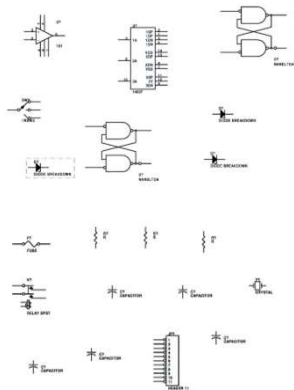


# PROGETTAZIONE

# PCB

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

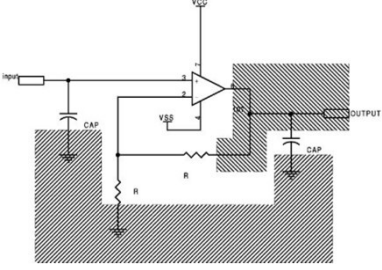
# Ciclo di Progettazione



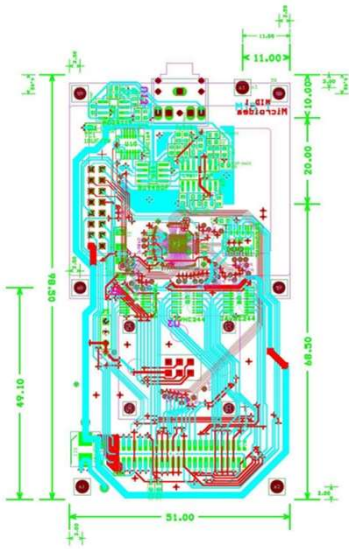
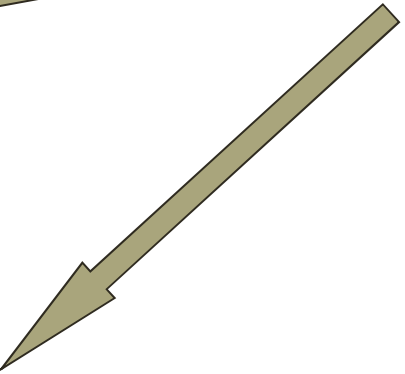
Simboli



Lista collegamenti

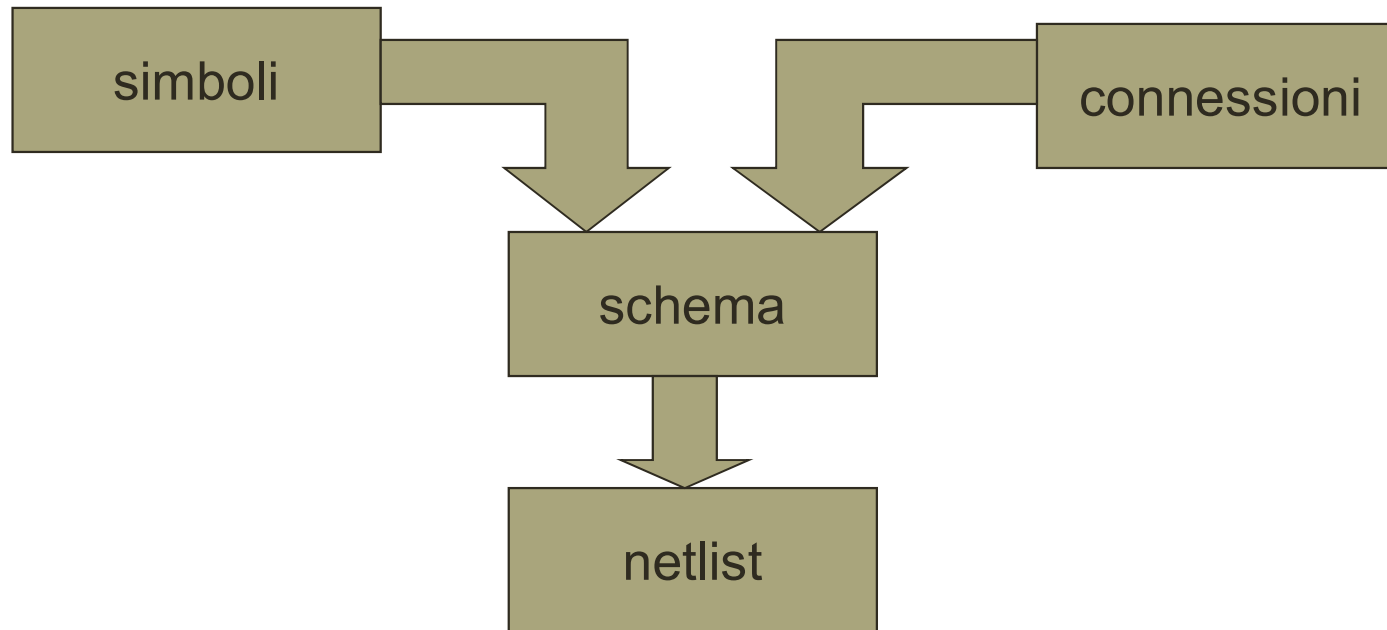


Schematico

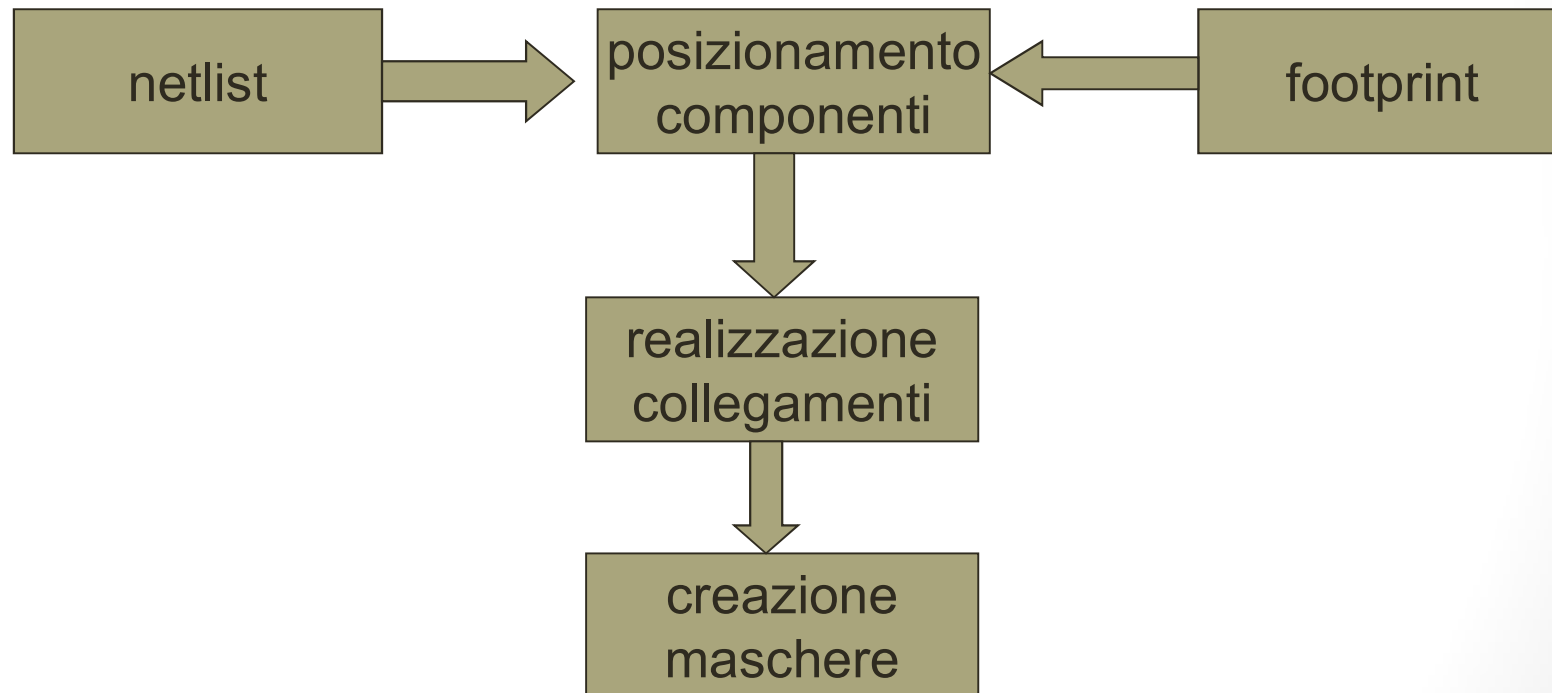


PCB

# Ciclo di Progettazione - Schematico



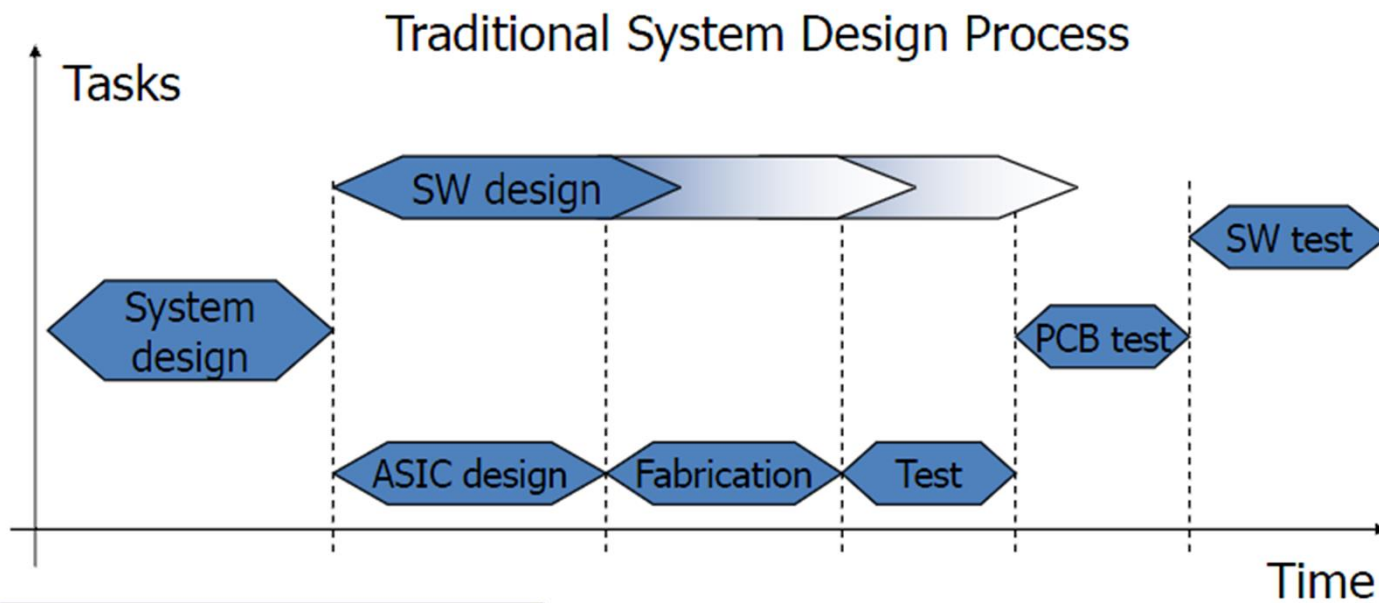
# Ciclo di Progettazione -LAYOUT



# Ciclo di Progettazione

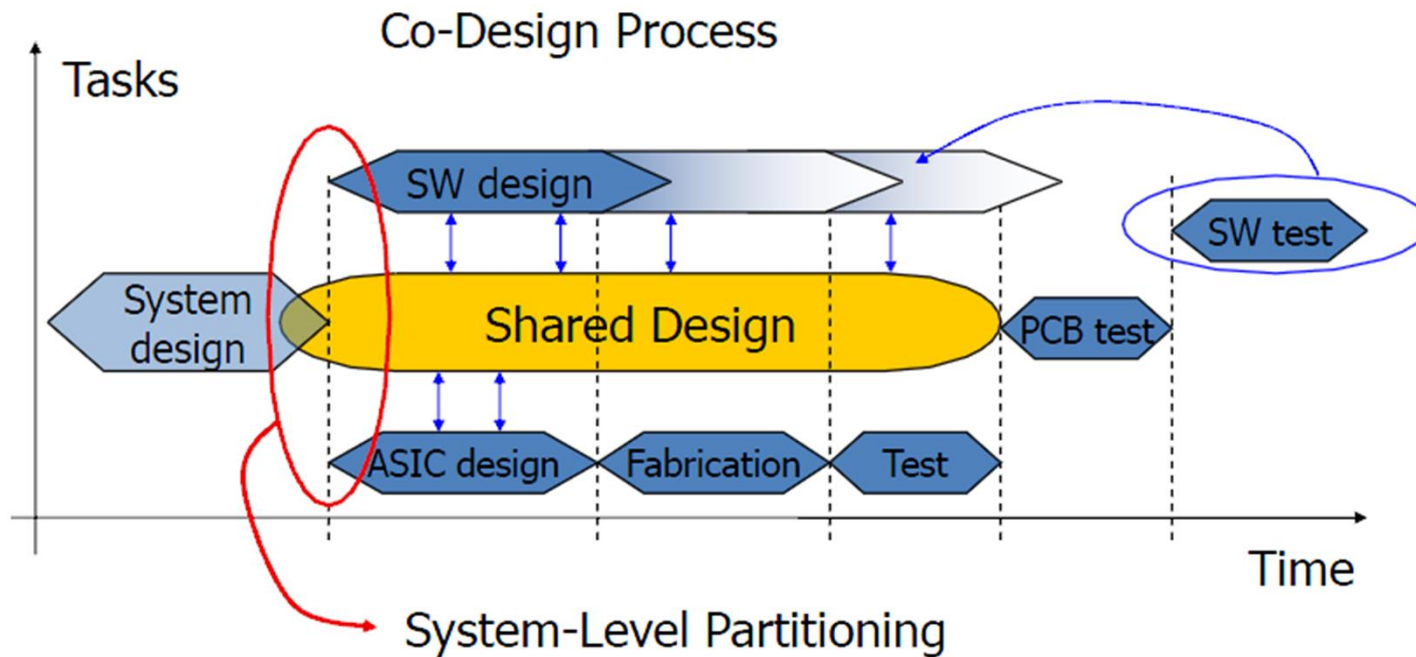


# Ciclo di Progettazione

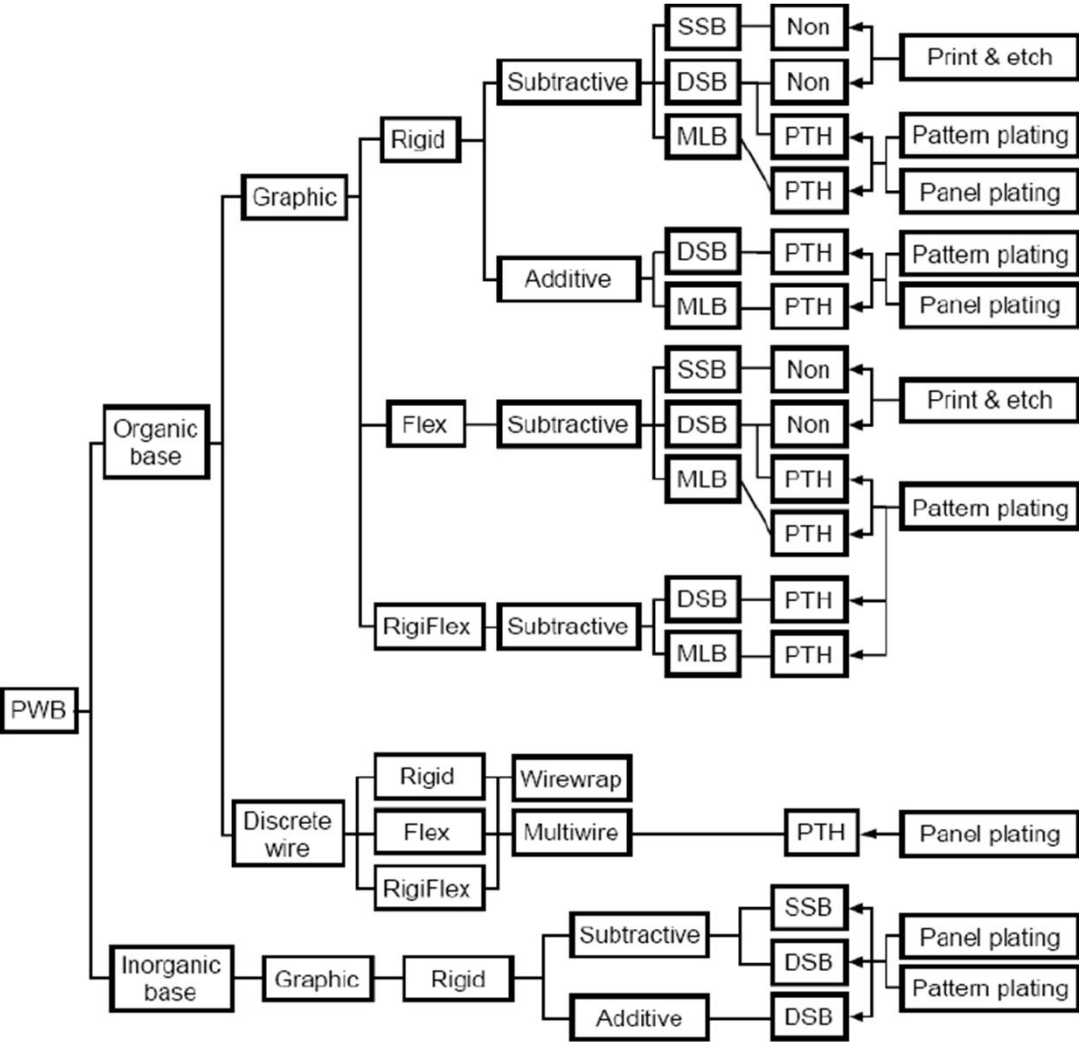


Copyright J. Madsen,  
some modifications applied

# Ciclo di Progettazione



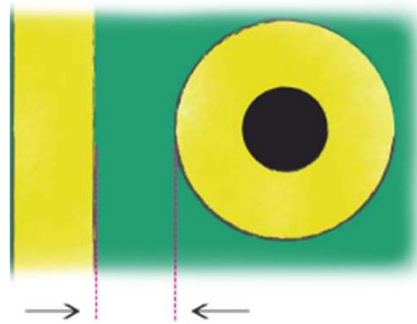
# Ciclo di Progettazione Produzione PCB



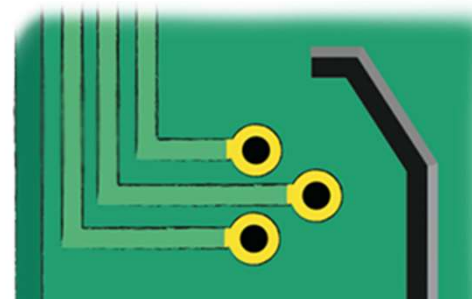


# Ciclo di Progettazione Produzione PCB

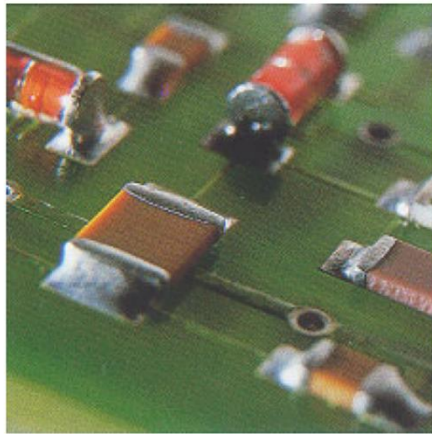
Vincoli Elettrici



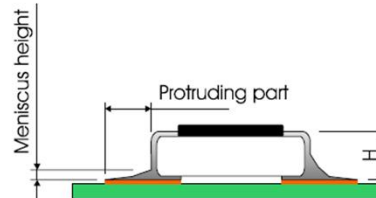
Vincoli Meccanici



# Ciclo di Progettazione Assemblaggio

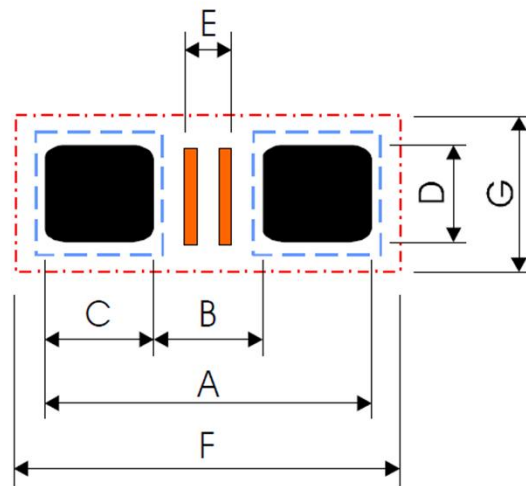


## Acceptable



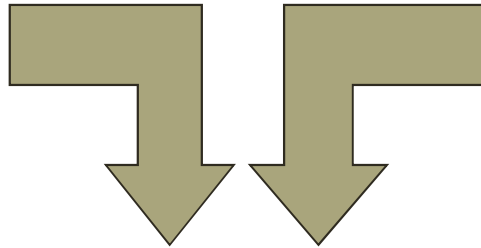
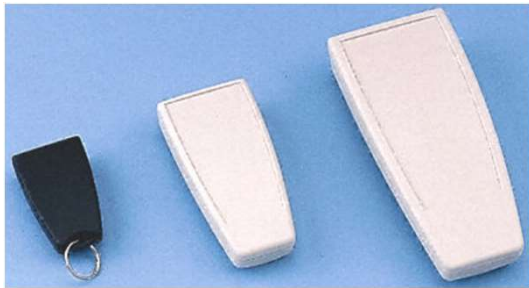
For Components with a termination height  $H < 1.2\text{mm}$  : The height of the meniscus must be at least  $1/3$  of the height  $H$

For components with a termination height  $H \geq 1.2\text{mm}$  : The height of the meniscus must be at least  $0.4\text{mm}$ .

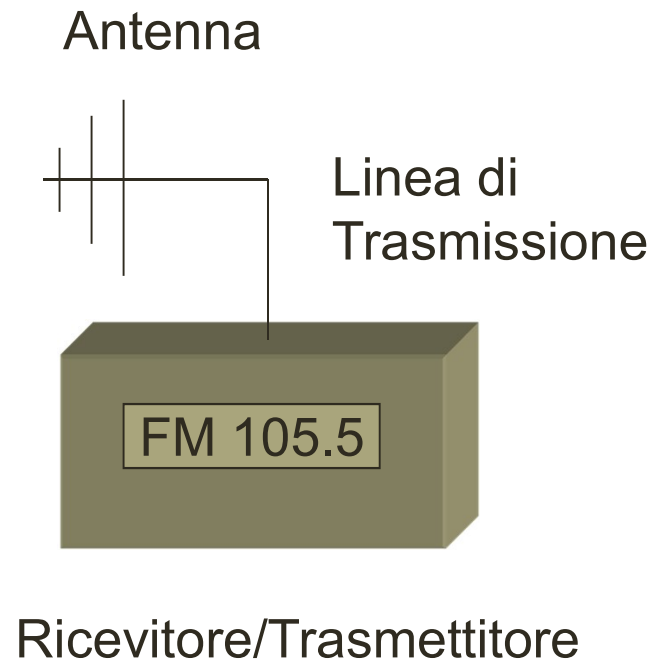
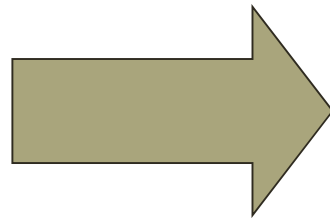


-  Occupied area
-  Solder land / Solder paste pattern
-  Solder resist pattern
-  Tracks or Dummy tracks (for wave soldering only)

# Ciclo di Progettazione Meccanici - Designer

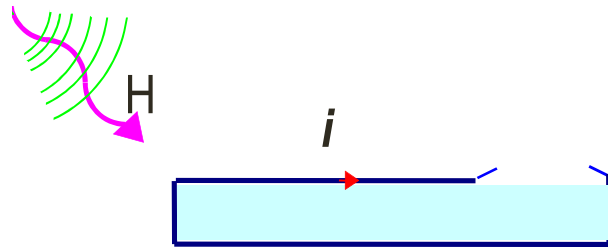


# Ciclo di Progettazione EMC



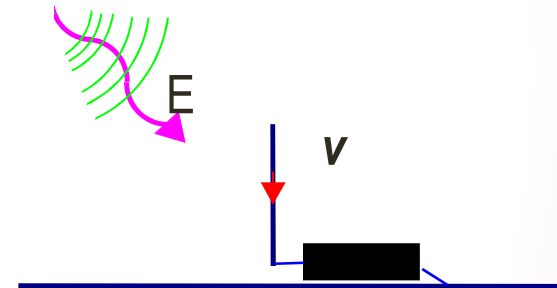
# Ciclo di Progettazione EMC

Accoppiamento  
Magnetico



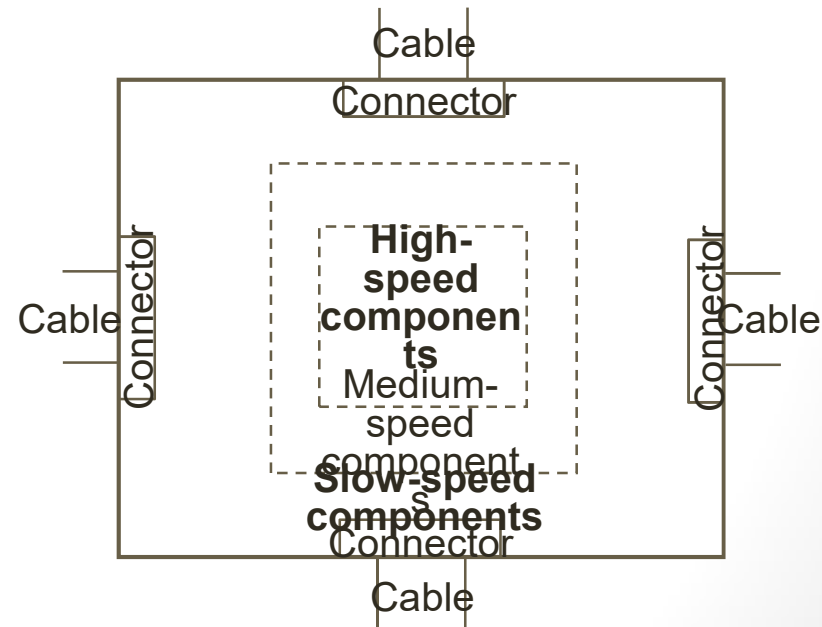
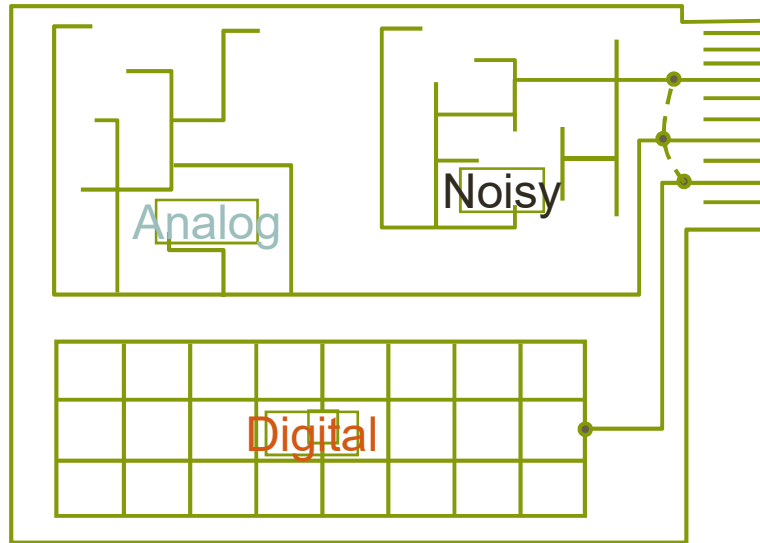
Generazione di corrente  $I$  per  
campo magnetico  $H$  (A/m)

Accoppiamento  
Elettrico

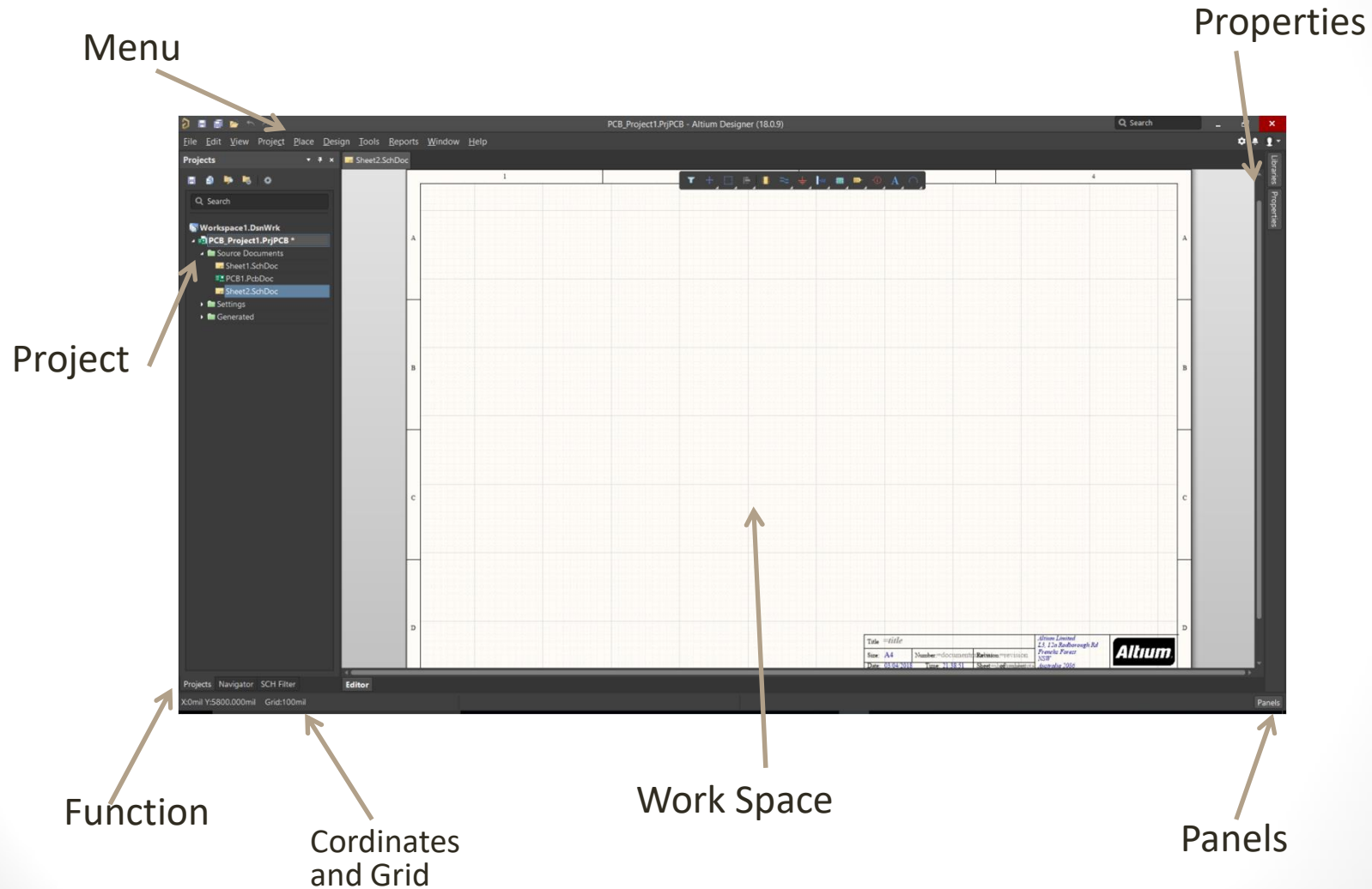


Tensione  $V$  generata per  
campo elettrico  $E$   
(V/m)

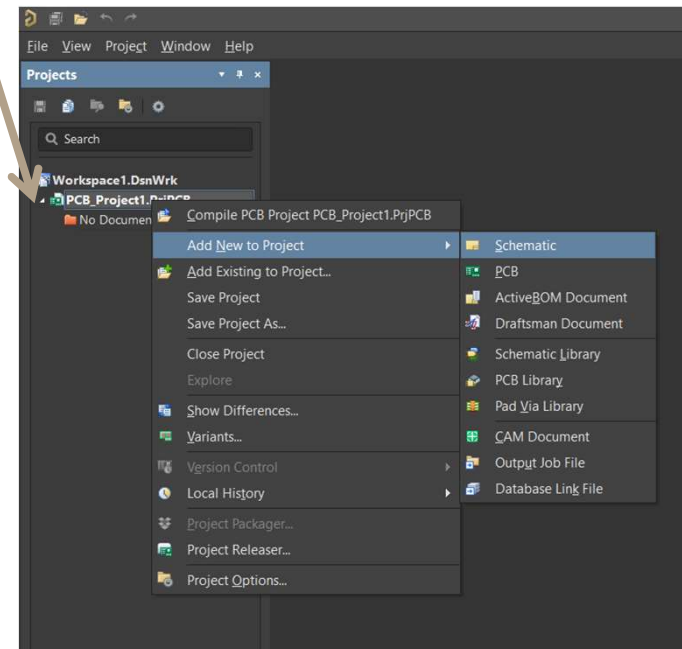
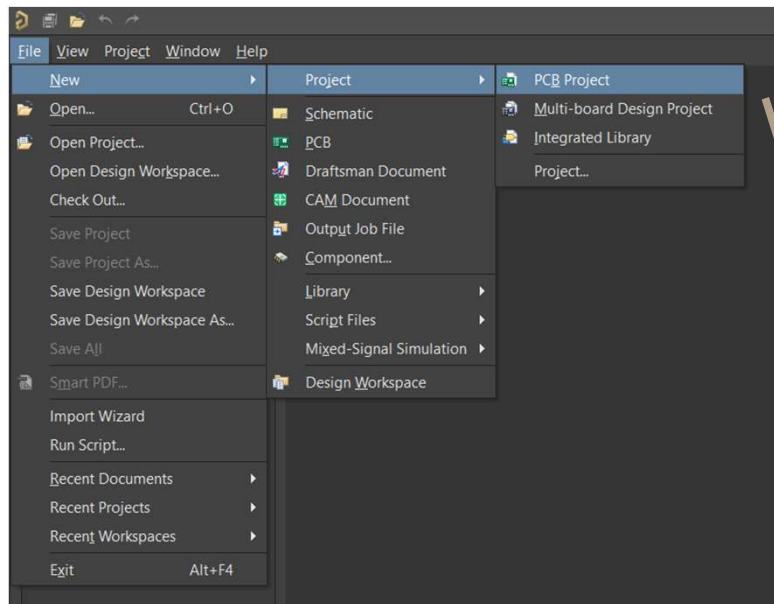
# Ciclo di Progettazione EMC



# Altium Designer - Schematics



# Schematics - New Project

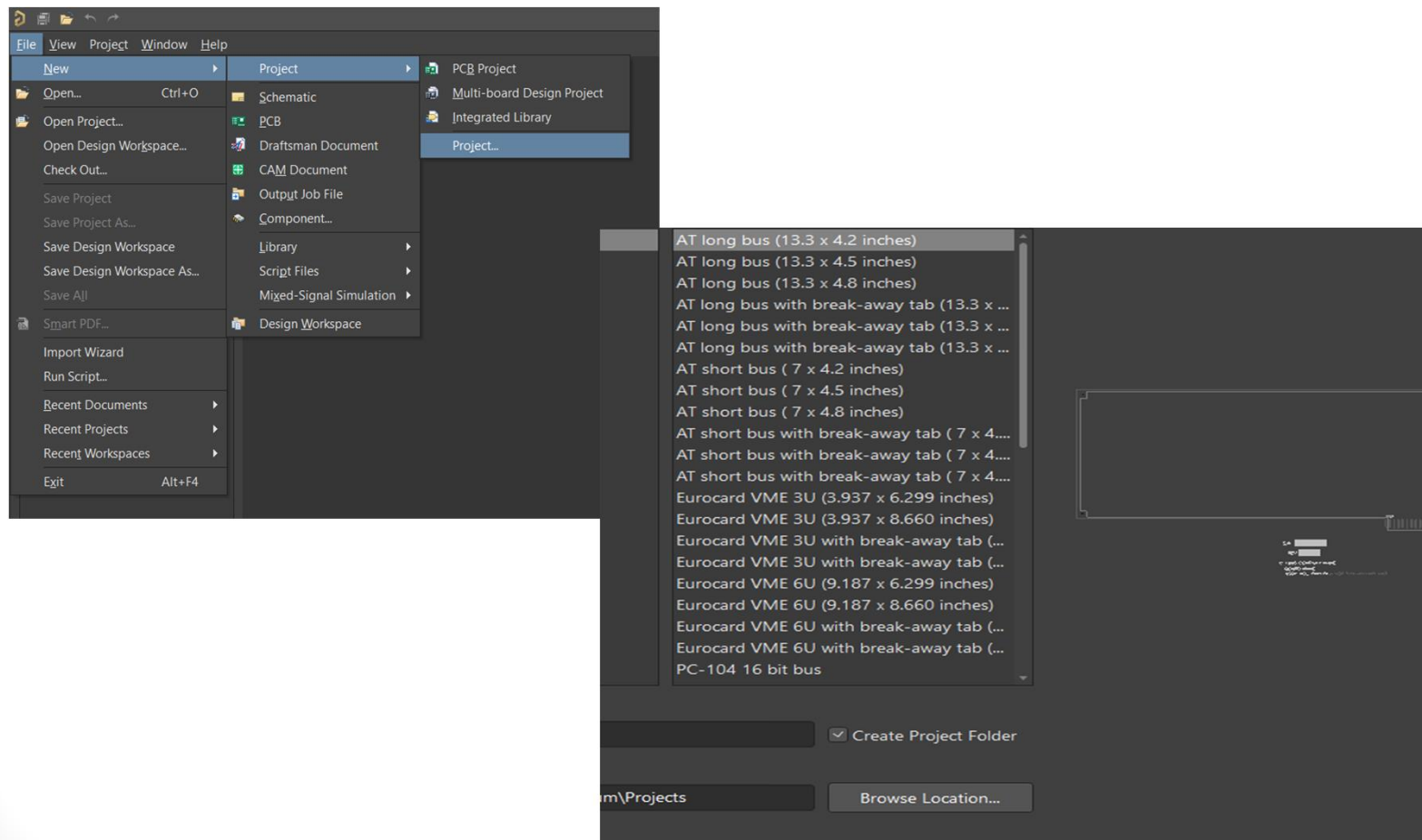


Nota:  
Un nuovo documento è utilizzabile solo dopo essere stato salvato.

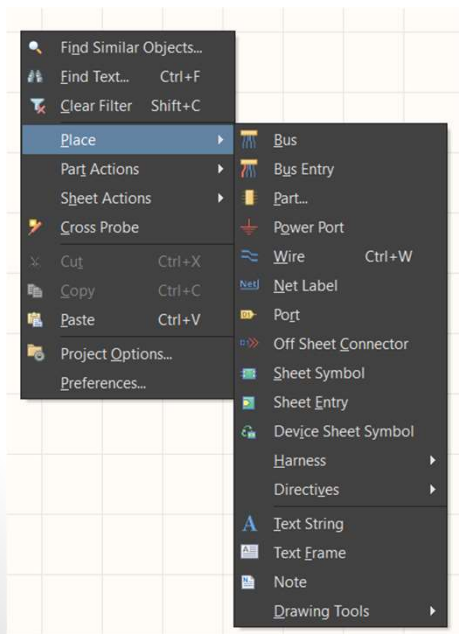
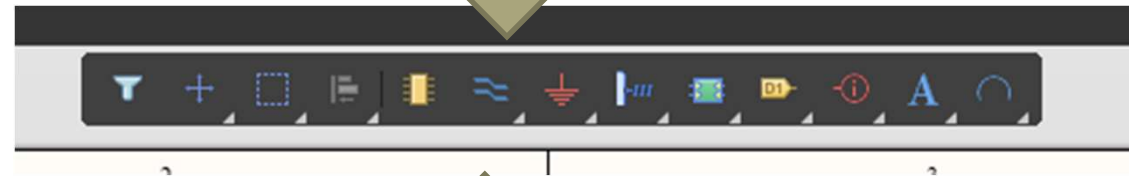
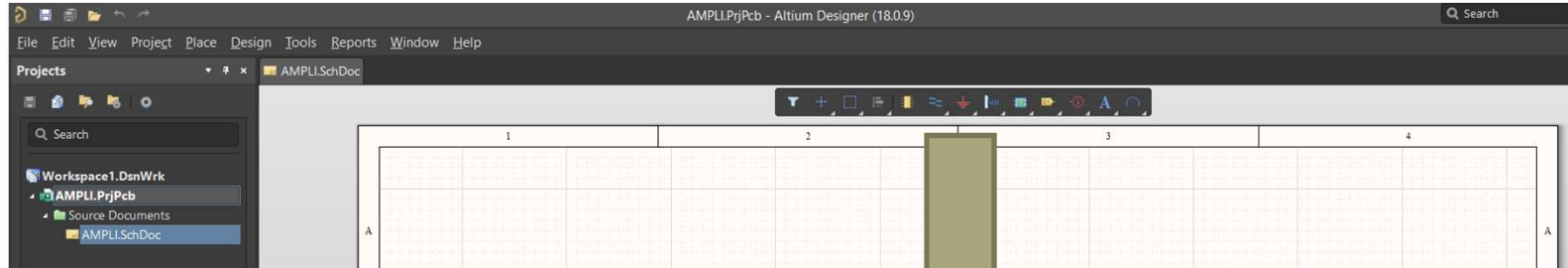


# Schematics

## New Project Source

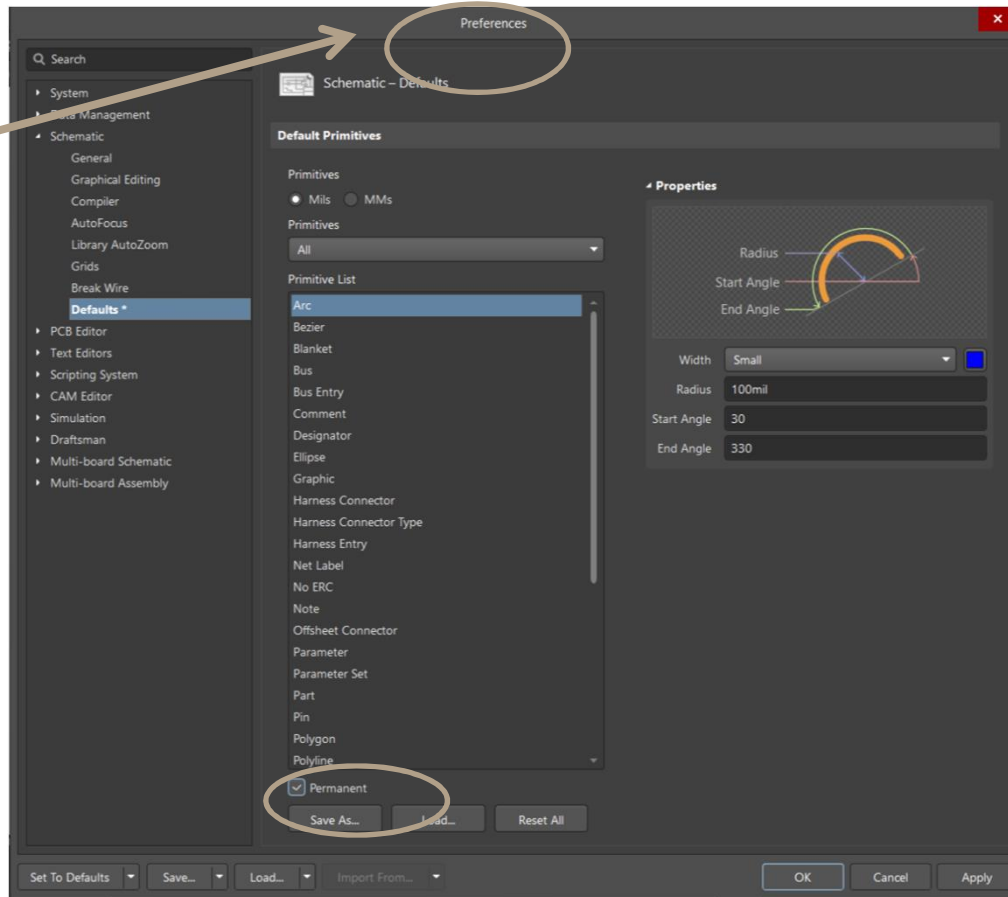


# Schematics Place



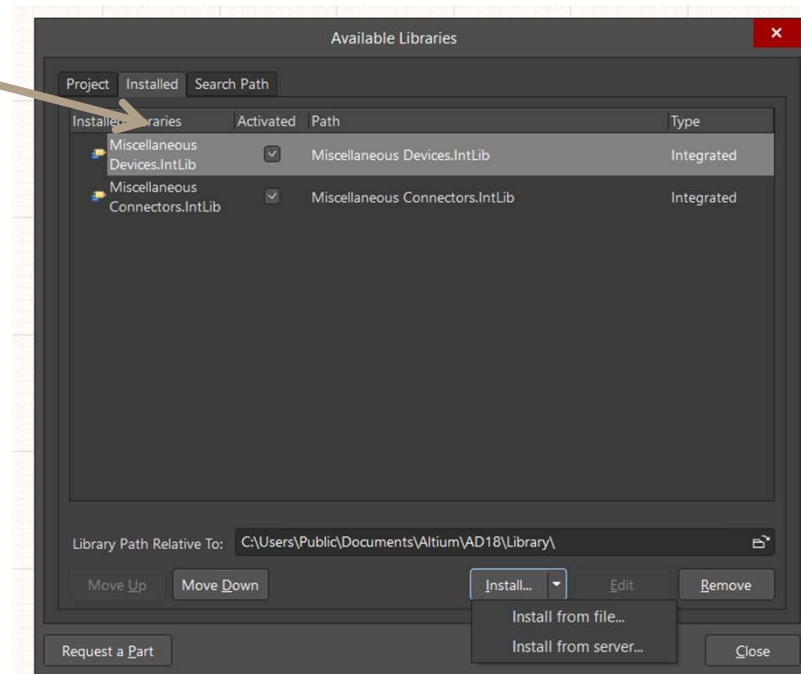
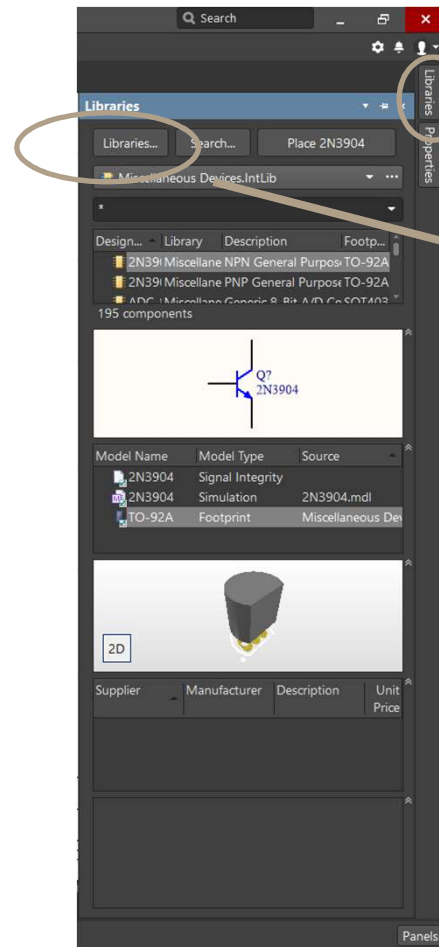
# Schematics Preferences

From  
TOOLS



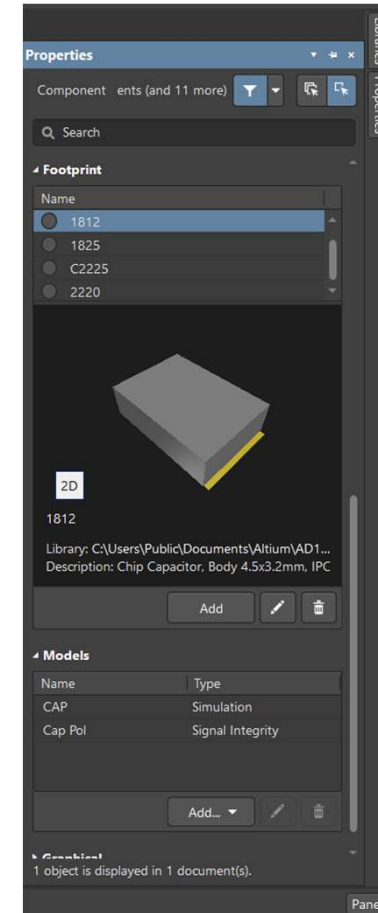
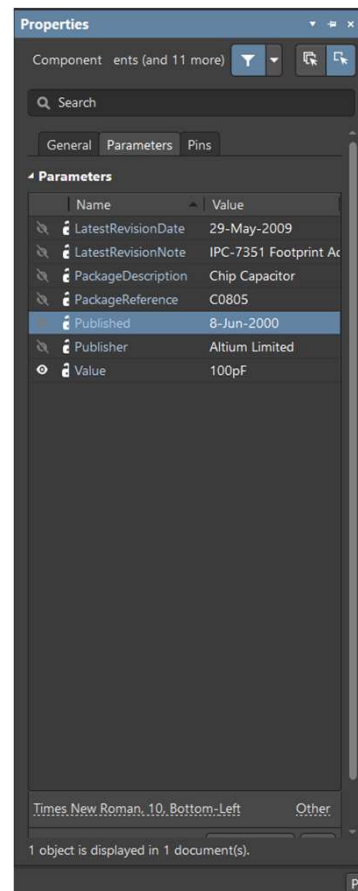
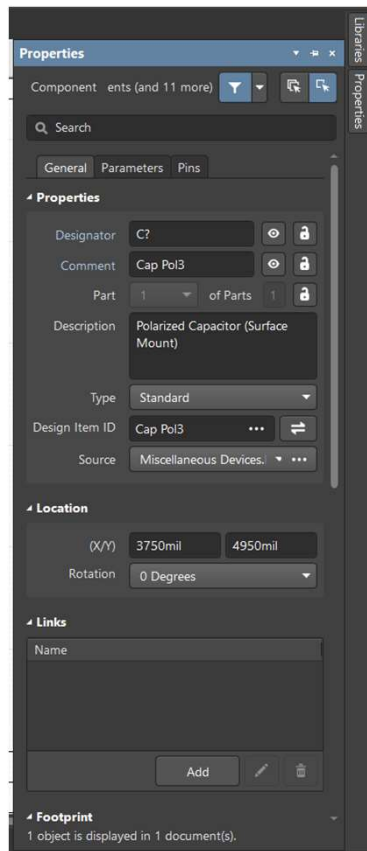
While attributes can be modified during placement (**Tab** to bring up associated properties dialog), bear in mind that these will become the default settings for further placement unless the **Permanent** option on the **Schematic – Default Primitives** page of the *Preferences* dialog is enabled. When 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

PCB\_Project1.PrjPCB - Altium Designer (18.0.9)

File Edit View Project Place Tools Reports Window Help

Projects Sheet1.SchDoc PCB1.PcbDoc Job1.OutJob Schlib1.SchLib

Supplier Links

Supplier	Description	Unit Price
Avnet	IC MAIN PWR CTRLR QUAD 32-QFN	
Mouser	IC MAIN PWR CTRLR QUAD 32-QFN	
Digi-Key	IC MAIN PWR CTRLR QUAD 32-QFN	

Keywords: ISL6236AIRZ-T-ND

Add Supplier Links

Supplier Description Unit Price

Intersil ISL6236AIRZ-T

Avnet IC MAIN PWR CTRLR QUAD 32-QFN

Mouser IC MAIN PWR CTRLR QUAD 32-QFN

Digi-Key IC MAIN PWR CTRLR QUAD 32-QFN

Total results 1

Order Quantity: 1

Parameters

Name	Value
Case/Package	QFN
Description	IC MAIN PWR CTRLR QUAD 32-QFN
Height	0.95mm

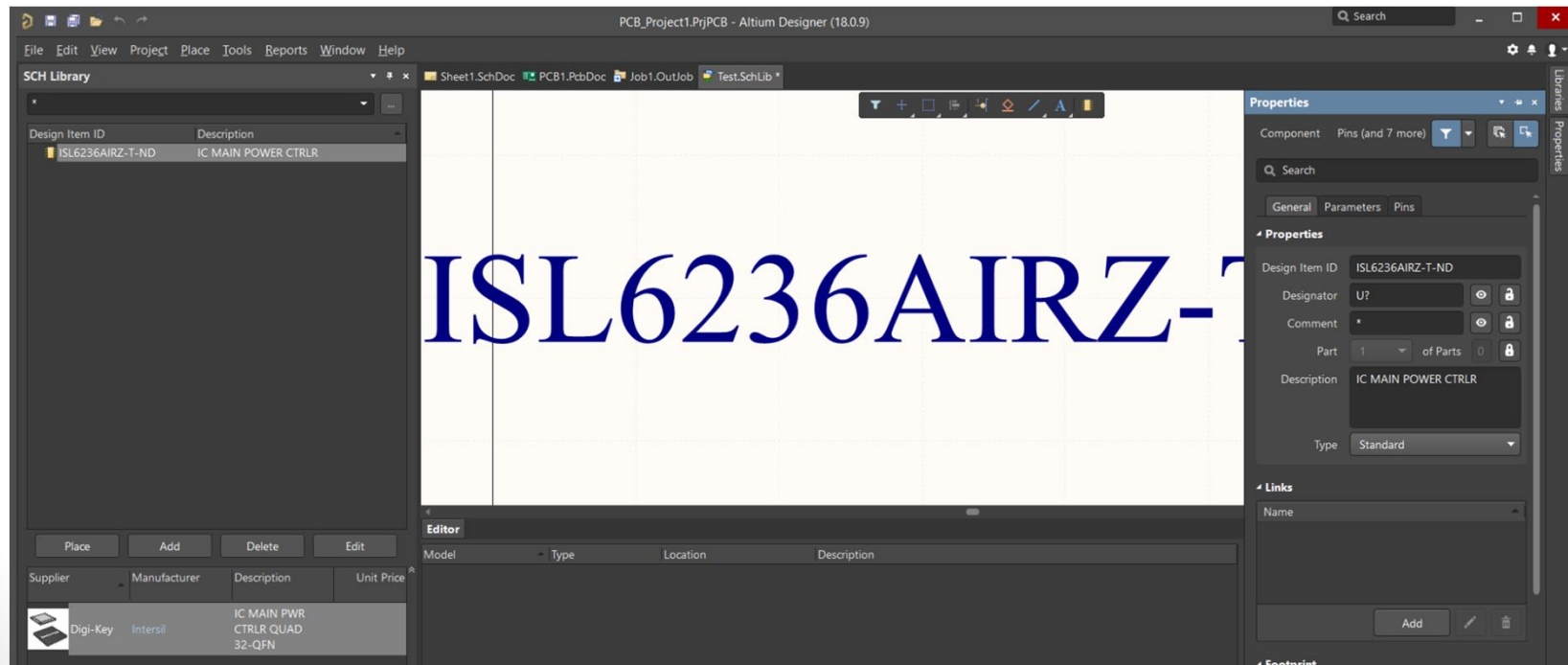
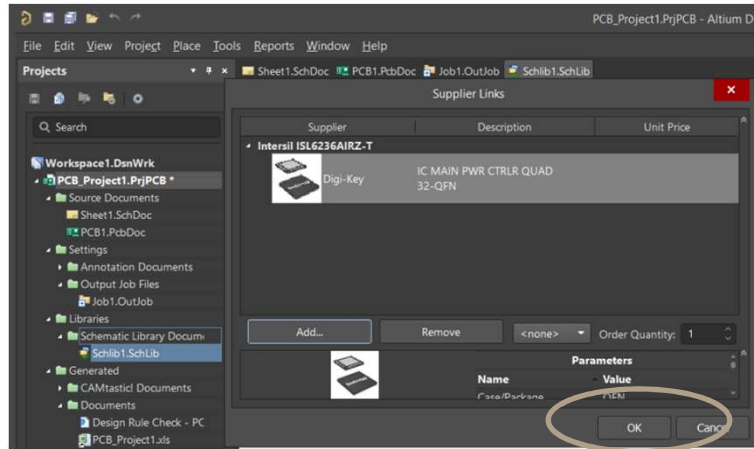
Editor

Model	Type	Location	Description
-------	------	----------	-------------

Projects Navigator SCH Library Add Signal Integrity Remove Edit...

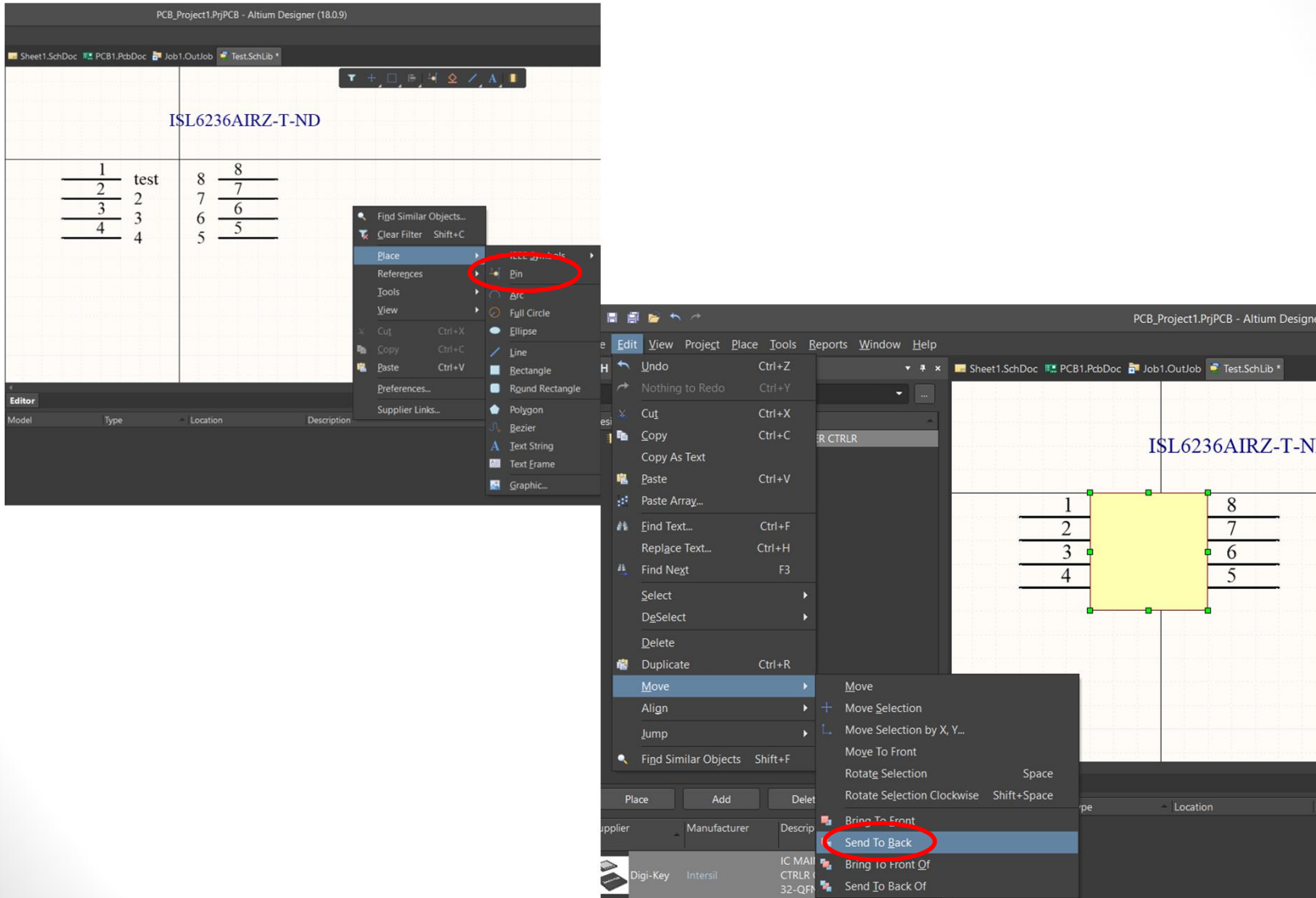
X:-1500mil Y:-100mil Grid:100mil

# New Components

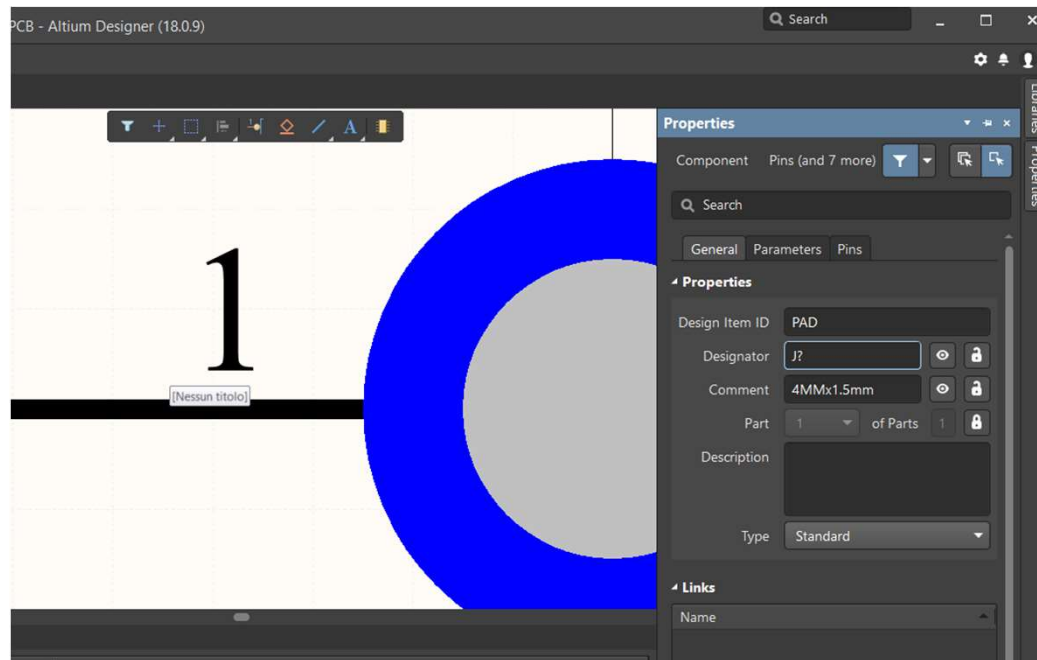




# PIN and Body



# 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

PCB\_Project1.PrjPCB - Altium Designer (18.0.9)

PCB1.PcbDoc Job1.OutJob Test.SchLib

Libraries

Place ISL6236AIRZ-T-ND

Test.SchLib

Design Item ID	Library	Description
ISL6236AIRZ-T-ND	Test.SchLib	IC MAIN POWER CTRLR

1 components

U7

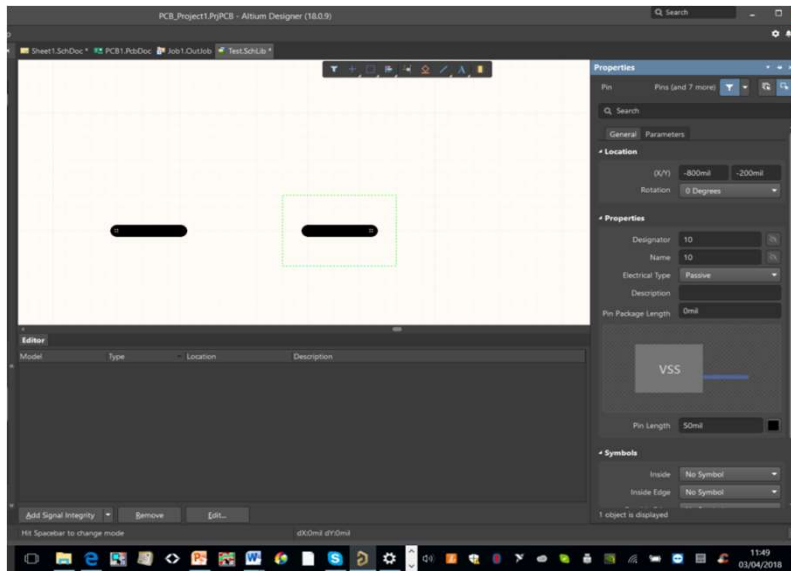
Model Name	Model Type	Source
No Preview Available		

Supplier	Manufacturer	Description	Unit Price
Digi-Key	Intersil	IC MAIN PWR CTRLR QUAD 32-QFN	

Documents

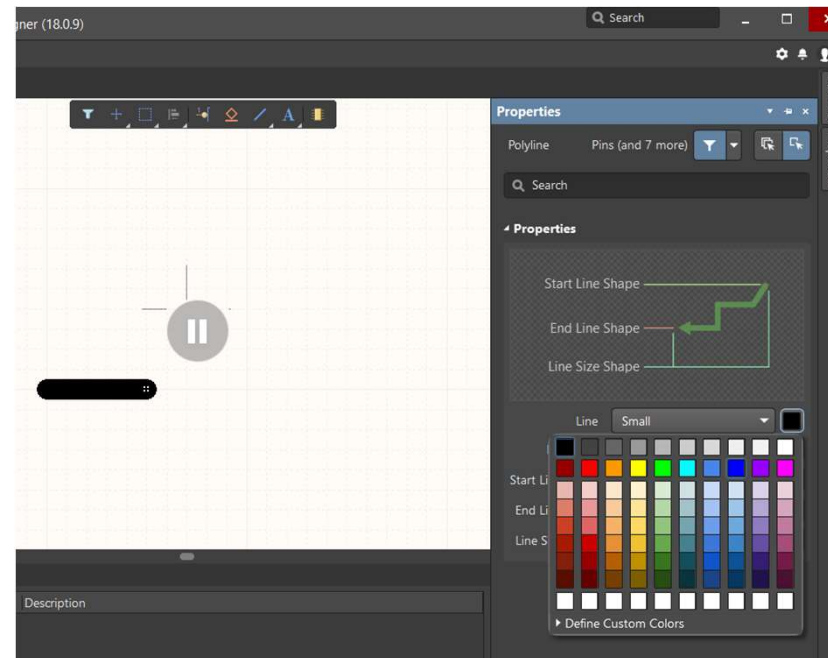
- fn6453-80078-24138323.pdf
- fn6453-2079351.pdf
- fn6453-3306213.pdf
- fn6453-25905352.pdf

# Examples : CAP and Inductor

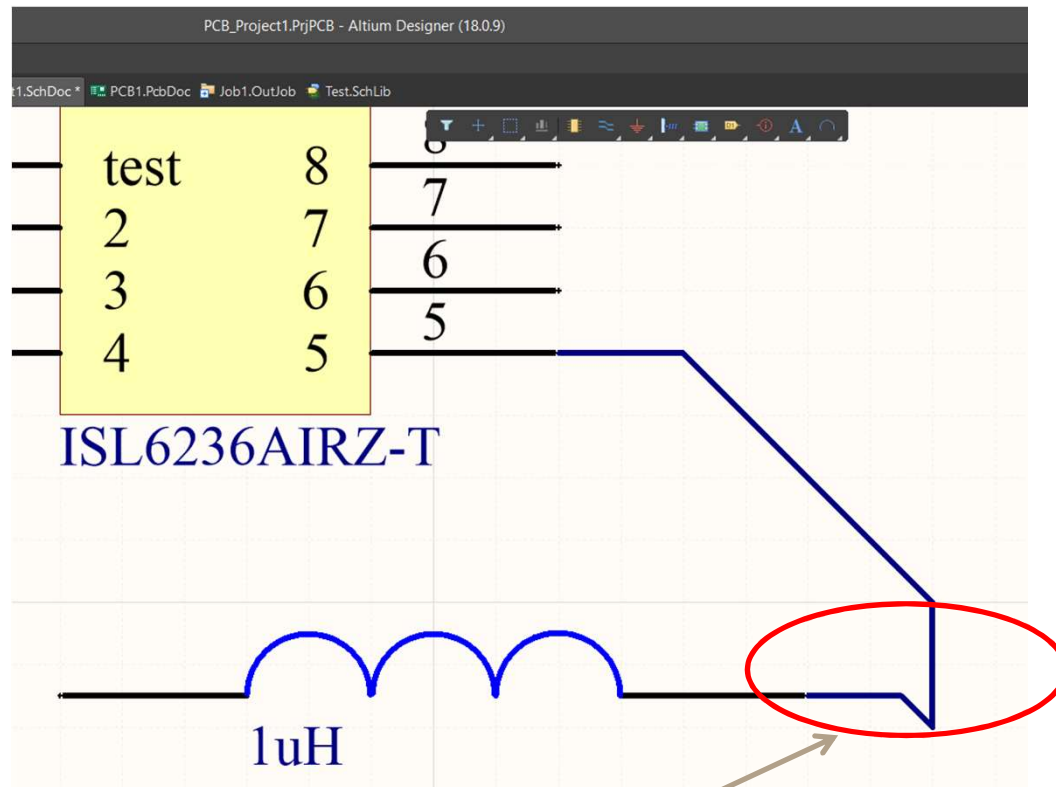


Con il TAB si aprono le proprietà

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



# GRIGLIA !!!!!!!!!!!!!!!!!!!!!



Attenzione alla griglia

# PIN UNCONNECTED

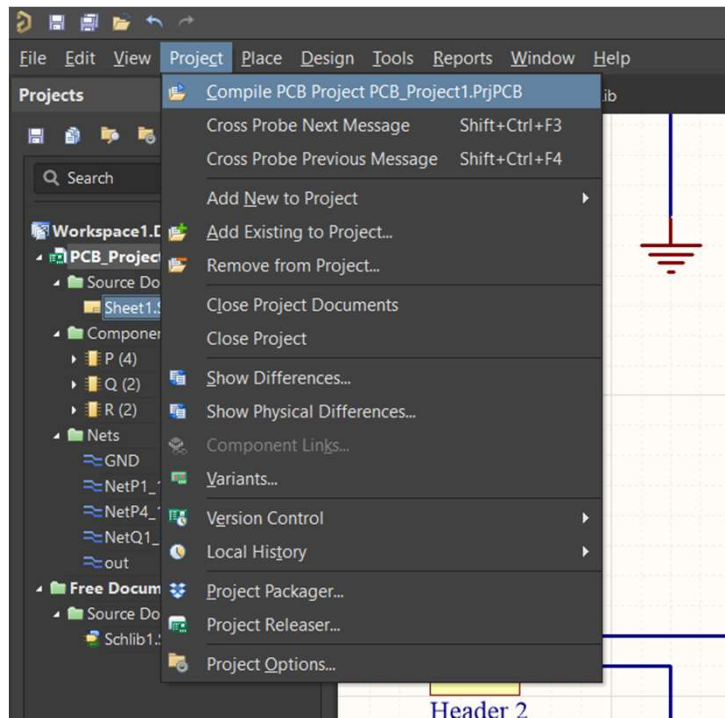
The image shows a screenshot of Altium Designer (18.0.9) with a circuit diagram and a component pinout table. The circuit diagram shows two headers (P?) connected to a MOSFET (Q?) through a resistor (R? Res3 1K). The MOSFET is connected to ground. The component pinout table is for the ISL6236AIRZ-T and shows unconnected pins 1, 2, 3, and 4 on the left side, and pins 5, 6, 7, and 8 on the right side. The pinout table is as follows:

Pin	Label	Pin	Label
1	test	8	8
2	2	7	7
3	3	6	6
4	4	5	5

The pinout table is labeled "U?" and "ISL6236AIRZ-T". The component name "ISL6236AIRZ-T" is also written below the table. The circuit diagram shows two headers (P?) connected to a MOSFET (Q?) through a resistor (R? Res3 1K). The MOSFET is connected to ground. The component pinout table is for the ISL6236AIRZ-T and shows unconnected pins 1, 2, 3, and 4 on the left side, and pins 5, 6, 7, and 8 on the right side. The pinout table is as follows:

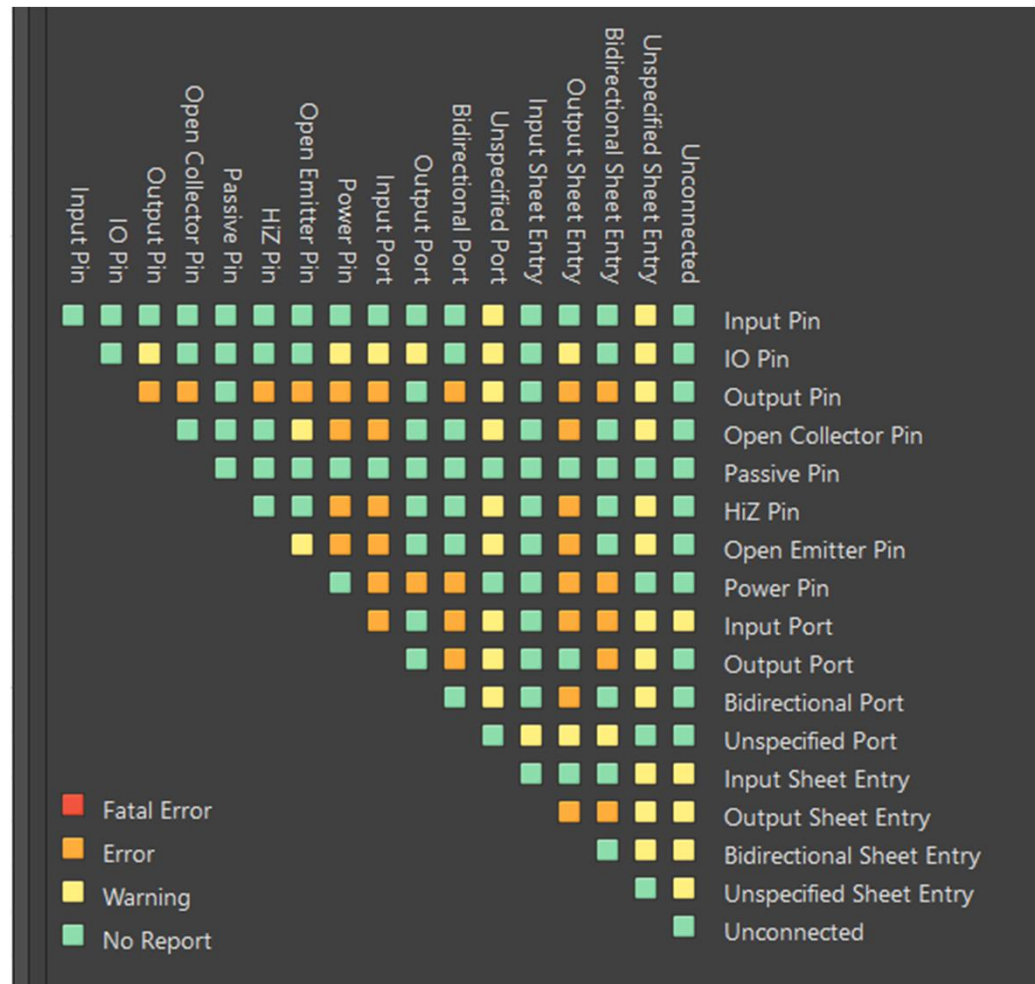
# Schematics Compile

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



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

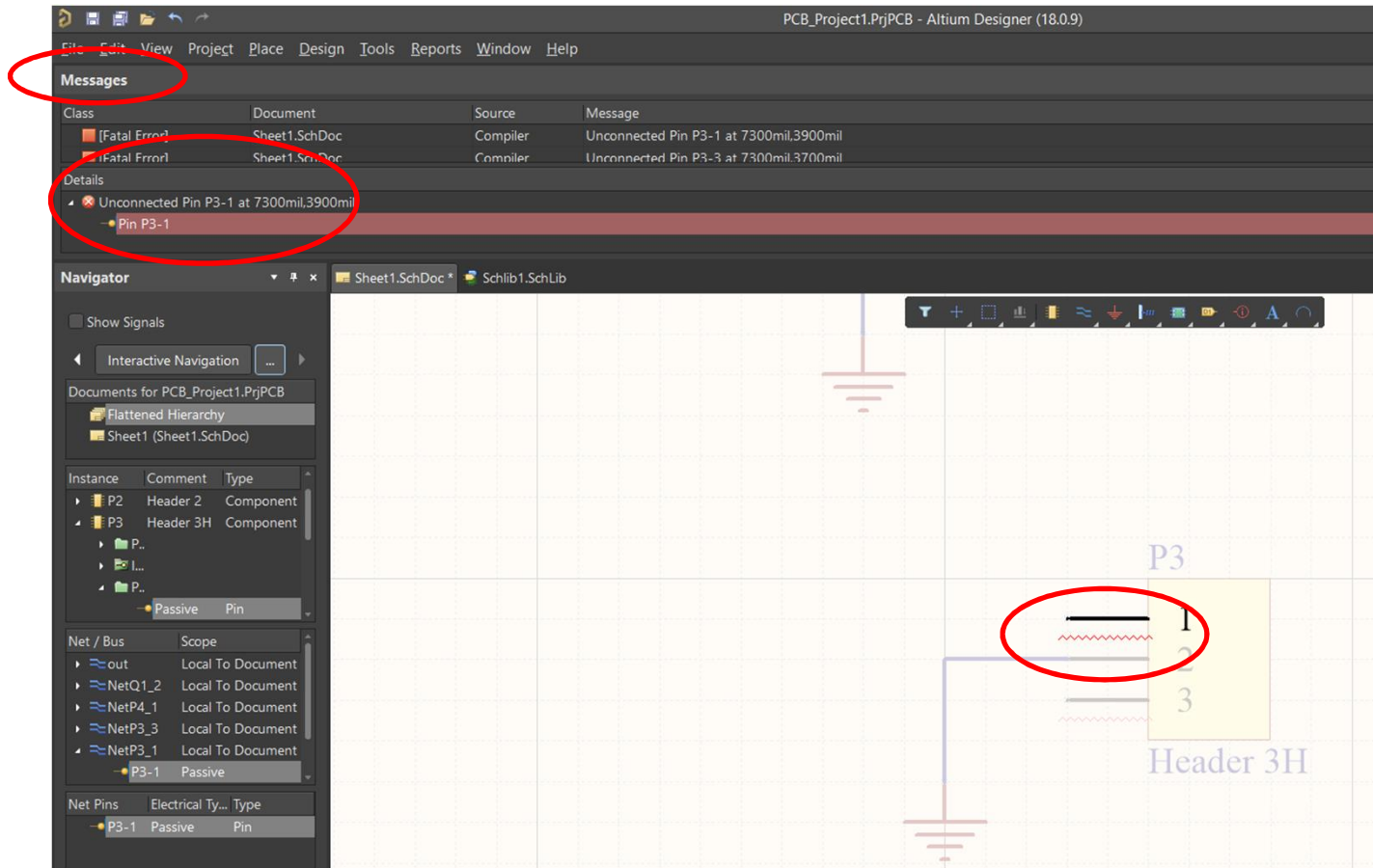
# Schematics Compile



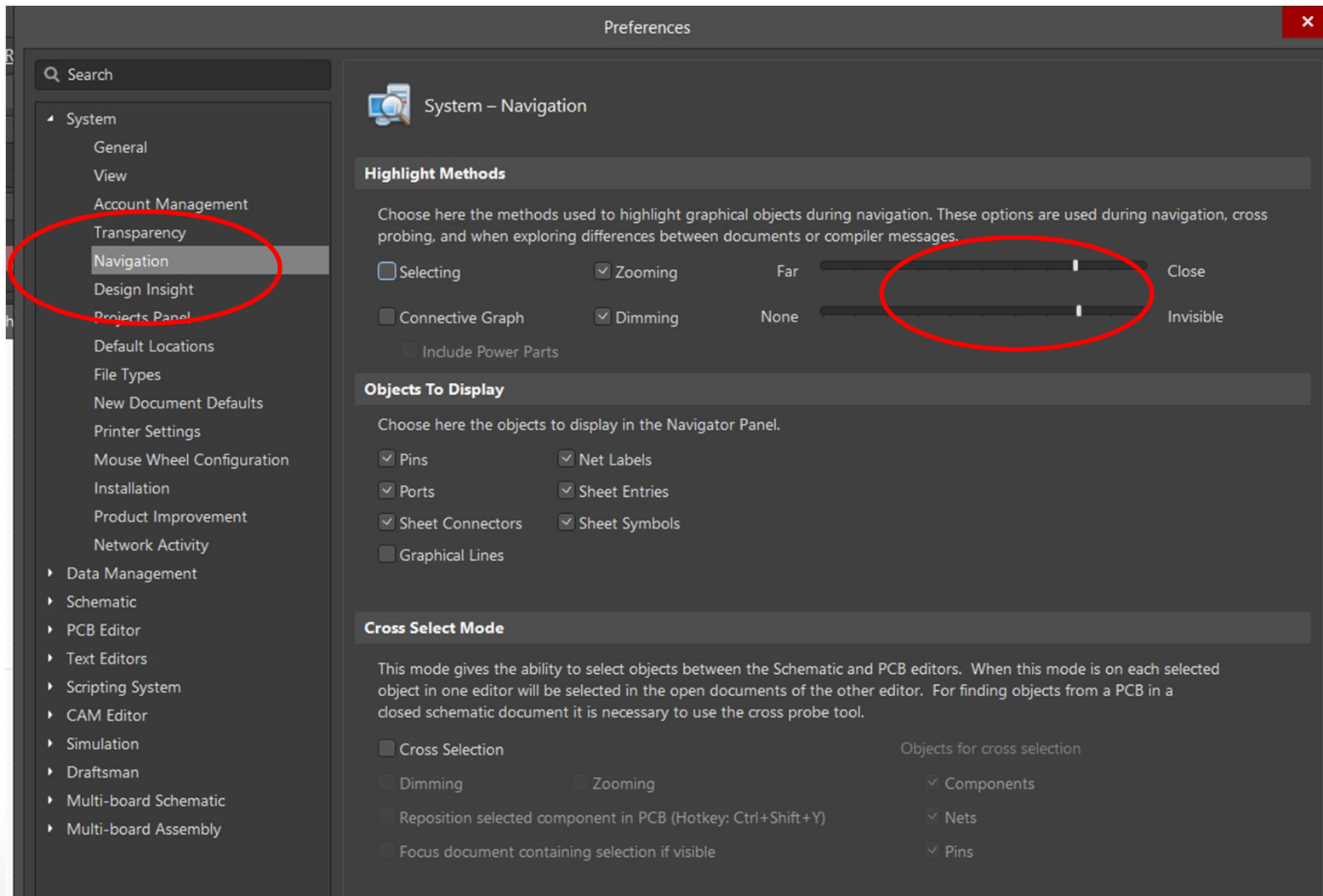
Con RIGHT CLICK posso cambiarli tutti in un a volta



# Schematics Compile



# Zoom su Errori



# Schematics Reports

The image shows the Altium Designer interface. The top menu bar has the 'Reports' menu highlighted with a red circle. A schematic diagram is visible in the background, featuring components P1, P4, and Res1. P1 and P4 are labeled as 'Header 2'. Res1 is a resistor with a value of 1K. The 'Bill of Materials For Project [PCB\_Project1.PrjPCB] (No PCB Document Selected)' dialog box is open in the foreground. It contains a table with the following data:

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

The dialog box also includes sections for 'Source Options', 'Supplier Options', 'Export Options', and 'Excel Options'. The 'Export Options' section shows 'File Format' set to 'Microsoft Excel Worksheet (\*.xls)'. The 'Excel Options' section shows 'Template' set to '<none>'. The 'Export...' button is visible at the bottom of the dialog.

# 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			

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

# Esempio / esercizio

