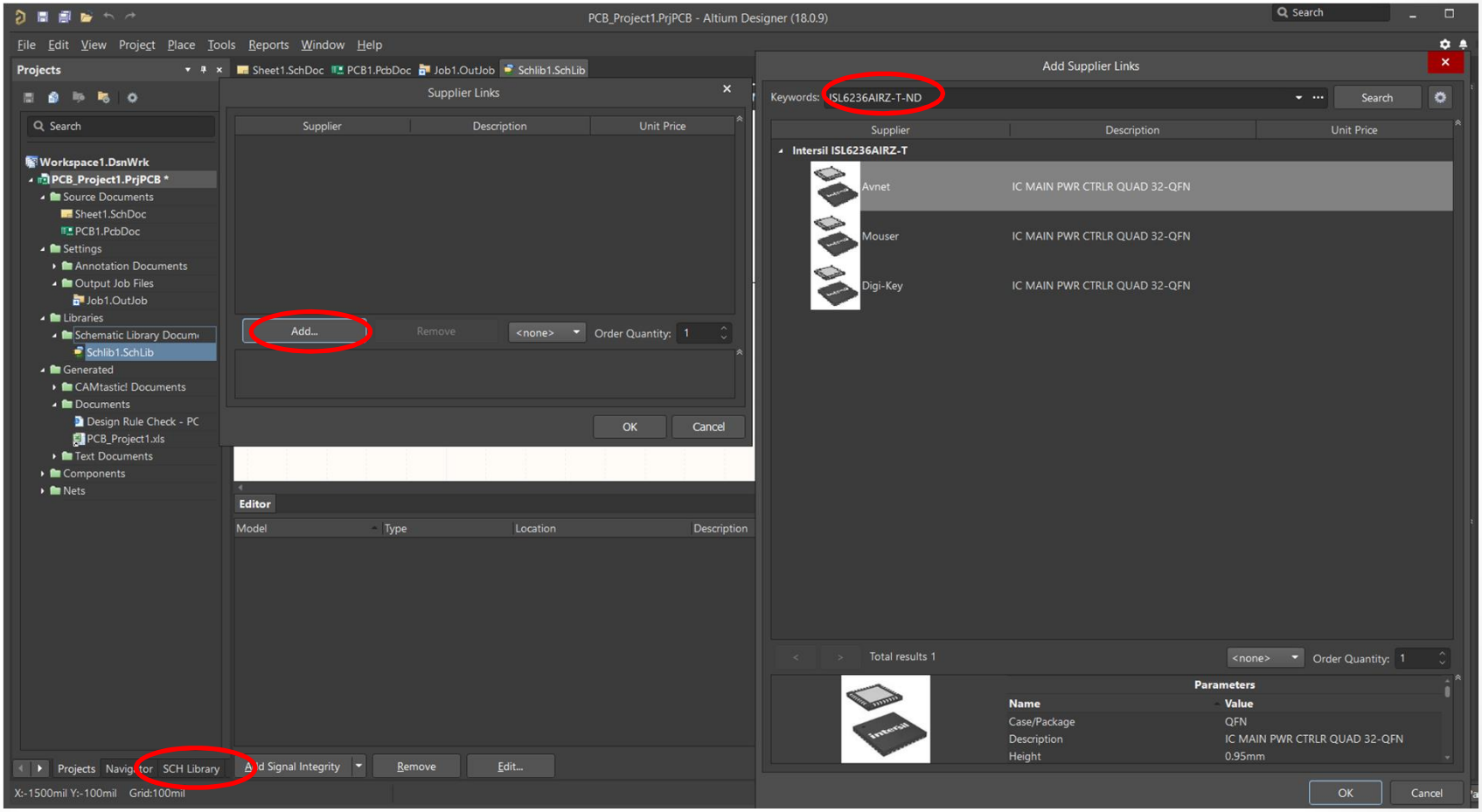
- PgUp e PgDown oppure CTRL e ruota mouse→ Zoom

- Mentre è Floating TAB → Apre le proprietà del componente

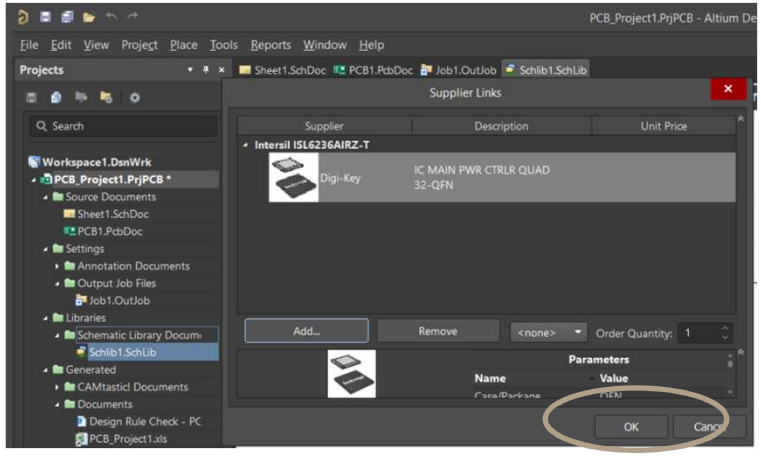
- Andare sui lati provoca il PAN se sto facendo qualche azione particolare

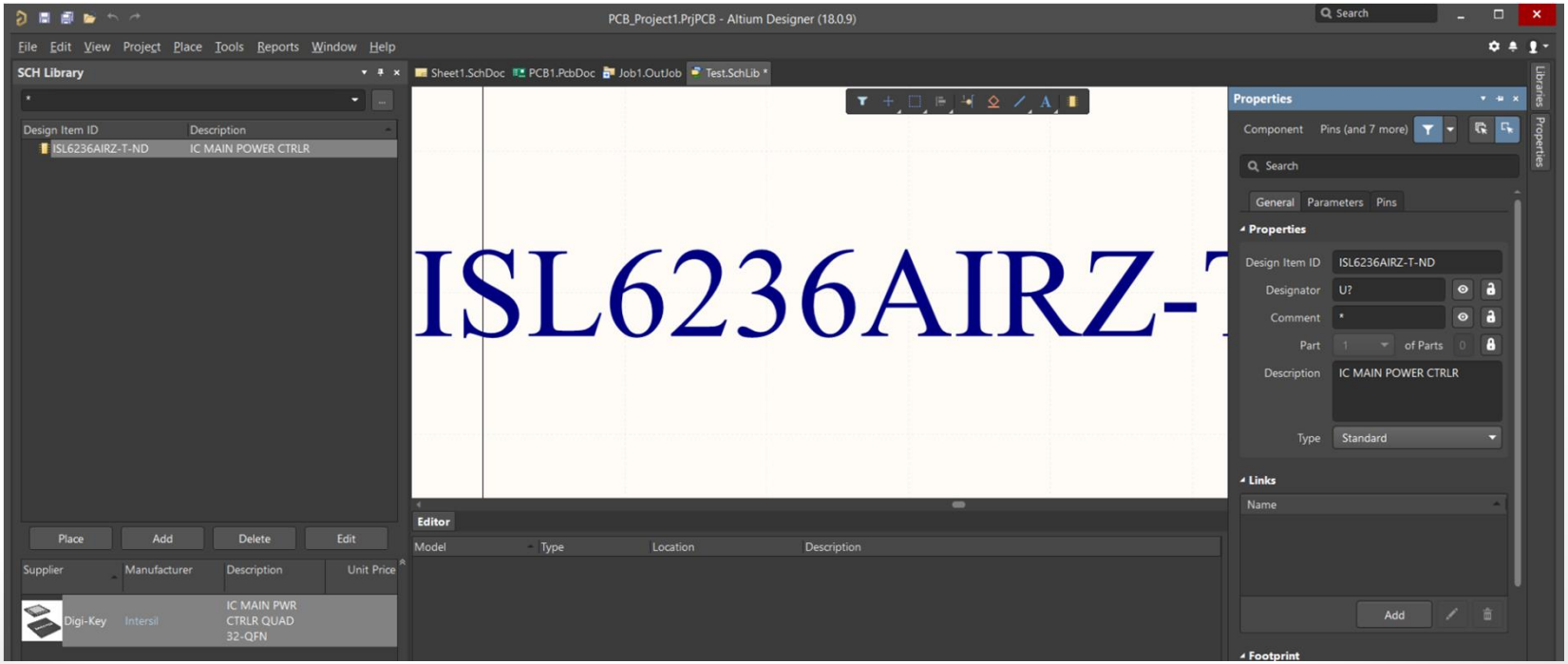
- V F → Fit della View

New Components

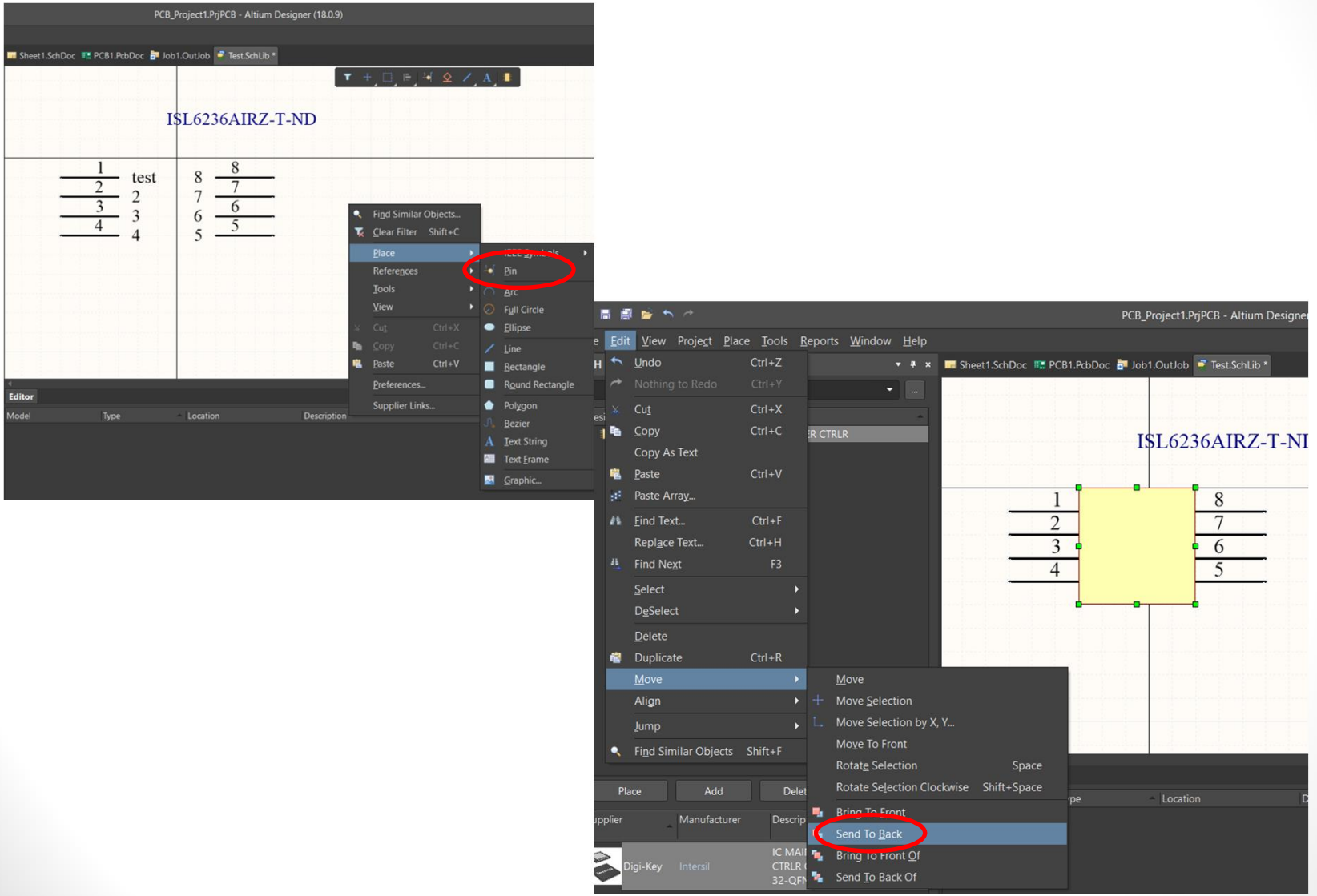


New Components

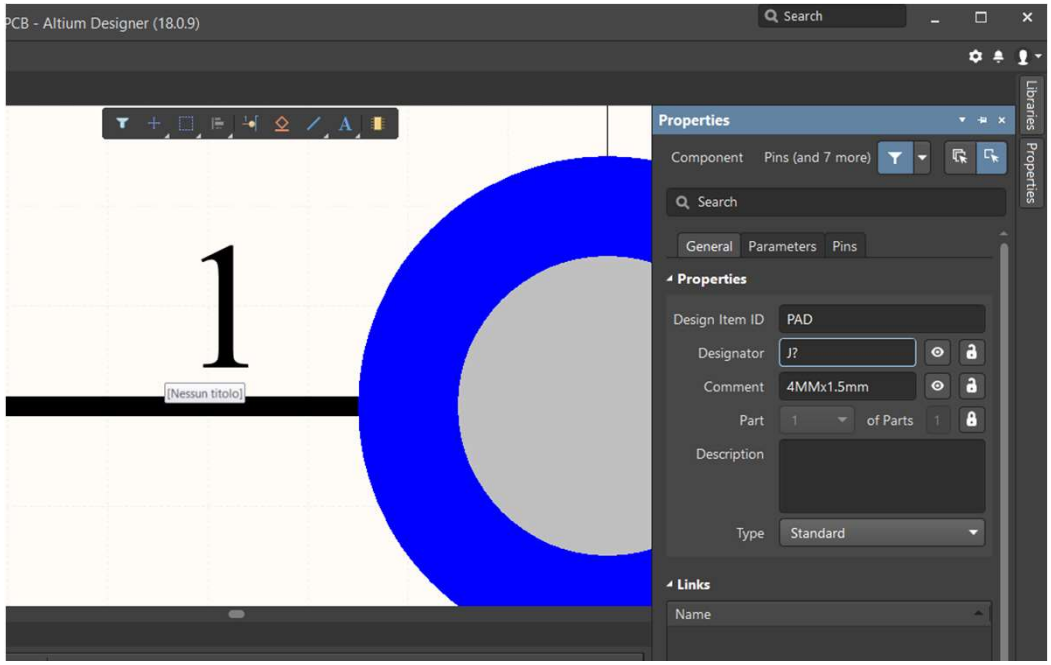




PIN and Body


PCB_Project1.PrjPCB - Altium Designer (18.0.9)
Sheet1.SchDoc
PCB1.PcbDoc
Job1.OutJob
Test.SchLib *
ISL6236AIRZ-T-ND
test
Find Similar Objects...
Clear Filter Shift+C
Place
References
Tools
View
Cut Ctrl+X
Copy Ctrl+C
Paste Ctrl+V
Preferences...
Supplier Links...
Pin
Arc
Full Circle
Ellipse
Line
Rectangle
Round Rectangle
Polygon
Bezier
Text String
Text Frame
Graphic...
Editor
Model
Type
Location
Description
Edit
View
Project
Place
Tools
Reports
Window
Help
Undo Ctrl+Z
Nothing to Redo Ctrl+Y
Cut Ctrl+X
Copy Ctrl+C
Copy As Text
Paste Ctrl+V
Paste Array...
Find Text... Ctrl+F
Replace Text... Ctrl+H
Find Next F3
Select
DeSelect
Delete
Duplicate Ctrl+R
Move
Align
Jump
Find Similar Objects Shift+F
Move
Move Selection
Move Selection by X, Y...
Move To Front
Rotate Selection Space
Rotate Selection Clockwise Shift+Space
Bring To Front
Send To Back
Bring To Front Of
Send To Back Of
ISL6236AIRZ-T-NI
Place
Add
Manufacturer
Digi-Key
Intersil

Designator and Comment

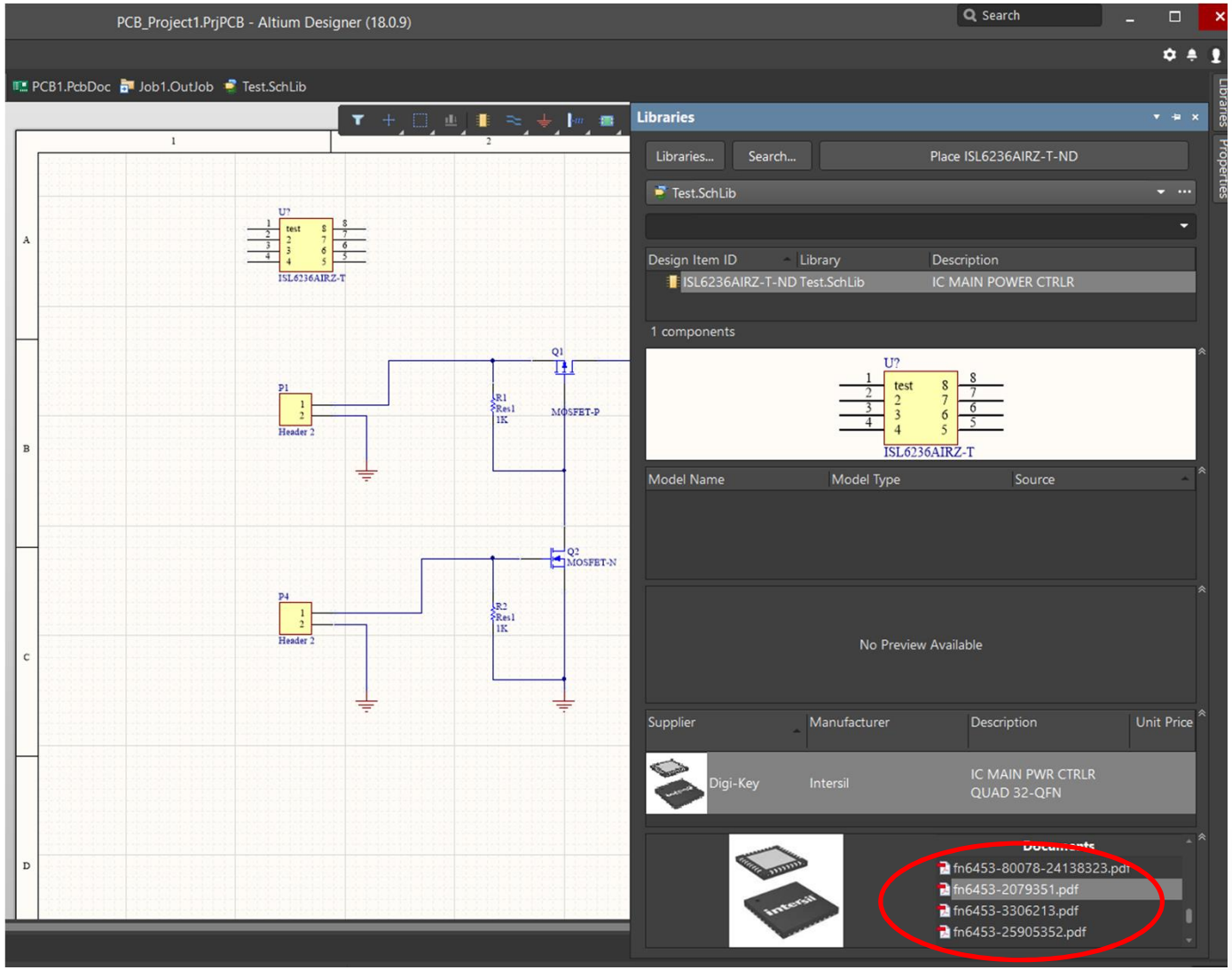


Designator → il tipo

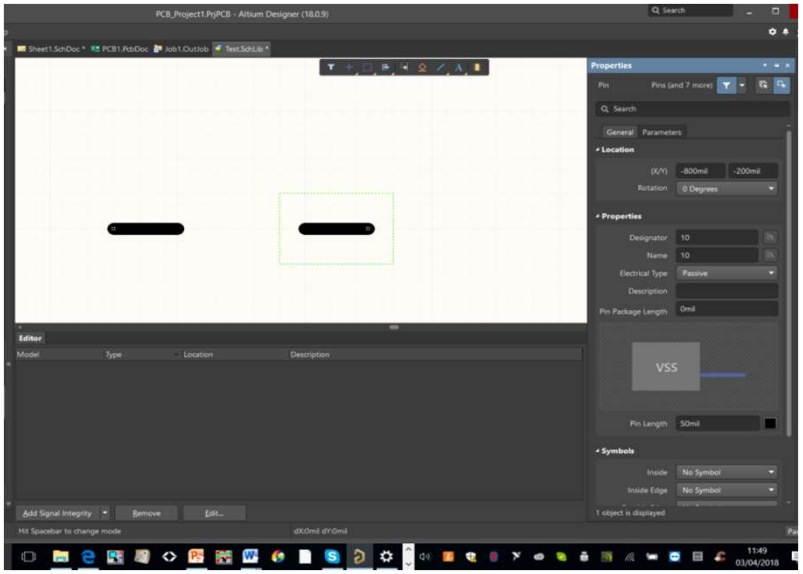
Comment →Value?

Se voglio poter fare la simulazione devo mettere il valore se è un parametro
Scrivendo '=Value'

New Components

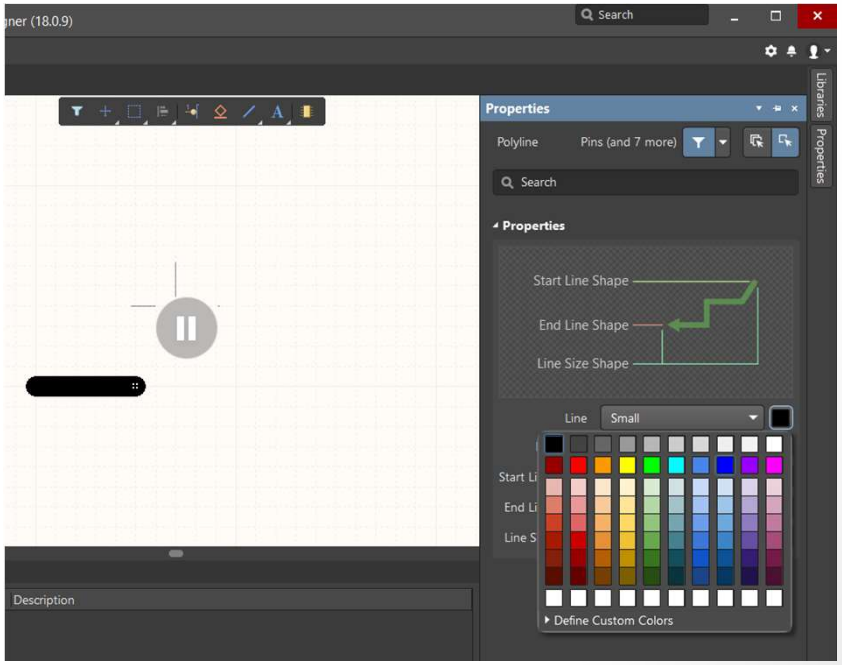


Examples : CAP and Inductor

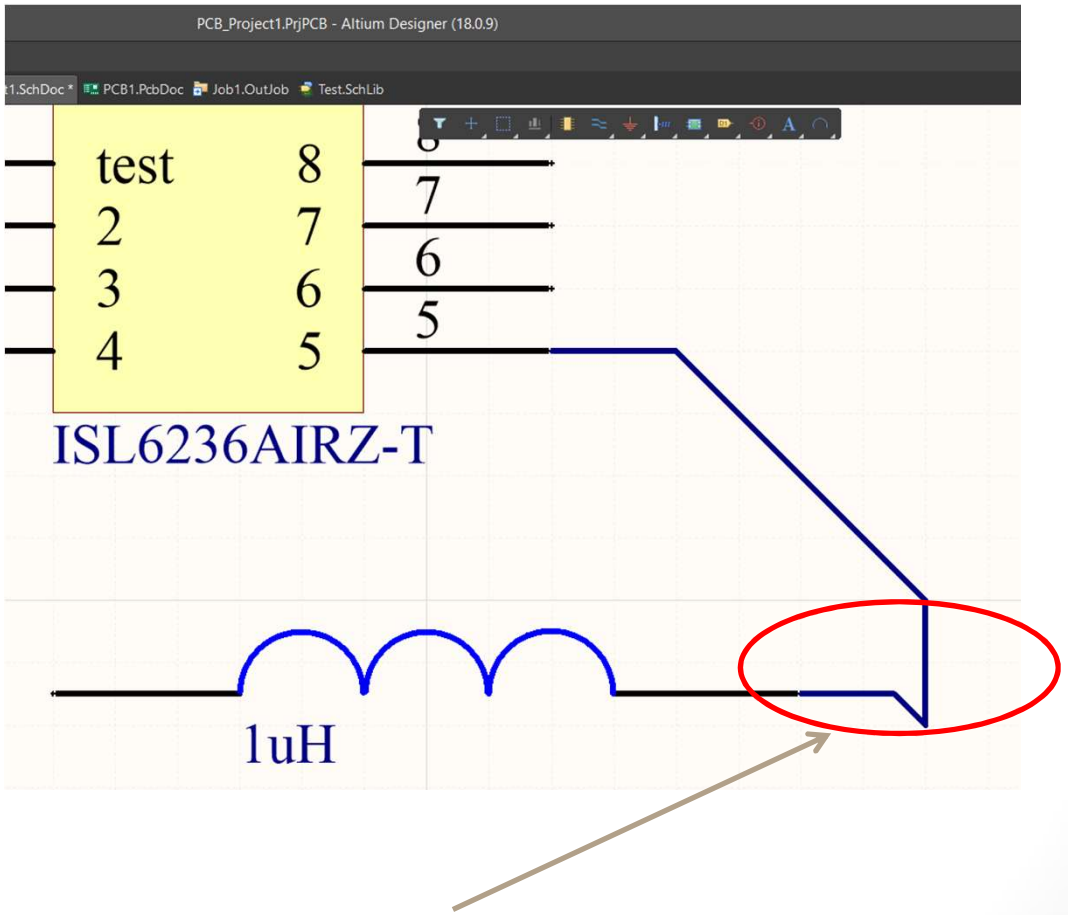


Con il TAB si aprono le proprietà

ATTENZIONE ALLA GRIGLIA
I PIN DEVONO SEMPRE ESSERE
NELLA GLIGLIA DELLO SCHEMATICO!

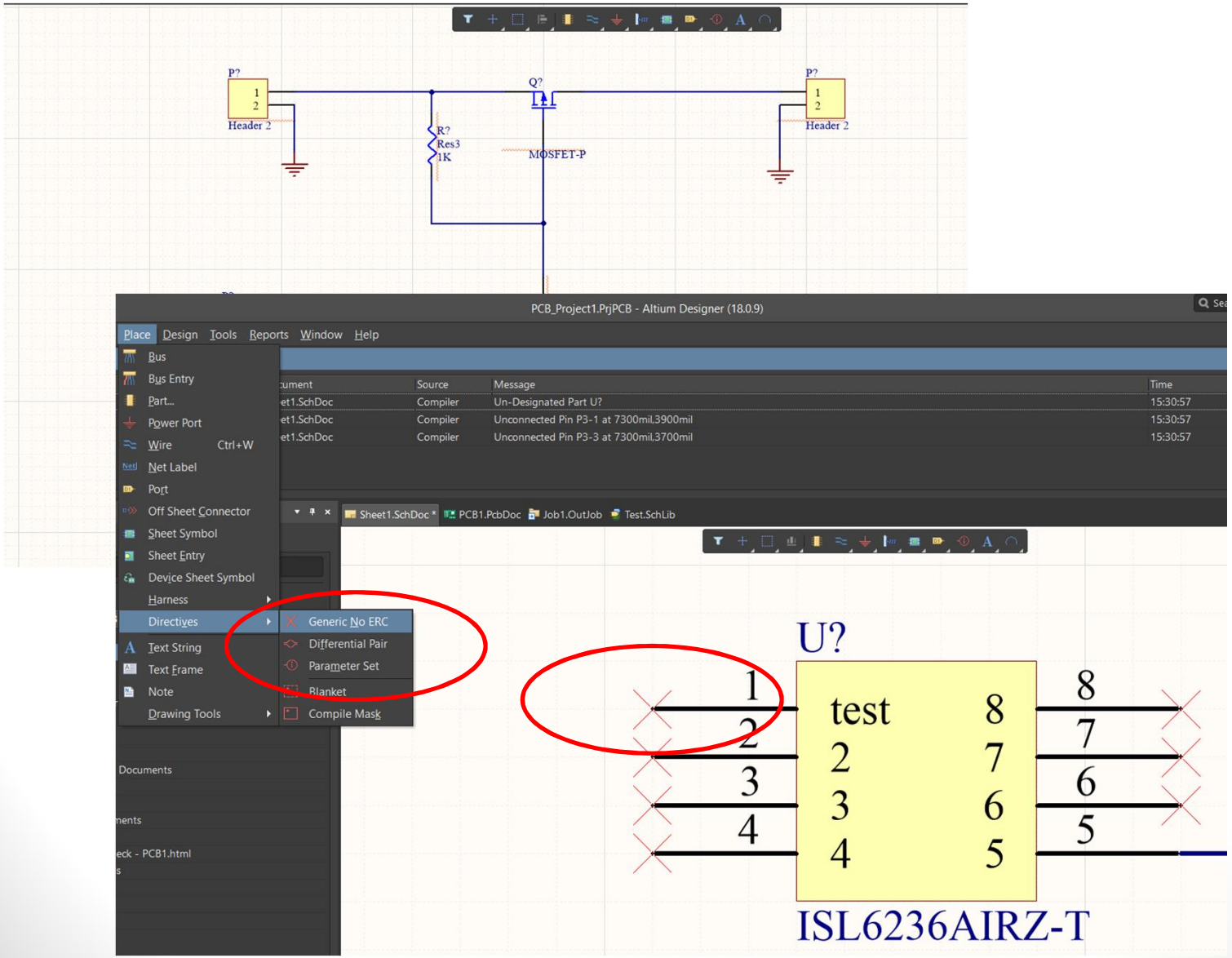


GRIGLIA !!!!!!!!!!!!!!!



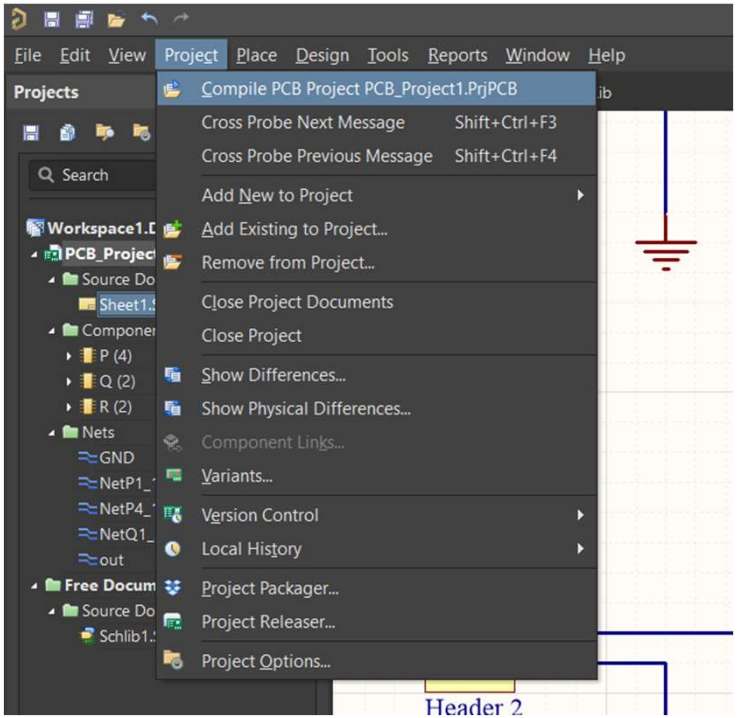
Attenzione alla griglia

PIN UNCONNECTED



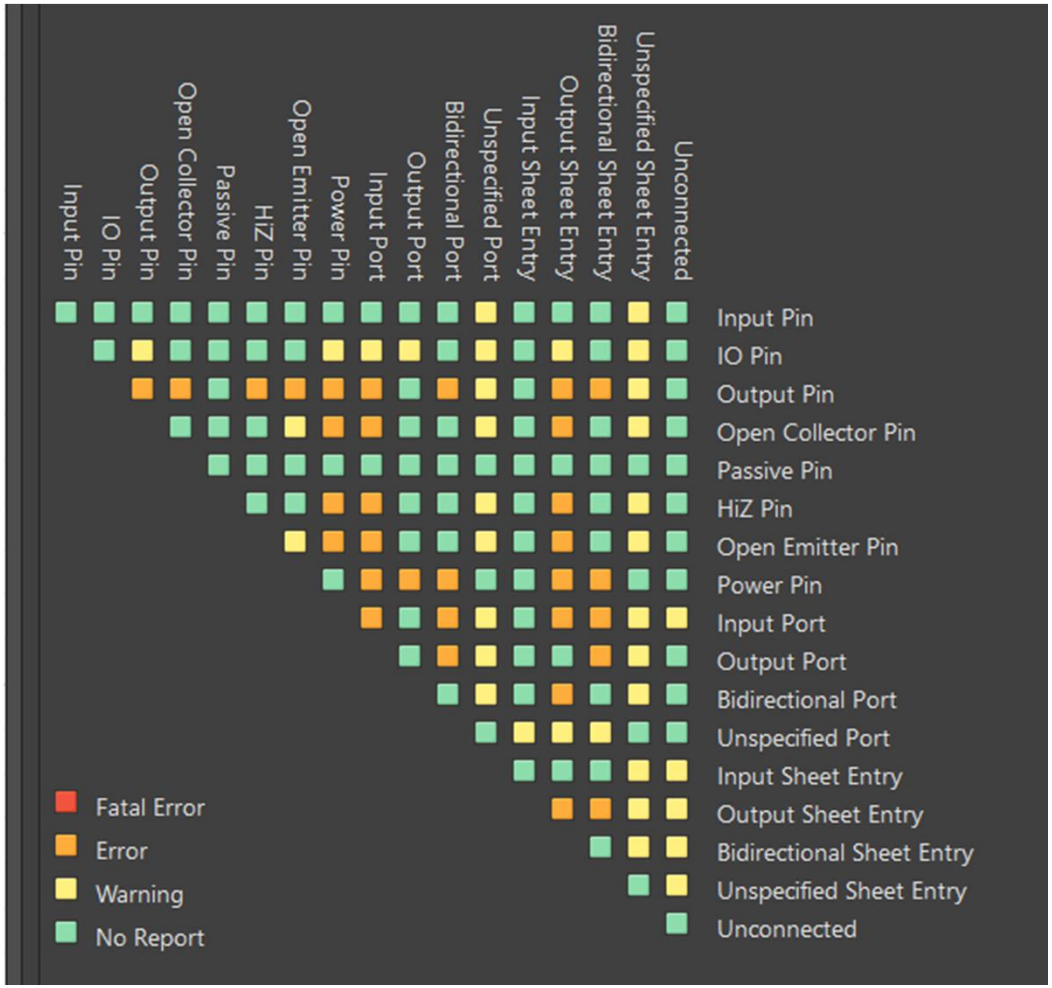
Schematics Compile

La verifica della correttezza dello schematico avviene attraverso la Compilazione del progetto



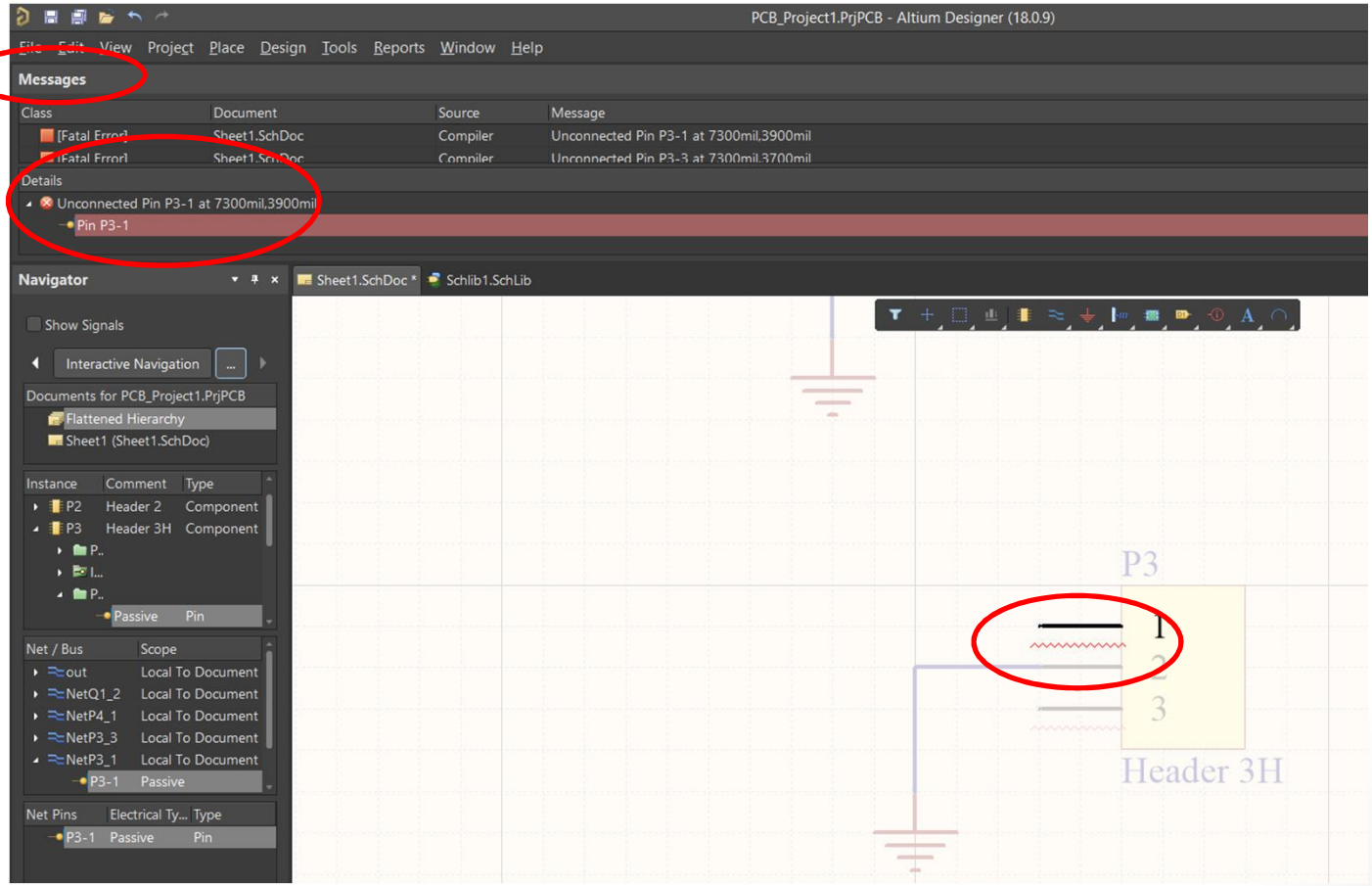
I vincoli che vengono verificati sono specificati in Project Option (da Project)

Schematics Compile

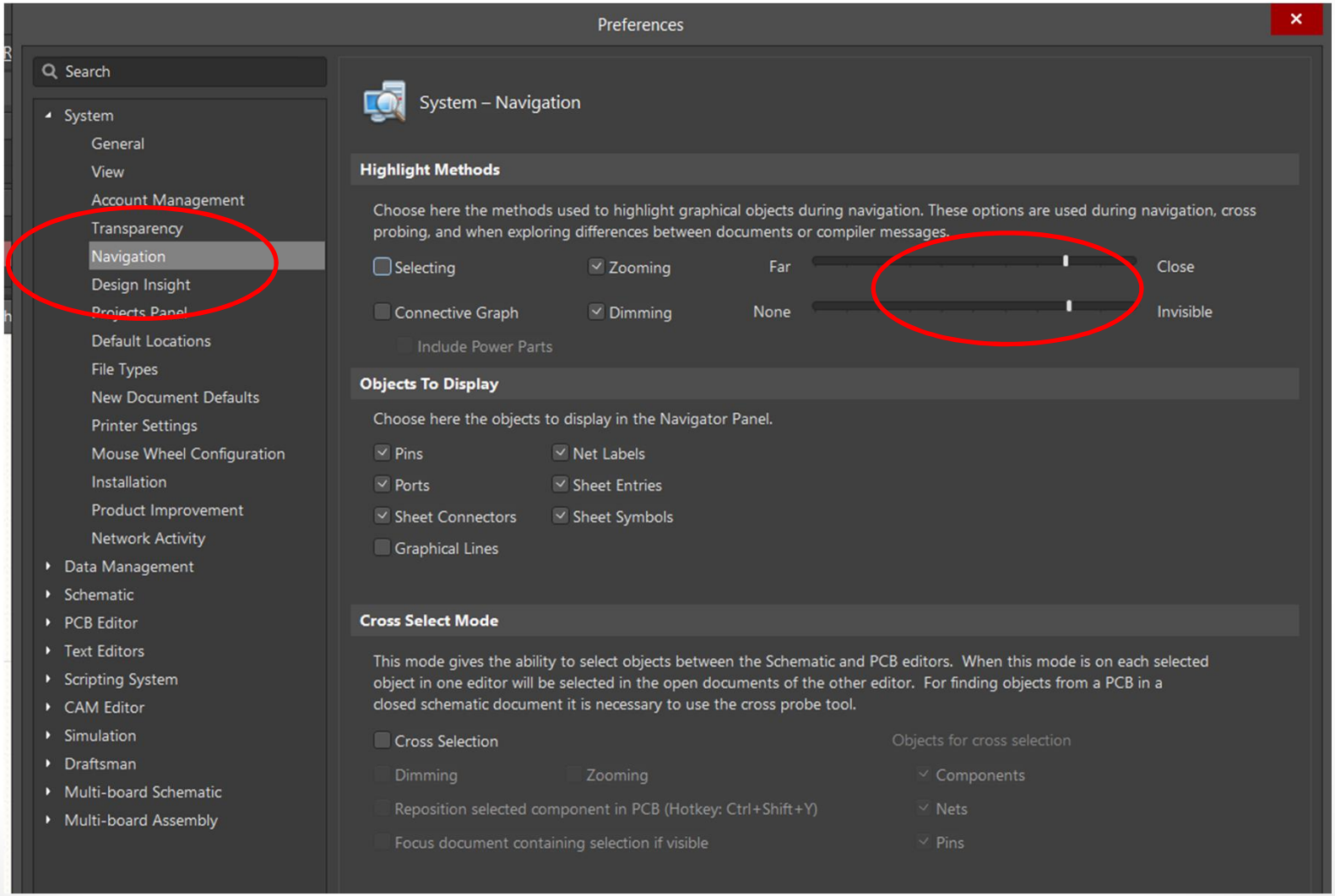


Con RIGHT CLICK posso cambiarli tutti in un a volta

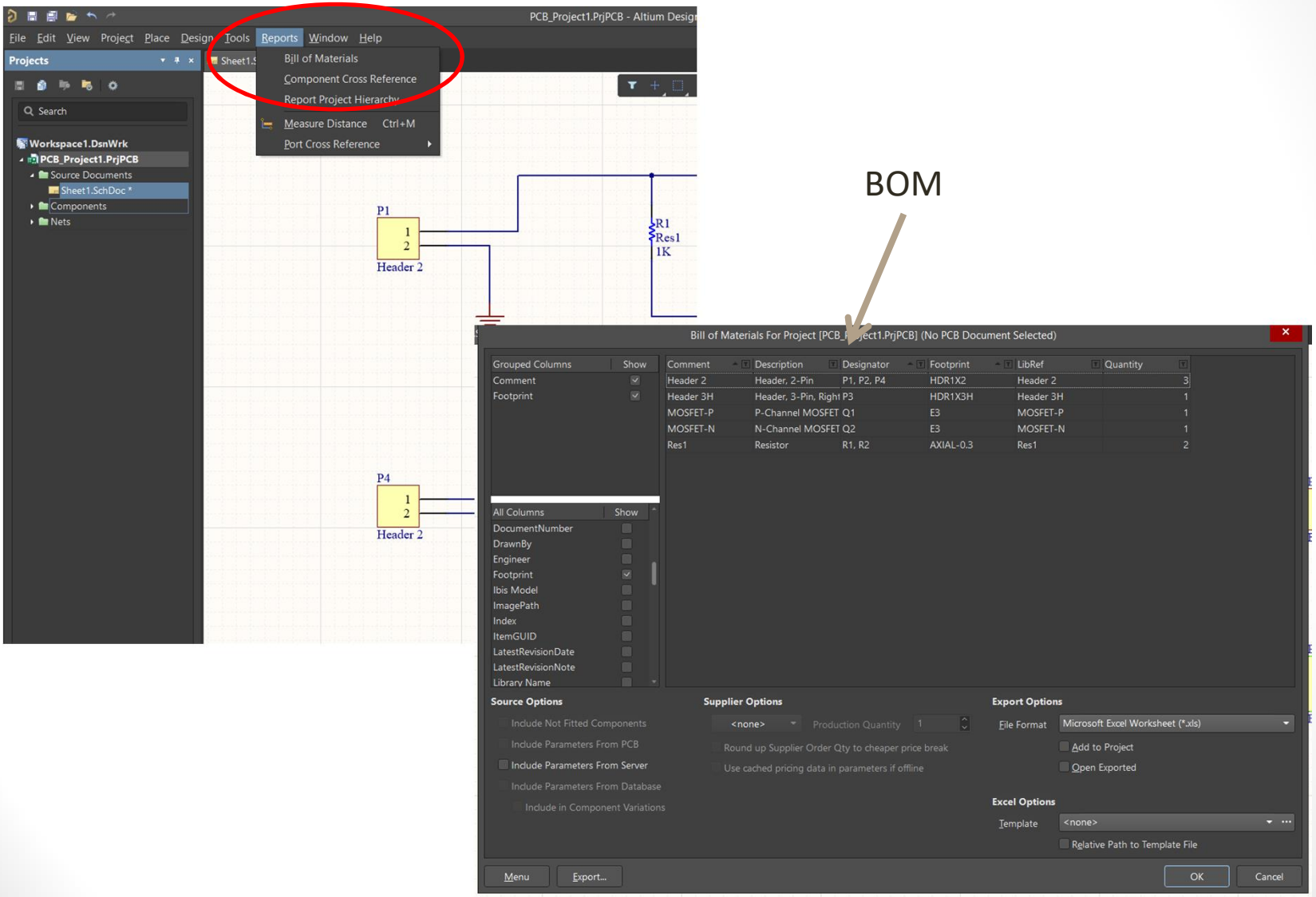
Schematics Compile



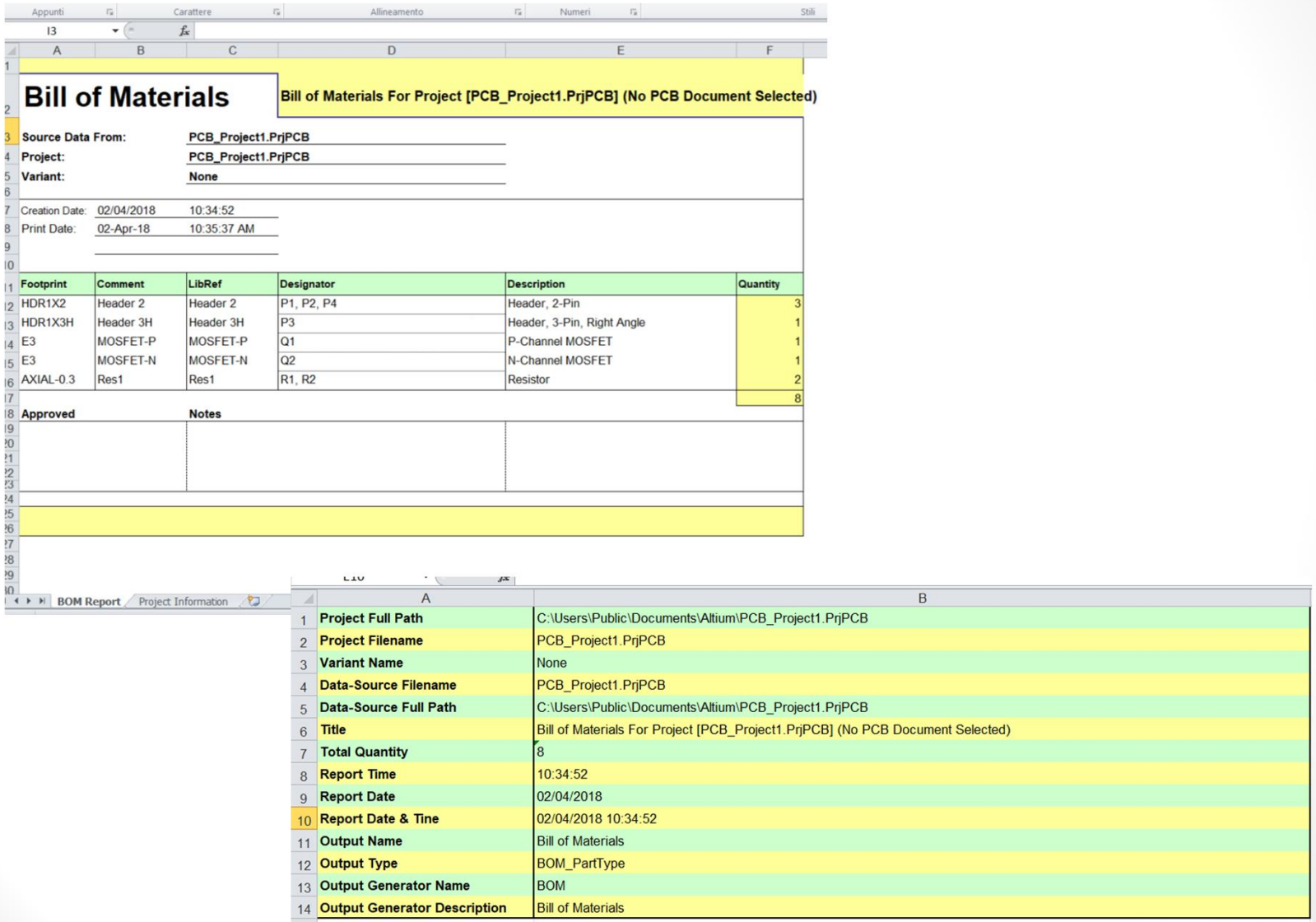
Zoom su Errori



Schematics Reports



Schematics Reports



Bill of Materials

Bill of Materials For Project [PCB_Project1.PrjPCB] (No PCB Document Selected)

Source Data From:	**PCB_Project1.PrjPCB**
Project:	**PCB_Project1.PrjPCB**
Variant:	**None**
Creation Date:	02/04/2018 10:34:52
Print Date:	02-Apr-18 10:35:37 AM

Footprint	Comment	LibRef	Designator	Description	Quantity
HDR1X2	Header 2	Header 2	P1, P2, P4	Header, 2-Pin	3
HDR1X3H	Header 3H	Header 3H	P3	Header, 3-Pin, Right Angle	1
E3	MOSFET-P	MOSFET-P	Q1	P-Channel MOSFET	1
E3	MOSFET-N	MOSFET-N	Q2	N-Channel MOSFET	1
AXIAL-0.3	Res1	Res1	R1, R2	Resistor	2
					8

Approved **Notes**

BOM Report | Project Information

	A	B
1	**Project Full Path**	C:\Users\Public\Documents\Altium\PCB_Project1.PrjPCB
2	**Project Filename**	PCB_Project1.PrjPCB
3	**Variant Name**	None
4	**Data-Source Filename**	PCB_Project1.PrjPCB
5	**Data-Source Full Path**	C:\Users\Public\Documents\Altium\PCB_Project1.PrjPCB
6	**Title**	Bill of Materials For Project [PCB_Project1.PrjPCB] (No PCB Document Selected)
7	**Total Quantity**	8
8	**Report Time**	10:34:52
9	**Report Date**	02/04/2018
10	**Report Date & Tine**	02/04/2018 10:34:52
11	**Output Name**	Bill of Materials
12	**Output Type**	BOM_PartType
13	**Output Generator Name**	BOM
14	**Output Generator Description**	Bill of Materials

Esempio / esercizio