

**PLECKO**

## KiCad Example

김병헌 Byeongheon Kim  
2025. 1Q  
benkim@plecko.biz

# 프로젝트 생성

The screenshot shows the KiCad 7.0 interface. The 'File' menu is open, and the 'New Project...' option is highlighted with a circled '1'. The 'Create New Project' dialog box is also open, showing the file path '로컬 디스크 (D:) > Work > KiCad > Education' and the file name 'Example' (circled '2'). The 'Save as type' is set to 'KiCad project files (\*.kicad\_pro)' (circled '3'). The checkbox 'Create a new folder for the project' is checked and highlighted with a red box (circled '4'). The 'Save' button is also circled '4'.

① File Edit View Tools Preferences Help

- New Project... Ctrl+N
- New Project from Template... Ctrl+T
- Open Demo Project...
- Open Project... Ctrl+O
- Open Recent >
- Close Project
- Save As... Ctrl+Shift+S
- Import Non-KiCad Project... >
- Archive Project...
- Unarchive Project...
- Quit Ctrl+Q

② File name: Example

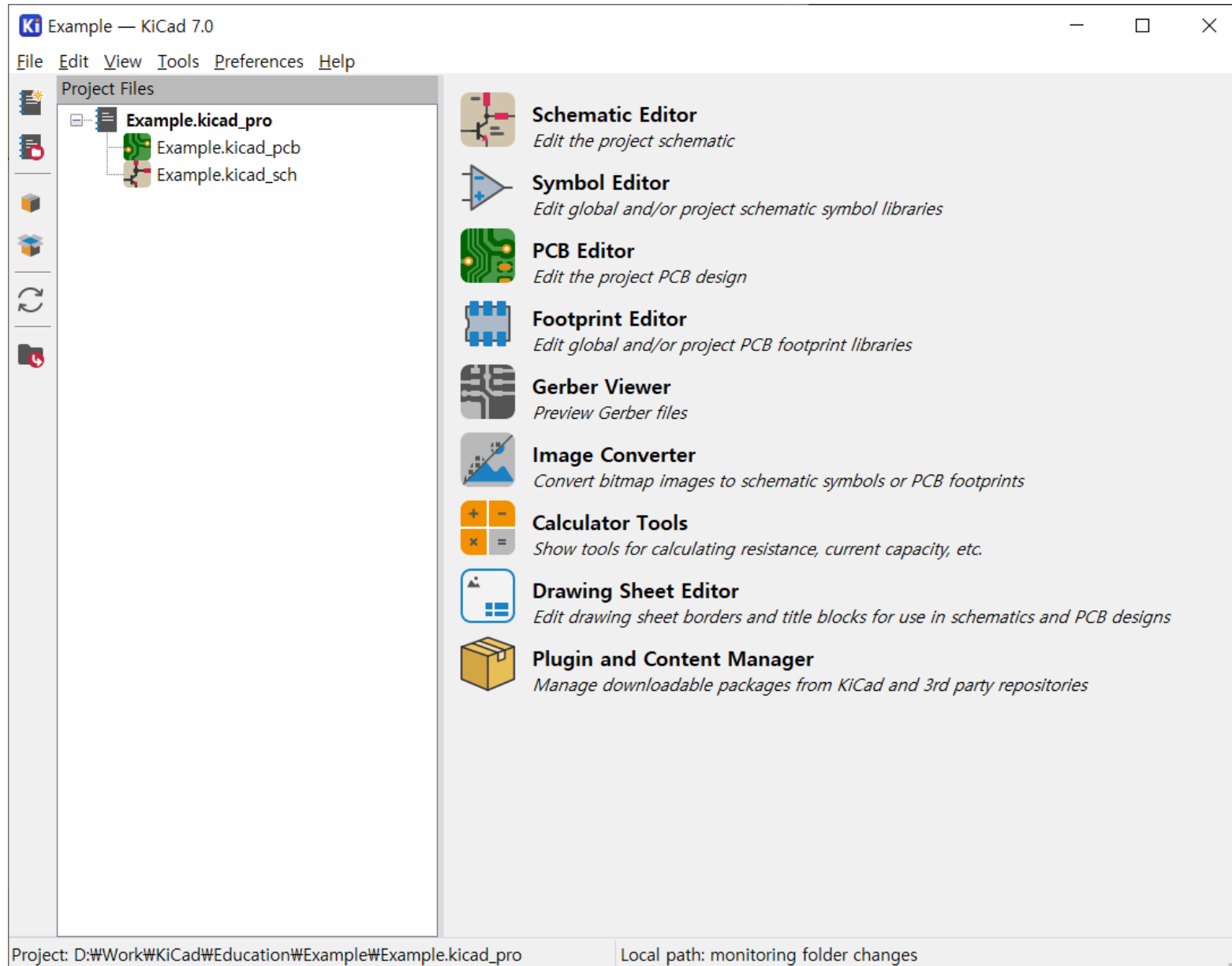
③ Save as type: KiCad project files (\*.kicad\_pro)

④ ☒ Create a new folder for the project

Save Cancel

Create new blank project (Ctrl+N) Local path: monitoring folder changes

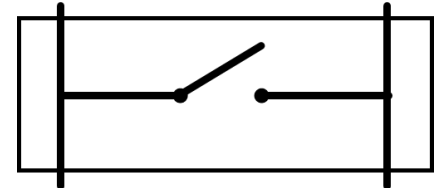
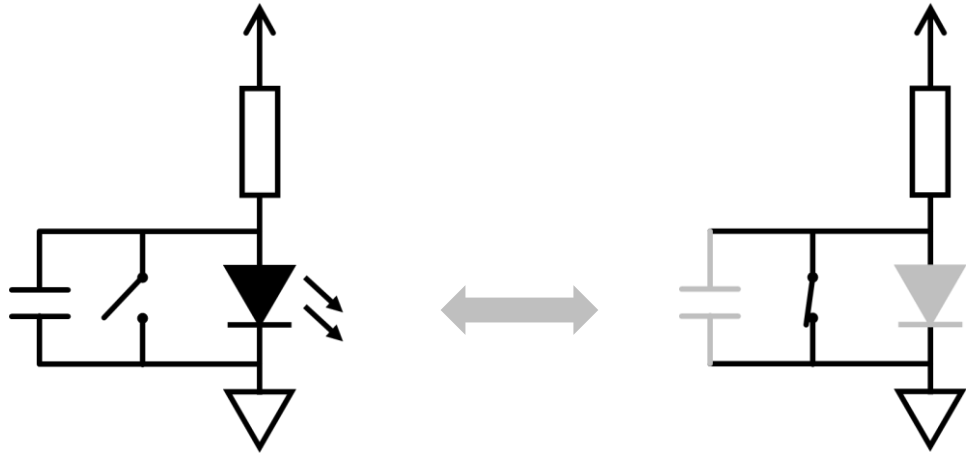
# 프로젝트 생성



# Design Circuit

## ❖ LED ON/OFF Circuit 예제

- ▶ LED: 2012 size, Chip LED, 20 mA, on drop 2 V
- ▶ 저항: 2012 size, 330 Ohm, 1/8 W
- ▶ 전원: 5 V 외부 전압
  - ✓ 커넥터: 2 pin connector (2.54 mm pitch)
- ▶ 스위치
  - ✓ 탭트 스위치
- ▶ 캐패시터: 2012 size, 1 nF, > 6V

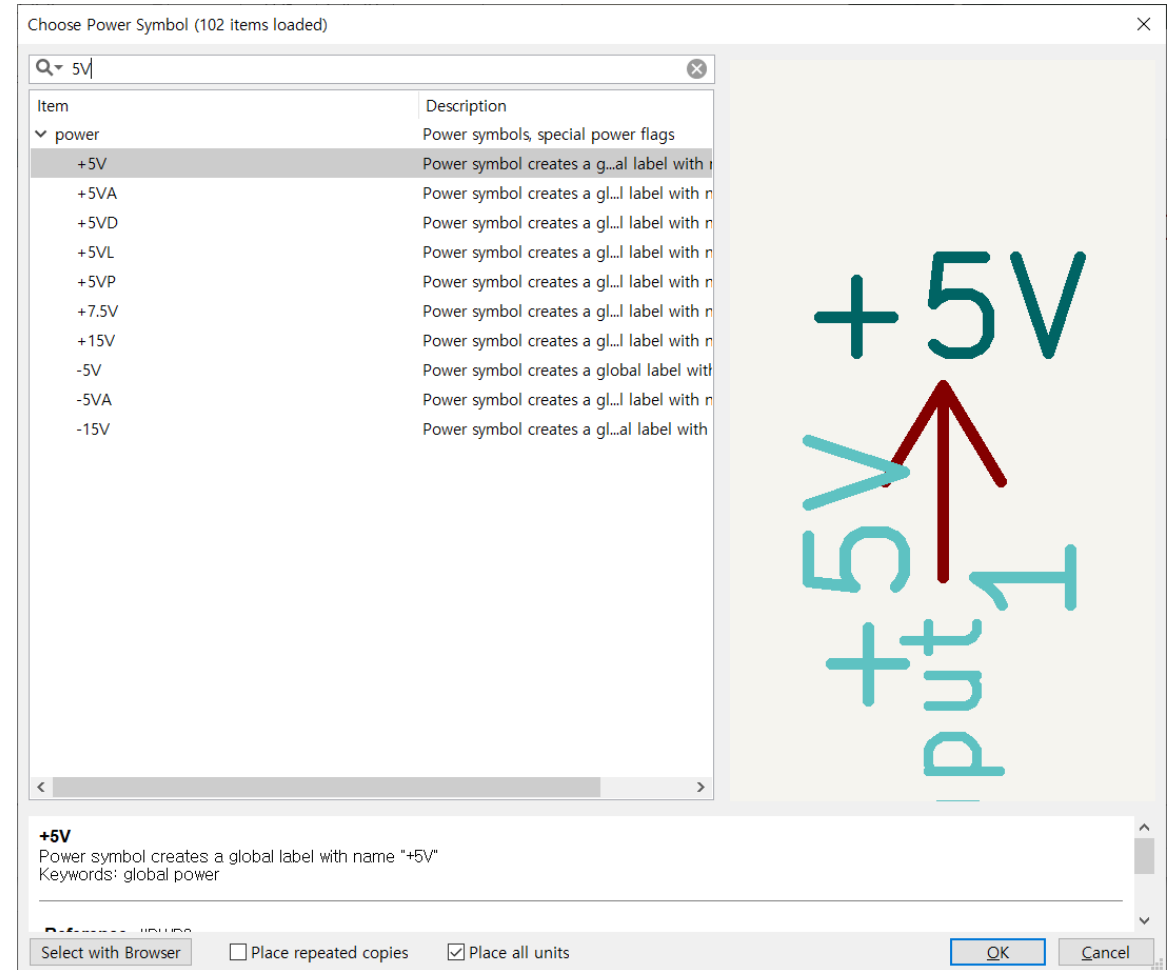
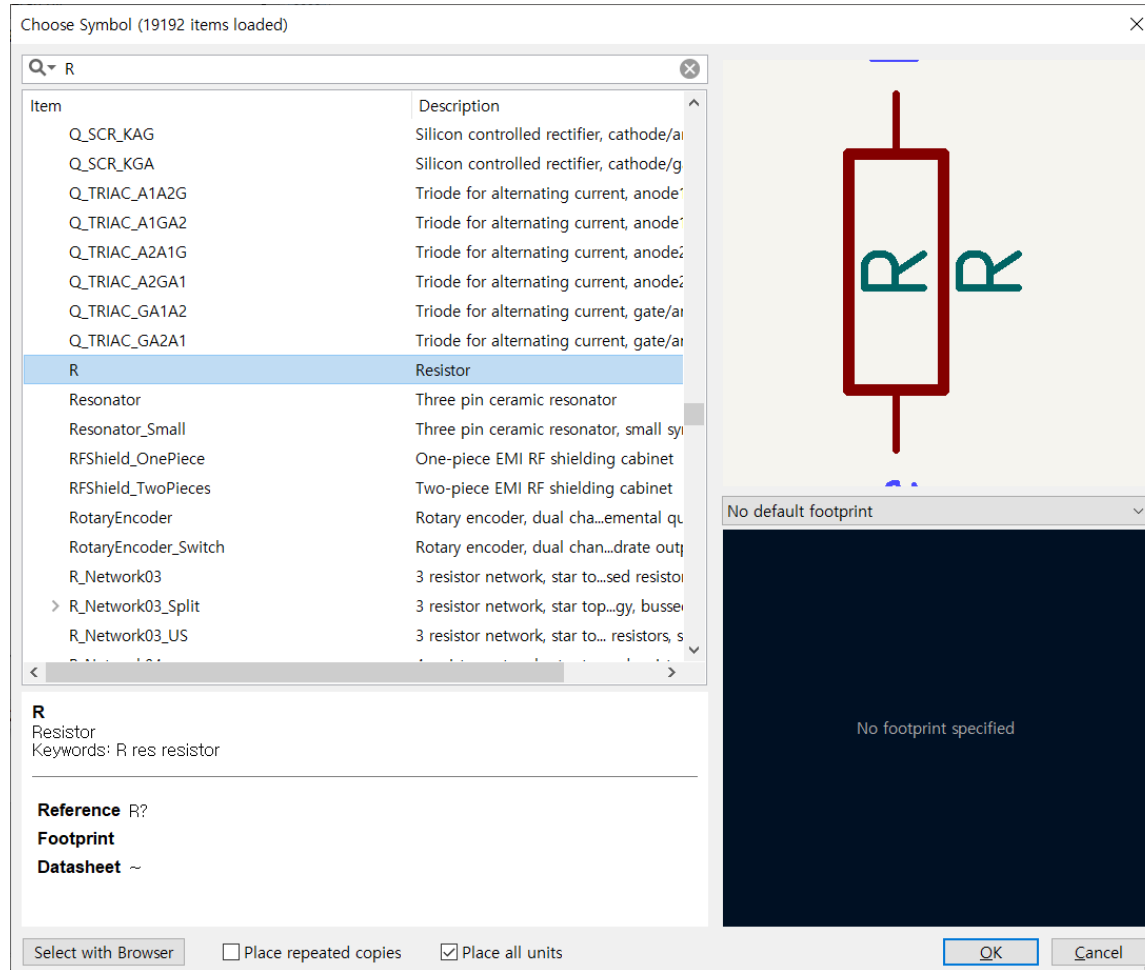


# Create Schematic

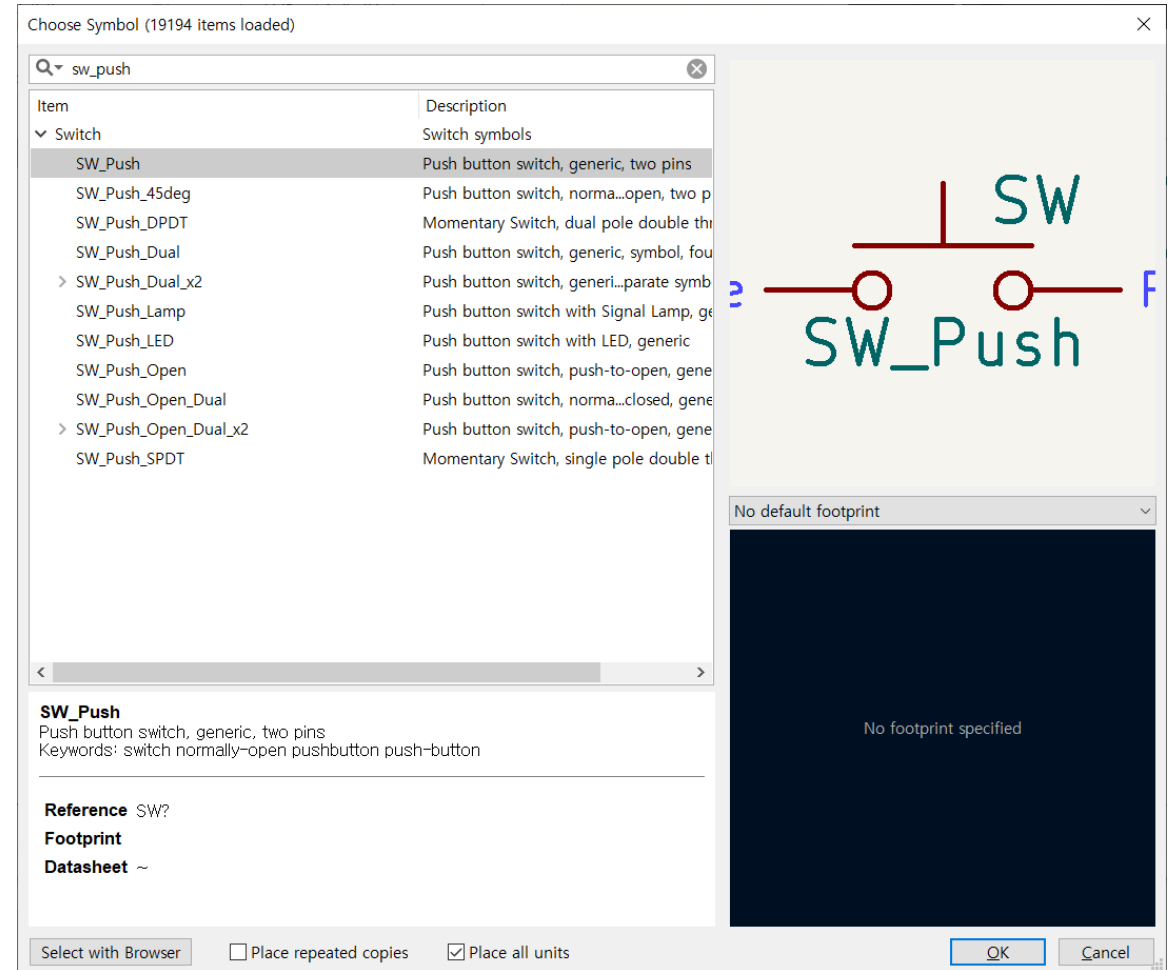
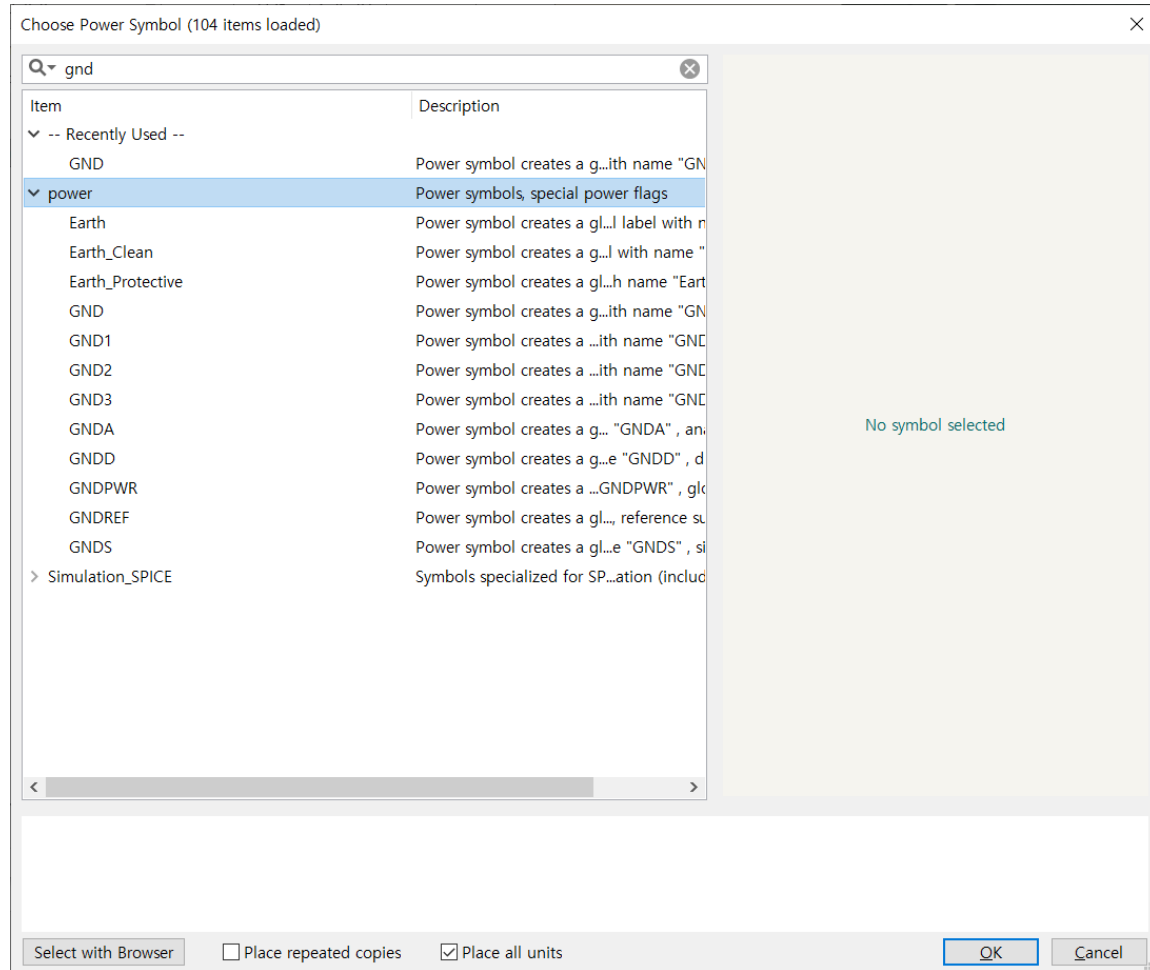
## ❖ LED ON/OFF Circuit 예제

- ▶ 커넥터와 Power pin 배치,
  - ✓ 커넥터: 2 pin connector (2.54 mm pitch)
- ▶ Ground 배치
- ▶ 저항 및 LED, 캐패시터, 스위치 배치
  - ✓ LED: 2012 size, Chip LED, 20 mA, on drop 2 V
  - ✓ 저항: 2012 size, 330 Ohm, 1/8 W
  - ✓ 캐패시터: 2012 size, 1 nF, > 6V
  - ✓ 스위치: SW\_Push 사용

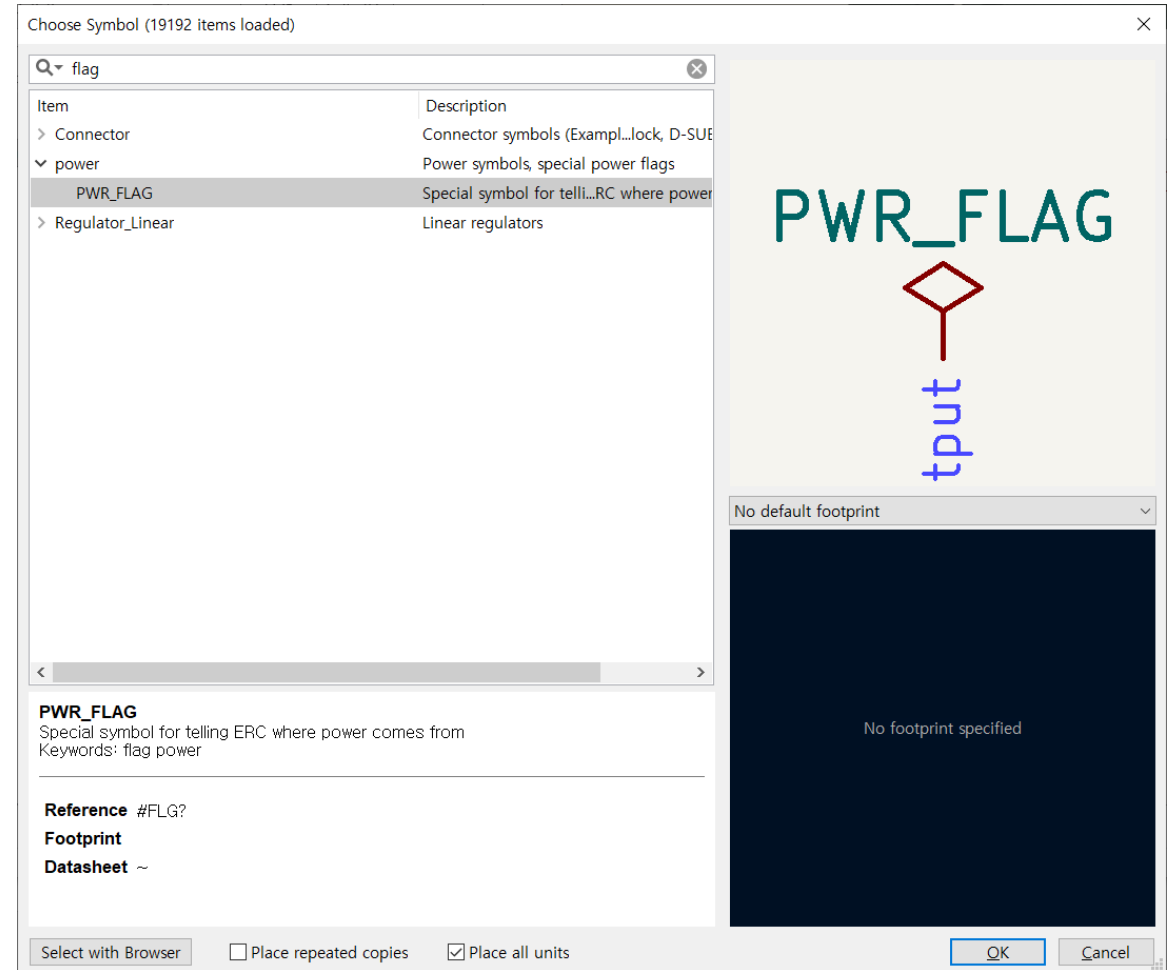
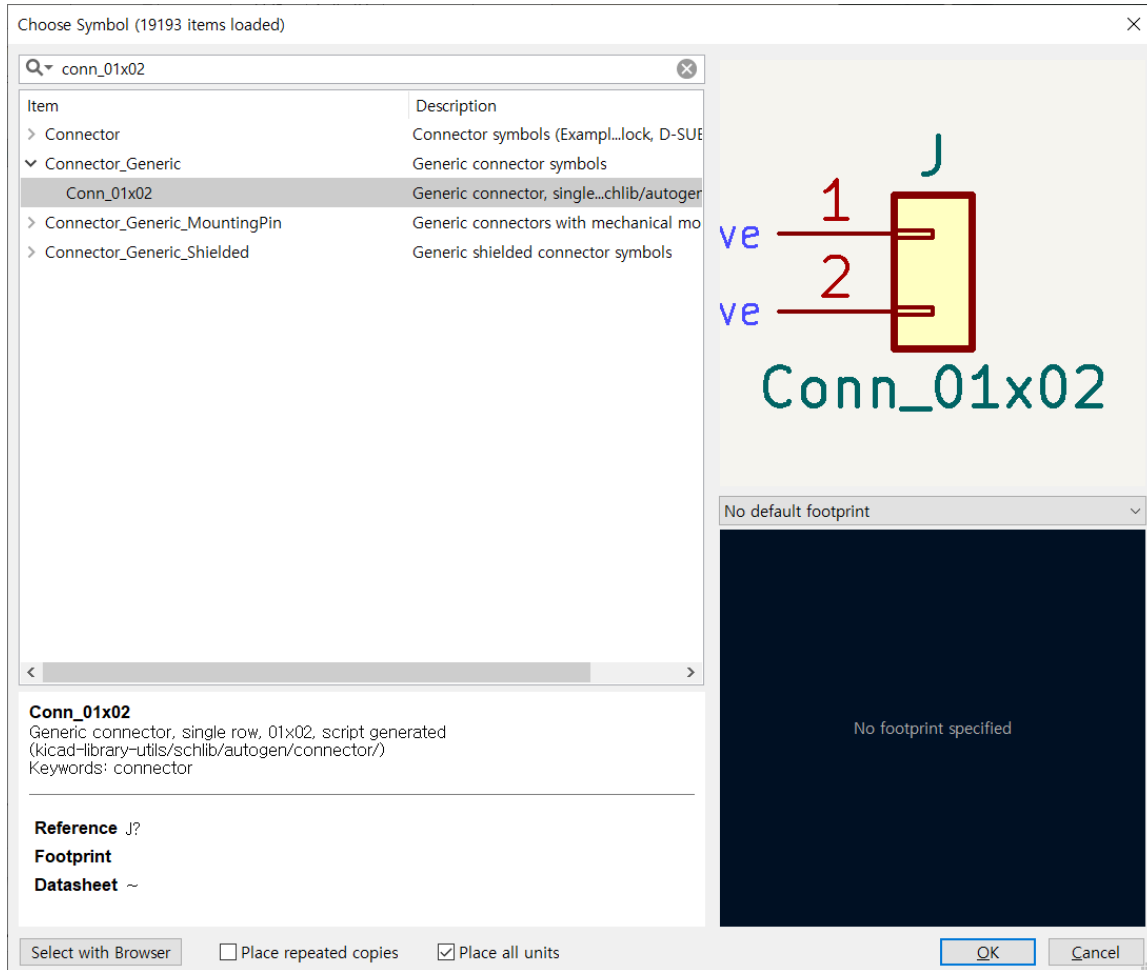
# Create Schematic



# Create Schematic



# Create Schematic





# Create Schematic

Example — Schematic Editor

File Edit View Place Inspect Tools Preferences Help

Schematic Hierarchy

- Root (page 1)

Electrical Rules Checker

Violations (0) Ignored Tests (0)

Show: ☒ All ☒ Errors 0 ☒ Warnings 0 ☒ Exclusions

Delete Marker Delete All Markers Run ERC Close

File 'D:\W... Z 1.53 X 52.8320 Y -13.4620 dx 52.8320 dy -13.4620 dist 54.5201 grid 0.2540 mm Select item(s)

The screenshot displays a schematic editor window titled 'Example — Schematic Editor'. The main workspace shows a circuit diagram with the following components: a capacitor C1 (1nF/6V), a switch SW1 (5W\_Push), a resistor R1 (330/2012), a diode D1 (LED), and a connector J1 (Conn\_01x02). The circuit is powered by a +5V source and connected to ground (GND). Two power flags, labeled PWR\_FLAG, are present. An 'Electrical Rules Checker' dialog box is open in the foreground, showing 'Violations (0)' and 'Ignored Tests (0)'. The dialog includes checkboxes for 'All', 'Errors', 'Warnings', and 'Exclusions', all of which are checked. The 'Errors' and 'Warnings' counts are both 0. Buttons for 'Delete Marker', 'Delete All Markers', 'Run ERC', and 'Close' are visible at the bottom of the dialog. The status bar at the bottom of the editor shows file information, coordinates, dimensions, grid size, and units.

# Create Schematic

Symbol Fields Table

Field	Show	Group By
Reference	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Value	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Footprint	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Datasheet	<input type="checkbox"/>	<input type="checkbox"/>

Add Field...

Remove Field...

Filter

☒ Group symbols


Reference	Value	Footprint	Qty
C1	1nF/6V	Capacitor_SMD:C_0805_2012Metric_Pad1.18x1.45mm_HandSolder	1
D1	LED	LED_SMD:LED_0805_2012Metric_Pad1.15x1.40mm_HandSolder	1
J1	Conn_01x02	Connector_PinHeader_2.54mm:PinHeader_1x02_P2.54mm_Vertical	1
R1	330/2012	Resistor_SMD:R_0805_2012Metric_Pad1.20x1.40mm_HandSolder	1
SW1	SW_Push	Button_Switch_SMD:SW_Push_1P1T_NO_CK_KSC6xxJ	1

Export as CSV...

Apply, Save Schematic & Continue

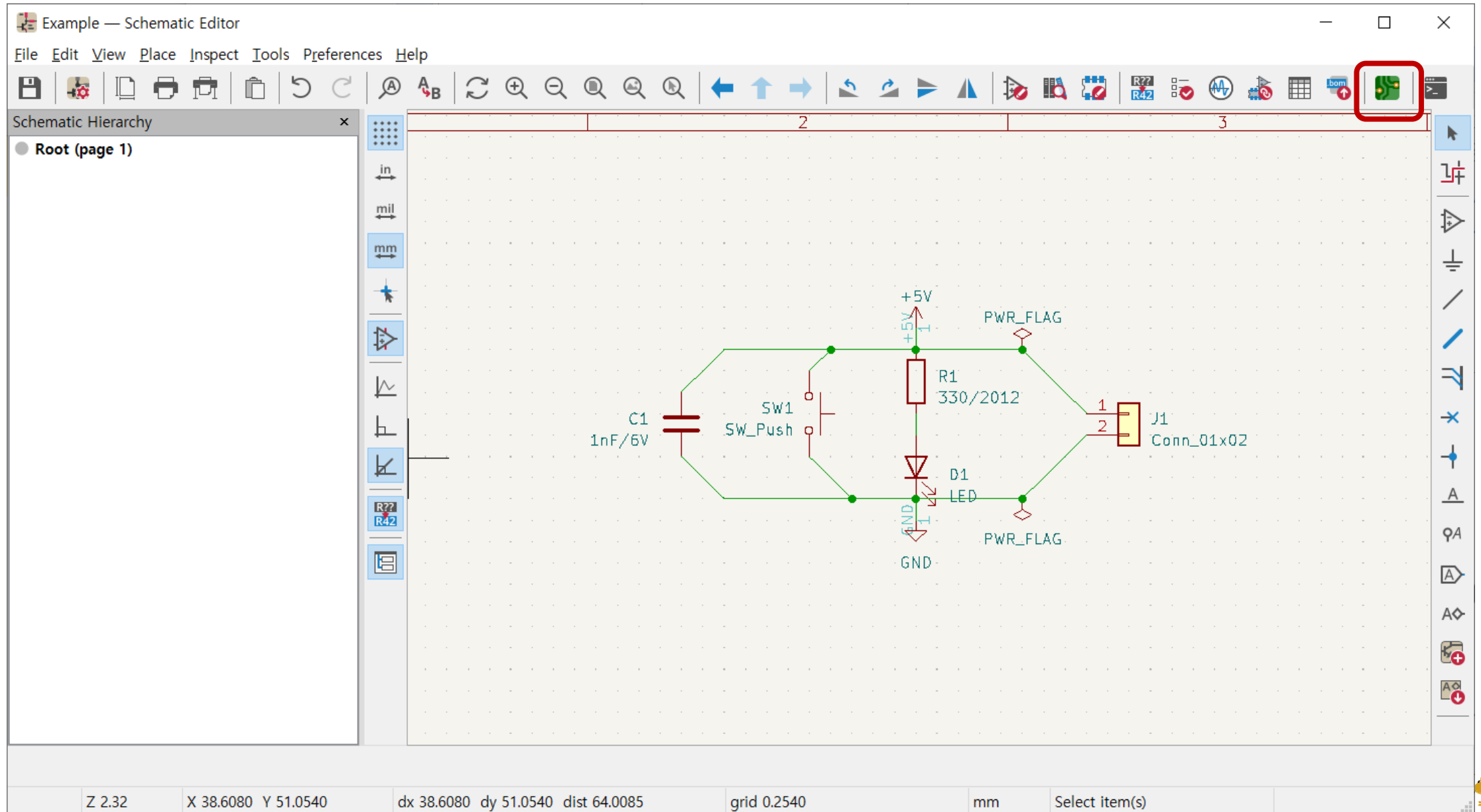
OK

Cancel

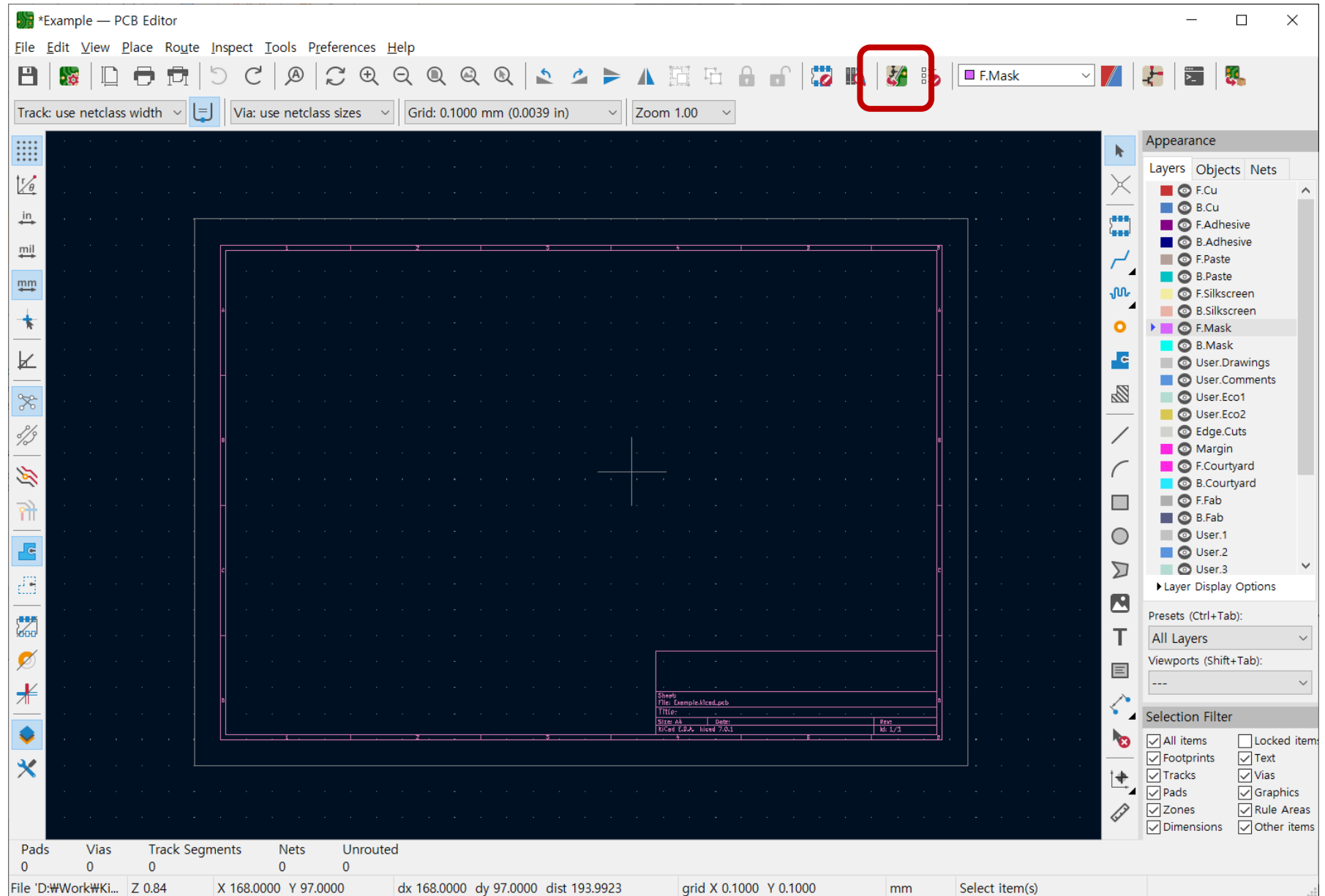


PLECKO

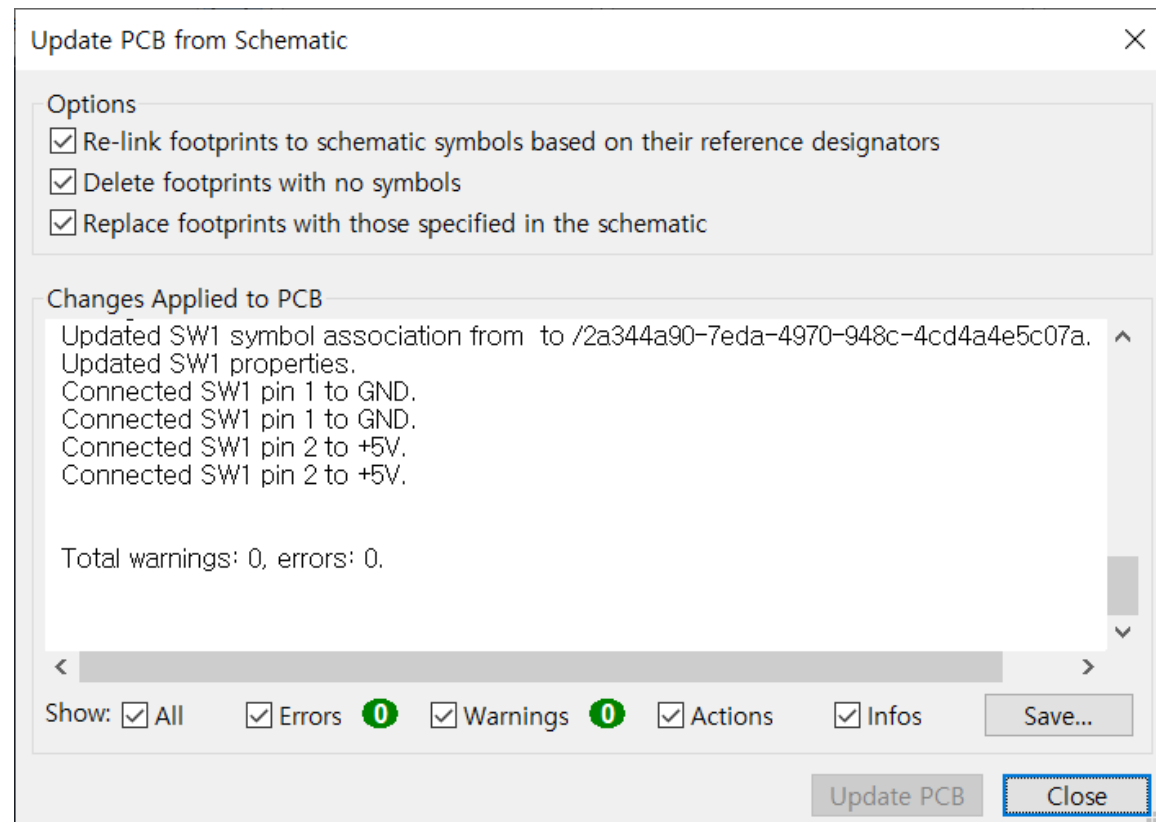
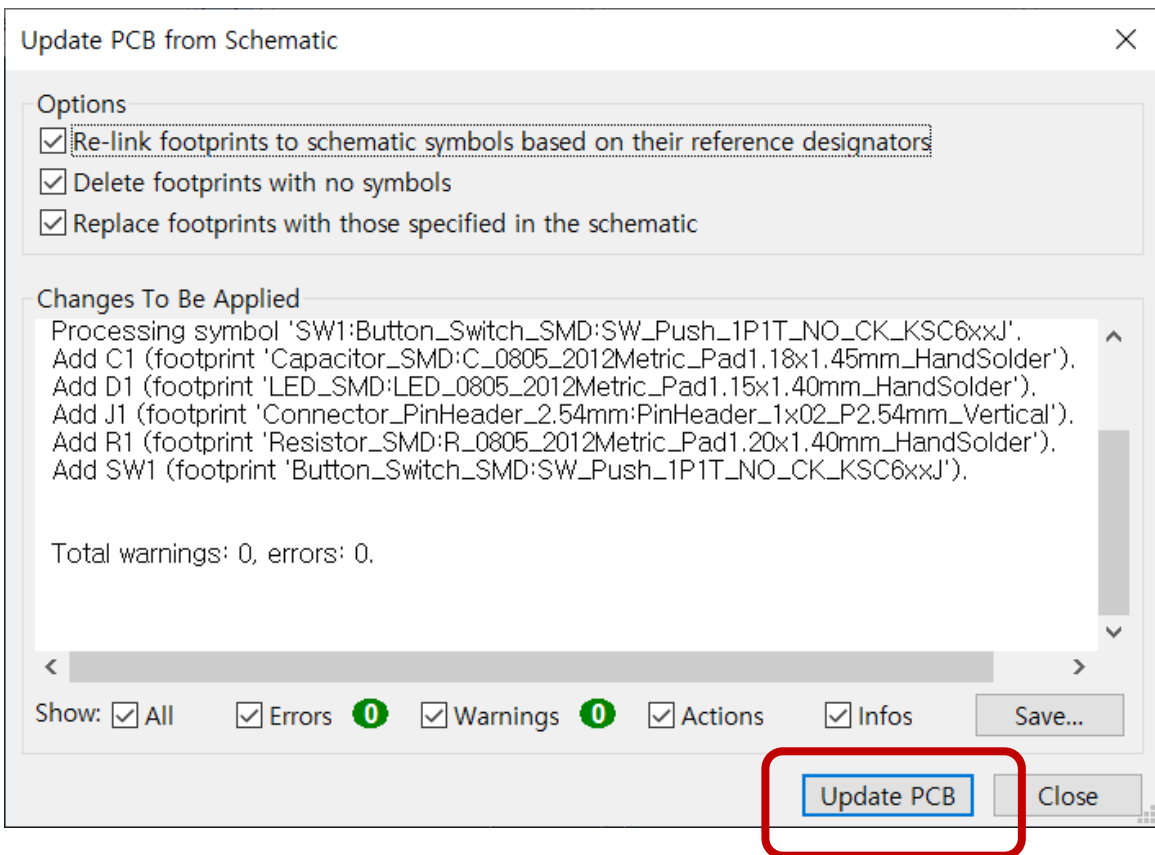
# Create Schematic



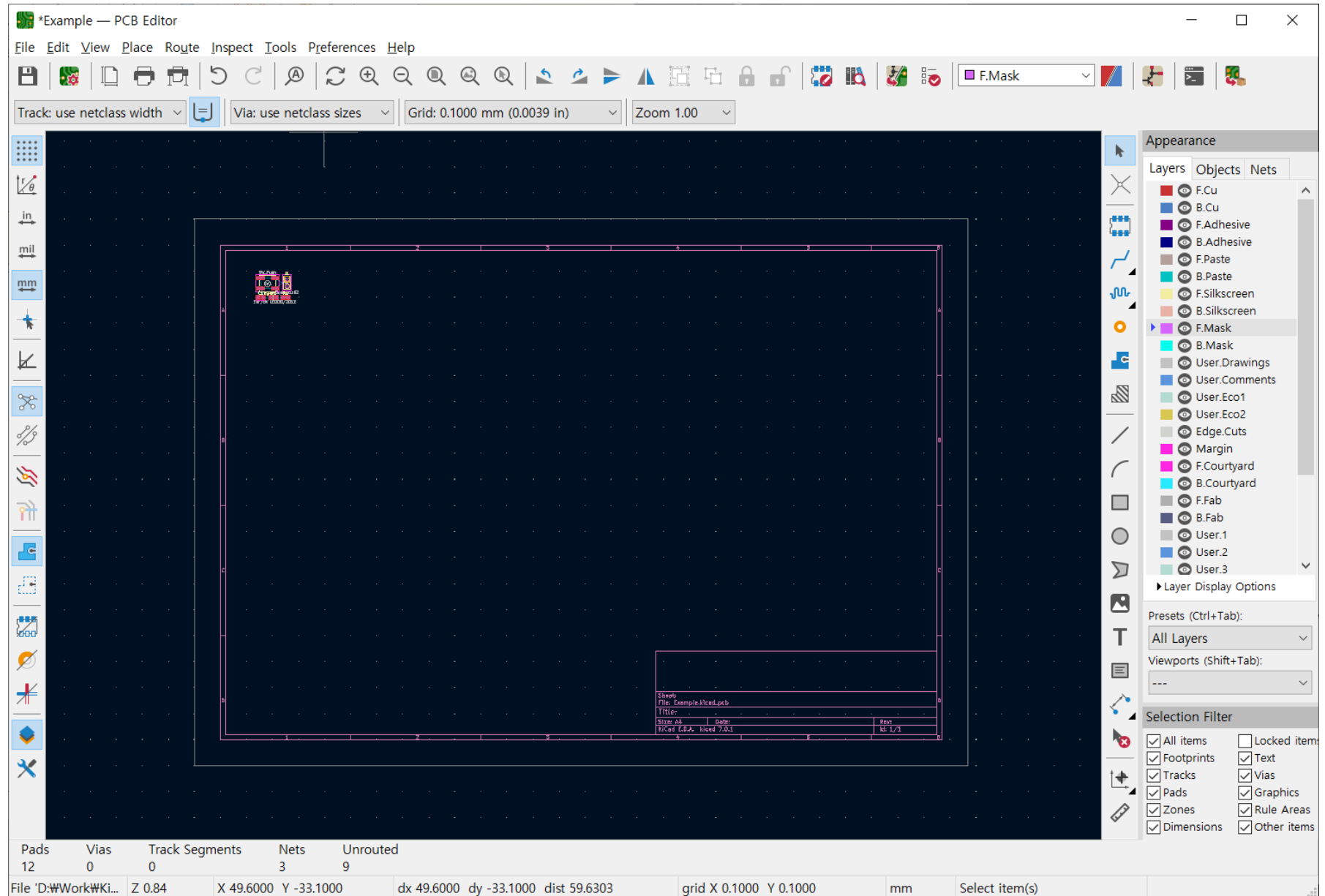
# Board Layout



# Board Layout



# Board Layout



# Board Layout: Board setup

Board Setup

Board Editor Layers

Physical Stackup

Board Finish

Solder Mask/Paste

Text & Graphics

Defaults

Formatting

Text Variables

Design Rules

Constraints

Pre-defined Sizes

Net Classes

Custom Rules

Violation Severity

Add User Defined Layer...

<input type="checkbox"/>	F.Courtyard	Off-board, testing
<input checked="" type="checkbox"/>	F.Fab	Off-board, manufacturing
<input checked="" type="checkbox"/>	F.Adhesive	On-board, non-copper
<input checked="" type="checkbox"/>	F.Paste	On-board, non-copper
<input checked="" type="checkbox"/>	F.Silkscreen	On-board, non-copper
<input checked="" type="checkbox"/>	F.Mask	On-board, non-copper
<input checked="" type="checkbox"/>	F.Cu	signal
<input checked="" type="checkbox"/>	B.Cu	signal
<input checked="" type="checkbox"/>	B.Mask	On-board, non-copper
<input checked="" type="checkbox"/>	B.Silkscreen	On-board, non-copper
<input checked="" type="checkbox"/>	B.Paste	On-board, non-copper
<input checked="" type="checkbox"/>	B.Adhesive	On-board, non-copper
<input checked="" type="checkbox"/>	B.Fab	Off-board, manufacturing
<input type="checkbox"/>	B.Courtyard	Off-board, testing
<input type="checkbox"/>	Edge.Cuts	Board contour
<input type="checkbox"/>	Margin	Edge_Cuts setback
<input type="checkbox"/>	User.Eco1	Auxiliary
<input type="checkbox"/>	User.Eco2	Auxiliary
<input checked="" type="checkbox"/>	User.Comments	Auxiliary
<input checked="" type="checkbox"/>	User.Drawings	Auxiliary
<input type="checkbox"/>	User.1	User defined layer
<input type="checkbox"/>	User.2	User defined layer

Import Settings from Another Board...

OK Cancel

# Board Layout: Board setup

Board Setup

Board Stackup

Board Editor Layers

Physical Stackup

Board Finish

Solder Mask/Paste

Text & Graphics

Defaults

Formatting

Text Variables

Design Rules

Constraints


Pre-defined Sizes

Net Classes


Custom Rules

Violation Severity


Copper

 Minimum clearance:


mm

 Minimum track width:


mm

 Minimum connection width:


mm

 Minimum annular width:


mm

 Minimum via diameter:

mm


 Copper to hole clearance:

mm


 Copper to edge clearance:

mm

Holes


 Minimum through hole:

mm


 Hole to hole clearance:

mm

uVias

 Minimum uVia diameter:

mm

 Minimum uVia hole:

mm

Silkscreen

Minimum item clearance:

mm

Minimum text height:

mm

Minimum text thickness:


mm

Arc/circle approximated by segments


Max allowed deviation:  mm

Note: zone filling can be slow when < 0.005 mm.

Zone fill strategy



☒ Allow fillets outside zone outline



Min thermal relief spoke count:

Length tuning

☒ Include stackup height in track length calculations

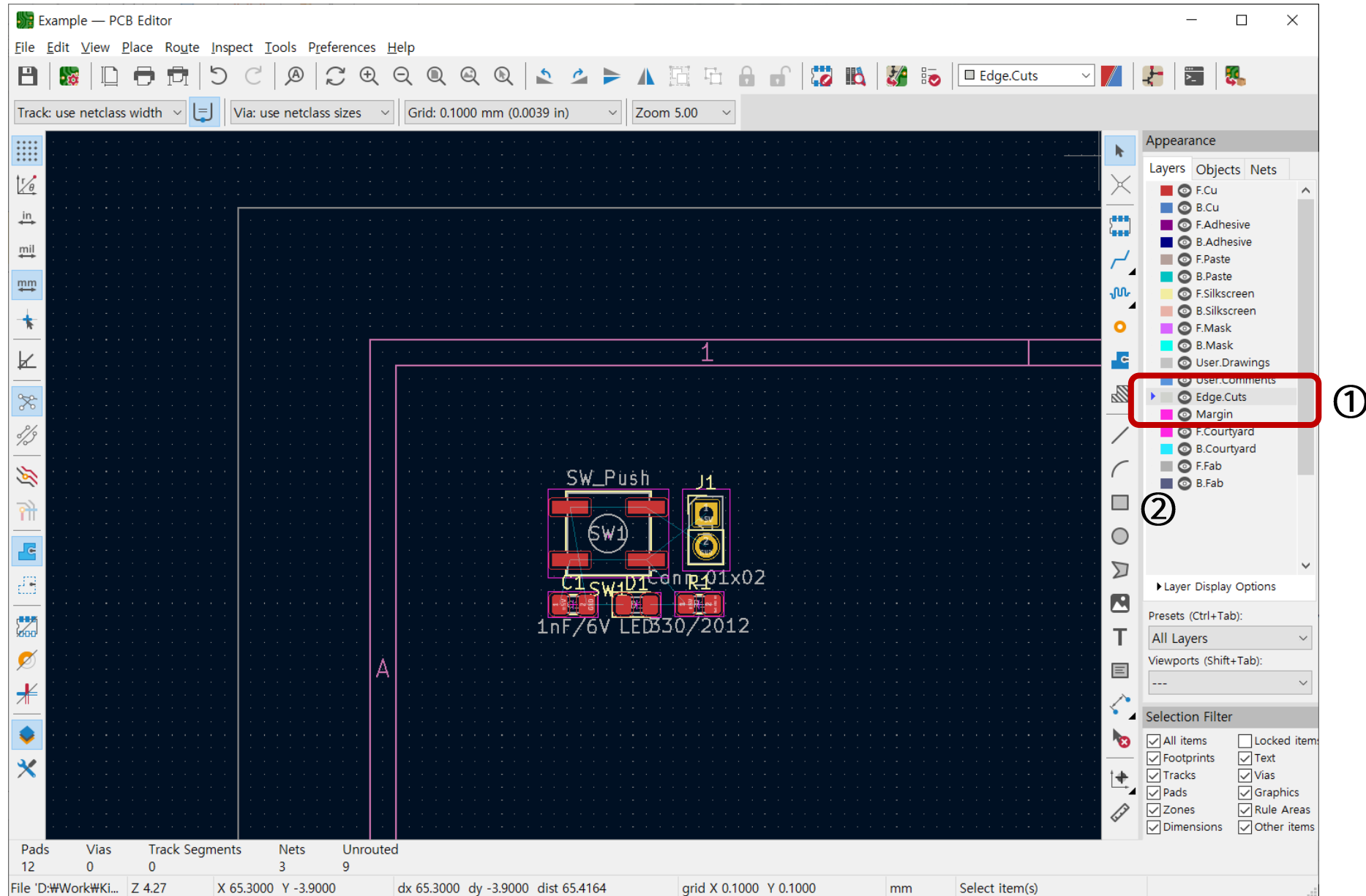
Import Settings from Another Board...

OK

Cancel



# Board Layout



# Board Layout: 경계선 긋기

The screenshot shows a PCB Editor window titled '\*Example — PCB Editor'. The main workspace displays a PCB layout with various components: SW\_Push, J1, C1, SW1, and R1. A purple rectangle is being drawn on the board. The 'Rectangle Properties' dialog box is open, showing the following settings:

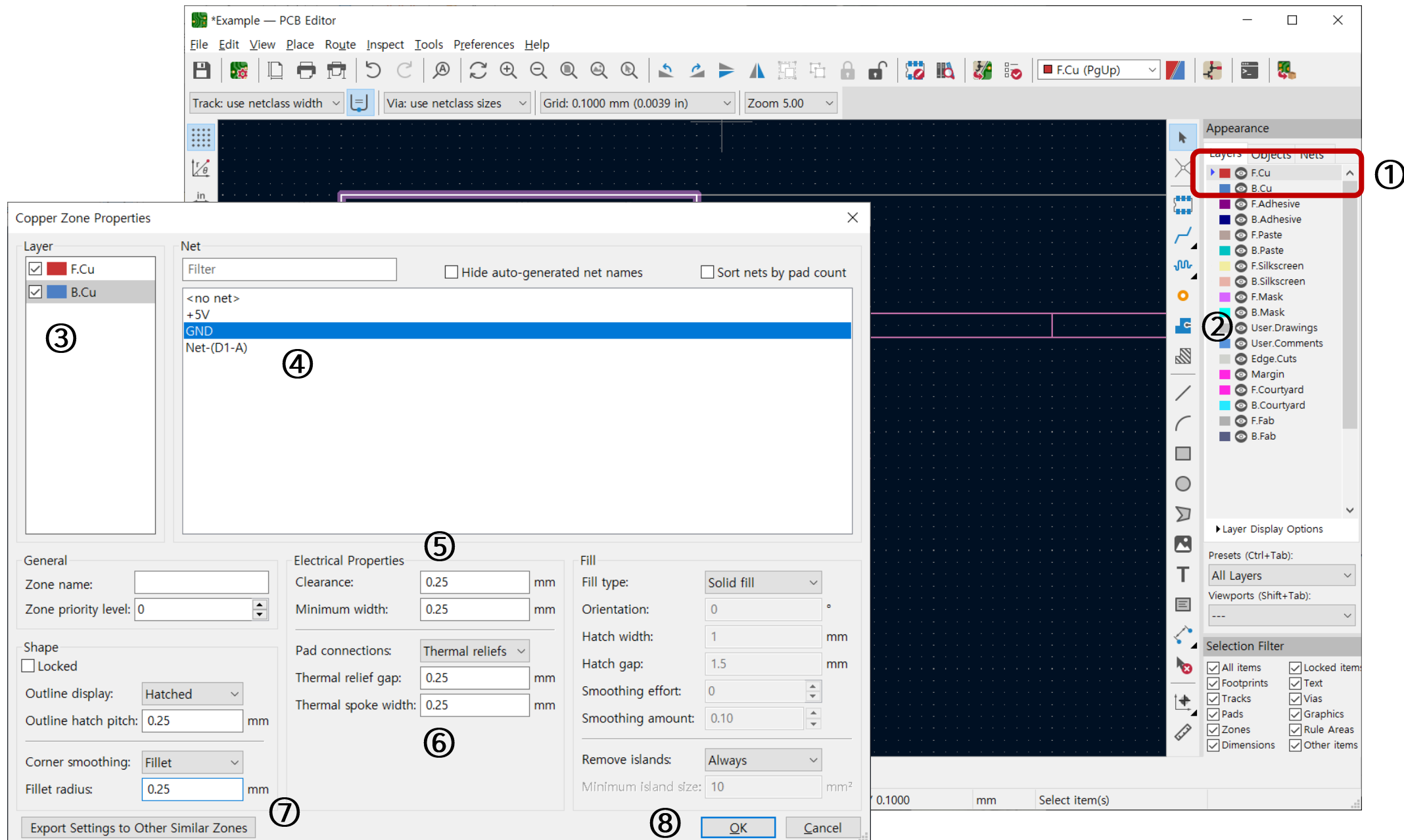
- Start Point: X: 0 mm, Y: 0 mm
- End Point: X: 30 mm, Y: 30 mm
- ☒ Locked
- ☐ Filled shape
- Line width: 0.1 mm
- Line style: Default
- Layer: ☐ Edge.Cuts

The dialog box has 'OK' and 'Cancel' buttons. The bottom status bar shows the following information:

Type	Status	Shape	Width	Height	Line Style	Line Width	Layer
Drawing	Locked	Rectangle	30.0000 mm	30.0000 mm	Default	0.1000 mm	Edge.Cuts

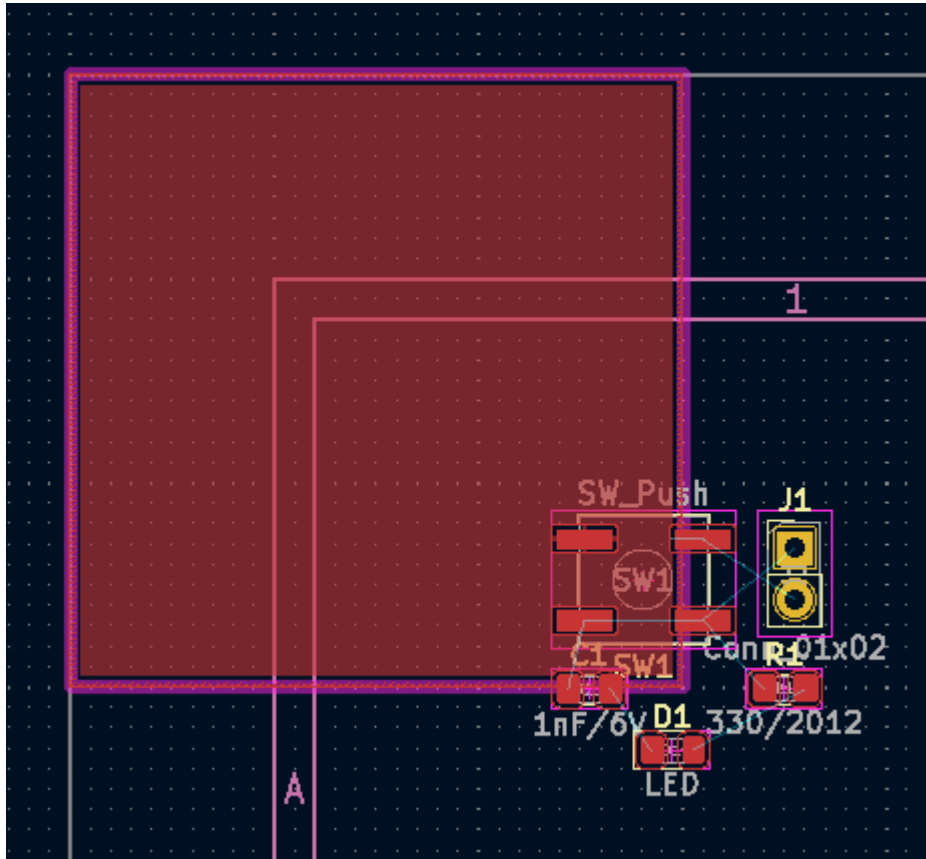
File 'D:\Work\Wki... Z 4.27' X 43.0000 Y -6.2000 dx 43.0000 dy -6.2000 dist 43.4447 grid X 0.1000 Y 0.1000 mm Select item(s)

# Board Layout: Pour 그리기



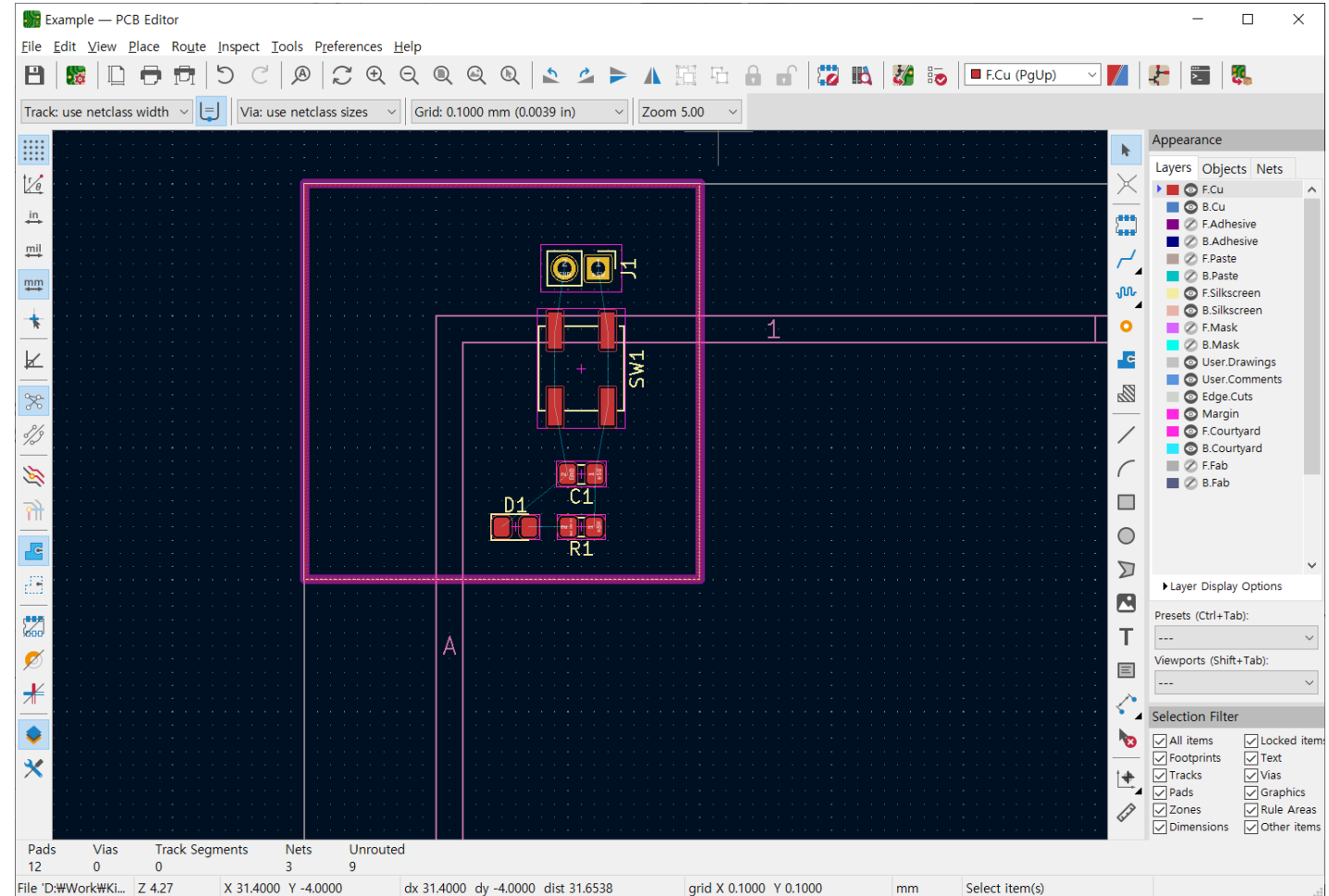
# Board Layout

단축키: B



Edgecut 경계선과 똑같은 크기로 그리더라도, board setup에서 지정한 copper to edge clearance가 적용됨.

단축키: Ctrl + B 후에 배치



# Board Layout: Routing

\*Example — PCB Editor

File Edit View Place Route Inspect Tools Preferences Help

Track: use netclass width Via: use netclass sizes Grid: 0.1000 mm (0.0039 in) Zoom 5.00

Appearance

Layers Objects Nets

- F.Cu
- B.Cu
- F.Adhesive
- B.Adhesive
- F.Paste
- B.Paste
- F.Silkscreen
- B.Silkscreen
- F.Mask
- B.Mask
- User.Drawings
- User.Comments
- Edge.Cuts
- Margin
- F.Courtyard
- B.Courtyard
- F.Fab
- B.Fab

Layer Display Options

Preset (Ctrl+Tab):

Viewports (Shift+Tab):

Selection Filter

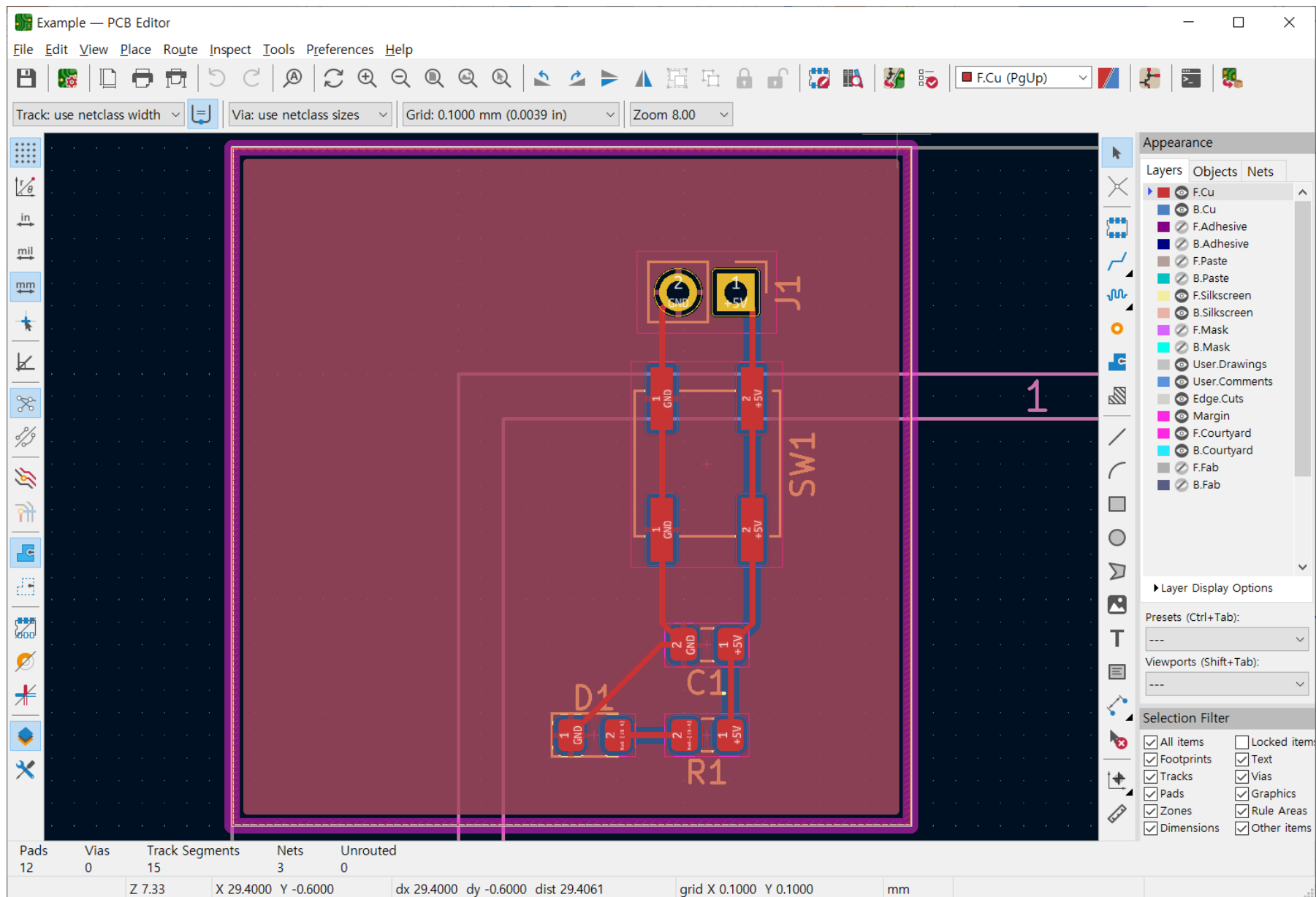
- ☒ All items
- ☒ Footprints
- ☒ Tracks
- ☒ Pads
- ☒ Zones
- ☒ Dimensions
- ☐ Locked items
- ☒ Text
- ☒ Vias
- ☒ Graphics
- ☒ Rule Areas
- ☒ Other items

자동 Routing  
모든 소자 선택 후 Shift + F  
배선 완료 후 단축키 B (Fill)

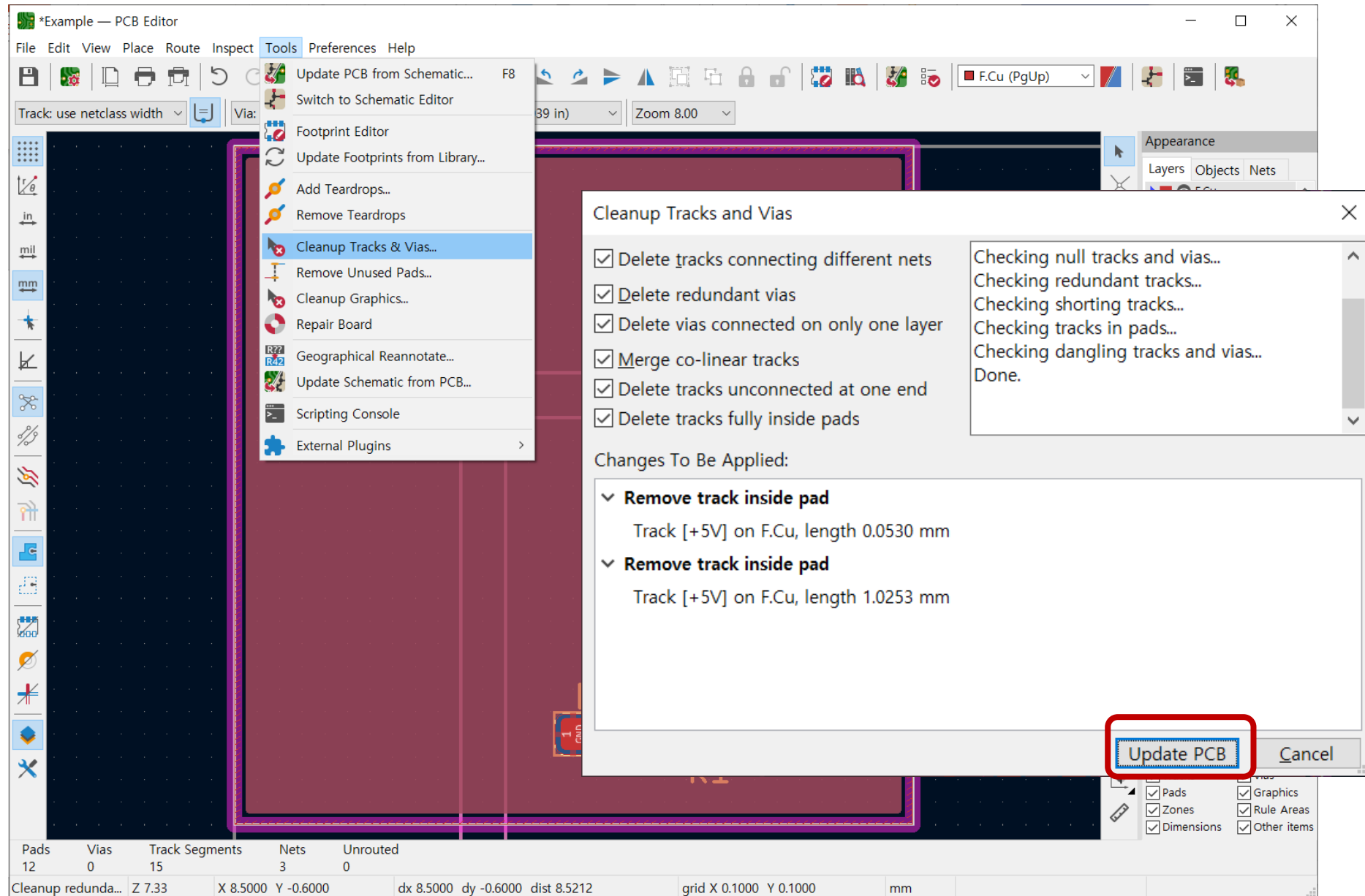
Pads	Vias	Track Segments	Nets	Unrouted
12	0	15	3	0

File 'D:\Work\WKI... Z 4.27 X -5.8000 Y 17.4000 dx -5.8000 dy 17.4000 dist 18.3412 grid X 0.1000 Y 0.1000 mm Select item(s)

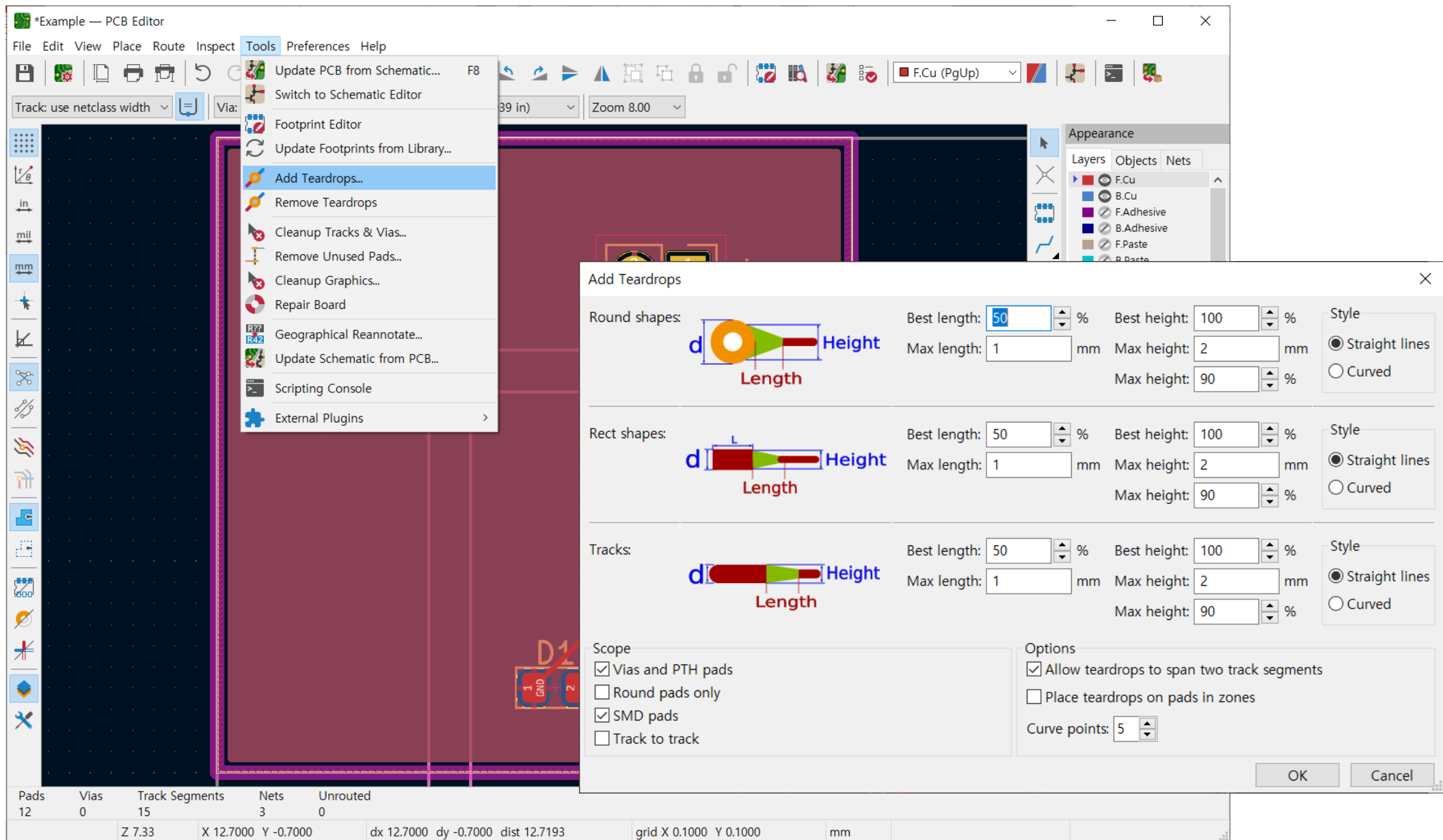
# Board Layout



# Board Layout

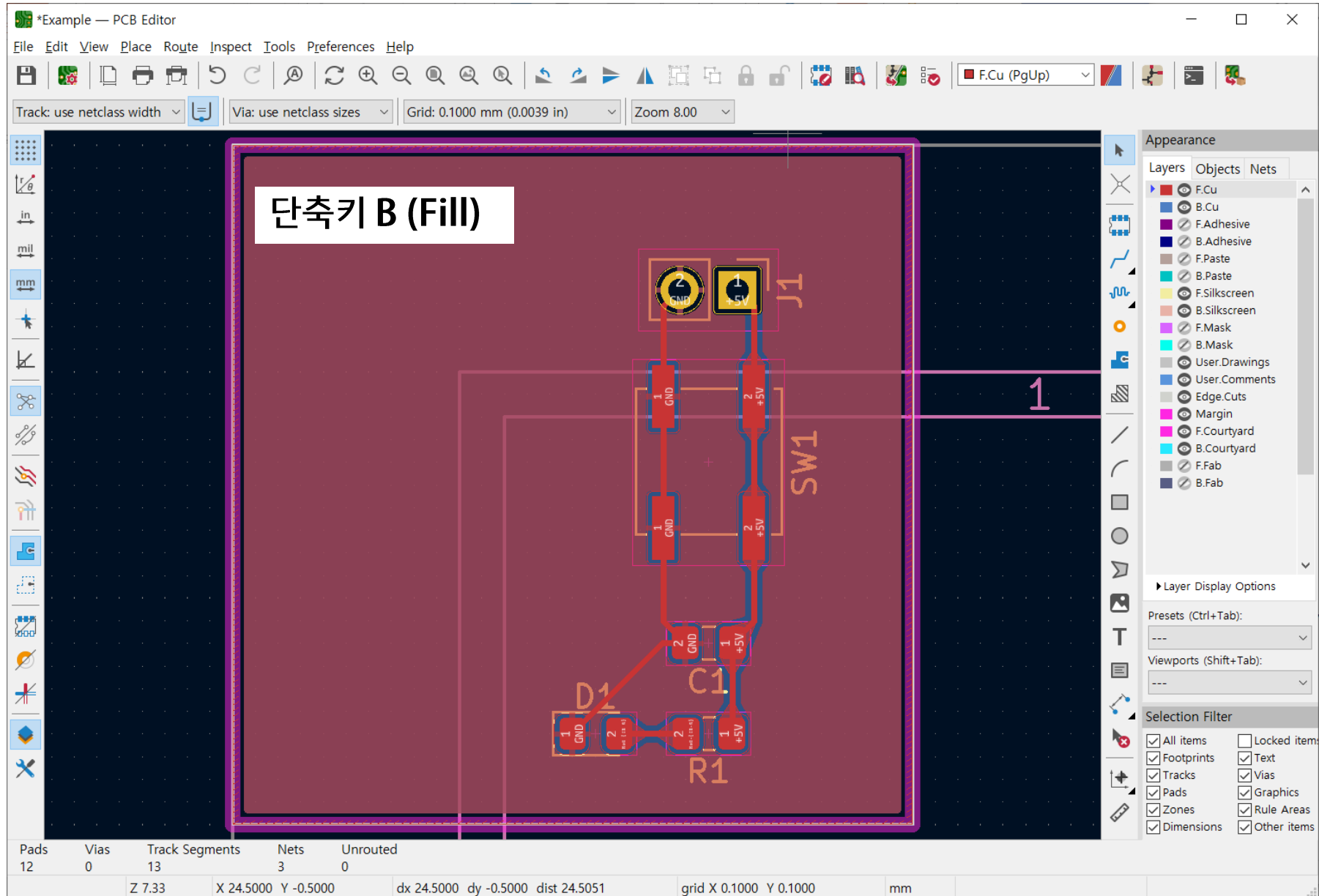


# Board Layout: 선택 사항

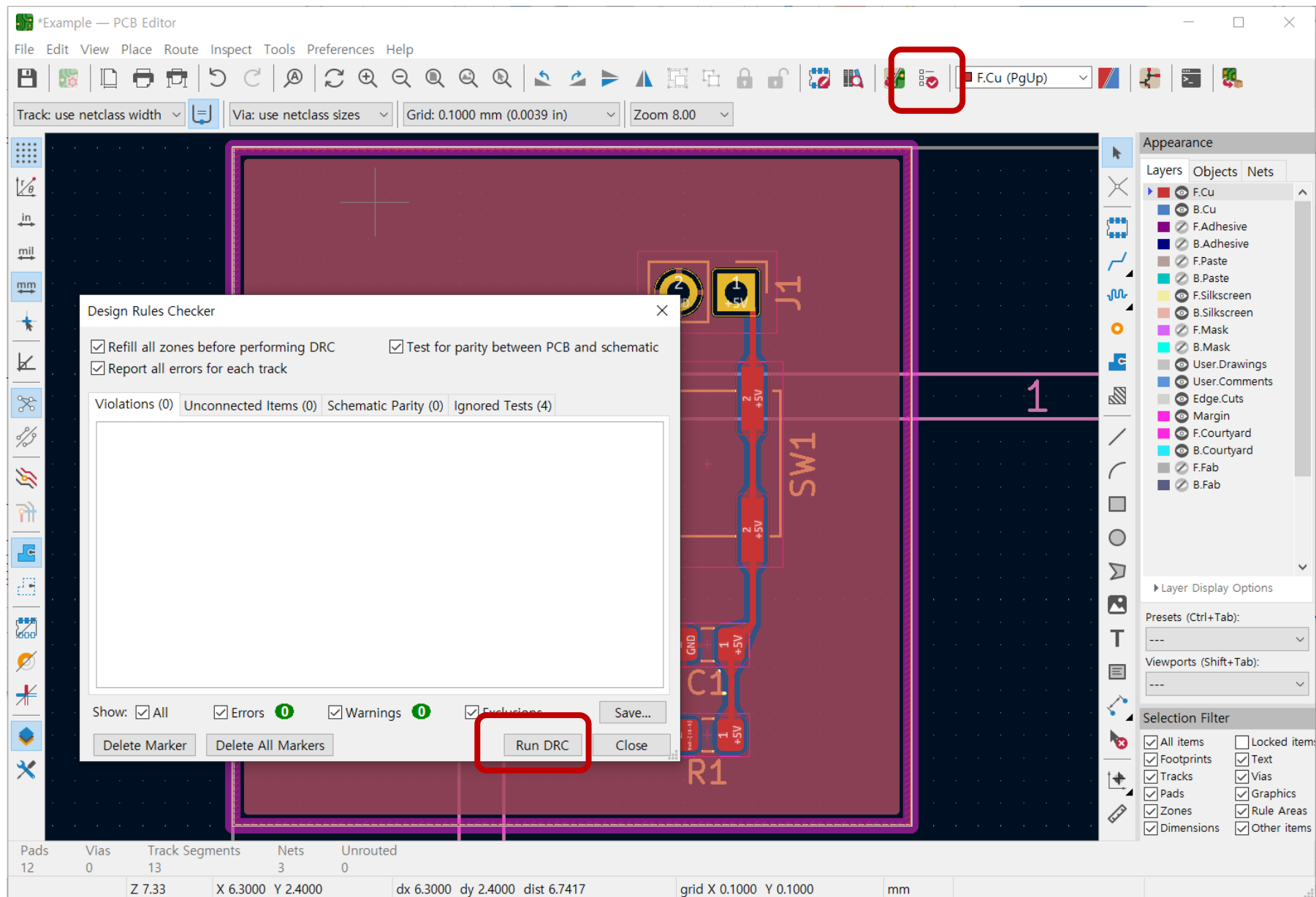




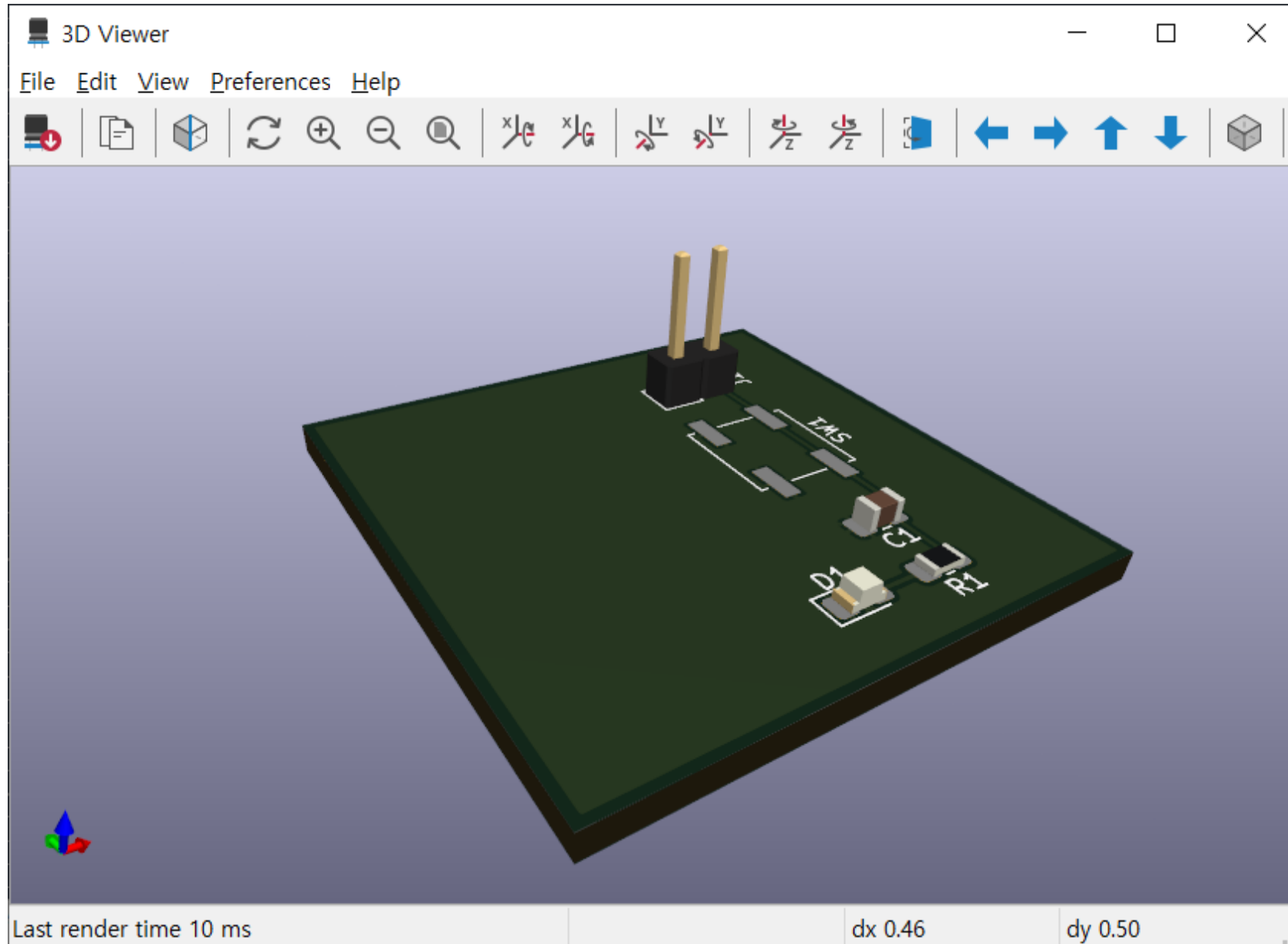
# Board Layout



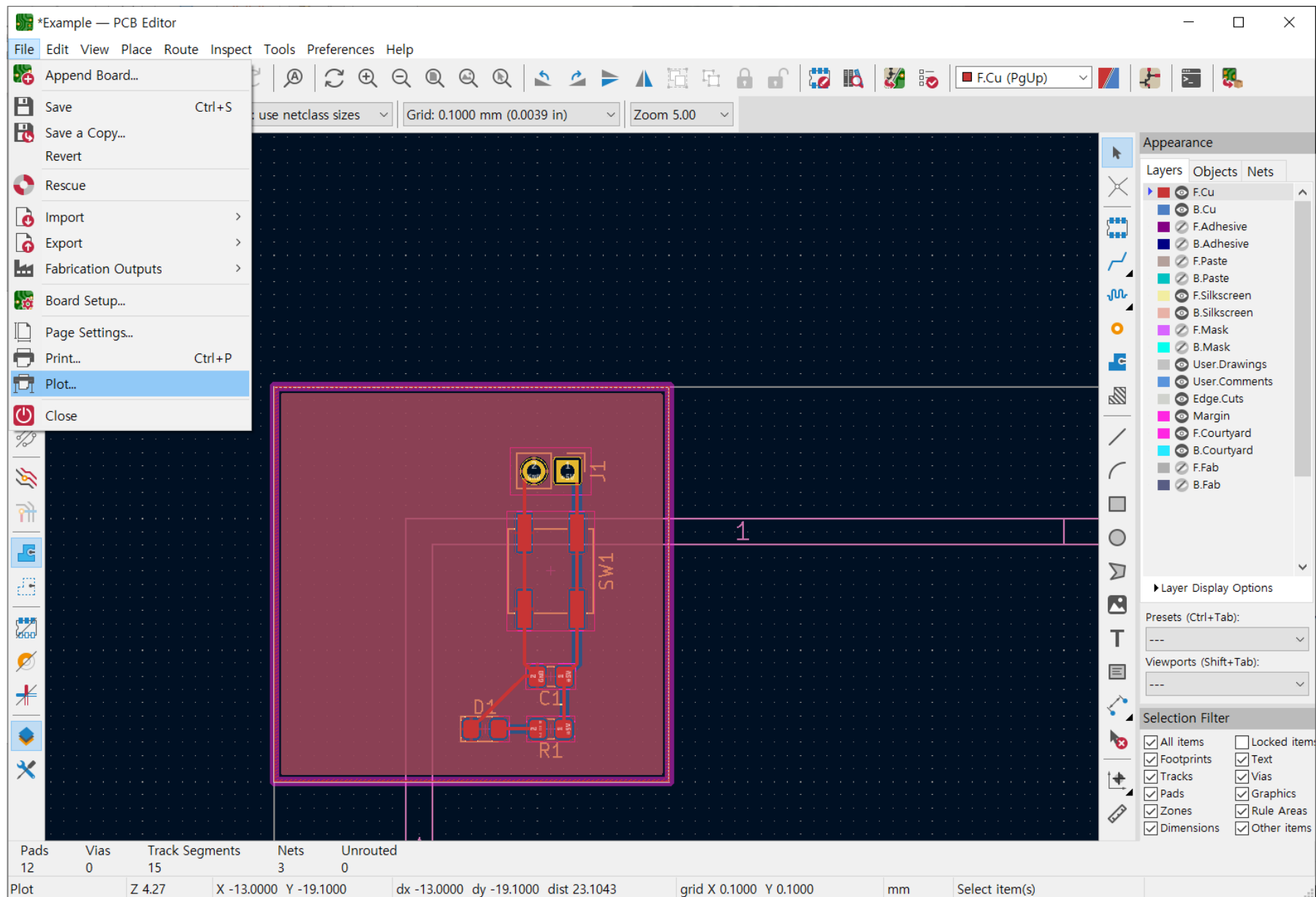
# Board Layout



# 3D Viewer



# Post processing



# Post processing

Plot

Plot format: Gerber Output directory: Gerber/

Include Layers

- ☒ F.Cu
- ☒ B.Cu
- ☐ F.Adhesive
- ☐ B.Adhesive
- ☒ F.Paste
- ☐ B.Paste
- ☒ F.Silkscreen
- ☒ B.Silkscreen
- ☒ F.Mask
- ☒ B.Mask
- ☐ User.Drawings
- ☐ User.Comments
- ☒ Edge.Cuts
- ☐ Margin
- ☐ F.Courtyard
- ☐ B.Courtyard
- ☐ F.Fab

Plot on All Layers

- ☐ F.Cu
- ☐ B.Cu
- ☐ F.Adhesive
- ☐ B.Adhesive
- ☐ F.Paste
- ☐ B.Paste
- ☐ F.Silkscreen
- ☐ B.Silkscreen
- ☐ F.Mask
- ☐ B.Mask
- ☐ User.Drawings
- ☐ User.Comments
- ☐ Edge.Cuts
- ☐ Margin
- ☐ F.Courtyard

General Options

- ☐ Plot drawing sheet
- ☒ Plot footprint values
- ☒ Plot reference designators
- ☐ Force plotting of invisible values / refs
- ☐ Mirrored plot
- ☐ Sketch pads on fabrication layers
- ☒ Check zone fills before plotting

Drill marks: None

Scaling: 1:1

Plot mode: Filled

- ☐ Use drill/place file origin
- ☐ Negative plot
- ☐ Do not tent vias

Gerber Options

- ☐ Use Protel filename extensions
- ☐ Generate Gerber job file
- ☐ Subtract soldermask from silkscreen

Coordinate format: 4,6, unit mm

- ☒ Use extended X2 format (recommended)
- ☒ Include netlist attributes
- ☐ Disable aperture macros (not recommended)

Output Messages

Show: ☐ All ☒ Errors 0 ☒ Warnings 0 ☒ Actions ☒ Infos

Run DRC... Plot Close Generate Drill Files...

Generate Drill Files

Output folder: Gerber/

Drill File Format

- ☒ Excellon
- ☐ Mirror Y axis
- ☐ Minimal header
- ☐ PTH and NPTH in single file
- Oval Holes Drill Mode
  - ☒ Use route command (recommended)
  - ☐ Use alternate drill mode
- ☐ Gerber X2

Map File Format

- ☒ PostScript
- ☐ Gerber X2
- ☐ DXF
- ☐ SVG
- ☐ PDF

Drill Origin

- ☒ Absolute
- ☐ Drill/place file origin

Drill Units

- ☐ Millimeters
- ☒ Inches

Zeros Format

- ☒ Decimal format (recommended)
- ☐ Suppress leading zeros
- ☐ Suppress trailing zeros
- ☐ Keep zeros

Precision: 2:4

Hole Counts

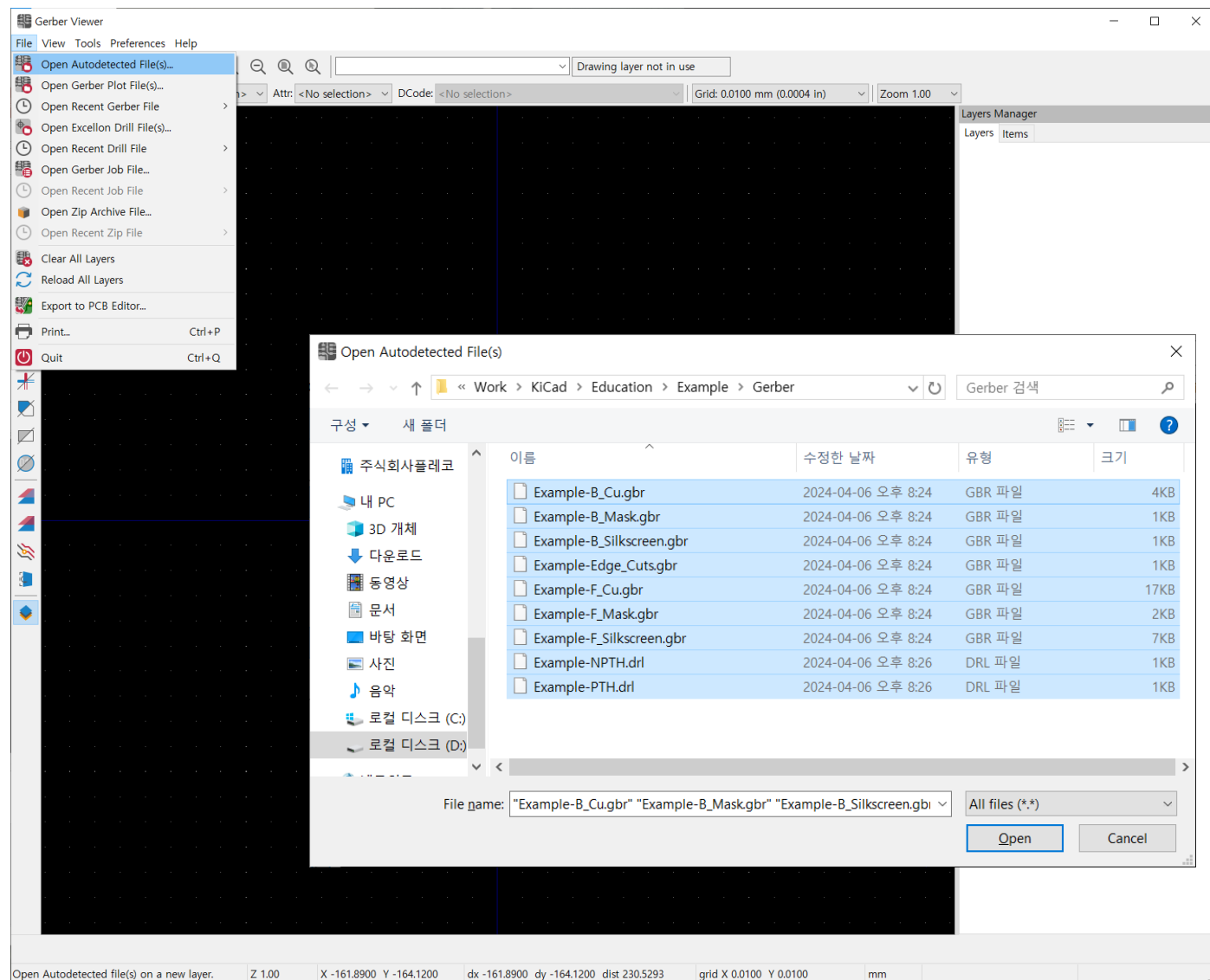
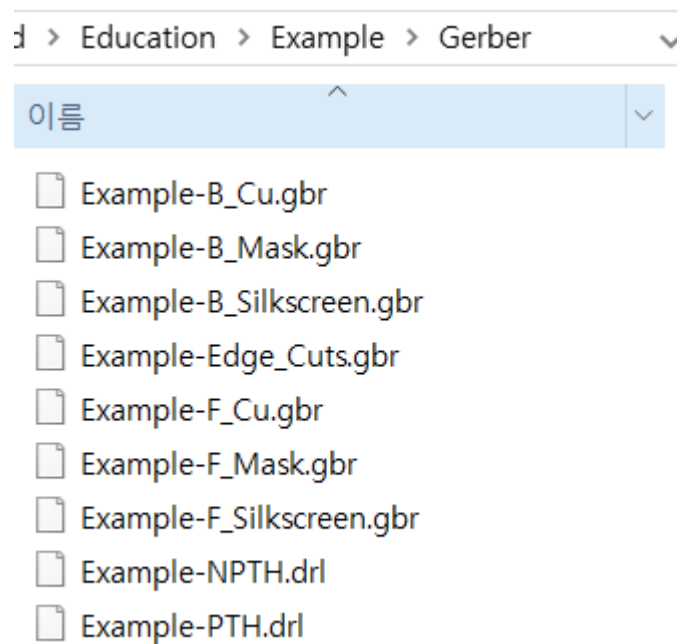
- Plated pads: 2
- Non-plated pads: 0
- Through vias: 0
- Micro vias: 0
- Buried vias: 0

Messages

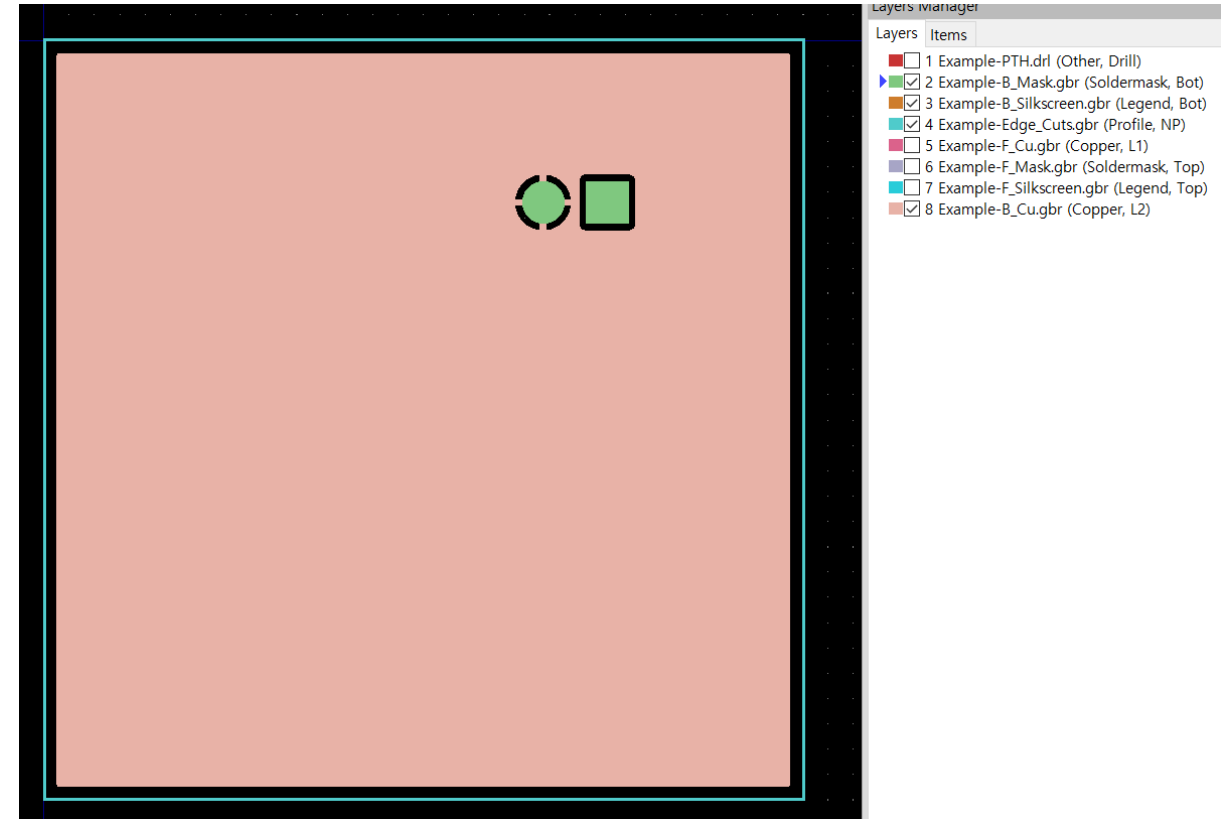
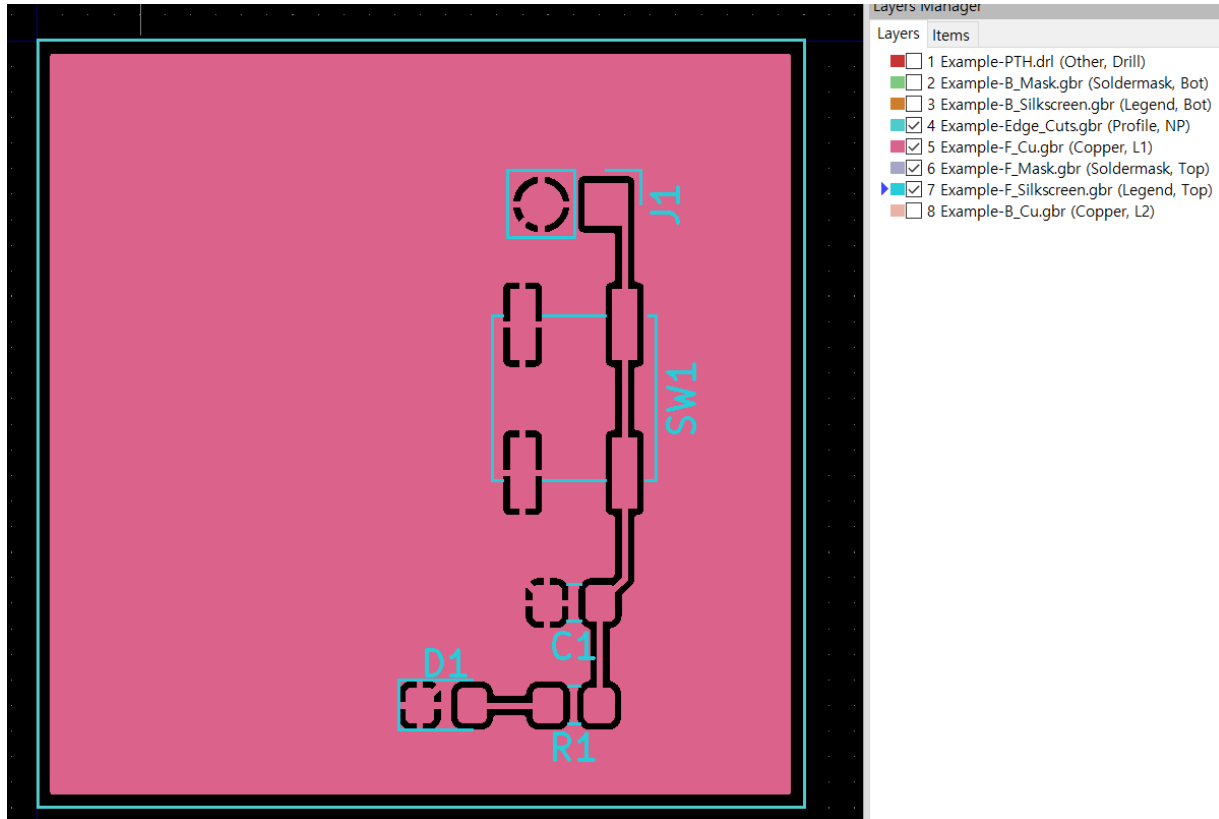
Created file 'D:\Work\KiCad\Education\Example\Gerber\Example-PTH.drl'  
Created file 'D:\Work\KiCad\Education\Example\Gerber\Example-NPTH.drl'  
Done.

Generate Report File... Generate Drill File Close Generate Map File

# Post processing



# Post processing: Check result



# Thank you!

benkim@plecko.biz



<https://www.plecko.biz>

@plecko\_company