

**FIRST STEPS**

# **ALTIUM DESIGNER**

BY KOUKI FEDI



**ALTIUM  
DESIGNER**

# QU'EST CE QU'ON A BESOIN ?



**DATASHEET**

[Link1](#)



**FOOTPRINT & SYMBOLE**

[Link2](#)



**3D MODEL**

[LINK3](#)

# Ouvrir Altium

Home Page - Altium Designer (21.1.1)

File View Project Window Help

Projects Home Page

Search Project Group 1.DsnWrk

Discover Learning Center 5 What's New 3

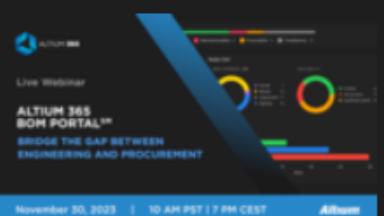
Altium Community Support Documentation

DESIGN SECRETS 0 of 225 WEBINARS IN YOUR REGION 0 CERTIFICATES 0 of 5 Training Catalog →

Welcome, fedi kouki

News & Updates

Updates Blog Video Webinar Filters: All

★ Webinar • Nov 23, 2023 •  Bridge the Gap Between Engineering and Procurement: Altium 365 BOM Portal<sup>SM</sup>  
Nov 30, 10:00 AM - 11:00 AM, PST, English  
Join our webinar and learn how to make informed parts decisions and mitigate supply chain risks with Altium 365 BOM Portal<sup>SM</sup>! Get a firsthand look at how this tool can help you improve your time to market and manage supply chain risks.  
[Register Now](#)

Open in Browser

Projects Navigator Panels

# Nouveau Projet

Home Page - Altium Designer (21.1.1)

**File** View Project Window Help

New

- Open... Ctrl+O
- Close Ctrl+F4
- Open Project...
- Open Project Group...
- Save Project Group As...
- Import Wizard
- Run Script...
- Recent Documents
- Recent Projects
- Recent Project Groups
- Exit Alt+F4

Project...

- Schematic
- PCB
- CAM Document
- Output Job File
- Component...

Library

- Design Center 5
- Script
- Mixed-Signal Simulation 3

Design Project Group

Altium Community

Support

Documentation

DESIGN SECRETS

0 of 225

WEBINARS IN YOUR REGION

0

CERTIFICATES:

0 of 5 Training Catalog →

Welcome, fedi kouki

News & Updates

Updates Blog Video Webinar

Filters: All ▾

★  Webinar • Nov 23, 2023 • [Watch](#)

**Bridge the Gap Between Engineering and Procurement: Altium 365 BOM Portal™**

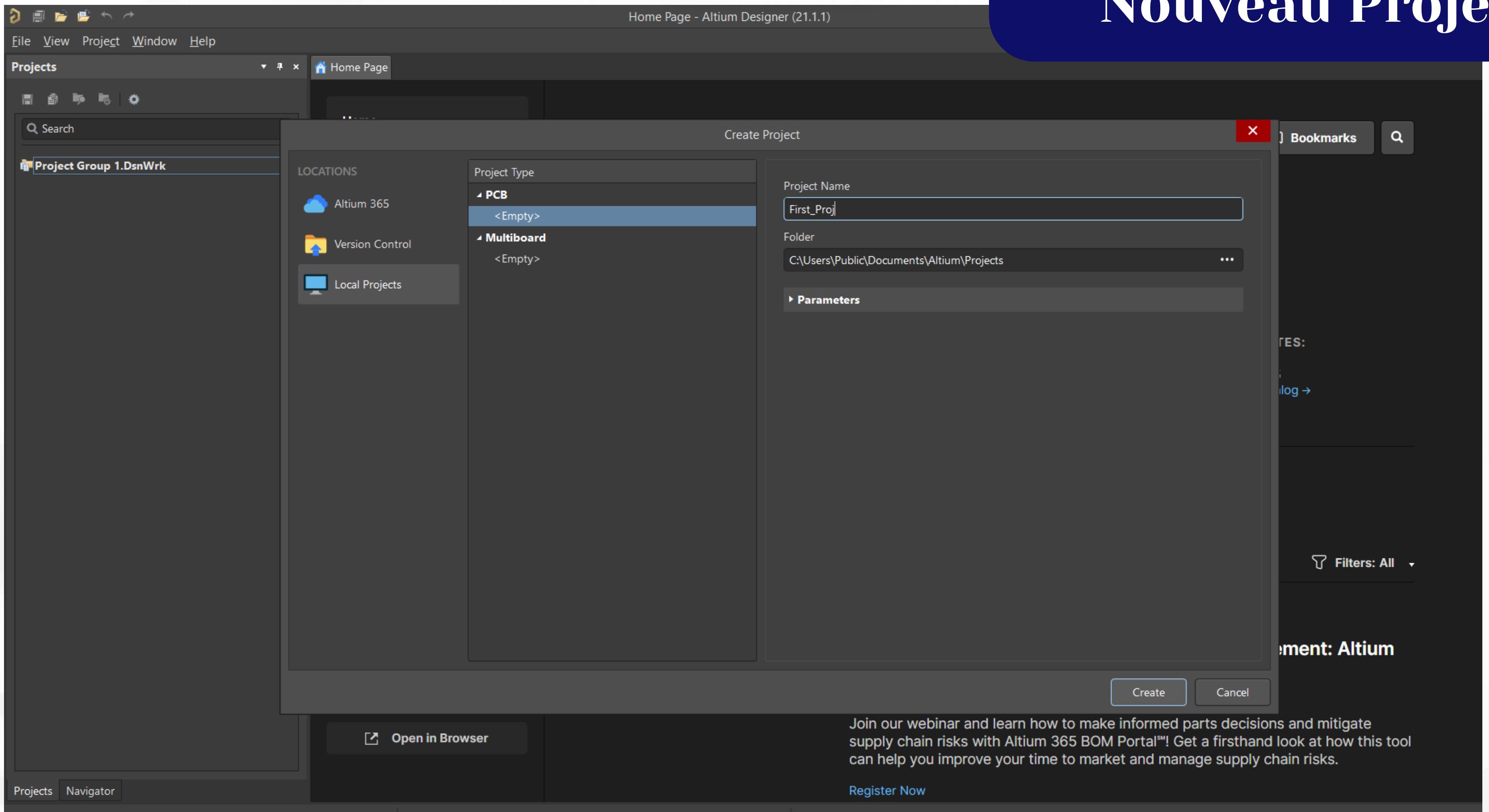
Nov 30, 10:00 AM - 11:00 AM, PST, English

Join our webinar and learn how to make informed parts decisions and mitigate supply chain risks with Altium 365 BOM Portal™! Get a firsthand look at how this tool can help you improve your time to market and manage supply chain risks.

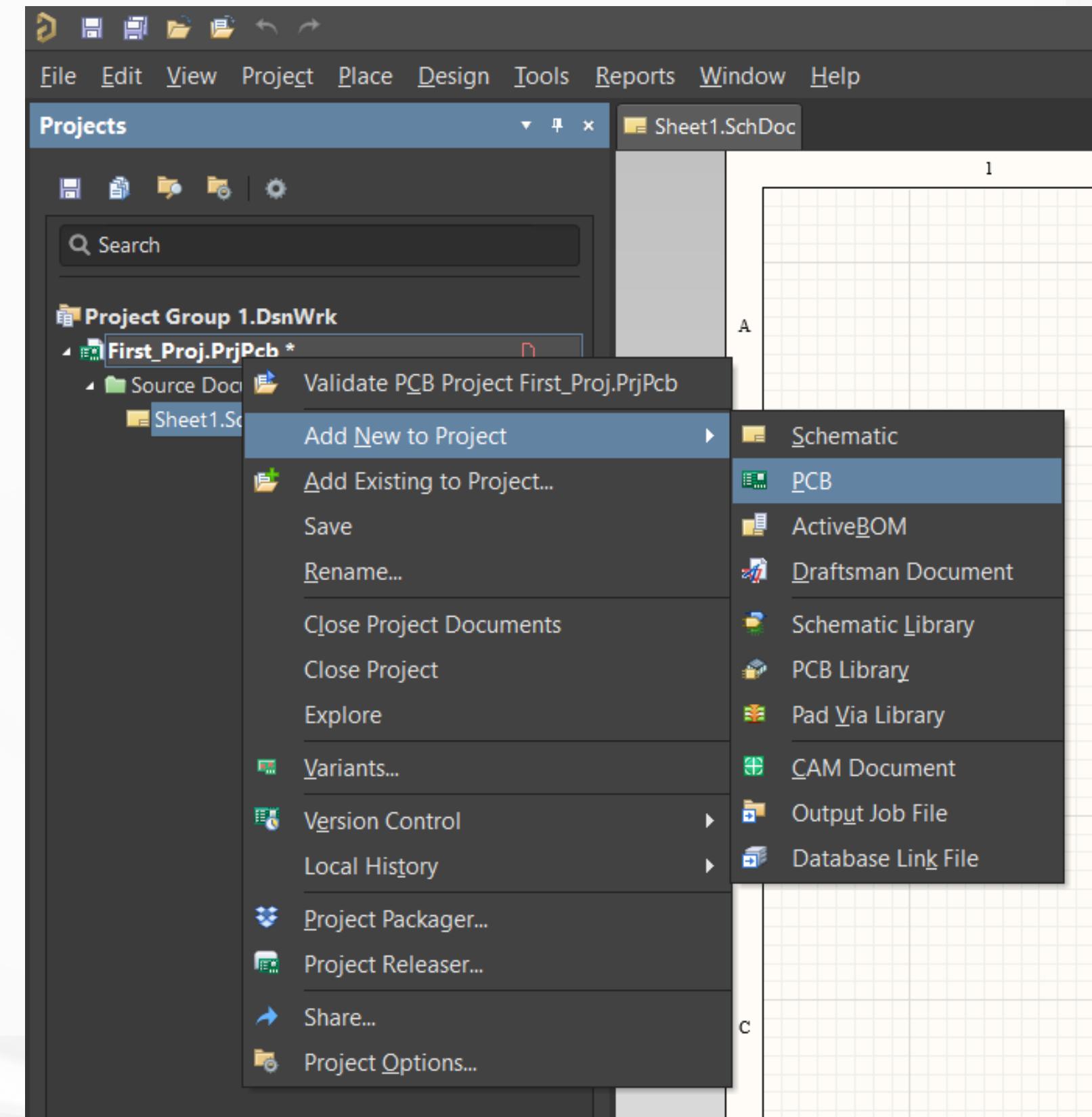
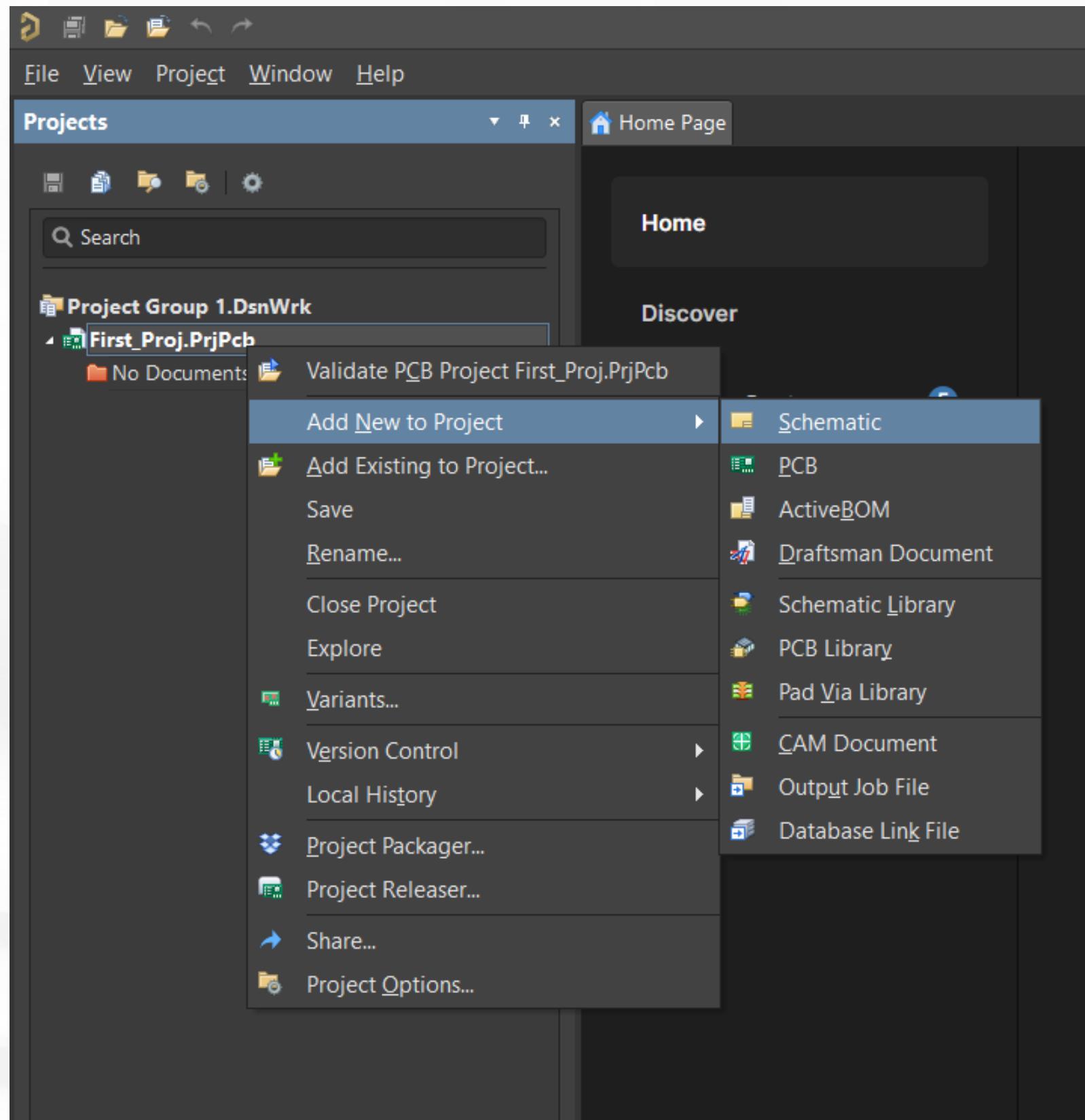
[Register Now](#)

Projects Navigator Panels

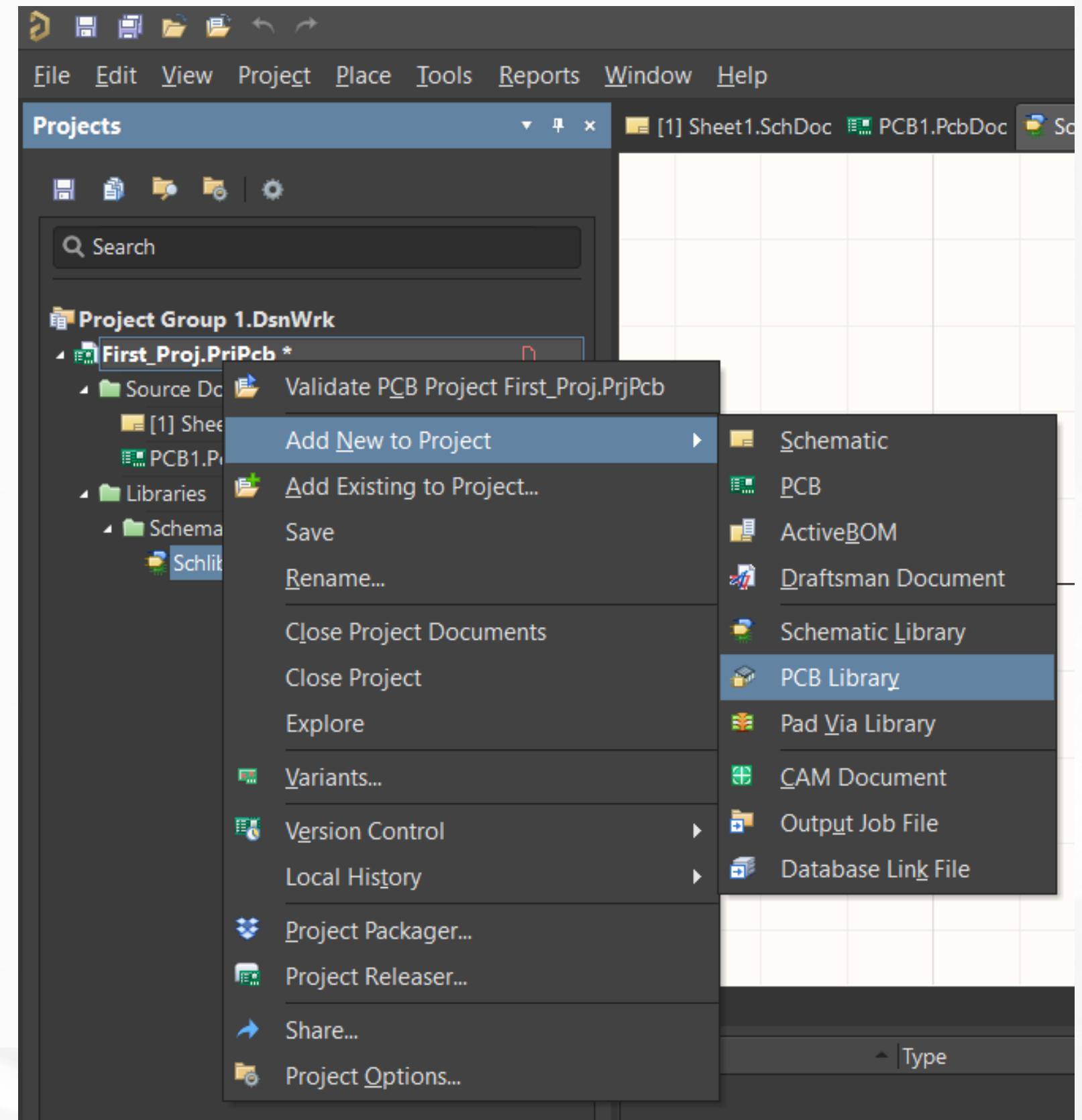
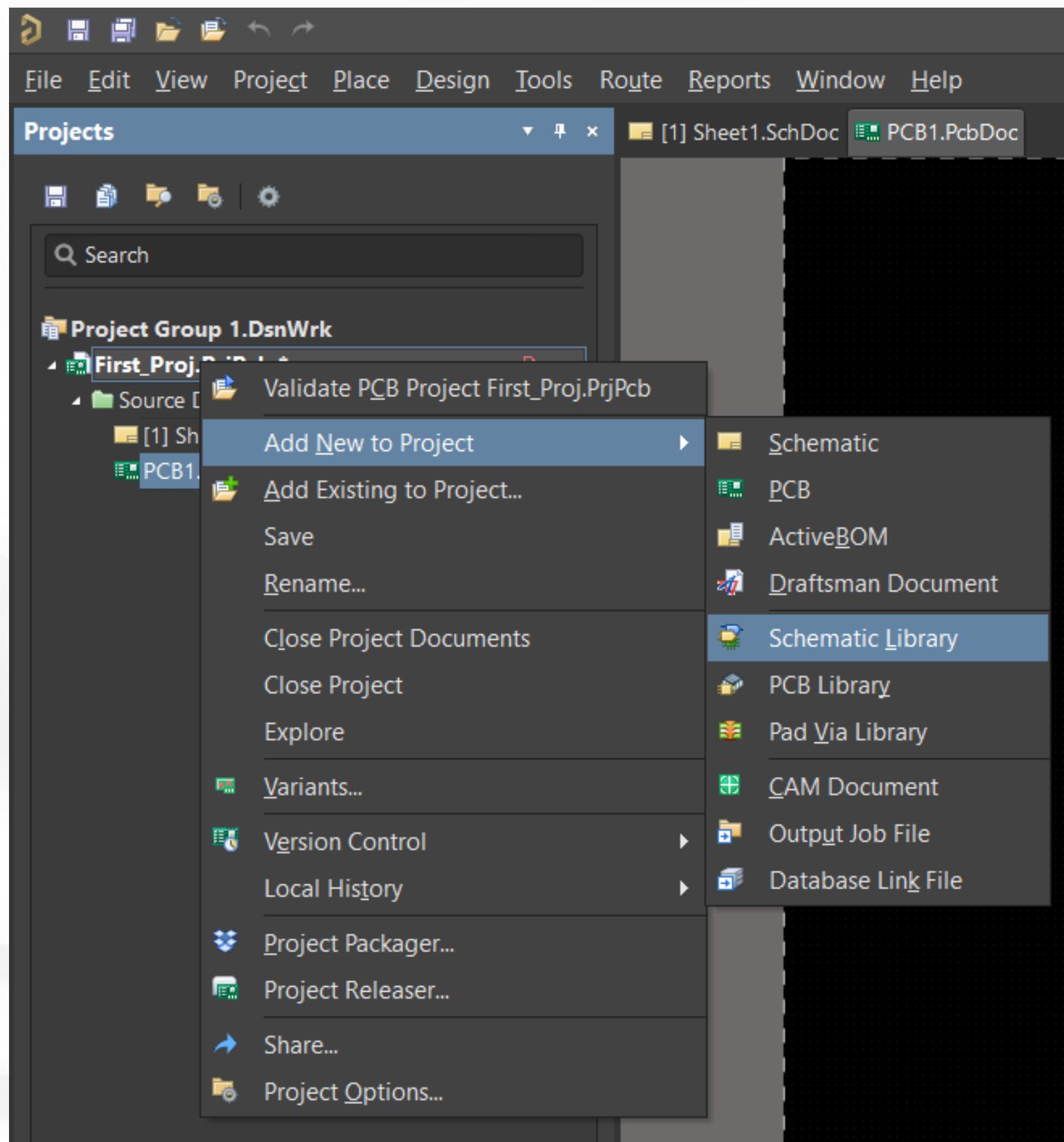
# Nouveau Projet



# creer les fichiers nécessaires



# creer les fichiers nécessaires



## Travail bien Structuré

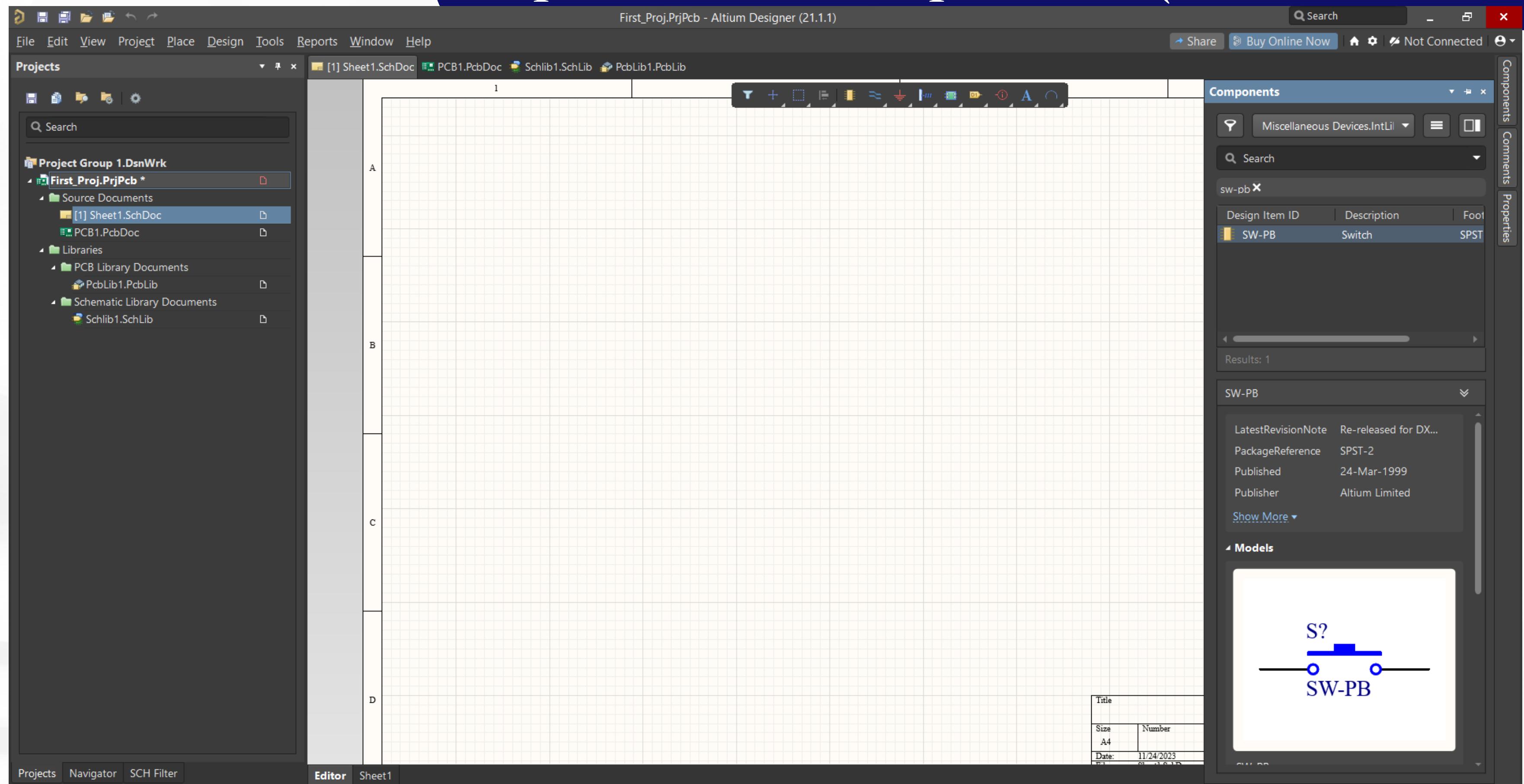
Pour que notre projet  
soit bien structurés

# POURQUOI AJOUTER LES SHEMATIC LIBRARY ET PCB LIBRARY

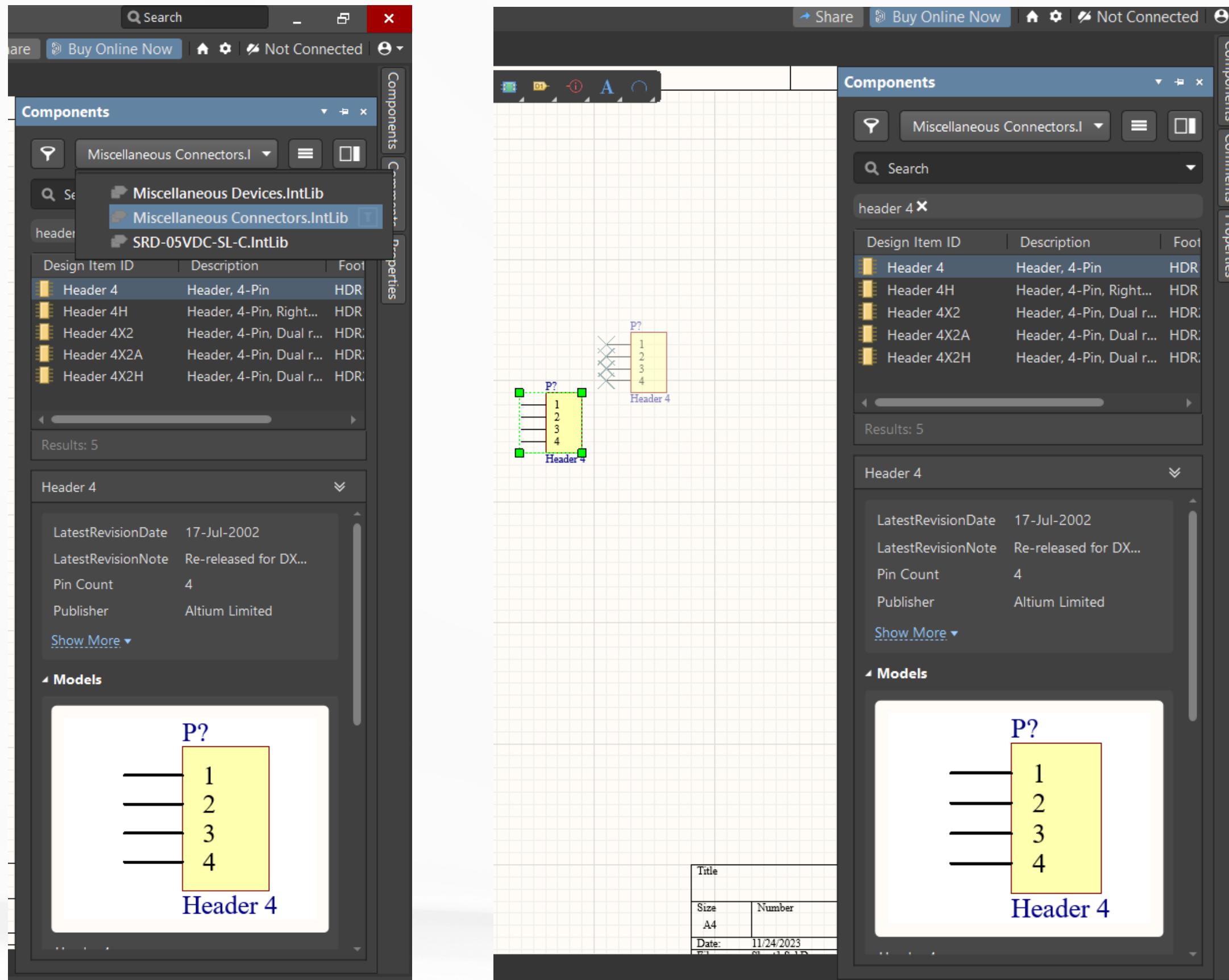
Intégrité

on garentie qu'il travail  
sur n'importe quelle  
autre machine

# importer des composants ( 1ère méthode )



# importer des composants ( 1ère méthode )



POUR INSÉRER UN COMPOSANT

**SELECTIONNER PUIS GLISSEZ  
DANS SCHEMATIC**

# importer des composants ( 2éme méthode )

The screenshot shows a web browser displaying the homepage of <https://www.snapeda.com/home/>. The page has a dark blue header with a navigation bar containing a back button, a refresh button, a lock icon, and the URL. To the right of the URL are settings and a font size selector. Below the header is a dark blue banner featuring a woman's face and the text "We're standing up for engineers! ⚡" followed by a "Learn more →" link. The main content area has a teal background. On the left, there's a "SnapMagic Search" logo and links for "Discover Parts" and "PCB Suppliers". A search bar in the center contains the word "diode". Below the search bar, a large white text area says "Let's make your design a snap." and "Focus on design with ready-to-use PCB footprints and schematic symbols". At the bottom, there are two buttons: "All Parts" and "PCB Suppliers", with "All Parts" being highlighted. A large white call-to-action button below them says "Search Over 25 Million Parts". At the very bottom, there's a line of text: "Or see examples: [USB Type C](#) [SMA Connector](#)".

# importer des composants ( 2éme méthode )

Manufacturer	Image	Part	Package	Availability	Avg. Price (USD)	Description	Data Available
		BAT54S-7-F	SOT-23		\$0.02	Schottky Diode 2xSeries 200mA 30V SOT23   Diodes Inc BAT54S-7-F	

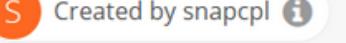
click 

**SnapMagic Search** Browse Parts PCB Suppliers Search Parts  Search InstaBuild  InstaPart  fedikk

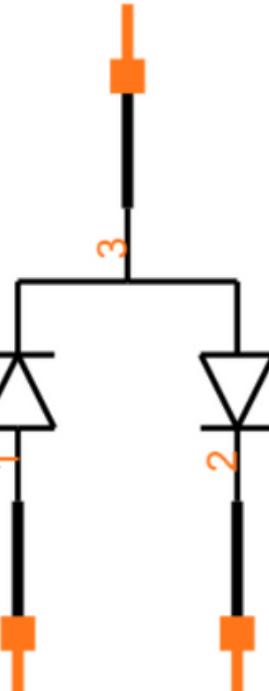
  
**BAT54S-7-F**  
Schottky Diode 2xSeries 200mA 30V  
SOT23 | Diodes Inc BAT54S-7-F  
Availability:  In Stock  
Package Type: SOT-23  
CAD Models: [Symbol](#), [Footprint](#), [3D Model](#)

 Add to Library  See Datasheet PDF  Download Symbol and Footprint  Auto-Import  Download Footprint

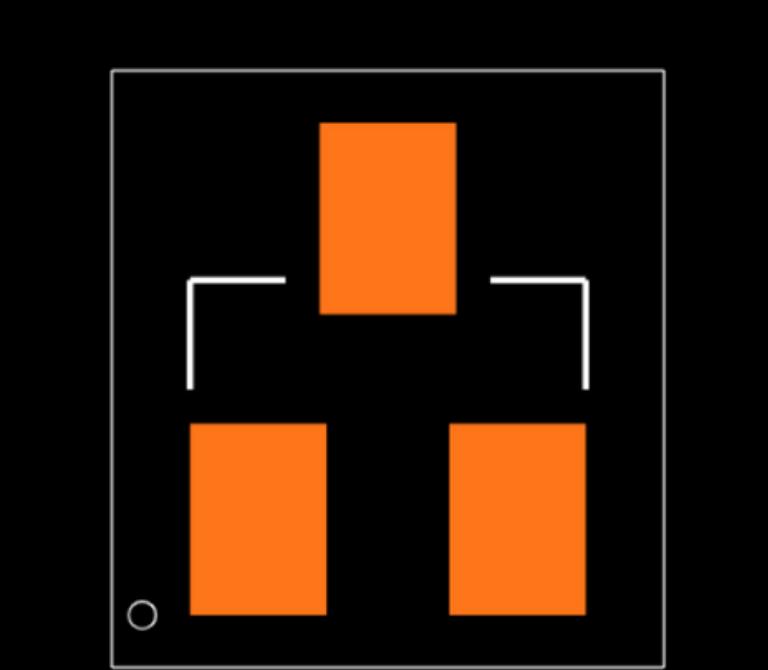
[https://www.snapeda.com/parts/BAT54S-7-F/Diodes Inc./view-part/?ref=search&t=diode#download-modal](https://www.snapeda.com/parts/BAT54S-7-F/Diodes%20Inc./view-part/?ref=search&t=diode#download-modal)

**Symbol**



**Footprint** 

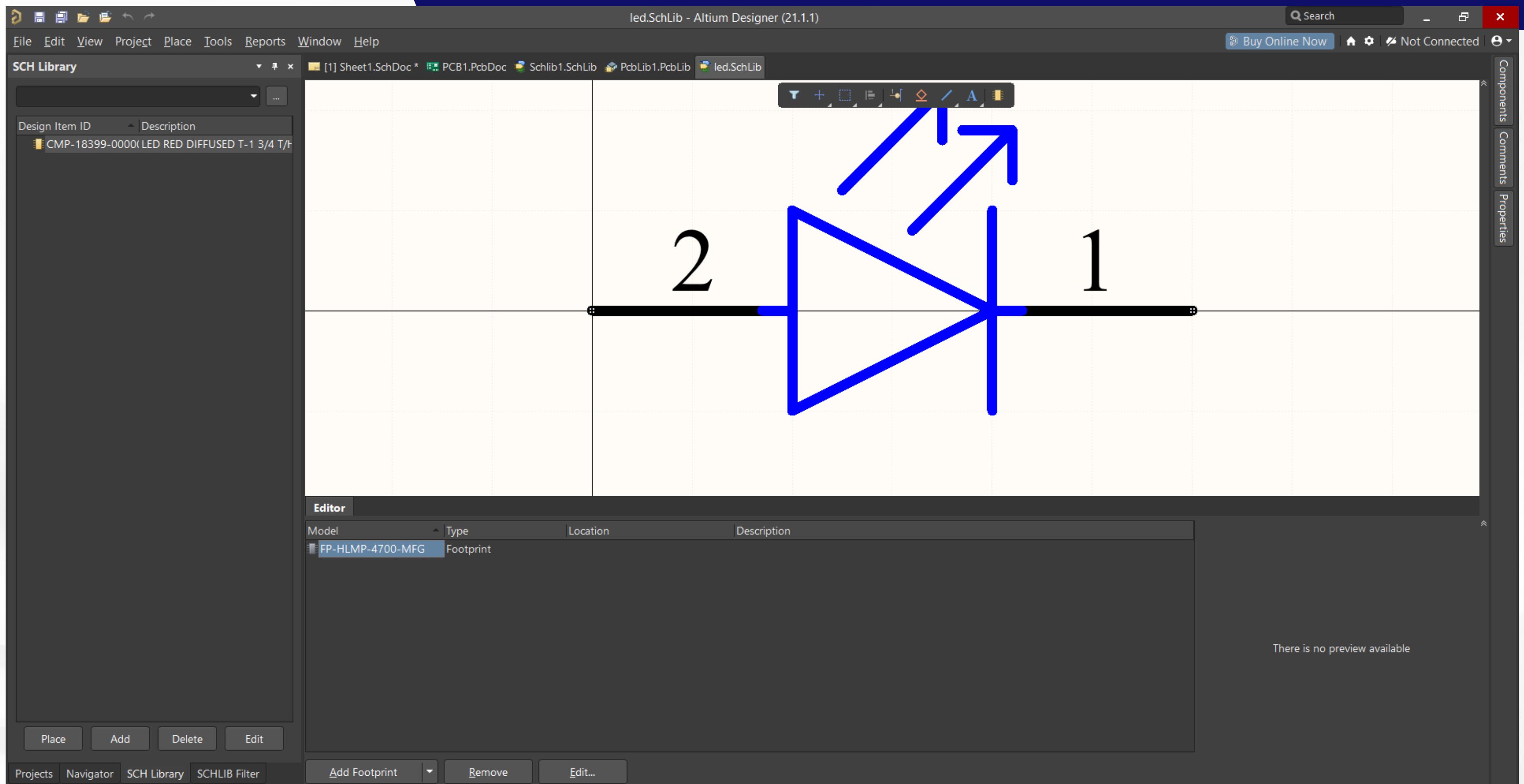


# importer des composants ( 2éme méthode )

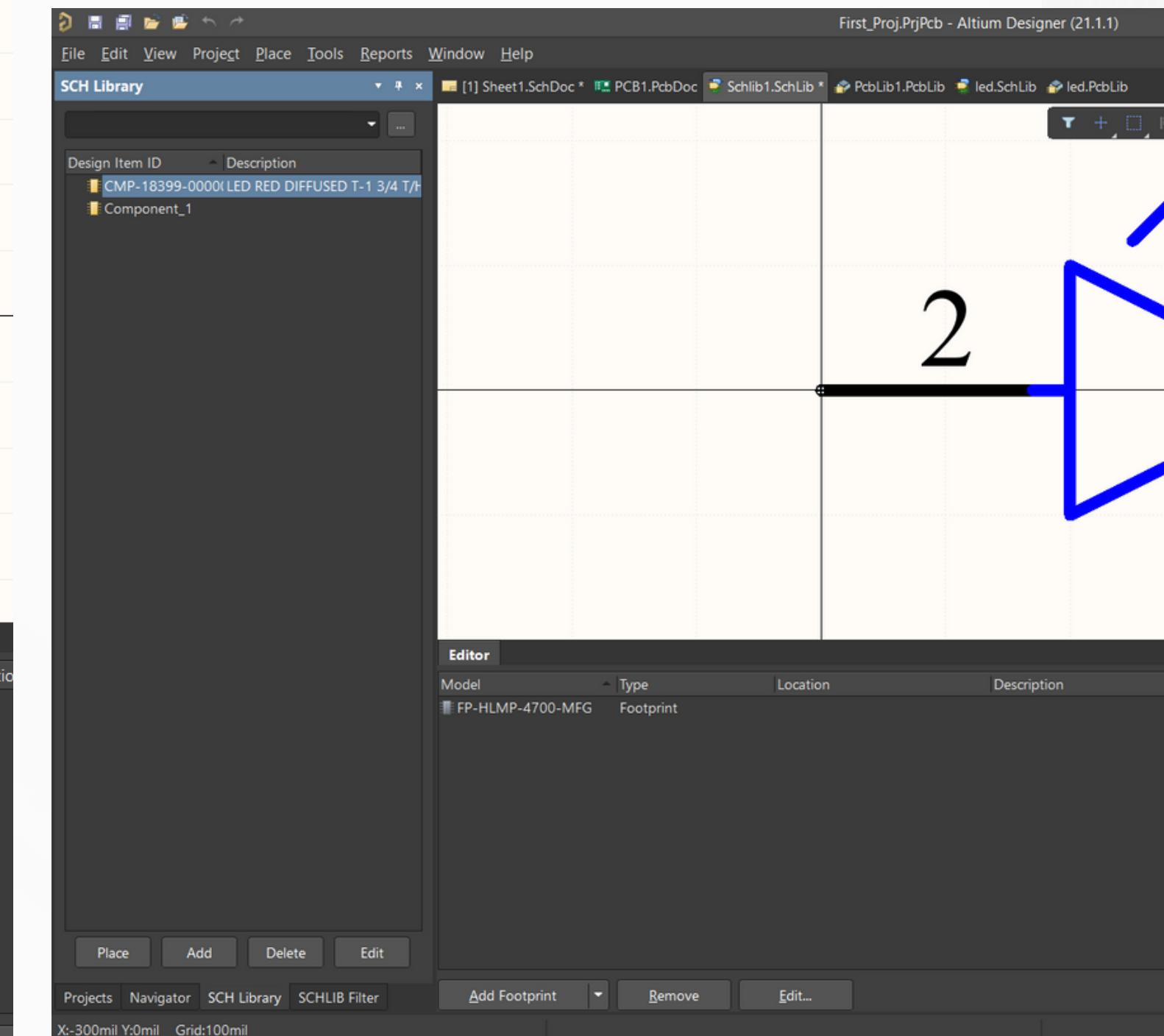
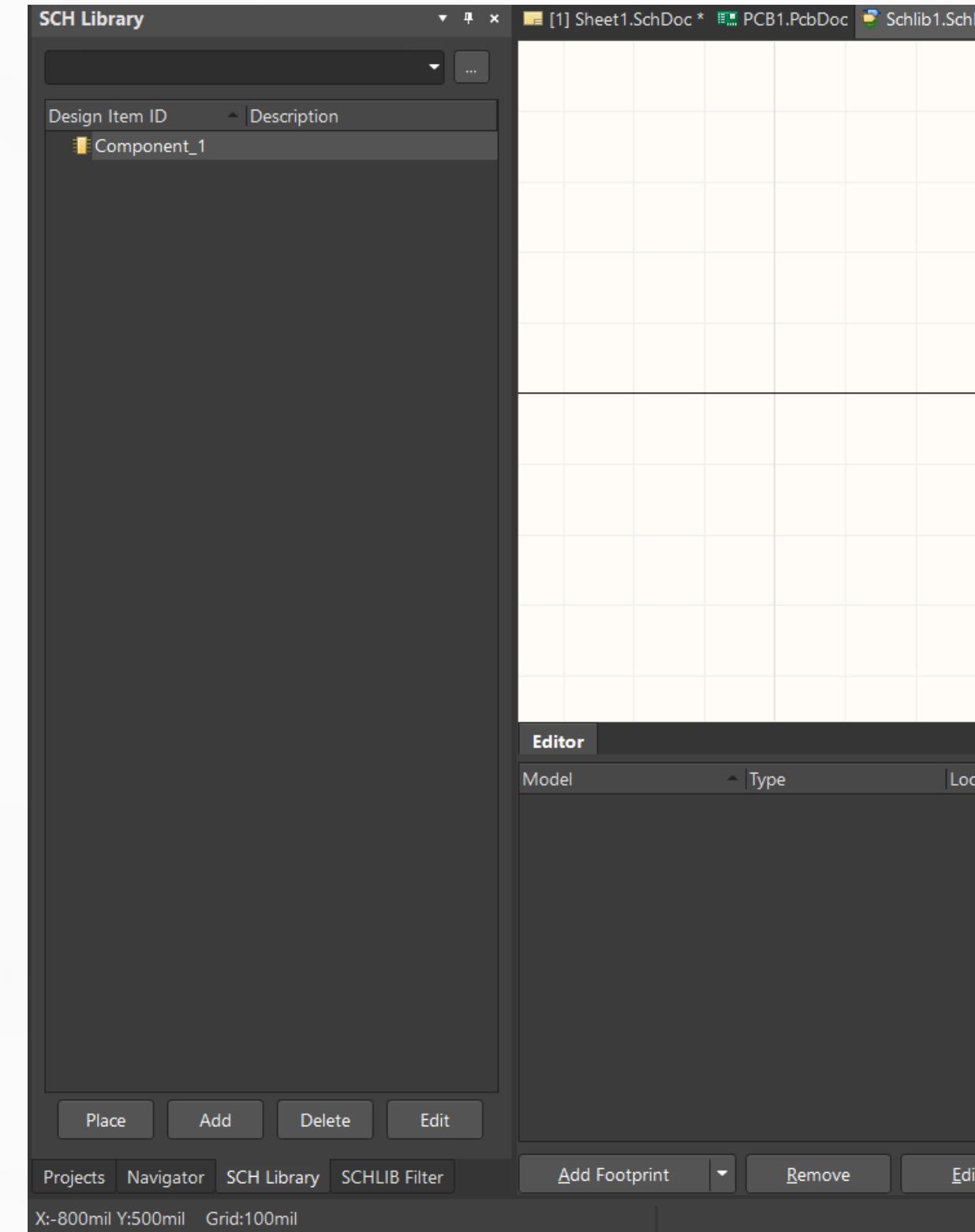
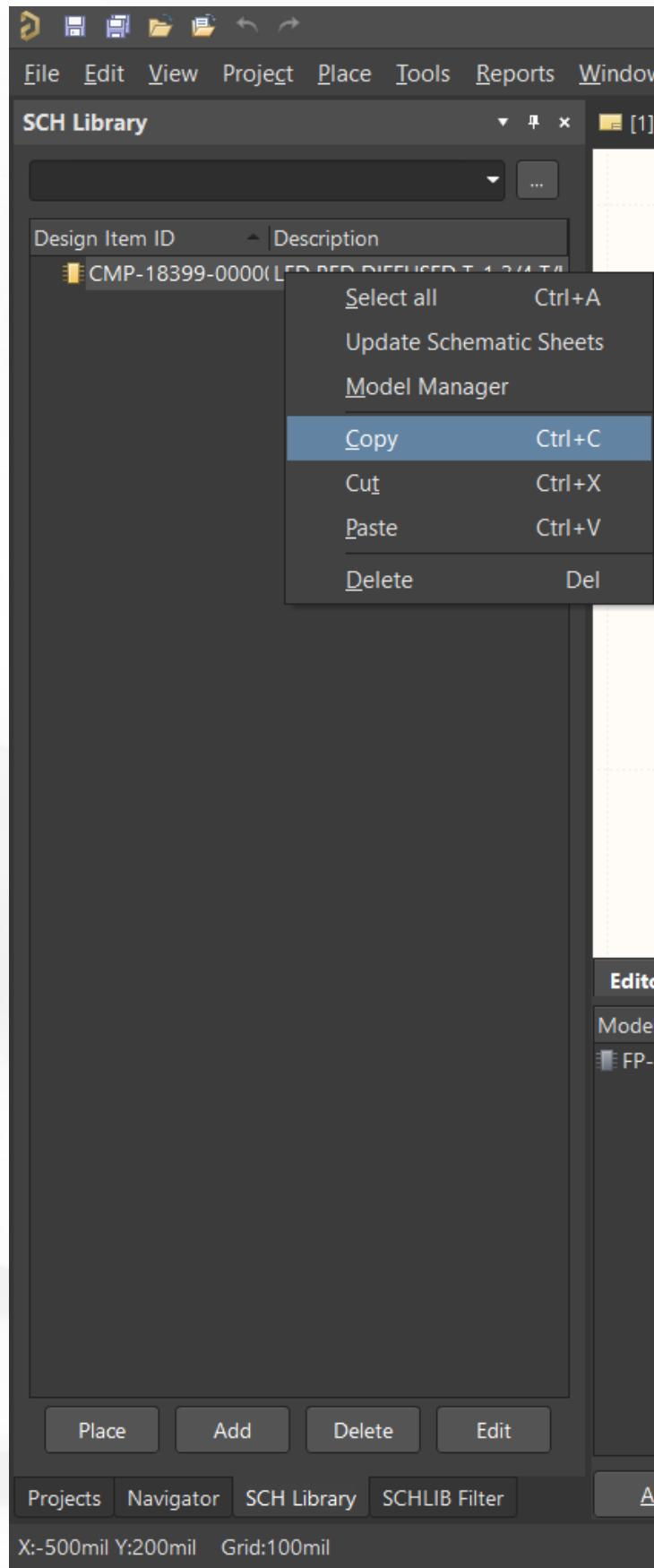
The image shows a software interface for component import. On the left, a 'Choose Download Format' dialog is open, listing various formats: Altium, Circuit Studio, CR-8000/5000 (Beta), DesignSpark PCB, DipTrace, EAGLE, Easy-PC (Beta), eCADSTAR (Beta), ExpressPCB Plus, and Autodesk Fusion 360. The 'Altium' option is highlighted with a brown background. A large blue arrow points from this dialog to a file explorer window on the right. The file explorer shows a folder structure under 'led': 'Téléchargements', 'Documents', 'Images', 'Musique', 'Vidéos', 'Projet\_finale', 'datasheet' (which is selected), 'présentation', and 'Captures d'écran'. Inside 'datasheet', there are three files: 'led' (Altium Integrated Library ...), 'led' (Altium PCB Library), and 'led' (Altium Schematic Library). The status bar at the bottom of the file explorer indicates '3 élément(s)'. To the right of the file explorer, the text 'DIFFERENT PARTIES' is displayed, followed by a table mapping file types to component parts:

Fichier library shematic	SYMBOLE
Fichier library PCB	FOOTPRINT+3D MODEL
Fichier compresseur	PCB+Shematic

# On ouvre le fichier led.schlib



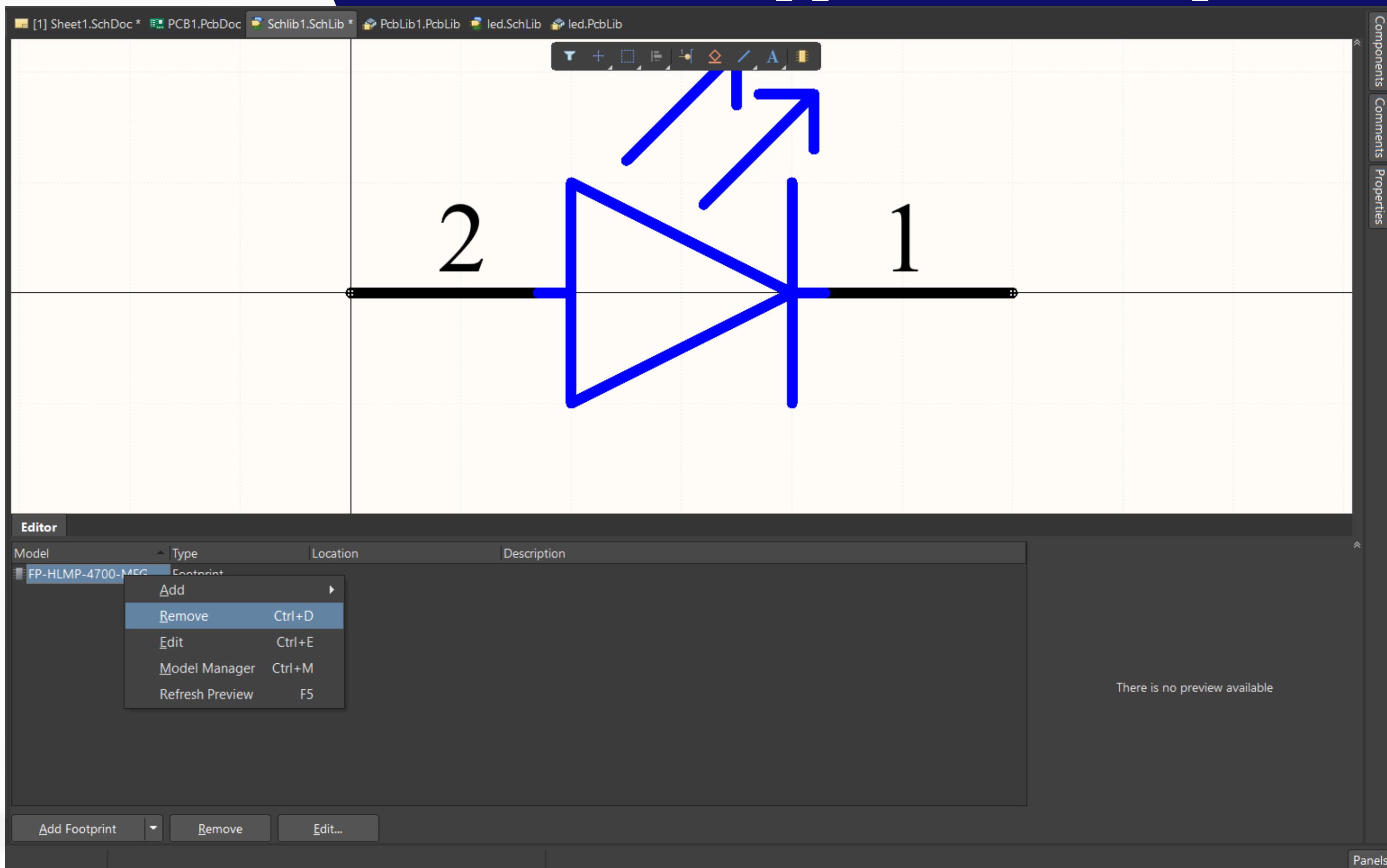
# copier le composant du led.schlib a shlib.shlib



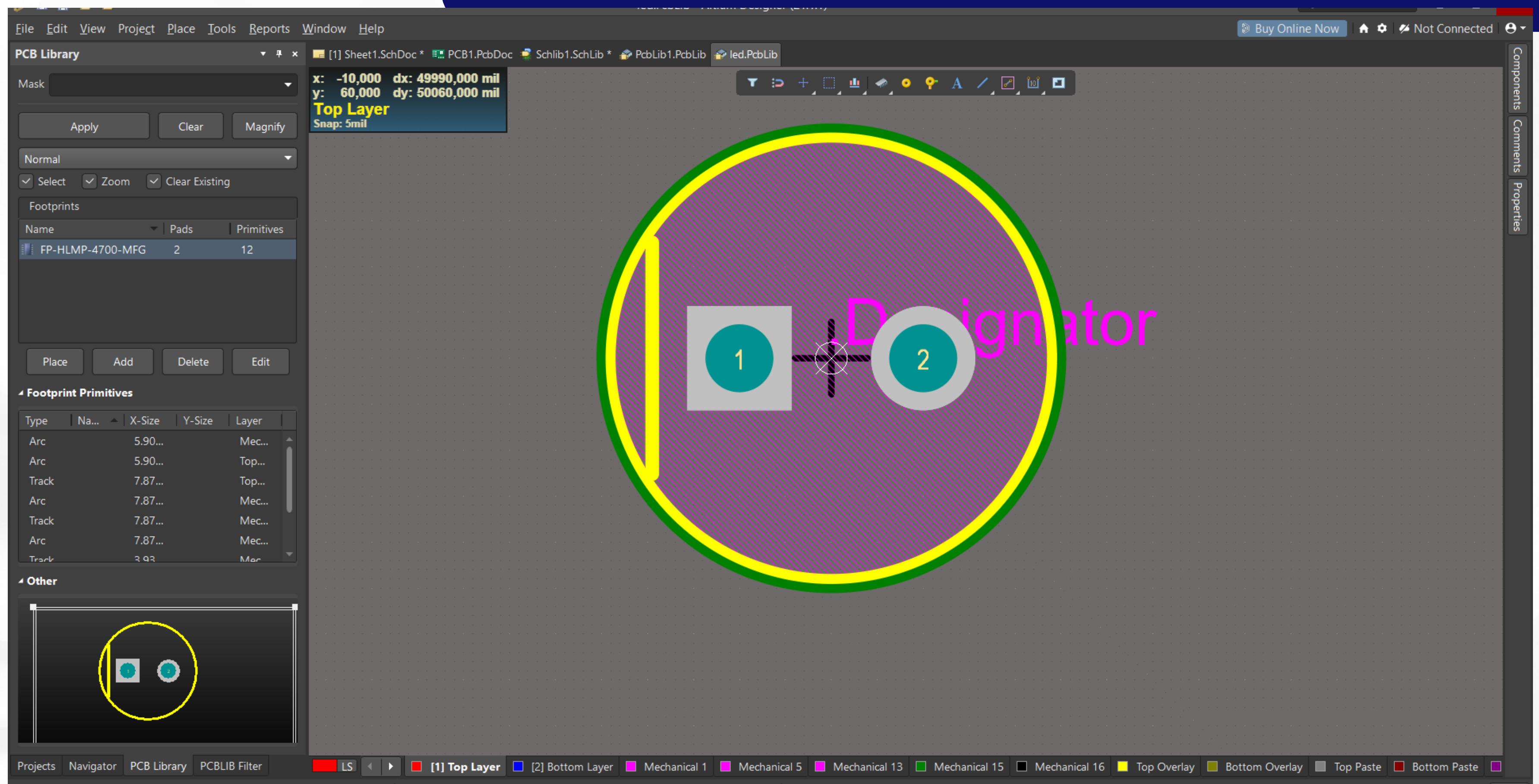
ouvrir schlib1.shlib

Ctrl+V

# On supprime le footprint

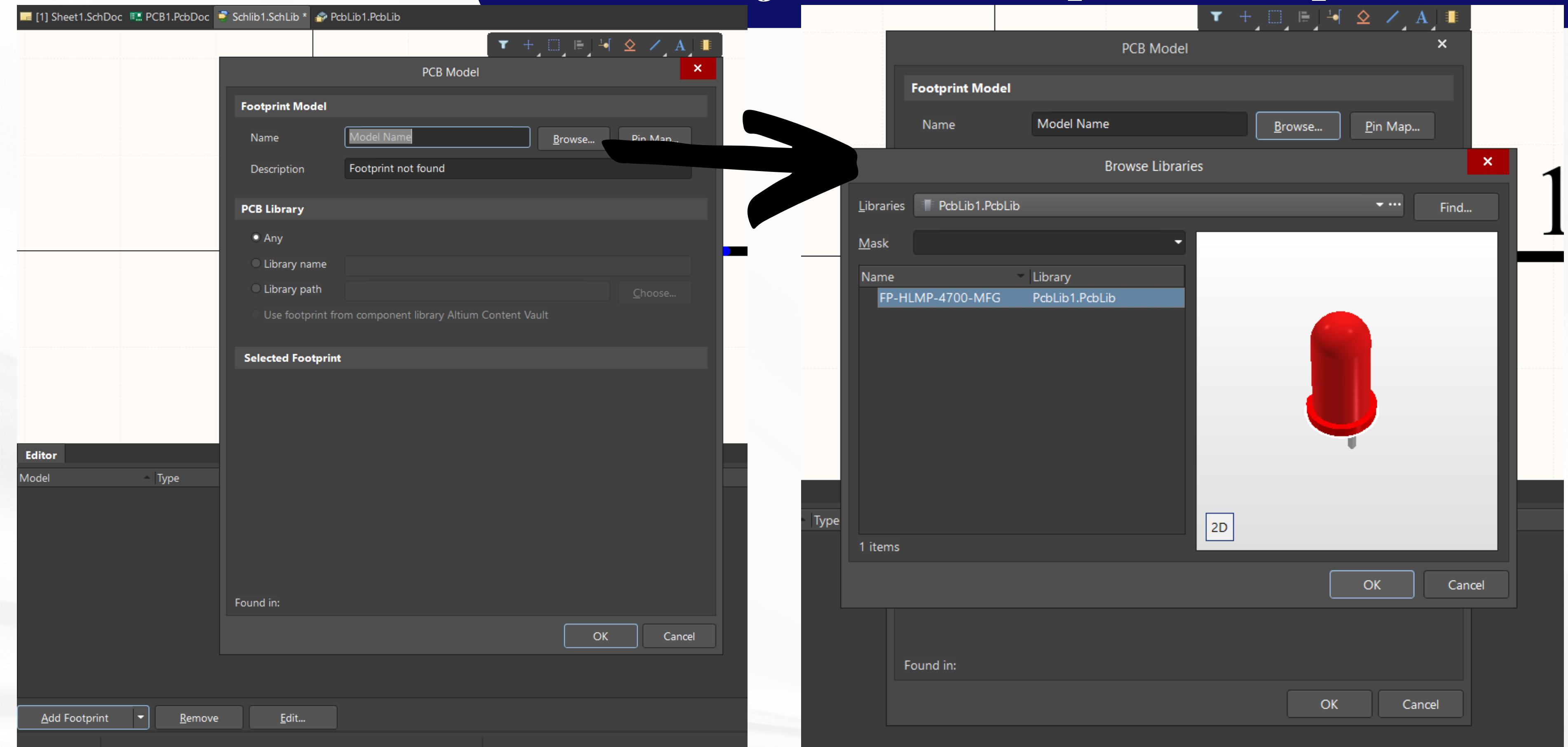


# On ouvre le fichier led.PCBLIB



**RECOPIER LE CONTENUE DU FICHIER DE LA  
MEME MANIÈRE QUE LE FICHIER SCHLIB**

# On ajoute le footprint adéquat



**ctrl + s**

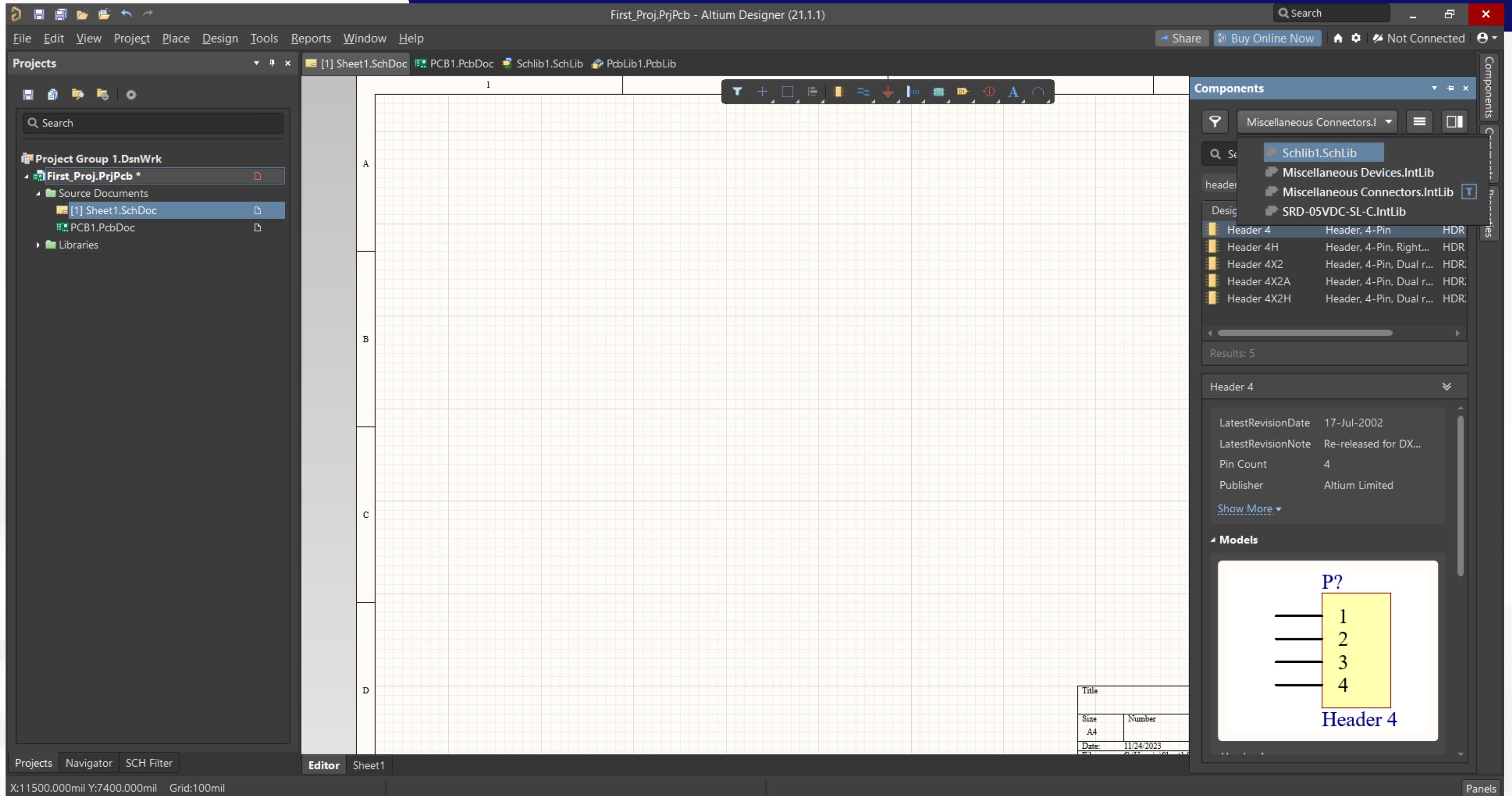
on oublie pas de  
sauvgarder

**ON RÉPETE  
CE  
PROCESSUS  
POUR LES  
AUTRES  
COMPOSANTS**

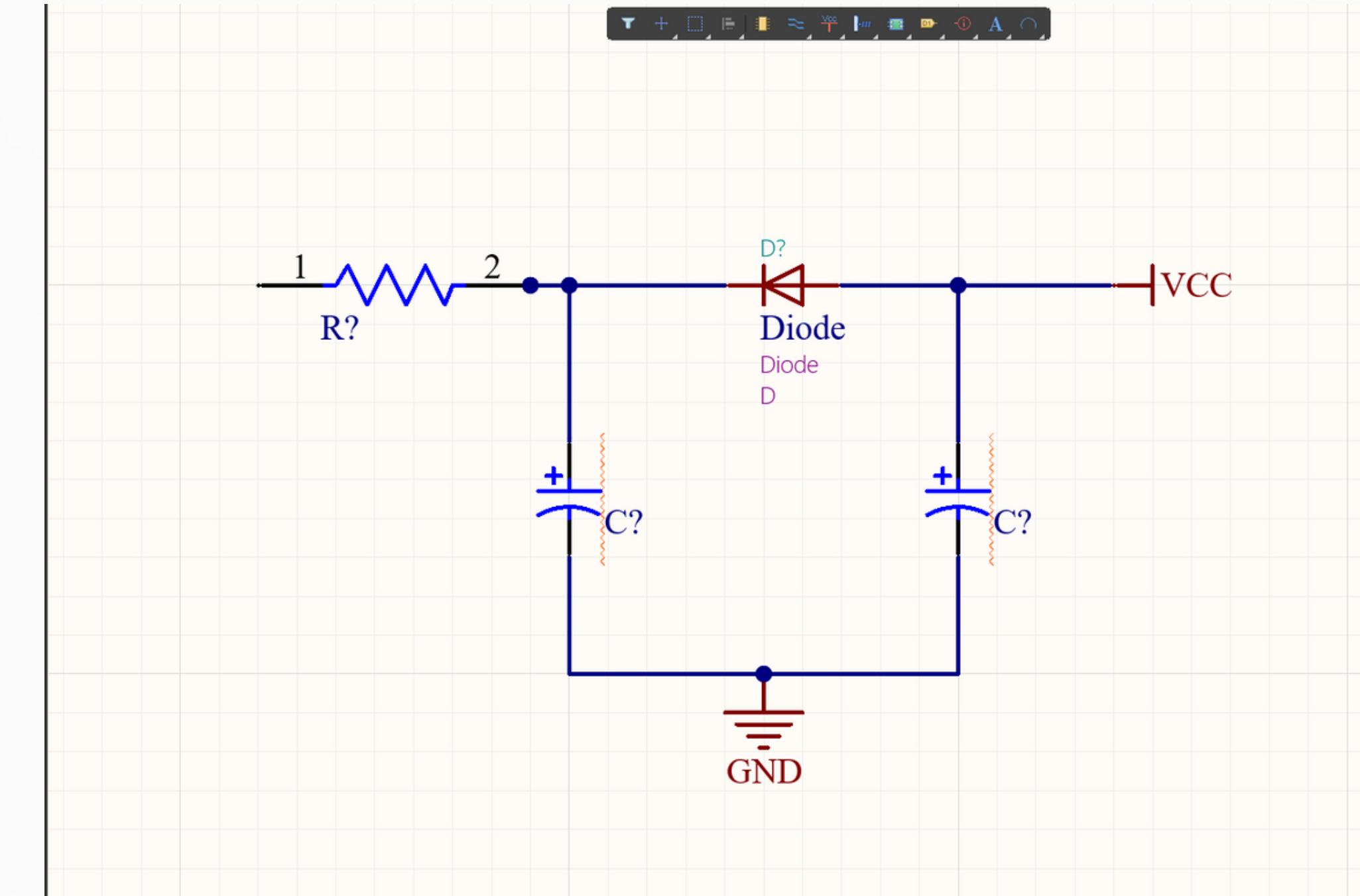
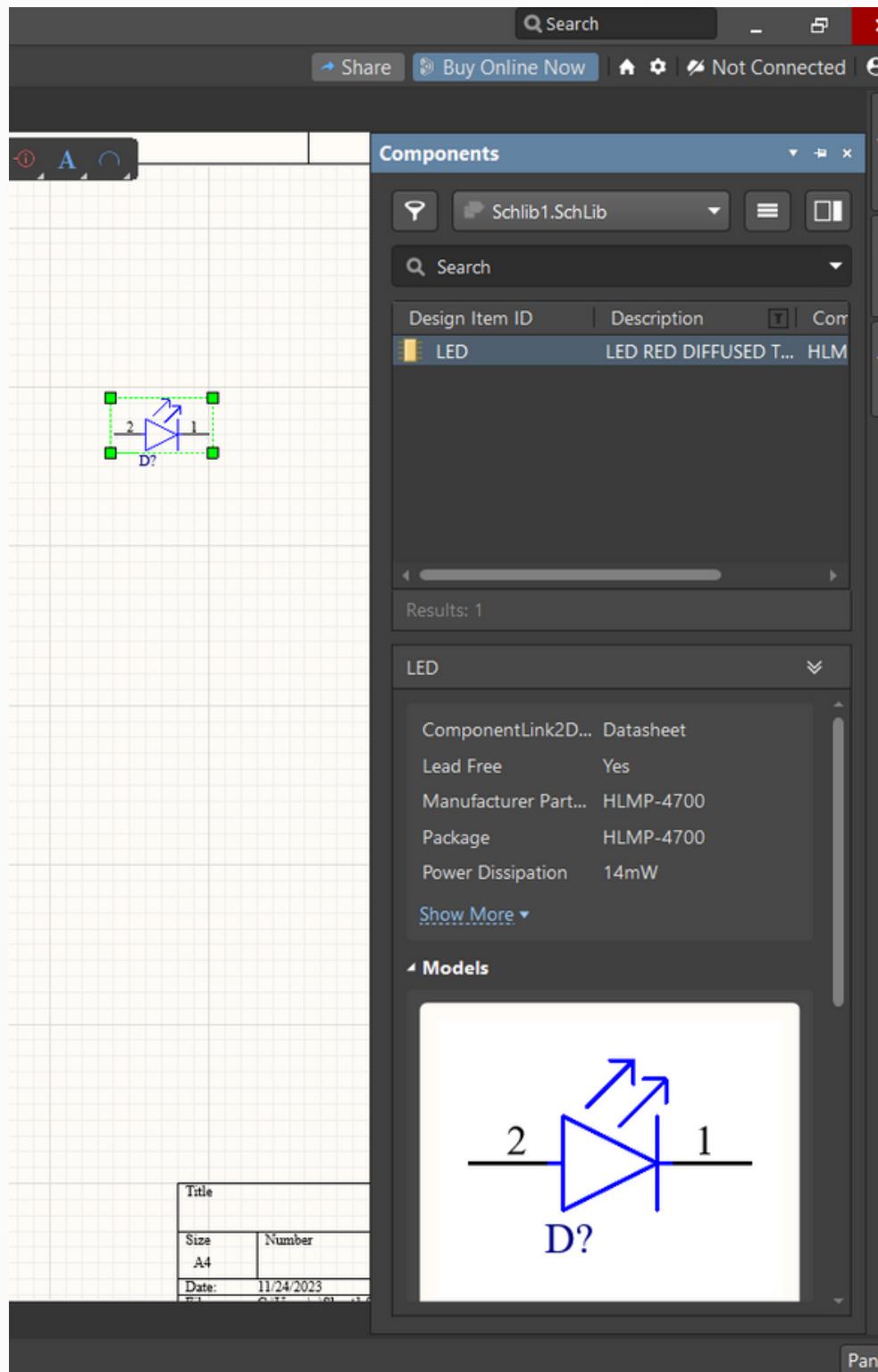
**rutilisibilité**

on peut faire une seule  
bibliothéque et on  
l'utilisera a chaque fois  
just il suffit de copier le  
contenu de fichier  
.schlib et .pcb

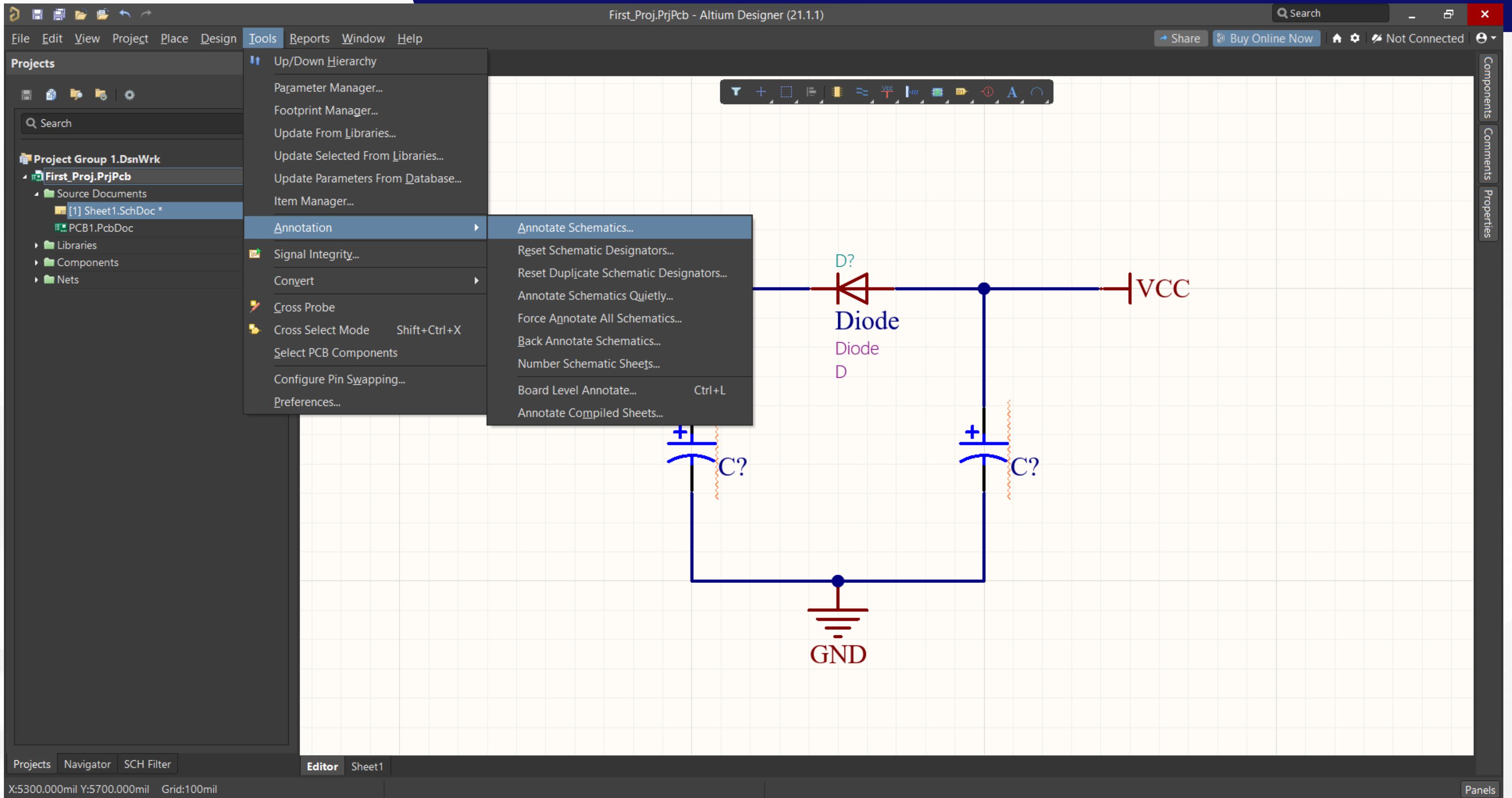
# ajouter nos composants dans schematic



# ajouter nos composants dans shematic



# Annotation du Circuit

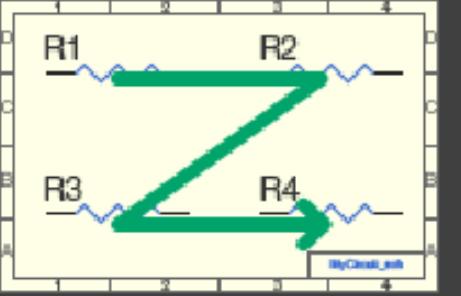


# cliquer update changes list puis accepter

Annotate X

### Schematic Annotation Configuration

**Order of Processing**: Across Then Down



**Matching Options**: Complete Existing Packages: None

**Component Parameter**: Strictly

- AC Ripple Current
- AE Capacitor Case
- Capacitance
- Capacitance Tolerance
- Capacitance Tolerance ±
- Capacitor Case Style
- Capacitor Mounting
- Capacitor Terminals
- Case Code (Imperial)
- Case Code (Metric)
- Case/Package
- Comment

**Process Location of**: Designator

**Replace Sub-Parts**: Off

**Schematic Sheets To Annotate**:

Schematic Sheet	Annotation Scope	Order	Start Index	Suffix
Sheet1.SchDoc	All	0	1	

### Proposed Change List

Current	Proposed	Location of Part		
<input checked="" type="checkbox"/> Designator	<input checked="" type="checkbox"/> Sub	Designator	Sub	Schematic Sheet
C?		C?		Sheet1.SchDoc
C?		C?		Sheet1.SchDoc
D?		D?		Sheet1.SchDoc
R?		R?		Sheet1.SchDoc

### Annotation Summary

Annotation is enabled for all schematic documents. Parts will be matched using 2 parameters, all of which will be strictly matched. (Under strict matching, parts will only be matched together if they all have the same parameters and parameter values, with respect to the matching criteria. Disabling this will extend the semantics slightly by allowing parts which do not have the specified parameters to be matched together.) Existing packages will not be completed. All new parts will be put into new packages.

All On All Off Update Changes List Reset All Back Annotate Accept Changes (Create ECO) Close

# Annotation du Circuit ( press validate changes)

Annotate

Schematic Annotation Configuration

Proposed Change List

Engineering Change Order

Modifications

Enable	Action	Affected Object	Affected Document	Status	Check	Done	Message
✓	Annotate Component(4)	C? -> C1 C? -> C2 D? -> D1 R? -> R1	In In In In	Sheet1.SchDoc Sheet1.SchDoc Sheet1.SchDoc Sheet1.SchDoc			

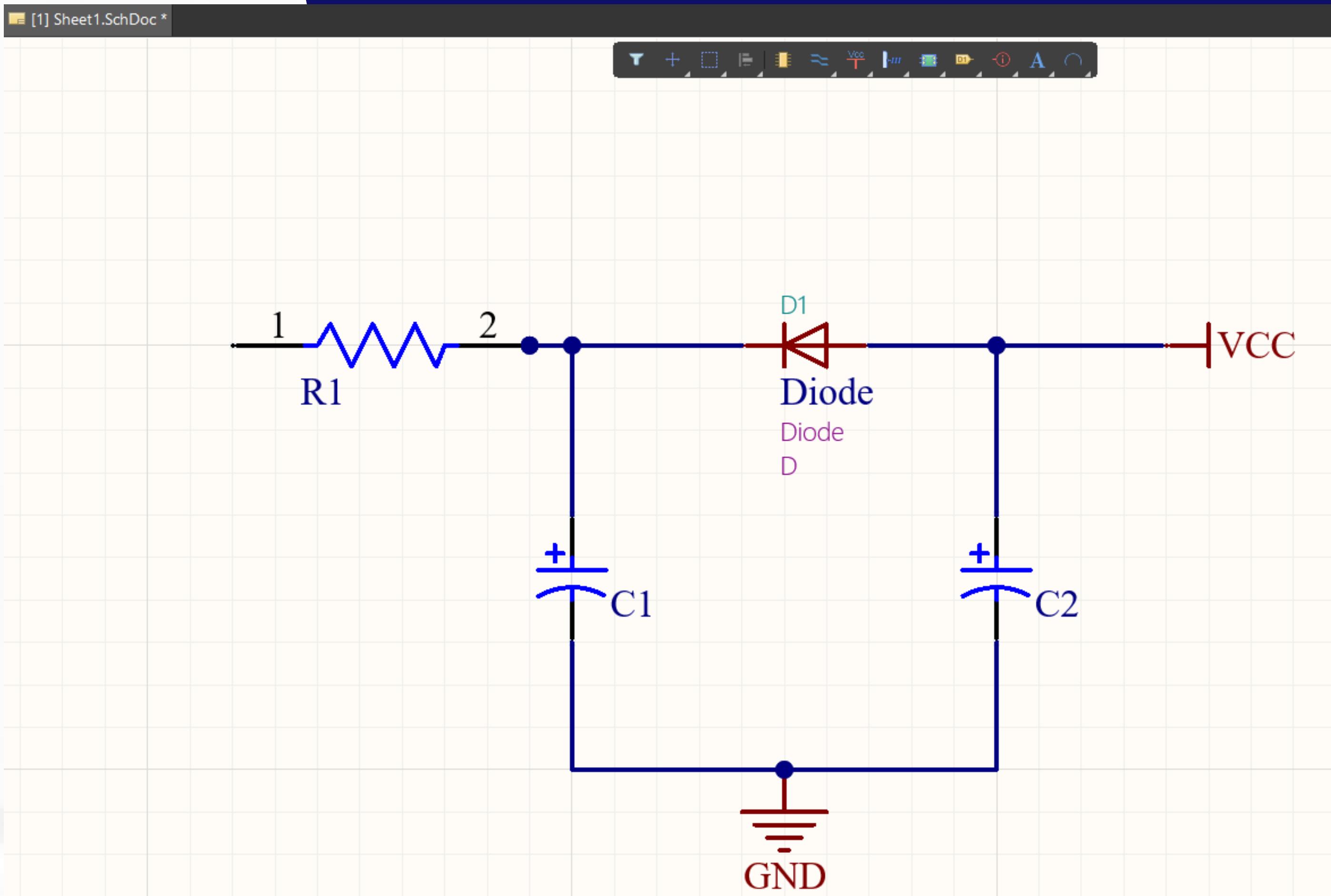
Validate Changes   Execute Changes   Report Changes...    Only Show Errors

All On   All Off   Update Changes List   Reset All   Back Annotate   Accept Changes (Create ECO)

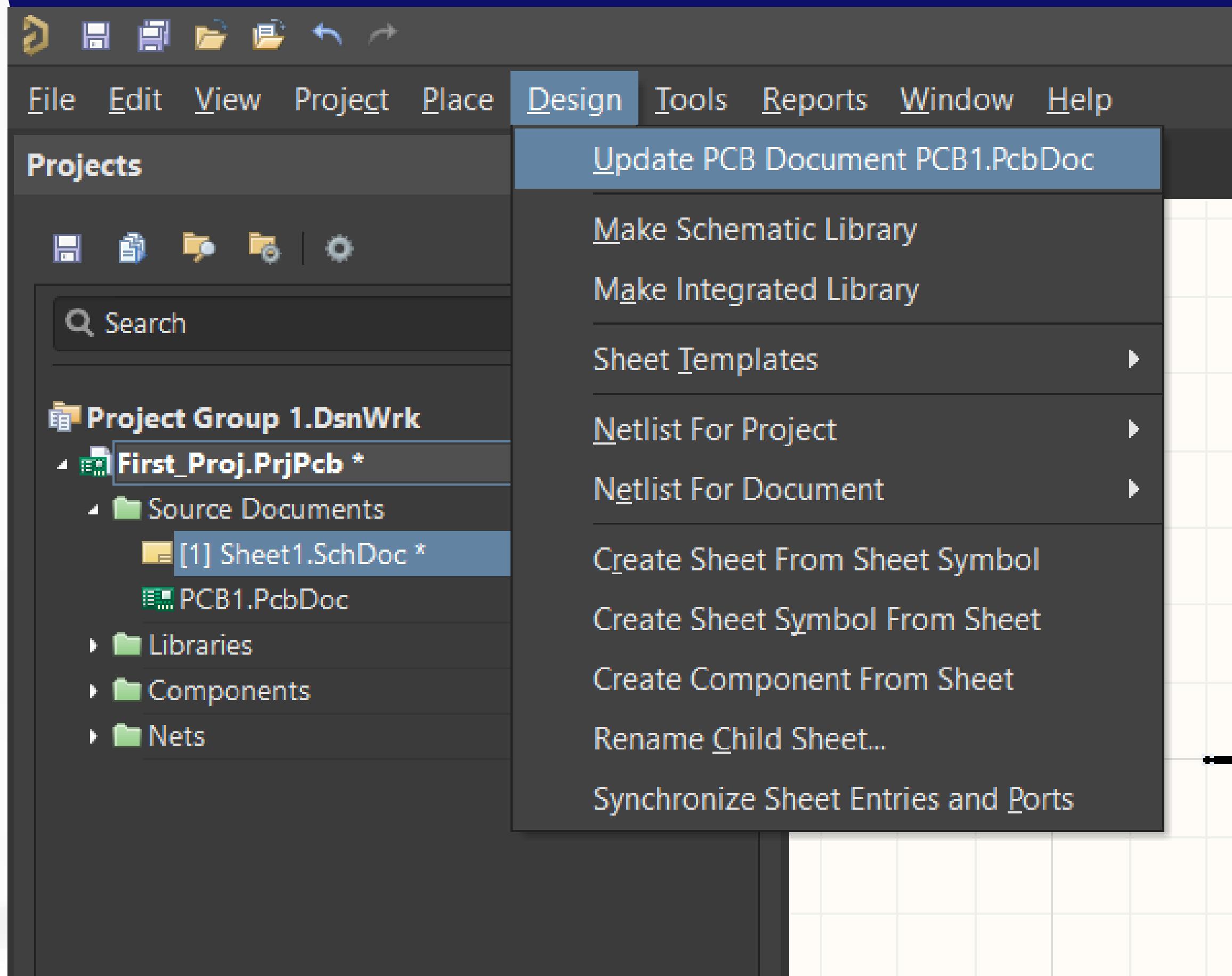
Close

Close

# Circuit annoté



# Creation du carte PCB



# Annotation du Circuit ( press validate changes)

Engineering Change Order

Modifications				Status		
Enable	Action	Affected Object	Affected Document	Check	Done	Message
	Add Components(4)	C1 C2 D1 R1	To To To To	PCB1.PcbDoc PCB1.PcbDoc PCB1.PcbDoc PCB1.PcbDoc		
	Add					
	Add					
	Add					
	Add Nets(3)	GND NetC1_1 VCC	To To To	PCB1.PcbDoc PCB1.PcbDoc PCB1.PcbDoc		
	Add					
	Add					
	Add Component Classes(1)	Sheet1	To	PCB1.PcbDoc		
	Add					
	Add Rooms(1)	Room Sheet1 (Scope=InComponentClass('She To		PCB1.PcbDoc		
	Add					

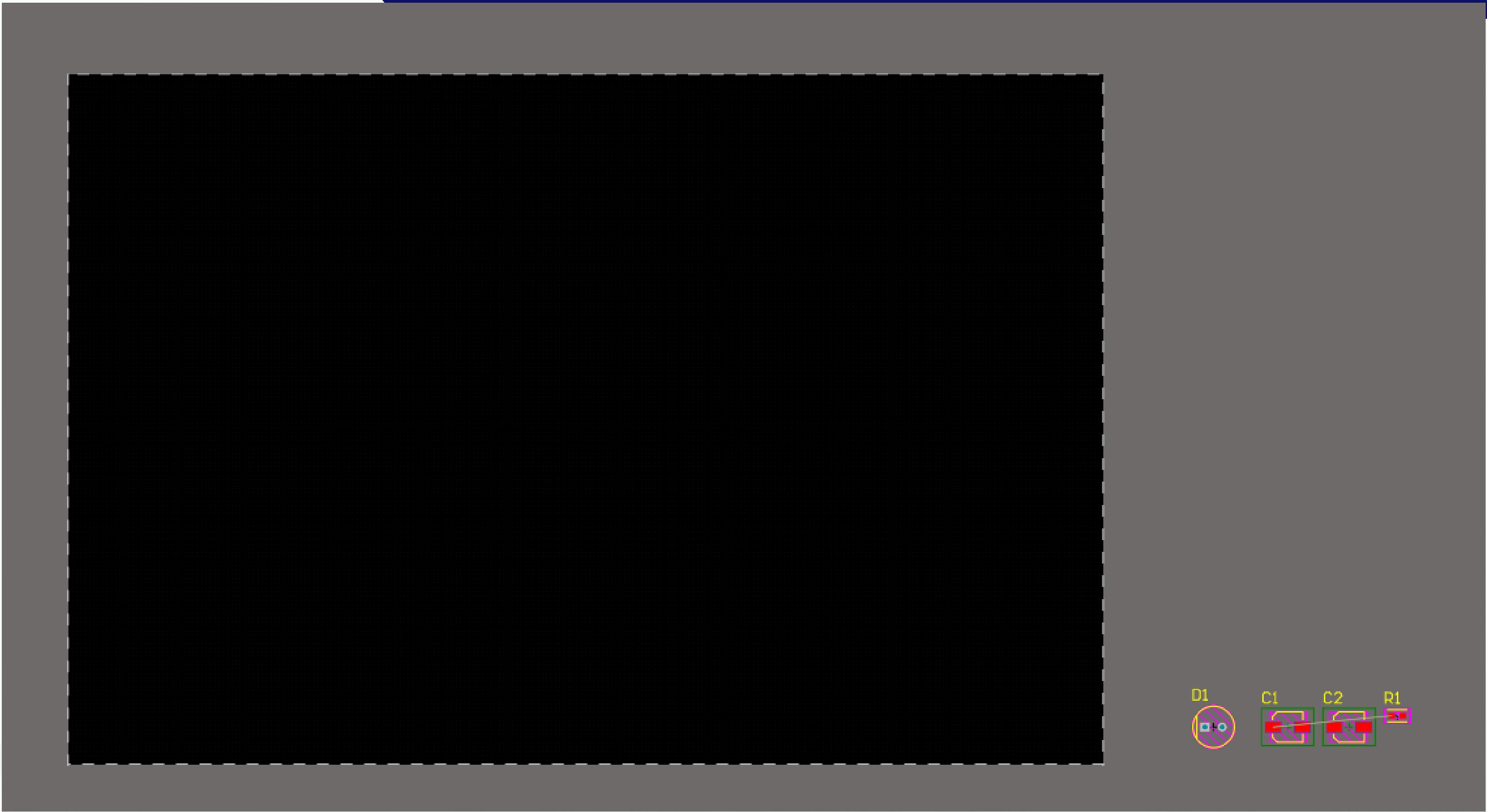
Validate Changes   Execute Changes   Report Changes...    Only Show Errors   Close

CLIQUEZ SUR VALIDATE CHANGES

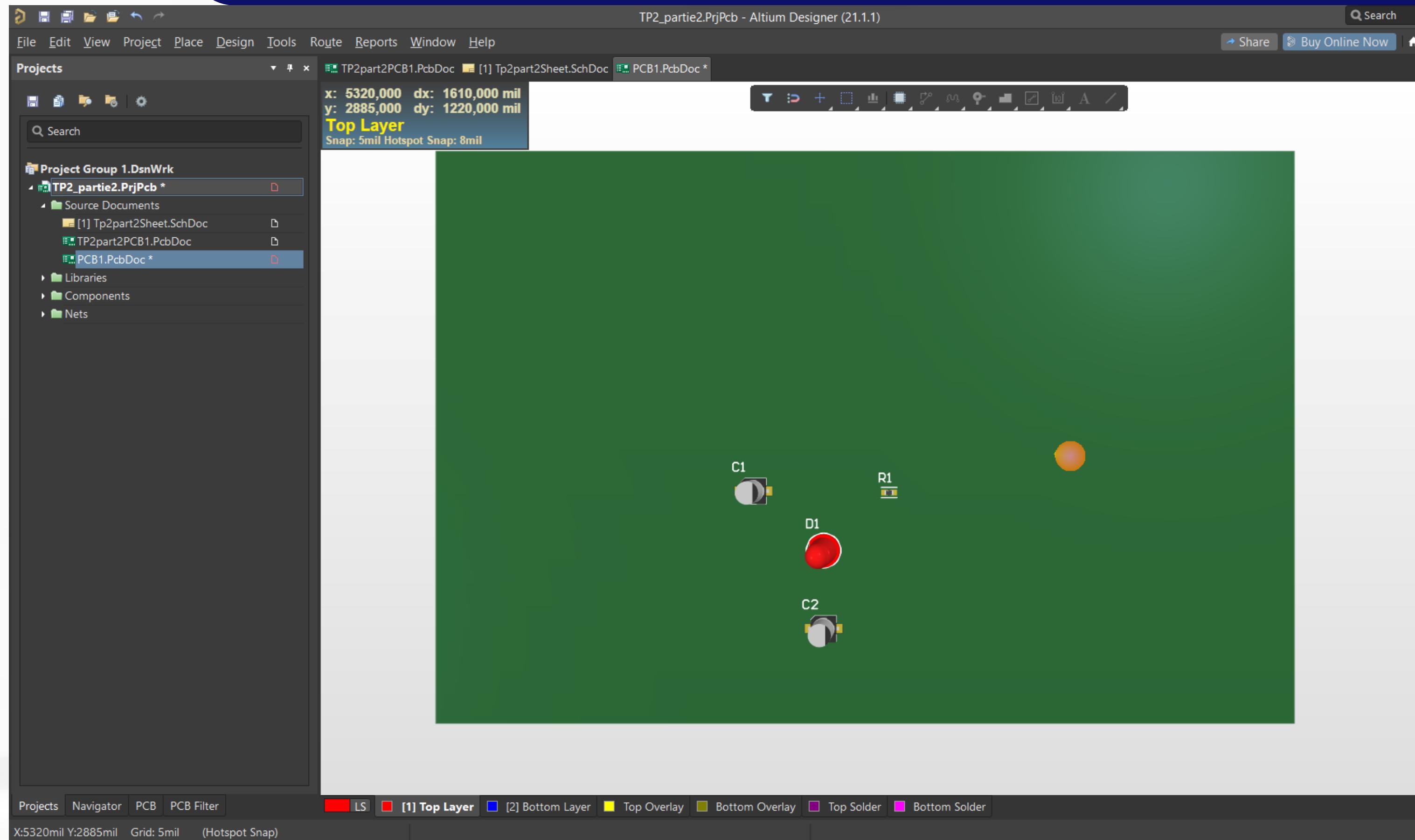


CLIQUEZ SUR EXECUTE CHANGES

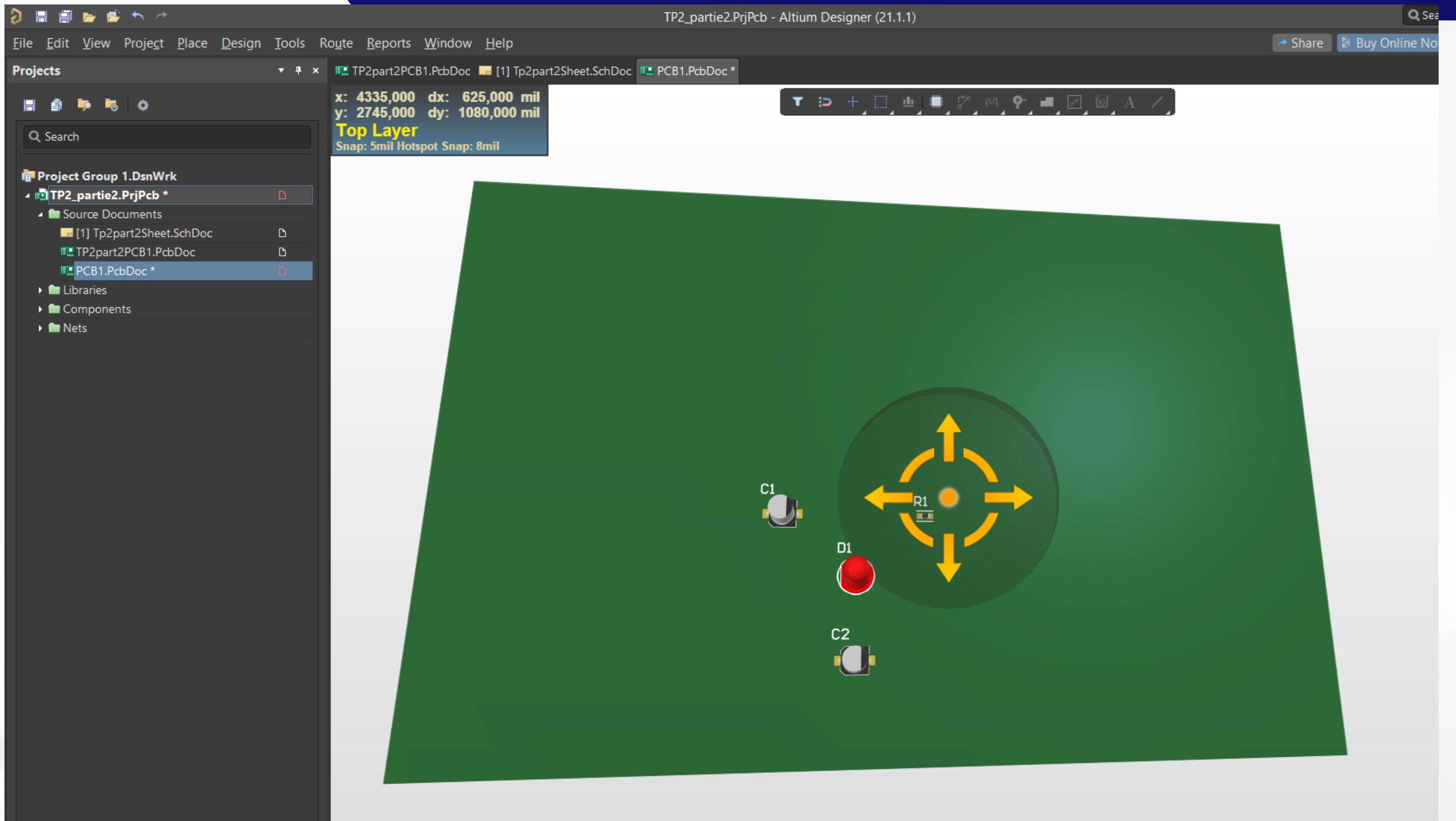
cliquer update changes list puis accepter



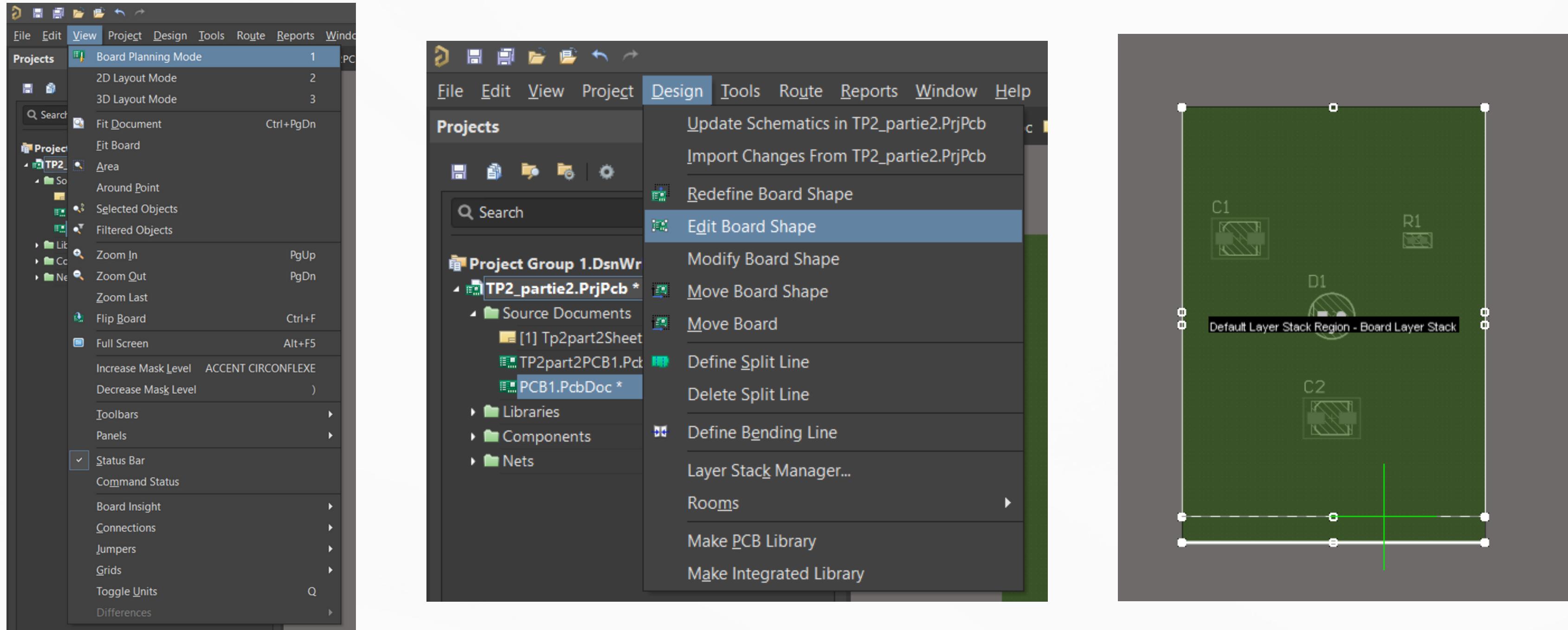
# on déplace les composants puis on appuis sur 3



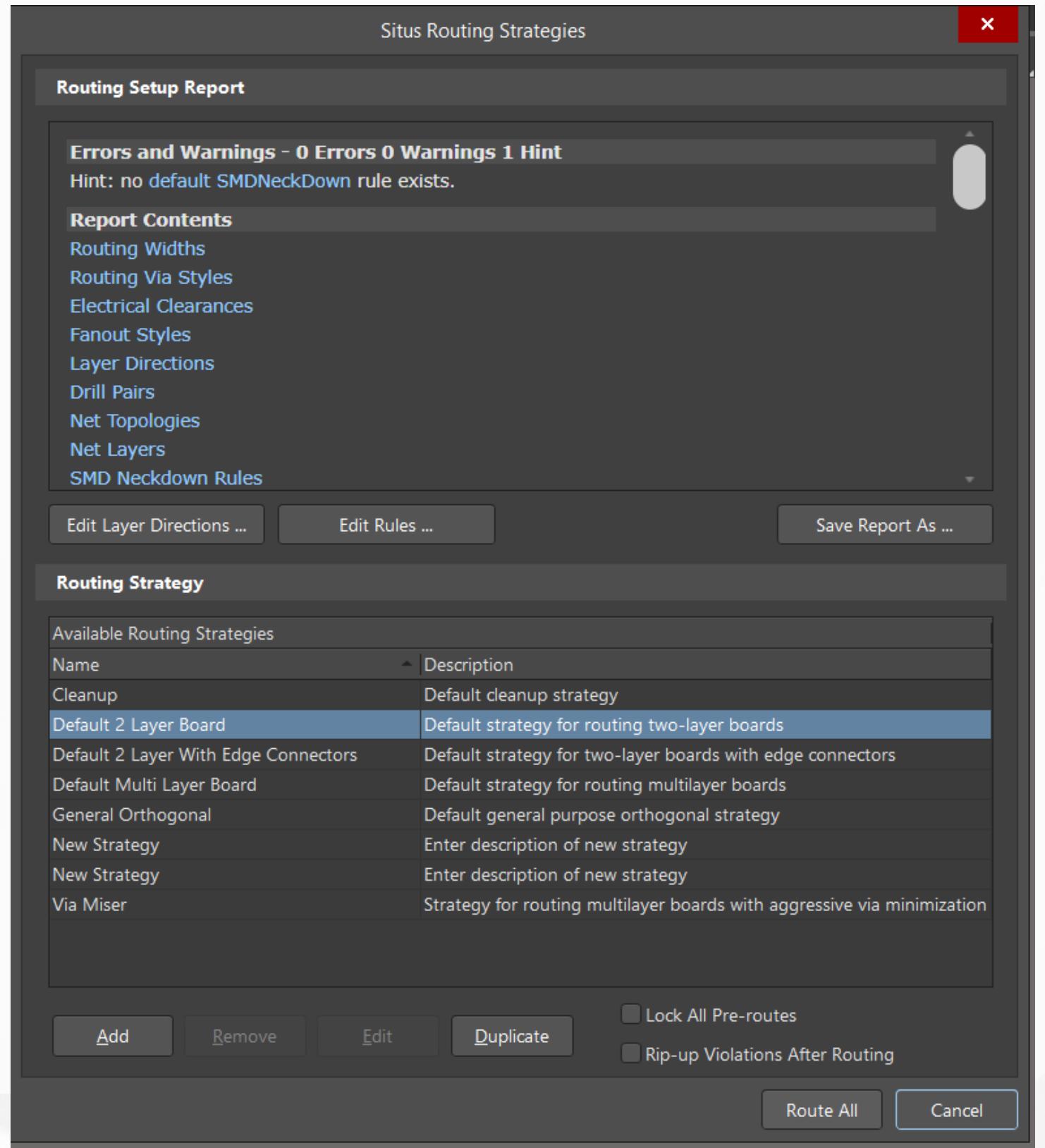
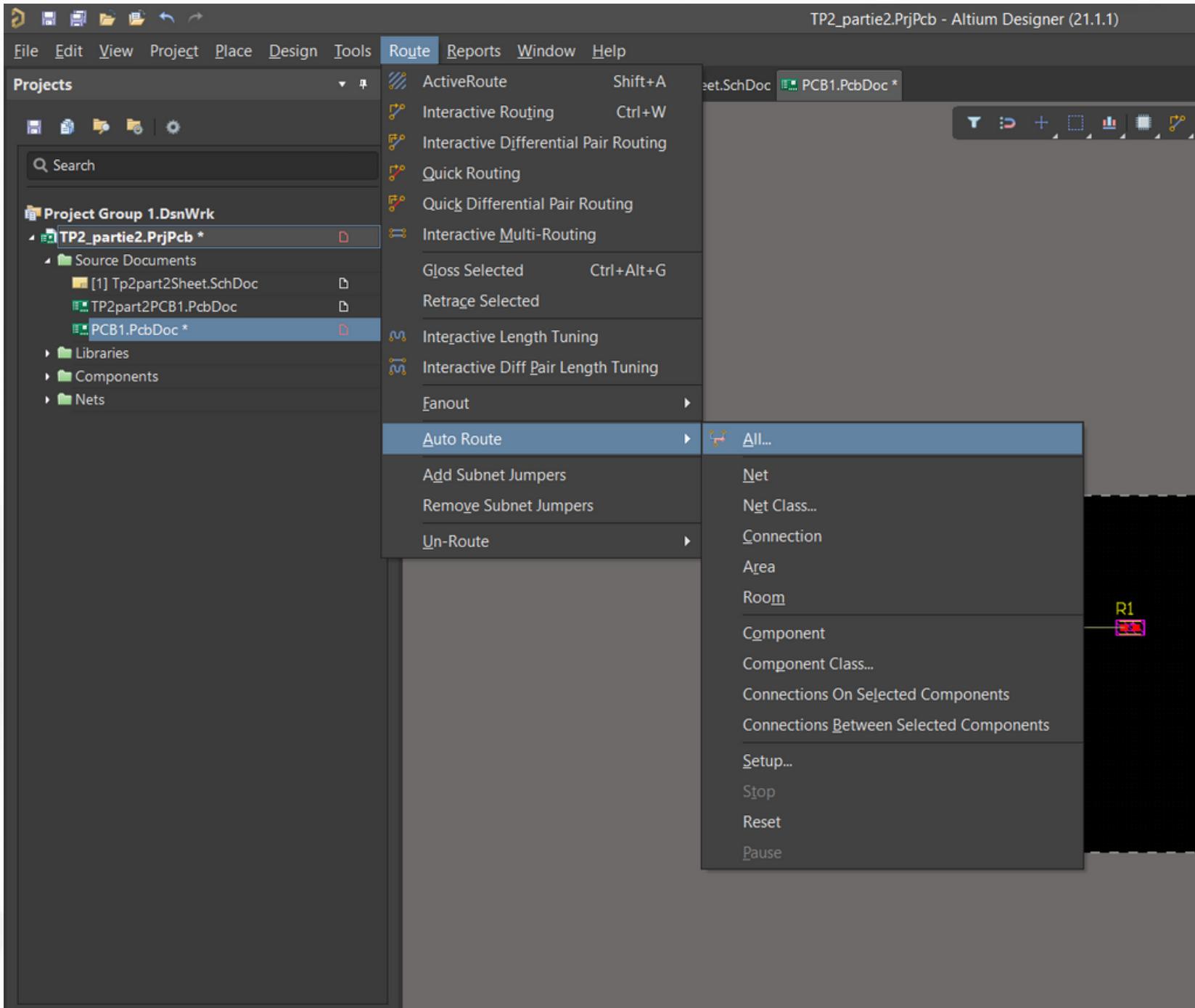
# hold shift + btn droite du souris



# réduire la taille du PCB



# Routage de la carte ( cliquer Route all)



# Félicitation

