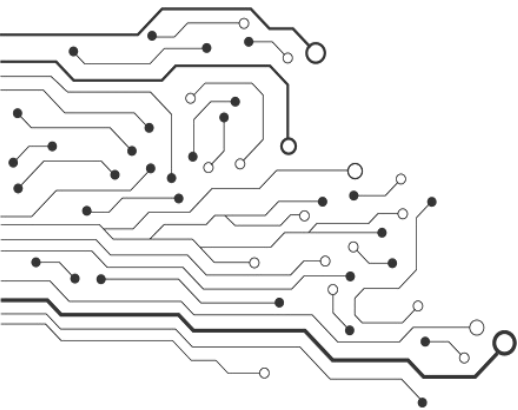
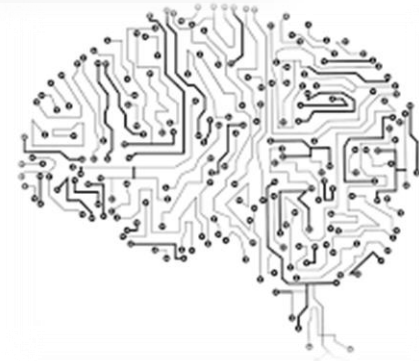


OrCAD - PCB Editor



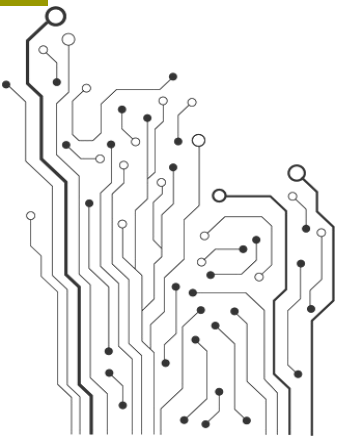
Dr. Sarwan Singh
NIELIT Chandigarh



Agenda



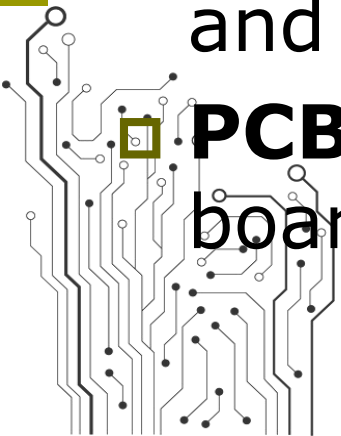
- ❑ Regulated Power Supply
 - Creating schematic, PSpice simulation
- ❑ Creating Netlist
- ❑ Footprint
- ❑ Component placement
- ❑ Routing





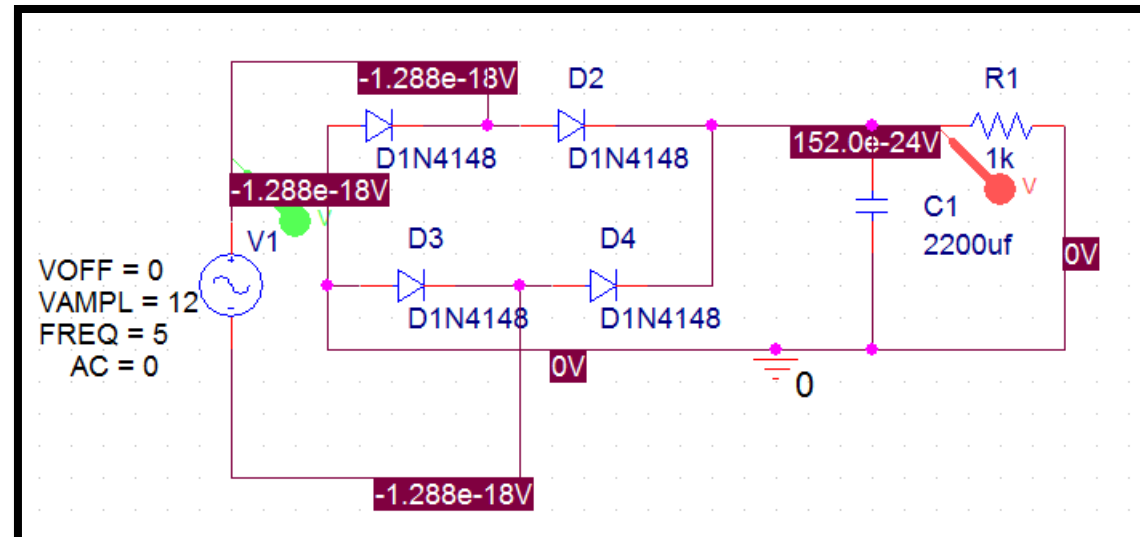
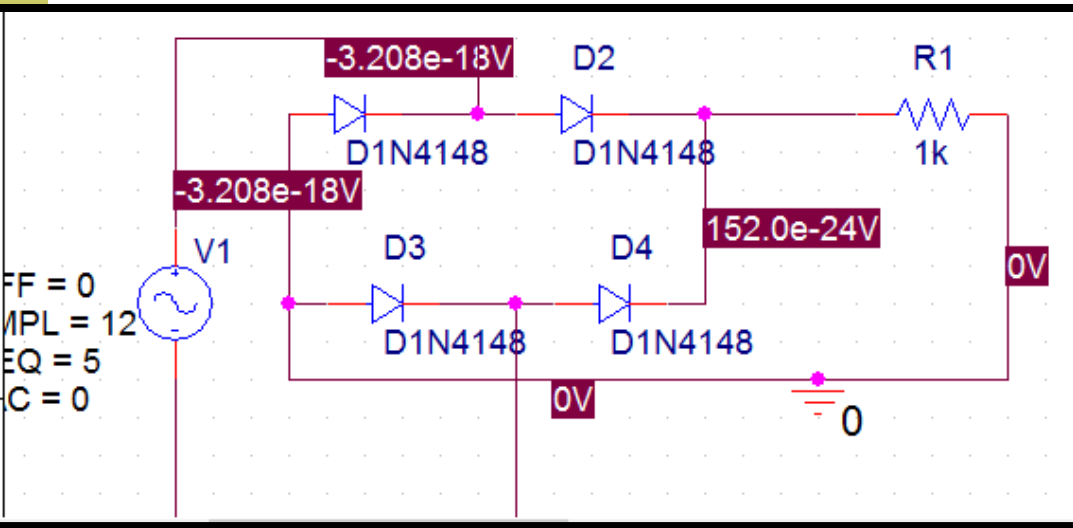
PSpice comprises three main applications

- ❑ **Capture** is used to draw a circuit on the screen, known formally as schematic capture. It offers great flexibility compared with a traditional pencil and paper drawing, as design changes can be incorporated and errors corrected quickly and easily
- ❑ **PSpice** simulates the captured circuit. You can analyse its behaviour in many ways and confirm that it performs as specified.
- ❑ **PCB Editor** is used to design printed circuit boards

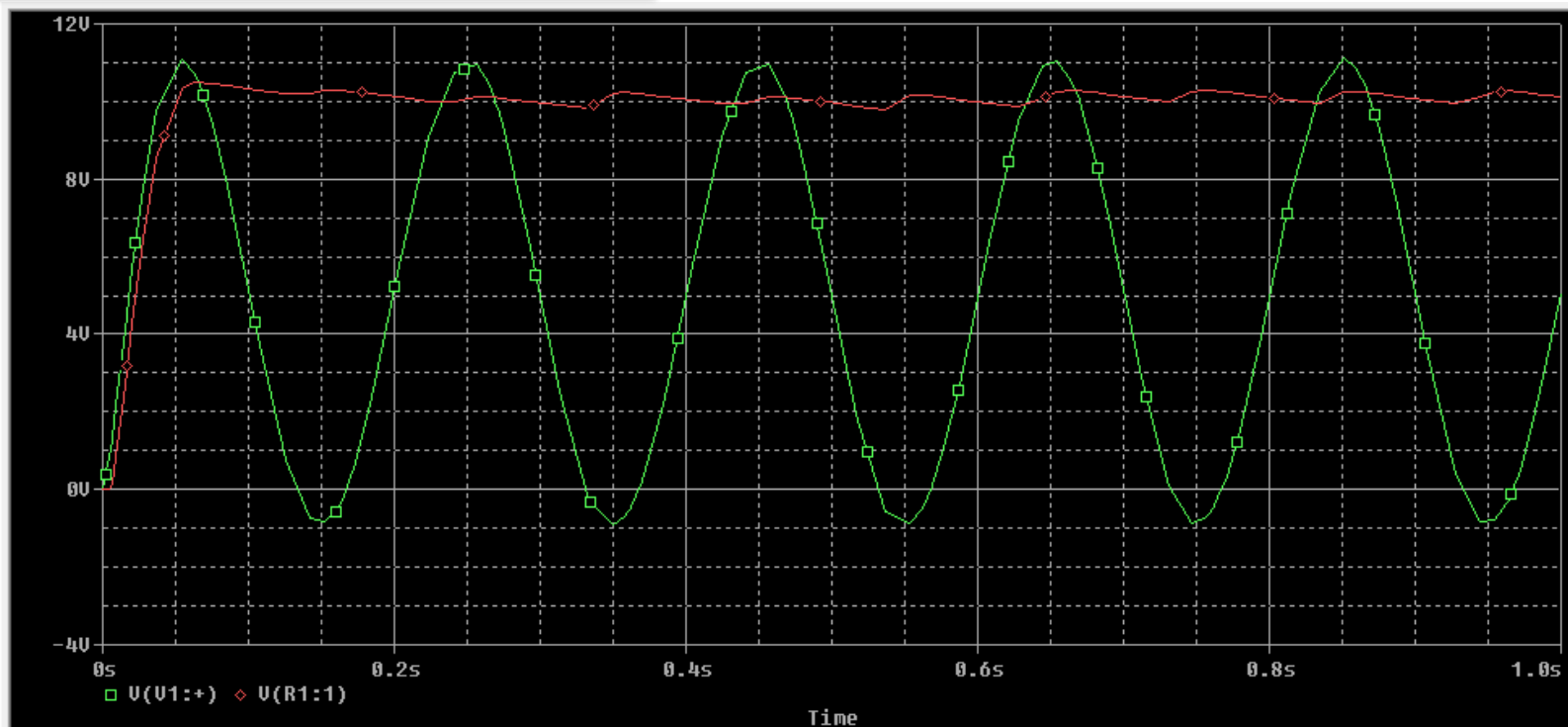
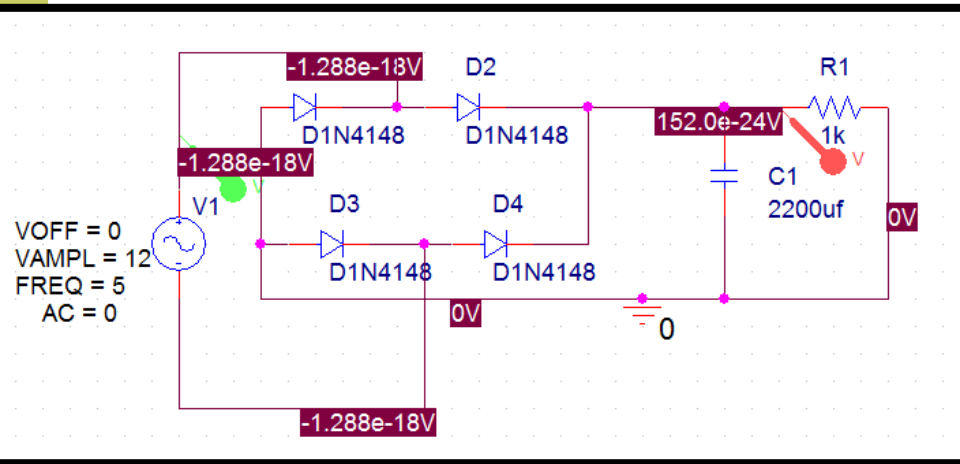




Bridge Rectifier-Power Supply



Capacitor as filter



Placing regulator LM7805

bridgeSup... * PAGE1* Start Page

Place Part

Part: LM7805C

Part List:

- LM723/OPAMP
- LM723C/OPAMP
- LM7301/NS/NAT_SEMI
- LM741/OPAMP
- LM741/NS/NAT_SEMI
- LM759CH/OPAMP
- LM759CP/OPAMP
- LM7805C/OPAMP

C:\CADENCE\SPB_17.2\TOOLS\CAPTURE\LIBRARY\PSICE\OPAMP.OLB

1_SHOT
7400
74AC
74ACT
74ALS
74AS

U? LM7805C

1 IN 2 OUT 3 GND

Packaging: Parts per Pkg: 1

Part: Type: Homogeneous

Normal Convert

+ Search for Part

INI File Location: C:\SPB_Data\cdssetup\OrCAD_Capture\17.2.0\capture.ini

Placing LED

bridgeSup... PAGE1* Start Page

U1
LM7805C
IN GND OUT
1 2 3

C1
2200uf

N4148

0

R1
1k

D5
MLED81

Place Part

Part
MLED81

Part List:

- MJH16004/TO/PWRBJT
- MJH16106/TO/PWRBJT
- MJH16110/TO/PWRBJT
- MJH6284/DARLNGTN
- mjh6284/ON/ON_BJT
- MJH6287/DARLNGTN
- mjh6287/ON/ON_BJT
- MLED81/OPTO**

Libraries:

- 1_SHOT
- 7400
- 74AC
- 74ACT
- 74ALS
- 74AS

Packaging
Parts per Pkg: 1

Part:

Type: Homogeneous

D?

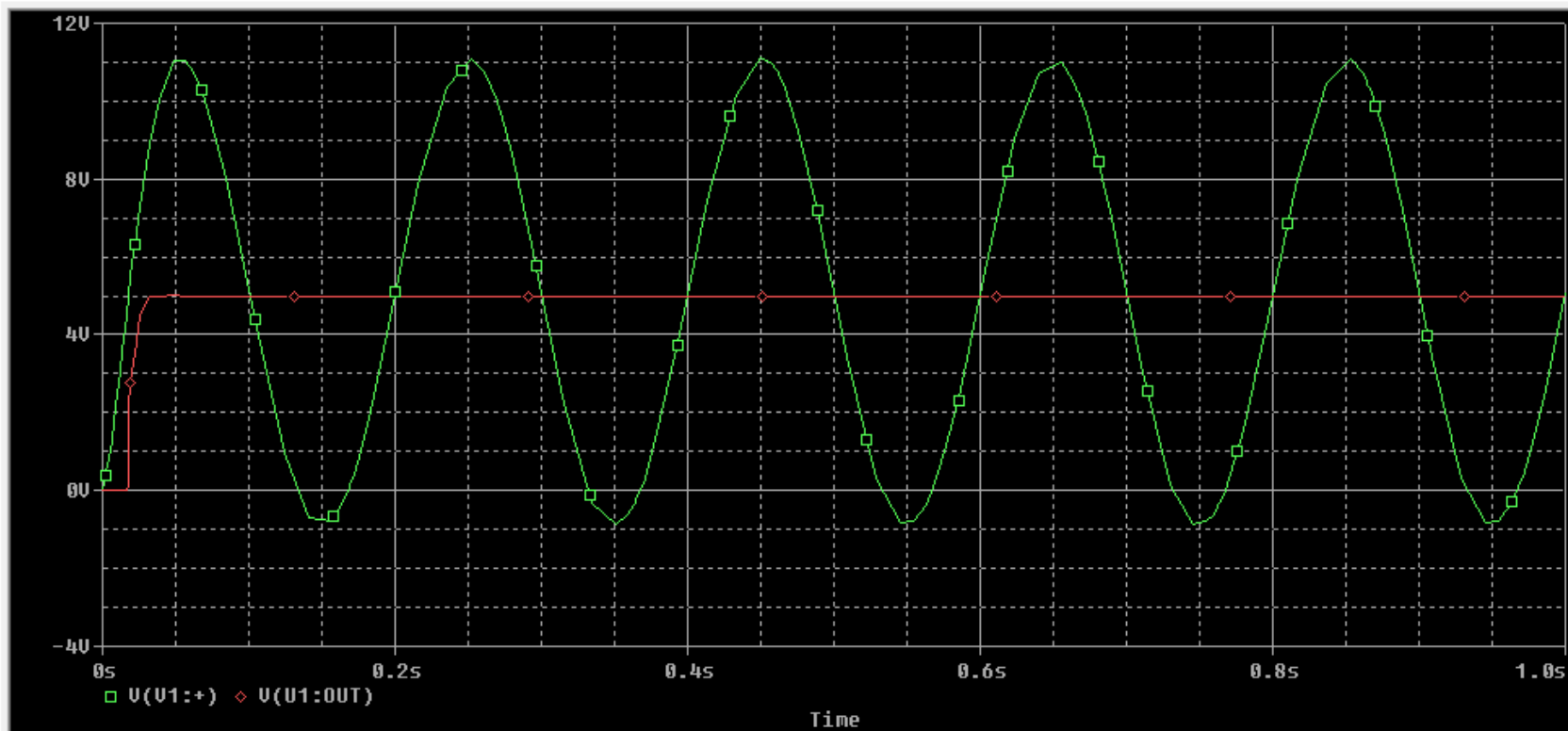
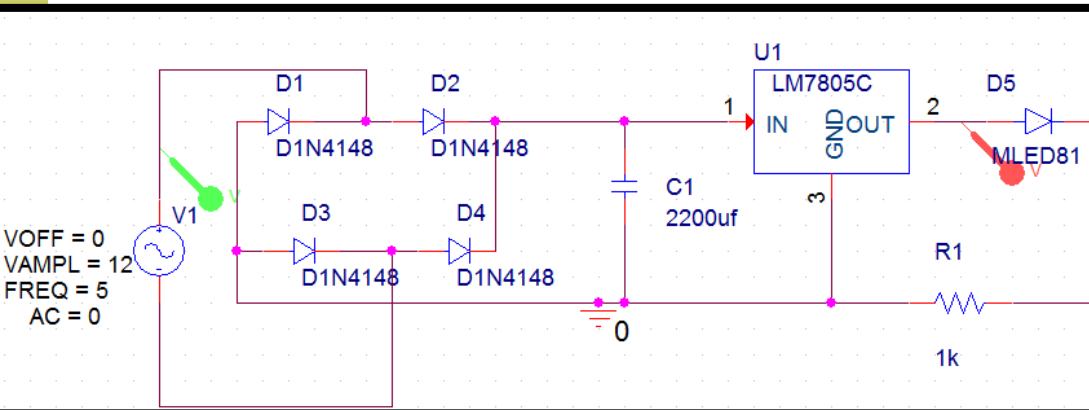
MLED81

☒ Normal ☐ Convert

+ Search for Part

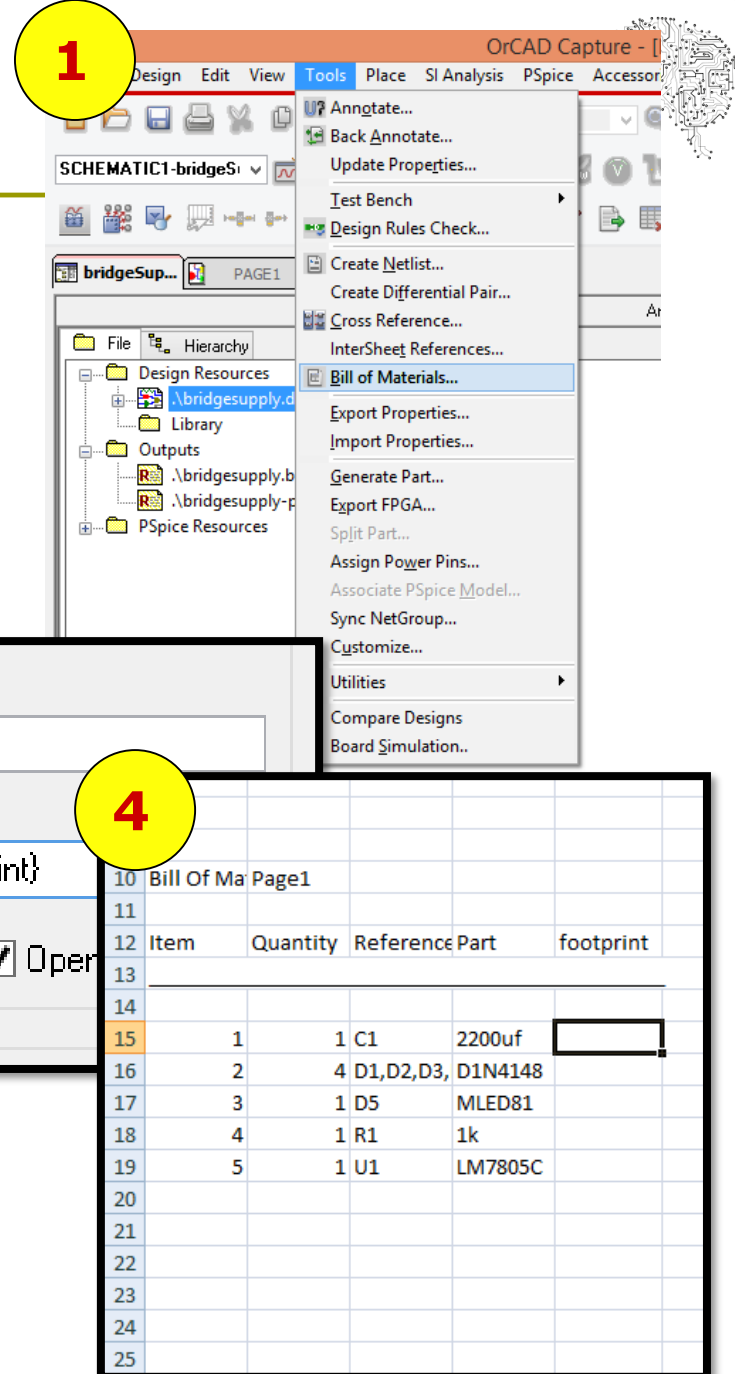
(5.30, 1.70)
INFO(ORCAP-2191): Creating PSpice Netlist
INFO(ORNET-1041): Writing PSpice Flat Netlist F:\Orcad\POWERsupply\bridgeSupply-PSpiceFile
INFO(ORNET-1156): PSpice netlist generation complete
INFO(ORCAP-2191): Creating PSpice Netlist
INFO(ORNET-1041): Writing PSpice Flat Netlist F:\Orcad\POWERsupply\bridgeSupply-PSpiceFile

Simulating Power Supply



Create Netlist

1



OrCAD Capture - I

Design Edit View Tools Place SI Analysis PSpice Accessory

Tools

- Annotate...
- Back Annotate...
- Update Properties...
- Test Bench
- Design Rules Check...
- Create Netlist...
- Create Differential Pair...
- Cross Reference...
- InterSheet References...
- Bill of Materials...**
- Export Properties...
- Import Properties...
- Generate Part...
- Export FPGA...
- Spjrit Part...
- Assign Power Pins...
- Associate PSpice Model...
- Sync NetGroup...
- Customize...
- Utilities
- Compare Designs
- Board Simulation...

SCHEMATIC1-bridgeS

bridgeSup... PAGE1

File Hierarchy

- Design Resources
 - \bridgesupply.d
 - Library
 - Outputs
 - \bridgesupply.b
 - \bridgesupply-p
 - PSpice Resources

2

Bill of Materials

Mode

☒ Process entire design

☐ Process selection

☒ Use instances (Preferred)

☐ Use occurrences

OK

Cancel

Help

Line Item Definition

Header:

Item\Quantity\Reference\Part

Combined property string:

{Item}\{Quantity}\{Reference}\{Value}

☐ Place each part entry on a separate line in Excel

Include File

☐ Merge an include file with report

Combined property string:

{Item}\{Quantity}\{Reference}\{Value}

Include file:

F:\ORCAD\POWERSUPPLY\BRIDGE

Report

Report File: ☐ View Output

F:\ORCAD\POWERSUPPLY\BRIDGE

3

Header:

Item\Quantity\Reference\Part\footprint

Combined property string:

{Item}\{Quantity}\{Reference}\{Value}\{footprint}

☐ Place each part entry on a separate line

☒ Open

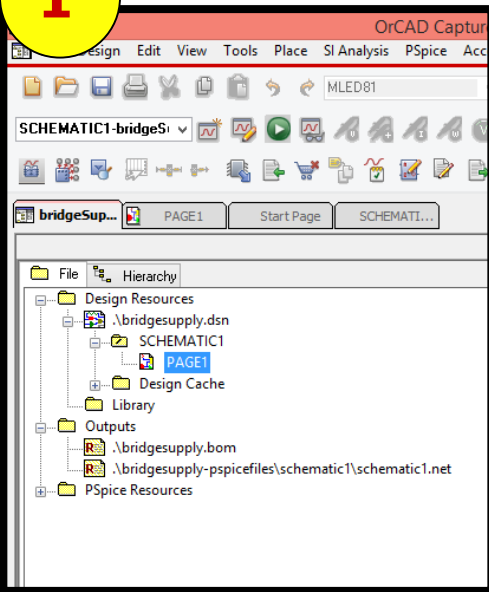
Include File

4

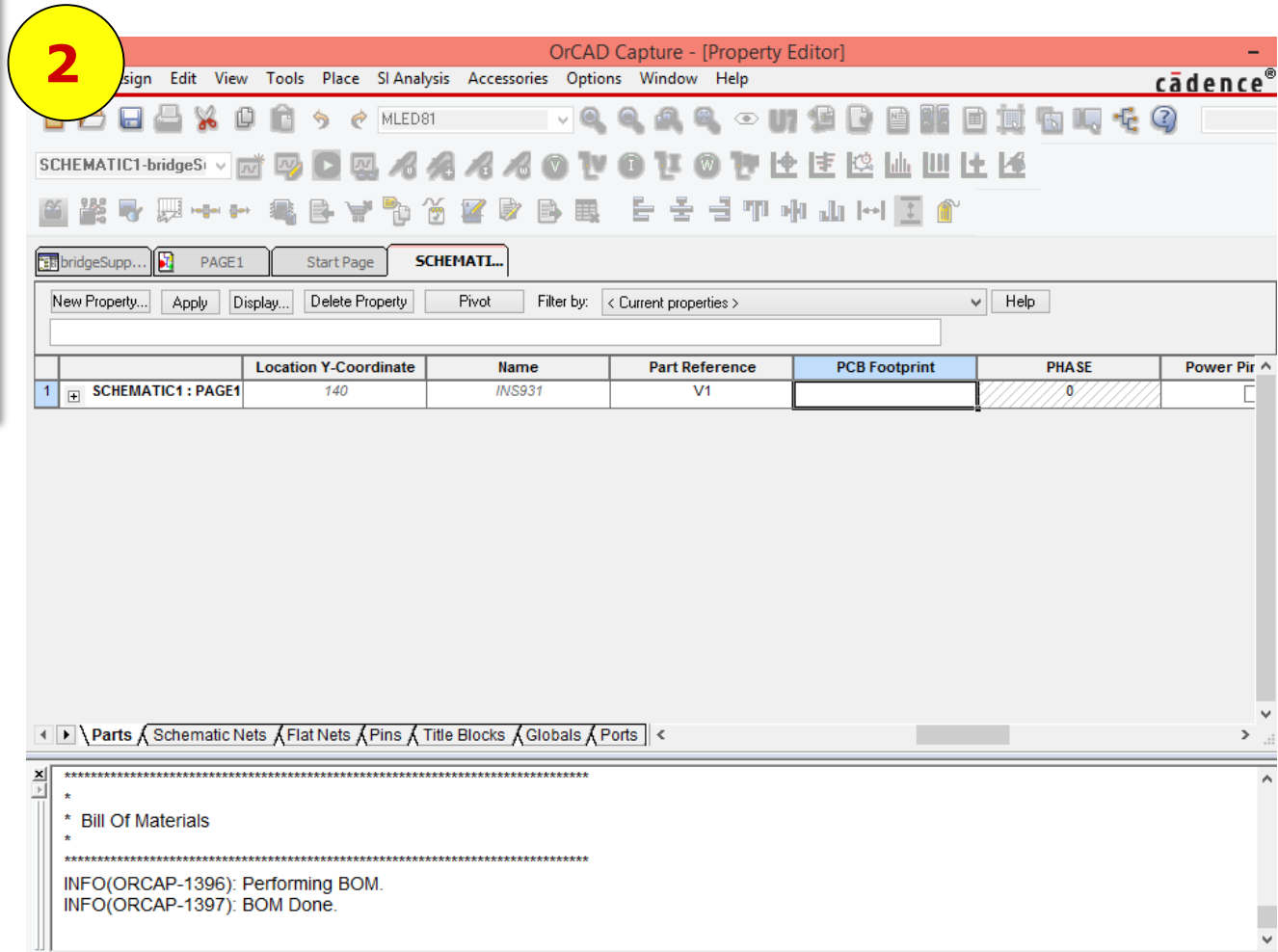
Item	Quantity	Reference	Part	footprint
1	1	C1	2200uf	
2	4	D1,D2,D3	D1N4148	
3	1	D5	MLED81	
4	1	R1	1k	
5	1	U1	LM7805C	

Schematic - footprint

1



2



PCB footprint

1

Bill of Materials

Scope: ☒ Process entire design ☐ Process selection

Mode: ☒ Use instances (Preferred) ☐ Use occurrences

OK Cancel Help

Line Item Definition

Header: Item\Quantity\Reference\Part\PCB footprint

Combined property string: {Item}\t{Quantity}\t{Reference}\t{Value}\t{PCB footprint}

☐ Place each part entry on a separate line ☒ Open in Excel

Include File

☐ Merge an include file with report

Combined property string: {Item}\t{Quantity}\t{Reference}\t{Value}

Include file: F:\ORCAD\POWERSUPPLY\BRIDGESUPPLY Browse...

Report

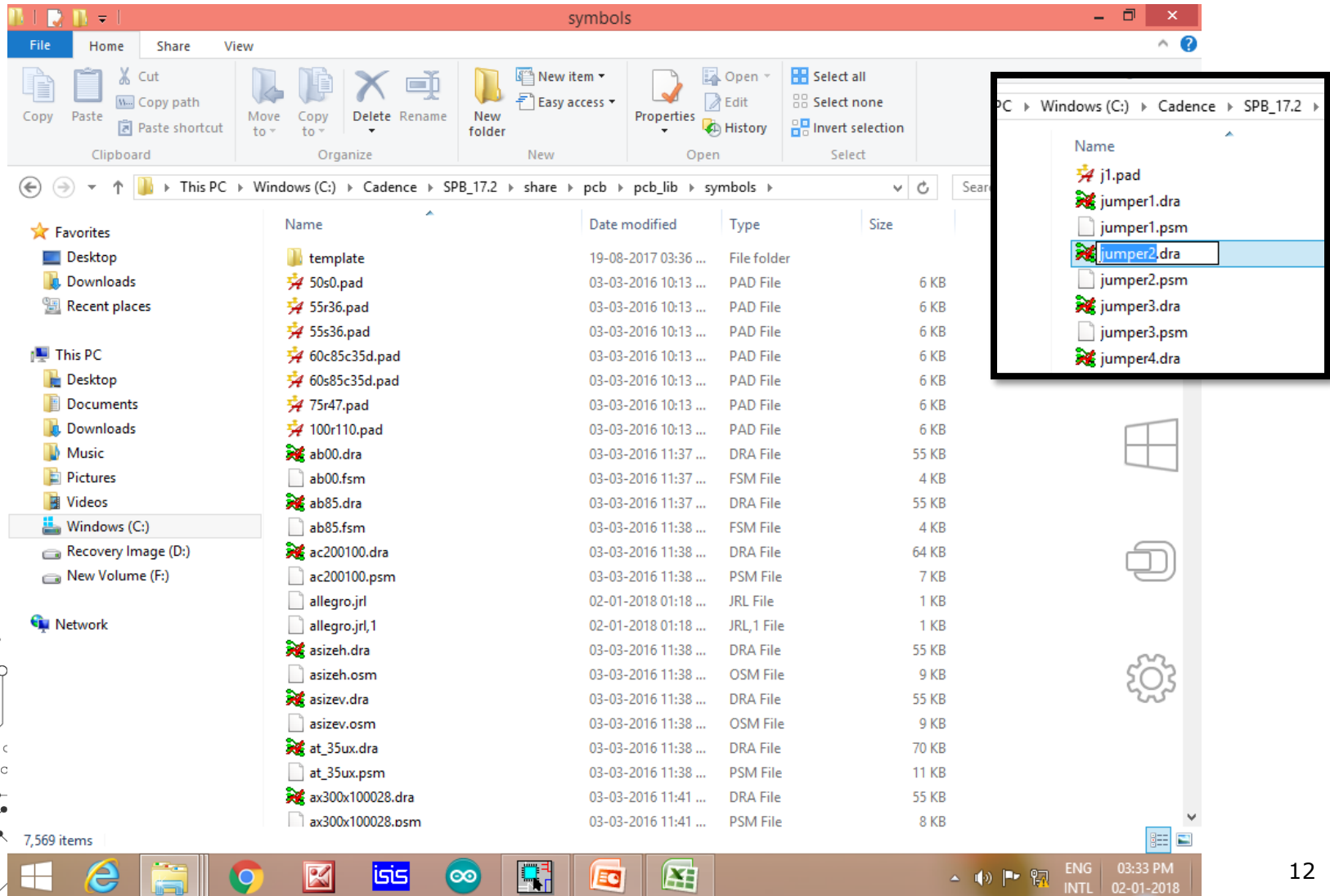
Report File: ☐ View Output F:\ORCAD\POWERSUPPLY\BRIDGESUPPLY.B Browse...

2

Bill Of Ma Page1				
Item	Quantity	Reference	Part	PCB footprint
1	1	C1	2200uf	cap196
2	4	D1,D2,D3,	D1N4148	dax2do35
3	1	D5	MLED81	dax1do35
4	1	R1	1k	AXRC05
5	1	U1	LM7805C	to3

Verifying footprint

C:\Cadence\SPB_17.2\share\pcb\pcb_lib\symbols

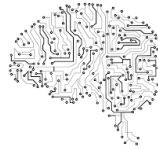


The screenshot shows a Windows File Explorer window titled 'symbols' with the address bar displaying the path: This PC > Windows (C:) > Cadence > SPB_17.2 > share > pcb > pcb_lib > symbols. The file 'jumper2.dra' is selected in the main list. An inset window on the right shows a detailed view of the selected file, listing the following files:

- j1.pad
- jumper1.dra
- jumper1.psm
- jumper2.dra** (selected)
- jumper2.psm
- jumper3.dra
- jumper3.psm
- jumper4.dra

The main window displays a list of files and folders with columns for Name, Date modified, Type, and Size. The list includes various files like 'template', '50s0.pad', '55r36.pad', '55s36.pad', '60c85c35d.pad', '60s85c35d.pad', '75r47.pad', '100r110.pad', 'ab00.dra', 'ab00.fsm', 'ab85.dra', 'ab85.fsm', 'ac200100.dra', 'ac200100.psm', 'allegro.jrl', 'allegro.jrl,1', 'asizeh.dra', 'asizeh.osm', 'asizev.dra', 'asizev.osm', 'at_35ux.dra', 'at_35ux.psm', 'ax300x100028.dra', and 'ax300x100028.psm'.

Diode footprint



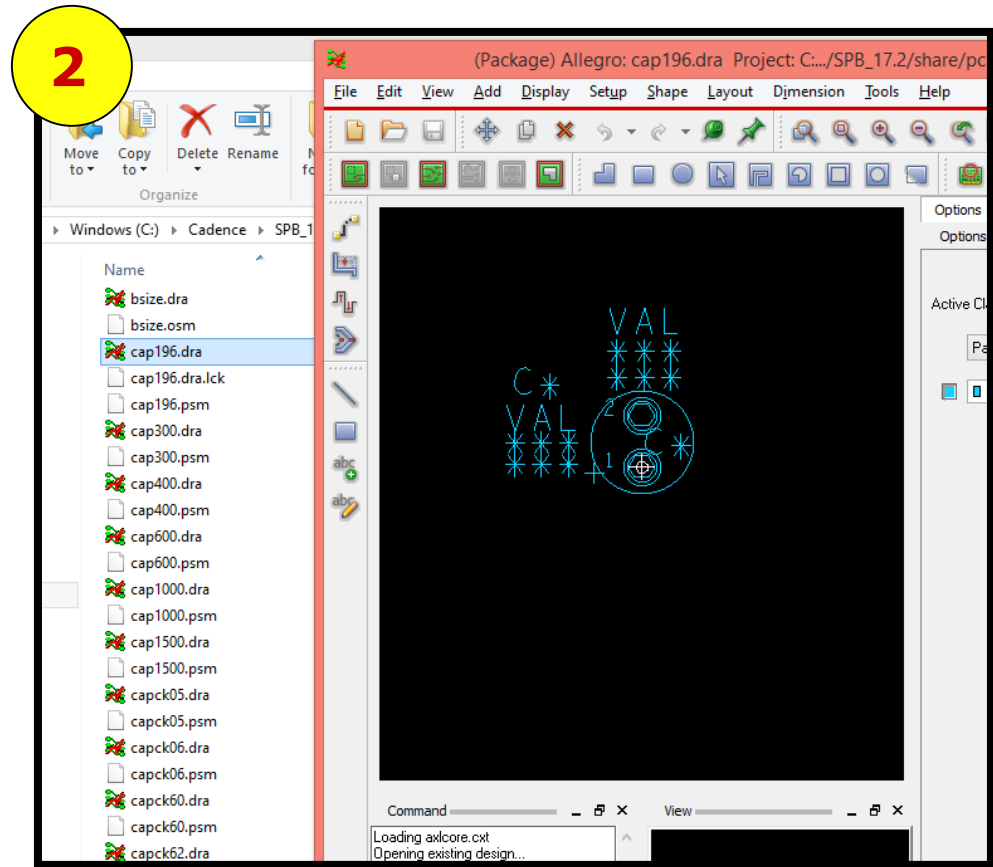
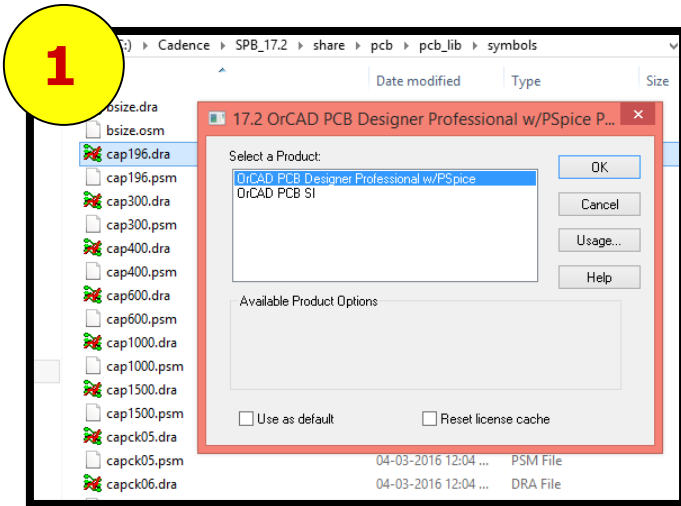
1

PAGE1* Star1 Page SCHEMATICL*					
	Apply	Display...	Delete Property	Pivot	Filter by: < Current properties > Help
	Location Y-Coordinate	Name	Part Reference	PCB Footprint	Power Pins Visible
SCHEMATIC1: PAGE1: D1	90	INS839	D1	dax2do35	<input type="checkbox"/>
SCHEMATIC1: PAGE1: D2	90	INS859	D2	dax2do35	<input type="checkbox"/>
SCHEMATIC1: PAGE1: D3	140	INS965	D3	dax2do35	<input type="checkbox"/>
SCHEMATIC1: PAGE1: D4	140	INS985	D4	dax2do35	<input type="checkbox"/>

2

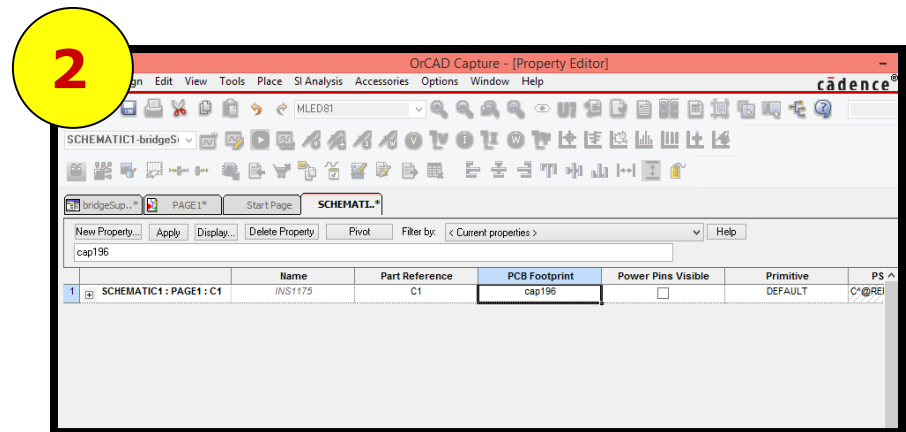
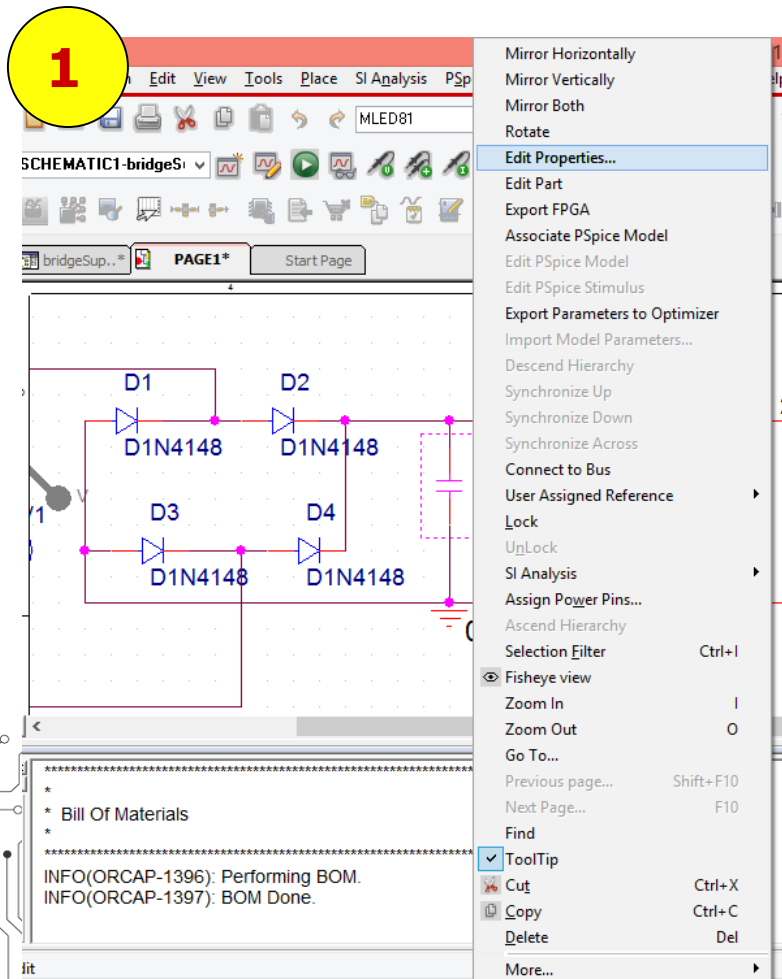
The screenshot shows the Allegro PCB Editor interface. The 'symbols' panel is open, displaying a list of files in the 'C:\Program Files\Allegro\lib\sym' directory. The file 'dax1do35.dra' is selected. The file explorer window shows the contents of the 'C:\Program Files\Allegro\lib\sym' directory, with 'dax1do35.dra' highlighted. The main workspace shows a diode footprint symbol.

Capacitor footprint



Changing footprint

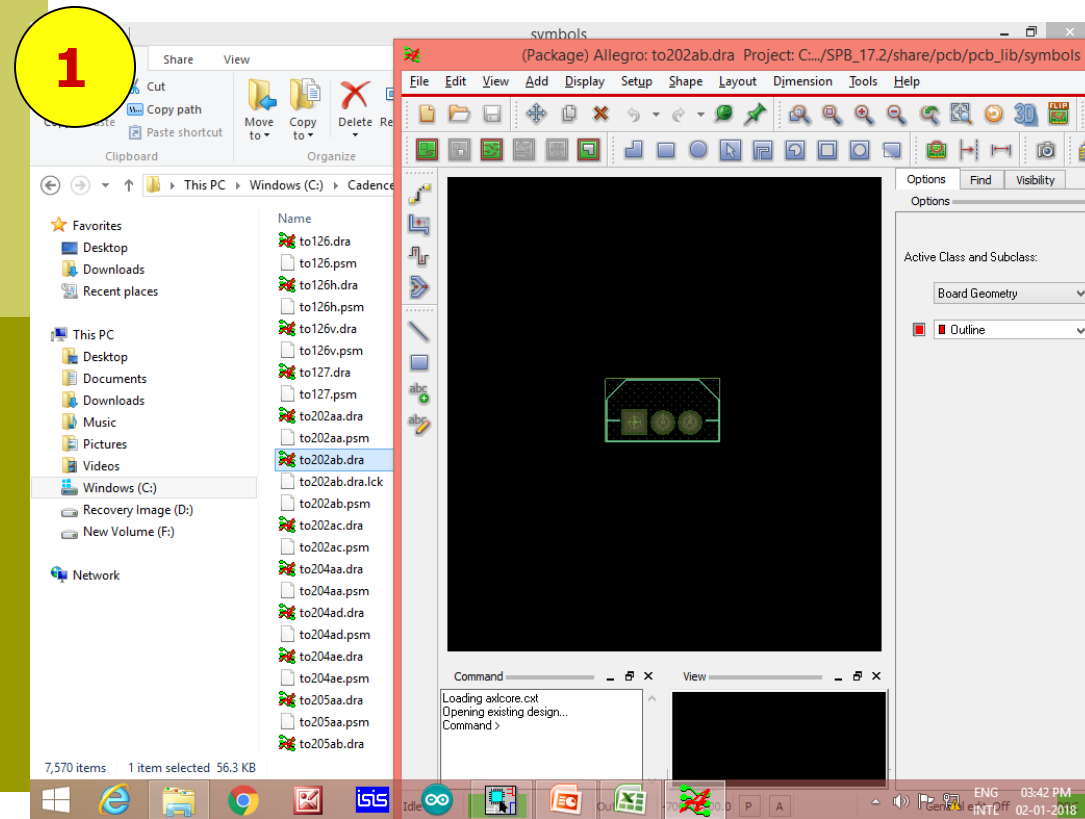
□ Right click on component > Edit properties



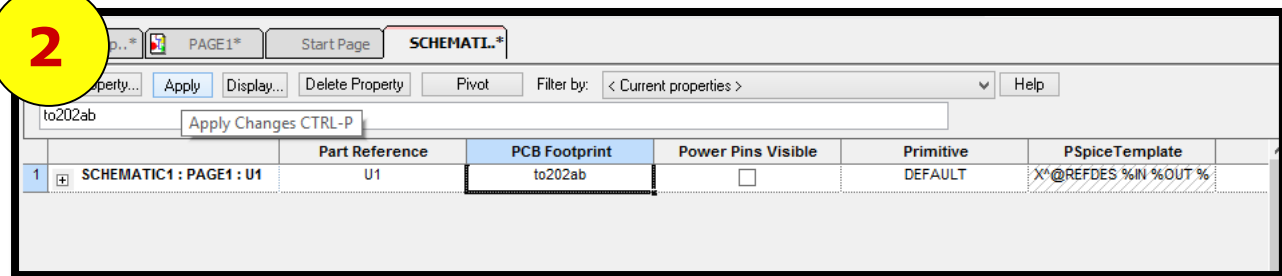
Footprint - 7805



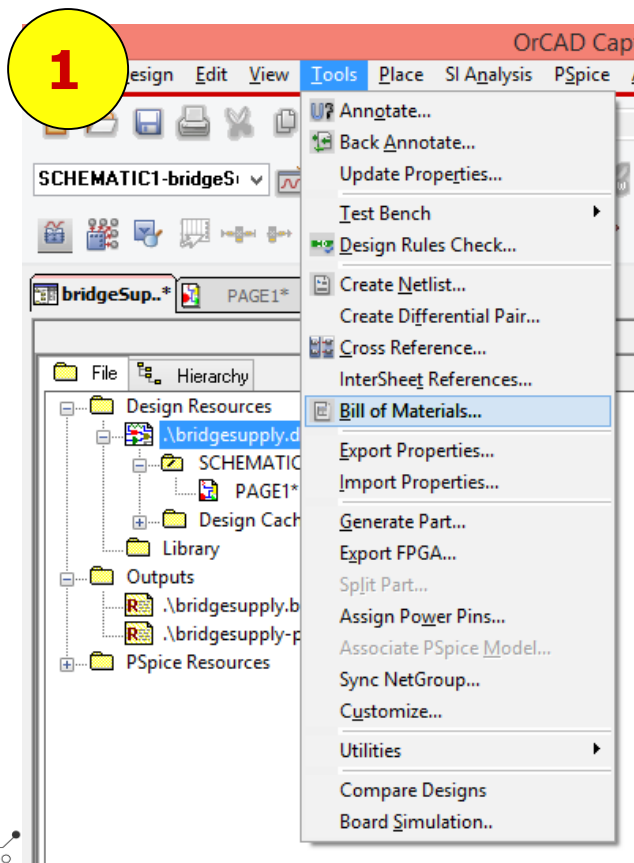
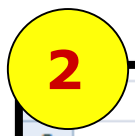
1



2



Update BoM with Footprint

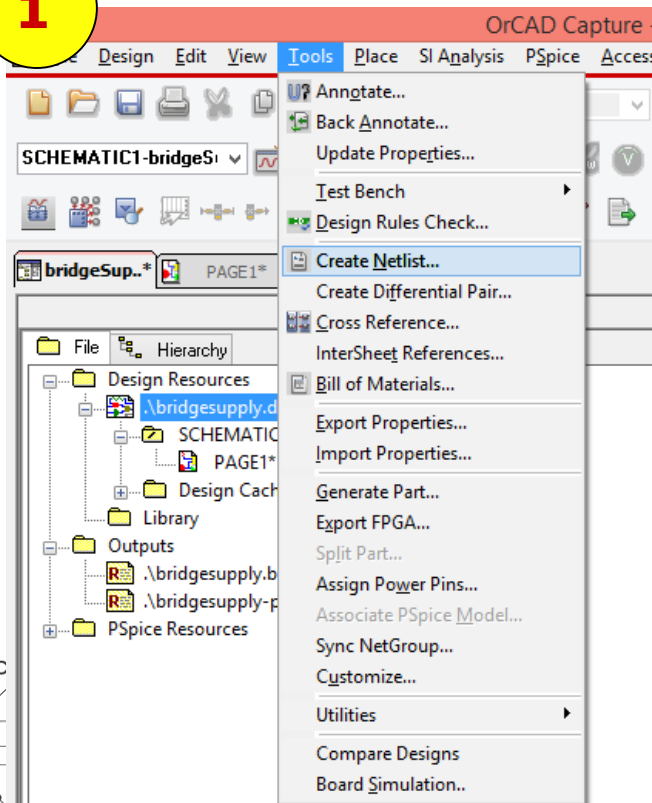



8				
9				
10	Bill Of Materials Page1			
11				
12	Item	Quantity	Reference Part	PCB footprint
13				
14				
15	1	1 C1	2200uf	cap196
16	2	4 D1,D2,D3,	D1N4148	dax2do35
17	3	1 D5	MLED81	dax1do35
18	4	1 R1	1k	AXRC05
19	5	1 U1	LM7805C	to202ab
20				
21				

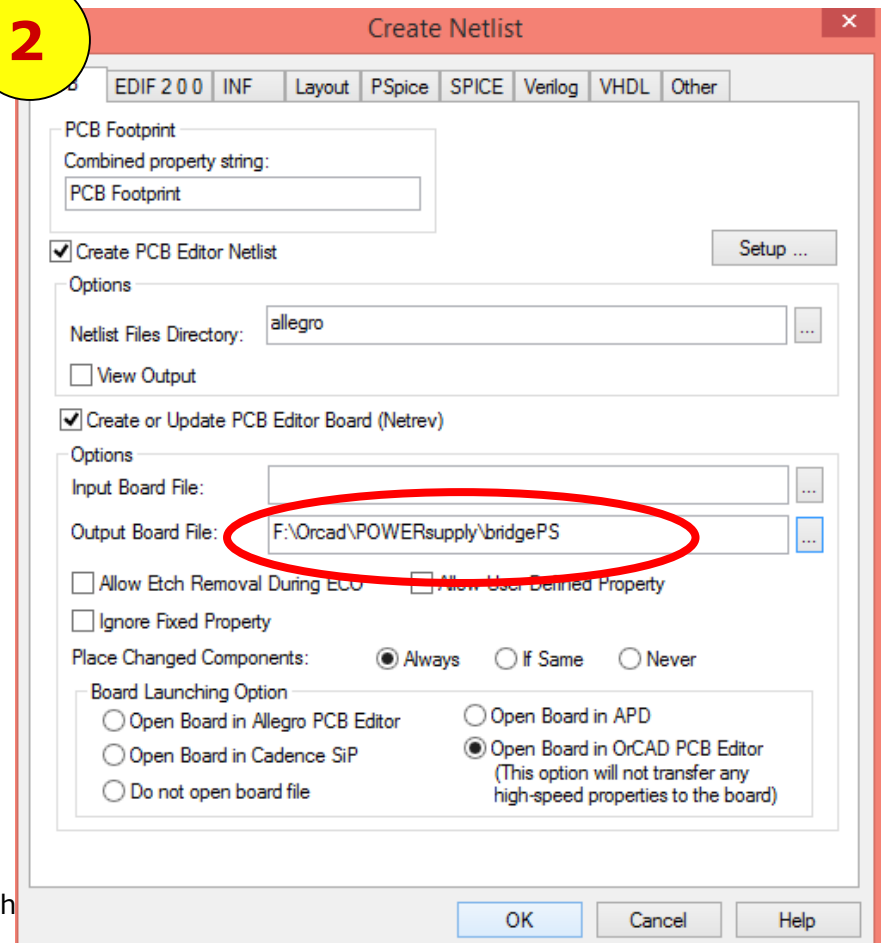
Creating Netlist

1. Tools > Create Netlist
2. check create or update PCB editor board

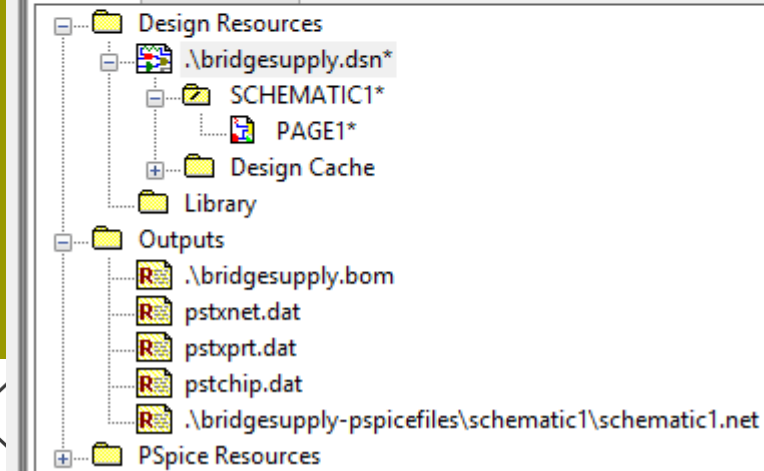
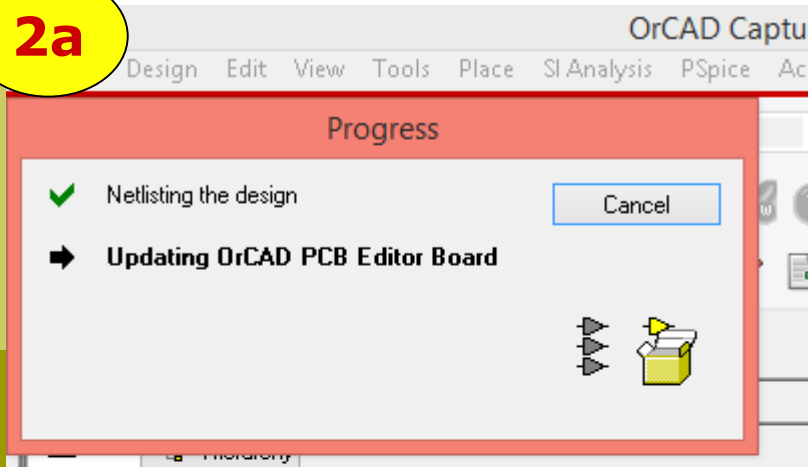
1



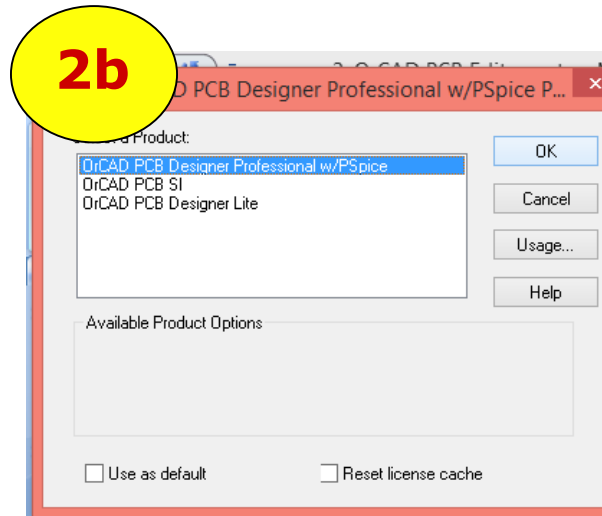
2



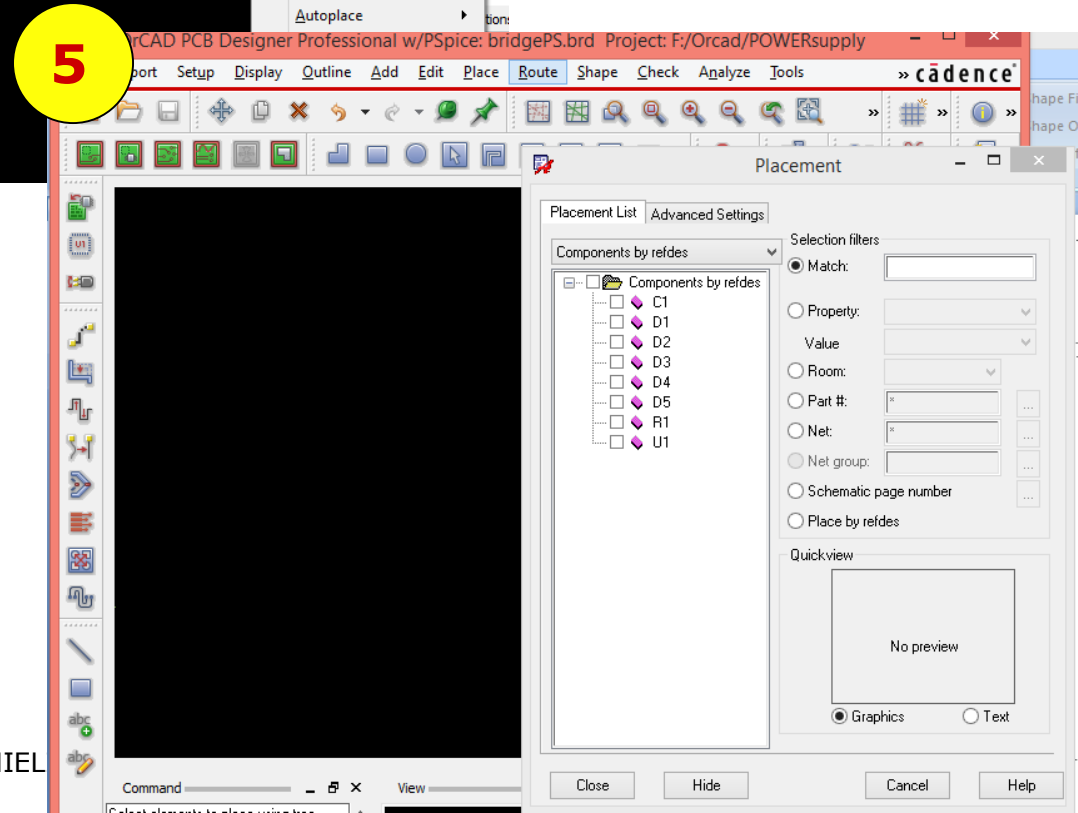
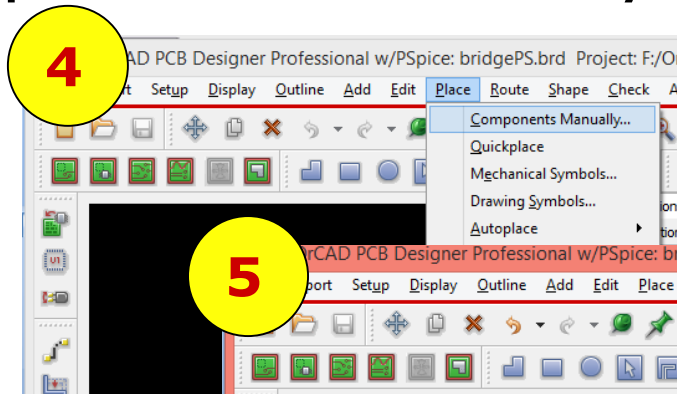
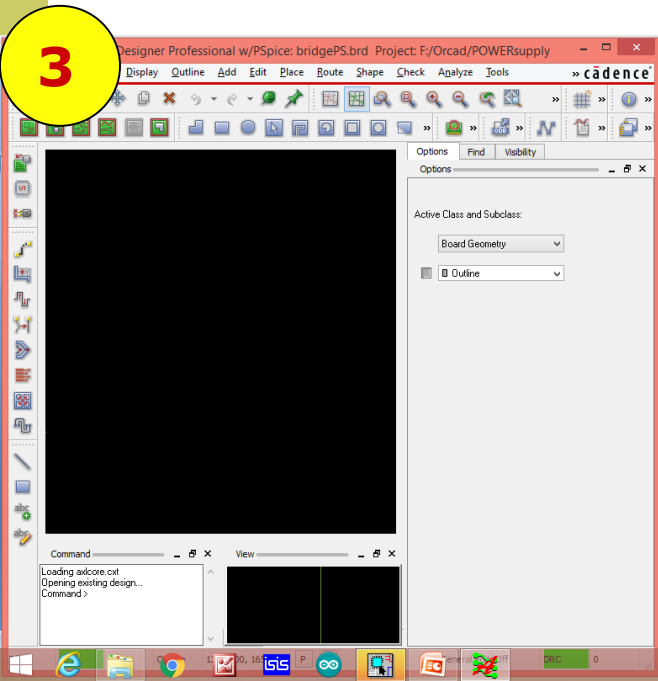
2a

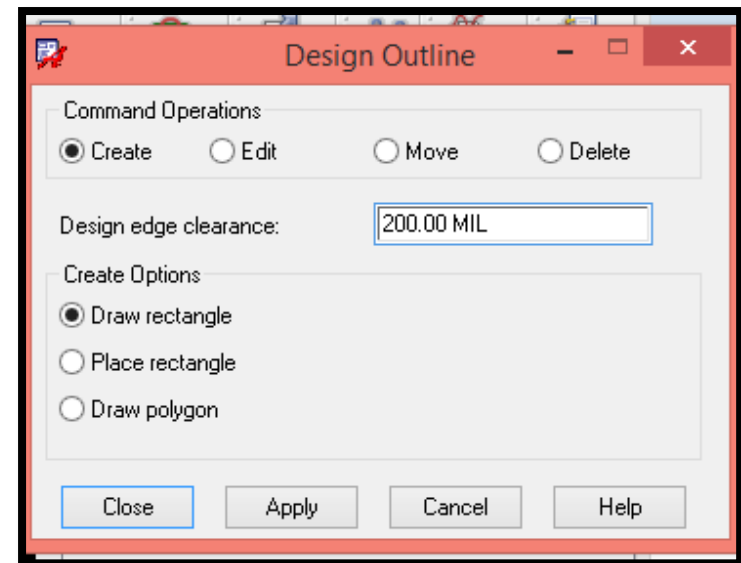
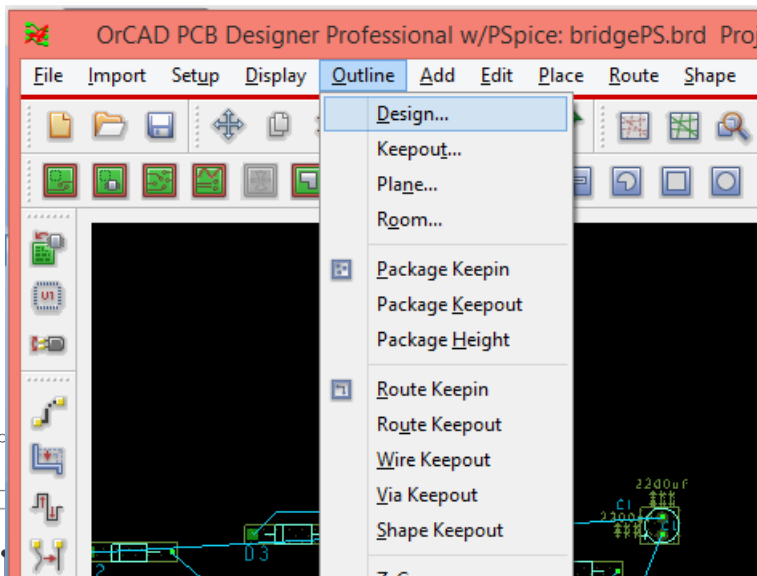
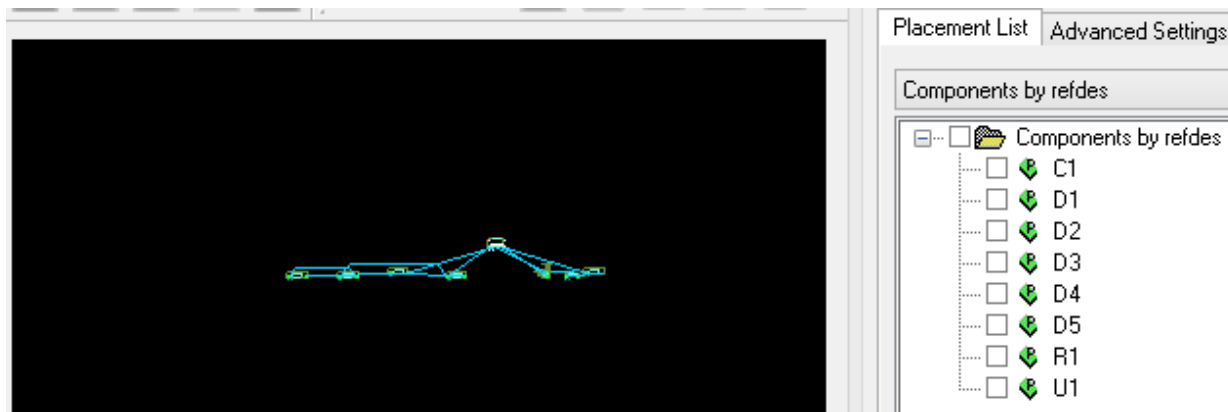


2b

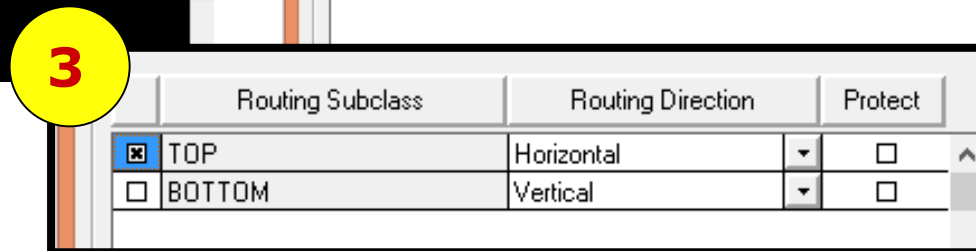
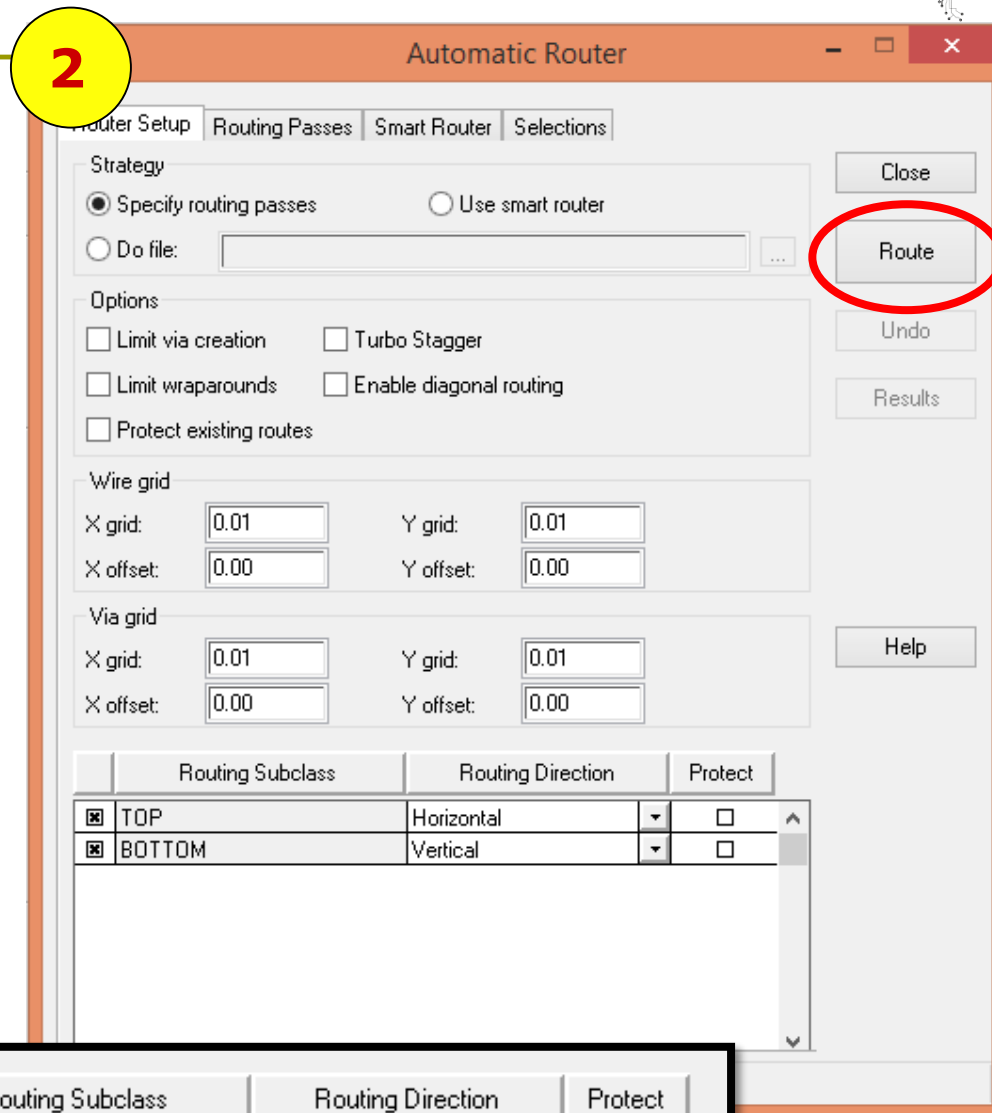
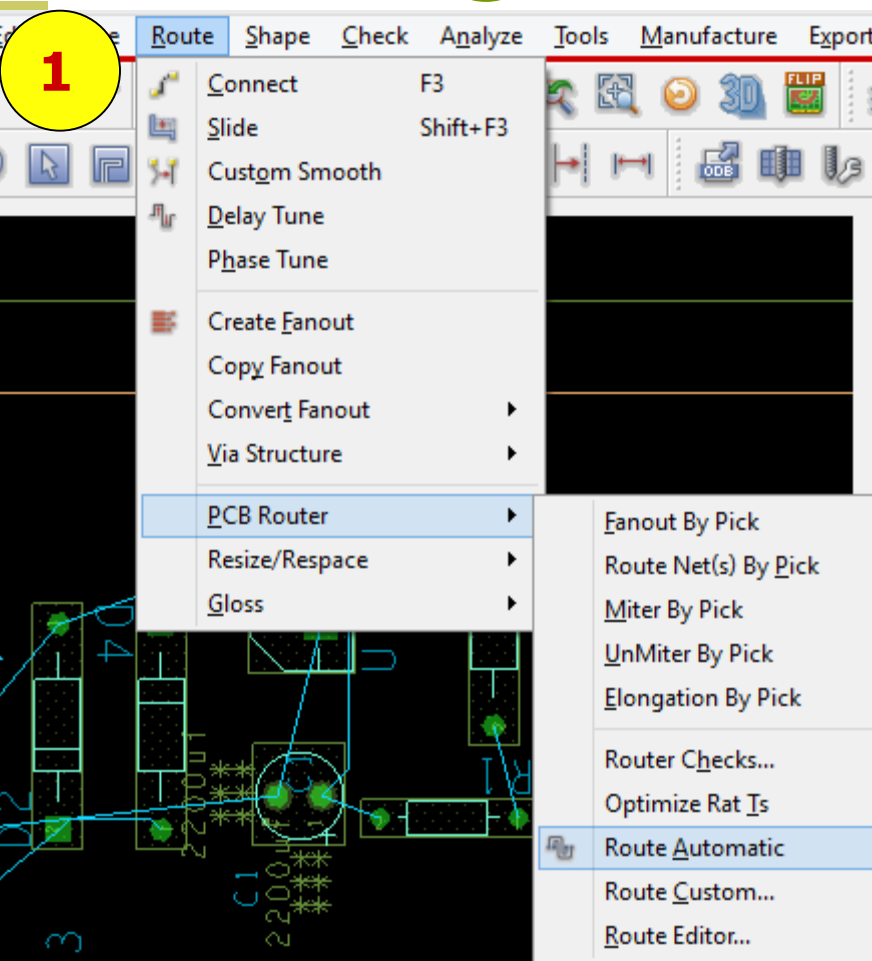
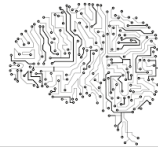


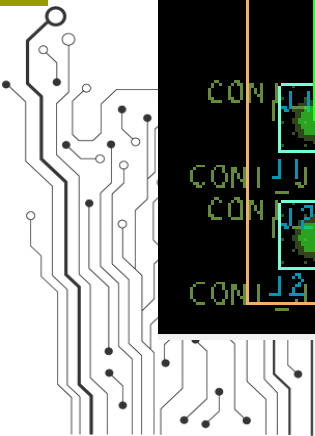
3. Blank PCB board created
4. Place > Components Manually





Routing







OrCAD PCB Designer Professional w/PSpice: bridgePS.brd Project: F:/Orcad/POWERsupply

File Import Setup Display Outline Add Edit Place Route Shape Check Analyze Tools Manufacture Export Help

Options Find Visibility

Options

Top Act
Bottom Alt
No available via Via

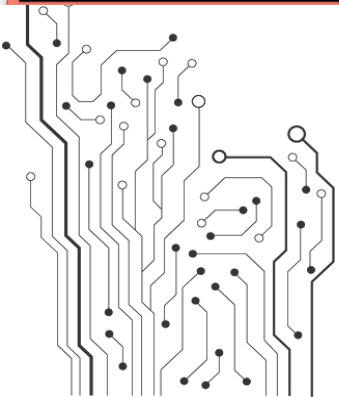
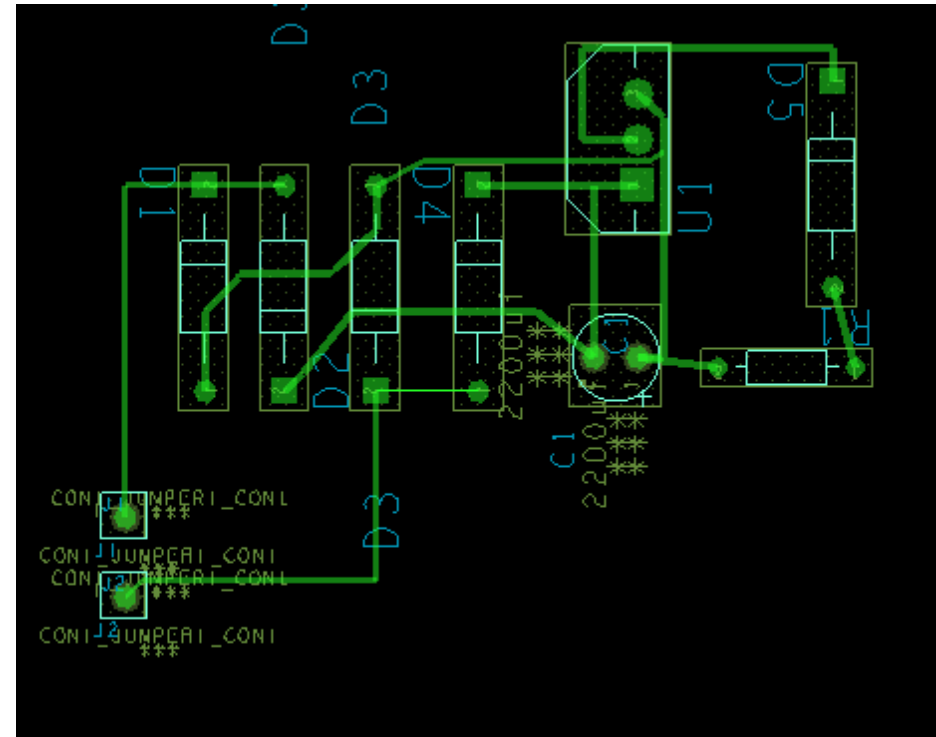
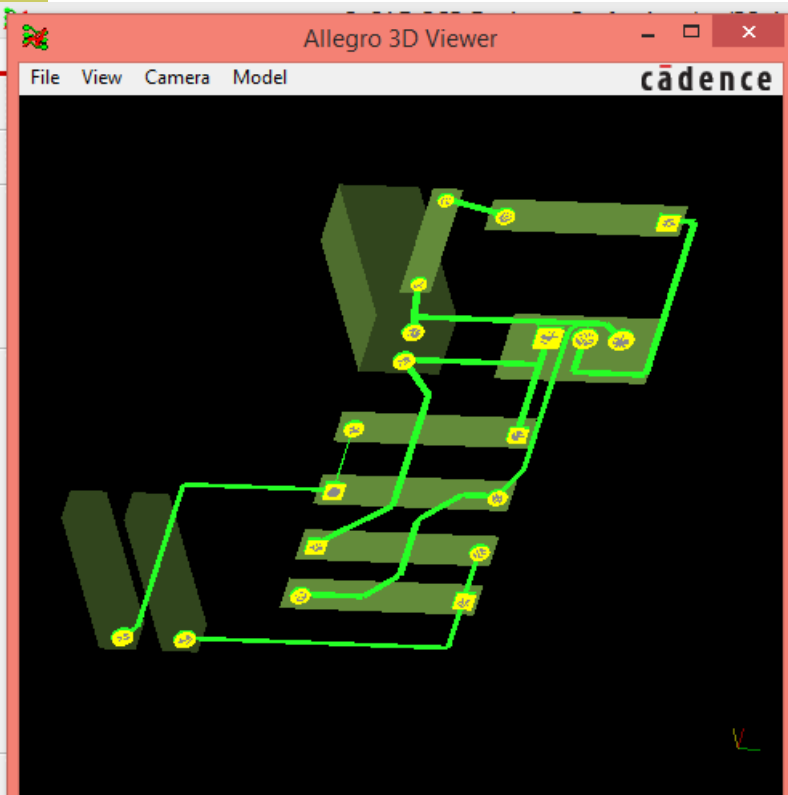
Net: Null Net

Line lock: Line 45
Miter: 1x width Min
Line width: 15.00
Bubble: Shove preferred
Shove vias: Off
☒ Gridless
☒ Clip dangling clines
Smooth: Minimal
☒ Snap to connect point
☒ Replace etch
☐ Auto-blank other rats

Command

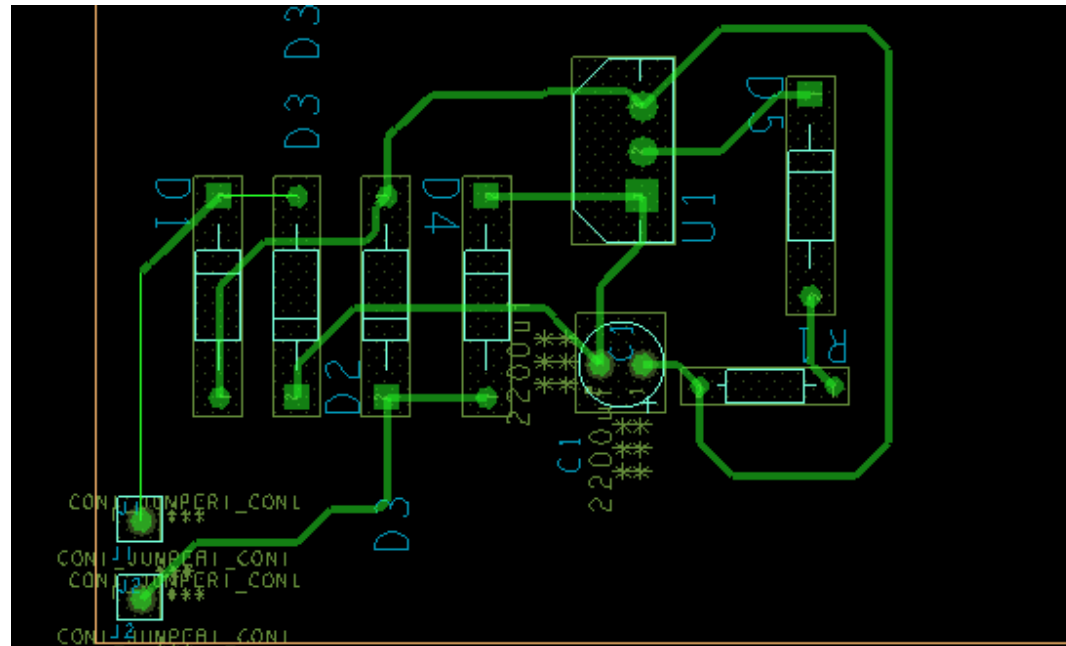
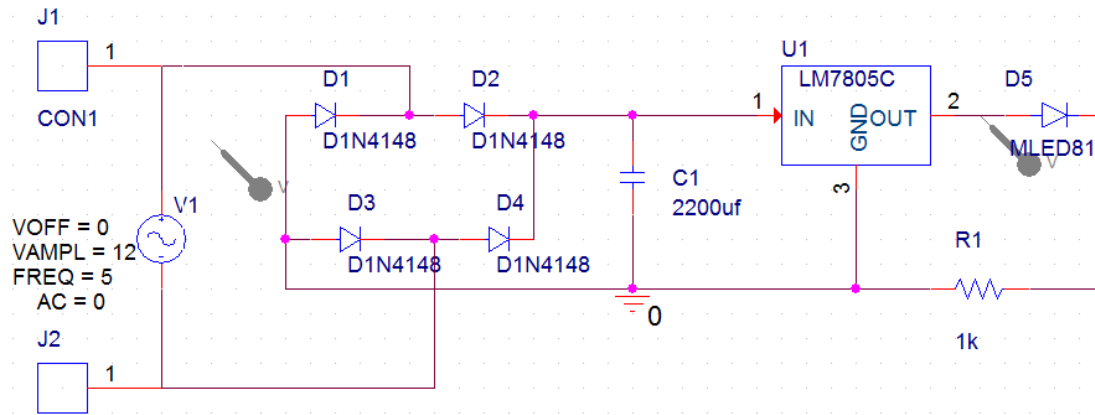
No DRC errors detected.
Pick first element.
last pick: 13225.00 10125.00
Pick first element.
last pick: 13225.00 10125.00
Pick first element.
Command > 21115

add connect Top 14450.00, 11100.00 P A General edit Off DRC 0

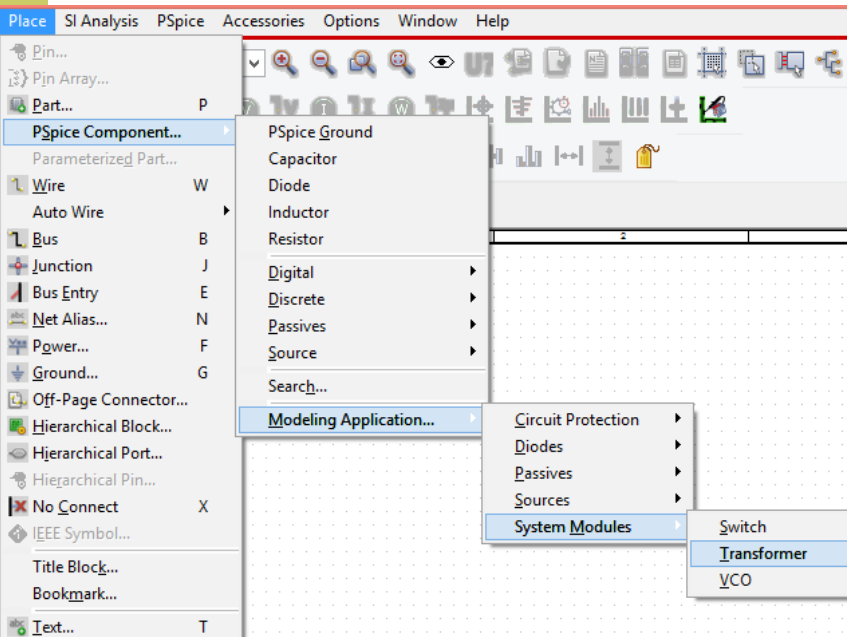




Adding jumper at Vin source



Center-tap transformer



PSpice Modeling Application: Single Phase Transformer With Linear Core

Select the appropriate transformer type and input transformer parameters accordingly to generate transformer model. You shall be able to place transformer directly into schematic and its model library file would be automatically included under simulation setting on placement of transformer.

☐ Two Winding ☐ Custom Tap ☒ Centre Tap ☐ Flyback ☐ Forward ☐ Forward with reset winding

This device would model a center tap transformer. You need to define primary winding inductance, primary winding resistance, leakage inductance, turn ratio and winding with tap. Turn ratio is defined as N_2/N_1 ; thus for step up transformer with step up ratio of 10, turn ratio should be 10 and for step down ratio of 10, it should be 0.1. For turn ratio calculation each TAP is considered as winding, i.e. 230v:23v-0-23v transformer turn ratio should be 10. All leakage inductance should be referred to winding with TAP and modeled as single leakage inductance. Use leakage inductance = 0, for modeling a transformer with ideal coupling between windings.

Parameter Name	Parameter Value
Model Name	CETAPS_Model
Primary Winding Inductance (LP)	1m
Primary Winding Resistance (Rp1)	10m
Secondary Winding Resistance (Rs1)	10m
Turn ratio N2/N1	10
Leakage Inductance	100n

☐ Primary Tap % ☒ Secondary Tap %

Place Close Help

Fullwave Rectifier

VOFF = 0
VAMPL = 60
FREQ = 10
AC = 60

