UNIVERSITY OF CALIFORNIA, BERKELEY
DEPARTMENT OF ELECTRICAL ENGINEERING AND COMPUTER SCIENCES

EE 113B/213B: Power Electronics Design
Module 8: Final PCB Design

Objectives

By the end of this module, you should be able to...

- Prepare for a PCB design review
- Generate final PCB files for manufacturing
- Confirm PCB files are compatible with manufacturer
- Order a PCB online

Pre-lab Assignment

3.4mm  trace of gnd

add 1 gnd near Vsw

PCB Final Touches

Once you have completed all routing, finalize* your PCB design and ensure the following:

1. Ensure your board is still 100x100 cm, even if you are not using all of the space.
2. Add four non-plated holes (typically in the corners) for standoffs. The standoffs in our parts library have size M3 screws.
3. Write your name in silkscreen somewhere on the top layer of your board. You are welcome to write other (class-appropriate) things in silkscreen, such as a title for your design, or additional labels.
4. Draw a rectangular silkscreen border encircling all parts of your design that will be considered when evaluating your converter's power density. This should include how widely the eligible traces expand on internal layers, including planes. You are welcome to shrink planes to make this area smaller as long as you don't disrupt the performance of your power converter. As a reminder, your converter's power density will be measured based on its box volume (i.e., the volume of the smallest rectangular box that could enclose the whole design) considering all internal power traces, signal traces, planes, and all components besides connectors, test points, and unused parts of the PCB.
5. Ensure all other silkscreen labels are positioned as desired on the PCB. You are not required to keep unwanted component labels.

*You should consider this the final "version 1" of your design. After your design review, you will implement the feedback you receive to create "version 2", which will be the version we order.

Design Review Slide Deck

Prepare PowerPoint slides for a PCB design review appointment with course staff. The purpose of these slides is to clearly show important features of your PCB layout with screenshots. For each aspect, you are welcome to use multiple screenshots if necessary, especially if multiple layers are relevant. The slides themselves do not need to be fancy, but the feature of focus should be very clear. You should organize your slides as follows:

1. (213B only) Topology: Show a schematic of your converter topology.
2. Critical Loops: Clearly show with screenshots all critical loops, one by one. This includes all commutation loop(s), gate loop(s), and any other high di/dt loops. For clarity, 213B students should also show each loop of focus on the schematic.
3. Critical Nodes: Clearly show with screenshots all critical nodes, one by one. This includes the switch node(s) and any other high dv/dt nodes. You should include the traces surrounding high dv/dt nodes in your screenshots to show the distance and/or shield between them. For clarity, 213B students should also show each node of focus on the schematic.
4. Power Stage Current Loops: Show screenshots of all power stage current loops, one by one. Show both the dc and ac current paths for each. For clarity, 213B students should also show each loop of focus on the schematic.
5. Analog Circuit Placement and Routing: Show screenshot(s) of how you have placed and routed analog circuits (e.g., sensing) to minimize interference and ensure minimal differences in ground potential. This includes showing the distance of such components from high di/dt loops and high dv/dt nodes, unless you are using shielding. This also includes showing the power rails and ground connections for these circuits.
6. Digital Circuit Placement and Routing: Show screenshot(s) of how you have placed and routed digital circuits (e.g., gate drive). This also includes showing the power rails and ground connections for these circuits.

7. Test Points: Provide a list of all nets for which you have included test points for probing voltage and loops for probing current. For clarity, 213B students should also show these nets on the schematic.

8. Silkscreen: Show a screenshot with your silkscreen layer and all four copper layers visible, to show that your silkscreen rectangle encloses all relevant traces and planes for power density. Also show that your name is written on the PCB in silkscreen.

Bill of Materials

Rather than providing a conventional Bill of Materials (BOM), open the BOM spreadsheet linked under this module on bCourses and locate the column with your name. In your column, enter the quantity of each part you will need for one build of your board (not including extra parts). For parts you do not plan to use, enter '0'. Make sure all parts on your PCB schematic are listed on this spreadsheet; this is the spreadsheet course staff will use to determine how many additional parts will need to be ordered. DO NOT modify any column but your own.

EE 213B: Enter parts not currently in our library under the "New Parts" heading at the bottom of the spreadsheet, and provide the part number, Digikey/Mouser link, and cost for a small quantity.

Assignment Feedback (Required)

How much dedicated time did you spend on this pre-lab? Add this at the end of your design review slides.

Submit a PDF of your design review slides to Gradescope.

Lab Assignment

PCB Design Review

Meet with course staff for a final review of your PCB design.

Check-off: Participating in a design review will count as a check-off for this module.

Generate Manufacturing Files

Make final changes to your PCB design based on the feedback you received during your design review. Be sure your design passes a Design Rule Check based on the rules implemented in Module 7 (i.e., minimum hole size, trace width/spacing, etc.). Then, generate manufacturing files as follows:

1. Copy and paste the '4 Layer PCBWay ALL.OutJob' file provided on bCourses into your PCB project folder.
2. Right click the .PrjPcb file in Altium and select 'Add existing to project' and choose the .OutJob file.
3. The .OutJob file should show up in the folder Settings/Output Job Files in Altium as shown in Fig. 1.
4. Open the .OutJob File in Altium.
5. To generate gerbers and NC drill files, click on 'Generate Content' under 'Folder Structure' as shown in Fig. 2. Altium will now generate the output files in the directory 'Project Outputs for xyz' in your file system where xyz is your project name as shown in Fig. 3.
6. Finally, .zip the Gerber and NC Drill files (selected in Fig. 3) and rename the .zip file to your board name as shown in Fig. 4. This is the file you will upload to the manufacturer's website.

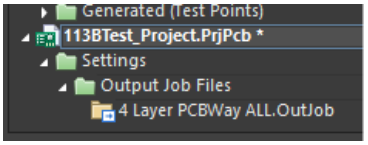


Figure 1: Output job folder.

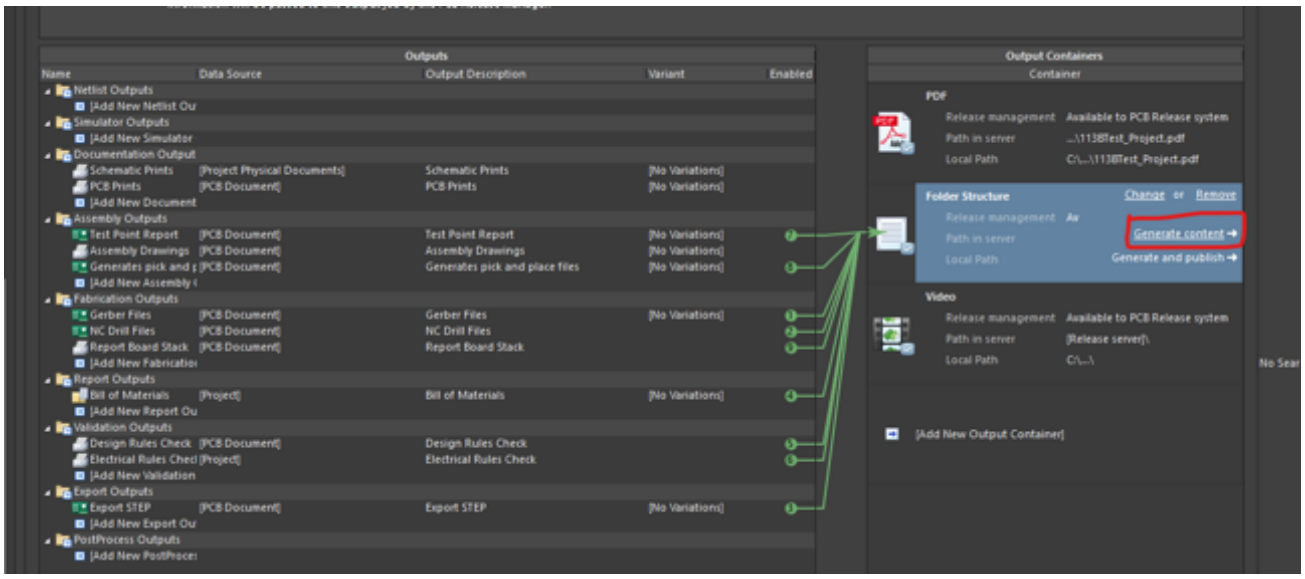


Figure 2: 'Generate Content' option.

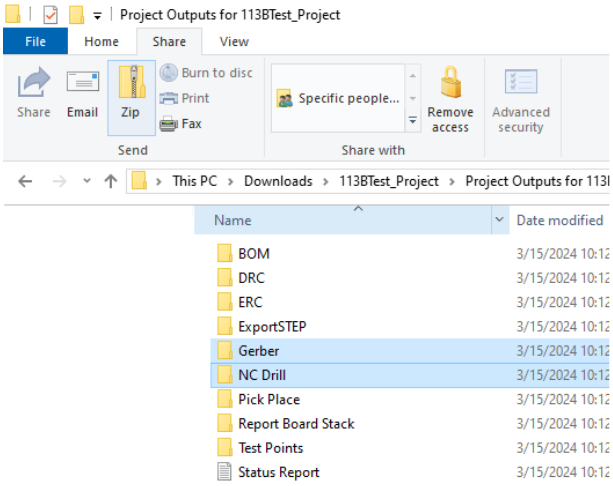


Figure 3: Gerber and NC Drill files.

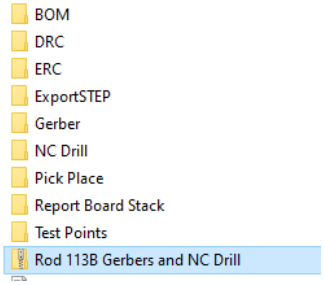


Figure 4: Final .zip file.

Confirm Manufacturer Compatibility

PCBs can be easily ordered online, and many manufacturers include a final design review as part of the ordering process to ensure your design is compatible with their fabrication capabilities. For this process, assume you will order from PCBWay, and follow these steps on PCBWay's website:

1. Click 'PCB Instant Quote' in the black toolbar at the top.
2. Complete the specification form according to your PCB design. Most of the default options here are fine, but be sure to enter your board size, number of layers, and select the minimum possible quantity. You may skip naming your layers when prompted.
3. Click 'Calculate' and then 'Save to Cart'.
4. Upload a .zip file containing your Gerber and NC drill files.
5. Click 'Submit Order Now'. You will be redirected to the 'My Orders' page, which will display your order as 'Under Review'. PCBWay will review your files to make sure your design is compatible with their fabrication capabilities. This process should take about ten minutes, and you can refresh the page to check its status.
6. Once your design passes the review, you will see your design progress from 'Under Review' to 'Awaiting Payment' on the My Orders page.
7. At this point, you would normally proceed to checkout and pay for your design to begin production. Take a screenshot of your 'My Orders' page showing that your design has passed the design review. You will submit this screenshot in the next section.

Submit Final Files

Your final file submission for this class will include (a) a .zip file containing your final PCB gerber/drill files, (b) your .PcbDoc file, and (c) a .zip file of your entire PCB project. Follow these steps to export your PCB project as a .zip file:

1. Select 'Project Packager' located until the 'Project' menu option.
2. Configure the package target as 'Package focused project'. Then, select 'Next'.
3. For the 'Zip File Options', ensure that generated files are not included. Also, make sure that all the 'Additional Items to Include' are unchecked. Click 'Next'.

4. The 'Select Files to Include' page displays all the files that will be included in your zip file. If you created additional .PcbDoc files for scratch work, uncheck the box next to the files. Only include your final .PcbDoc file. Click 'Next'.

5. The final page confirms that your project was successfully packaged. The zip file will be located at the path that was specified on the 'Zip File Options' page. The default path places the zip file in the project folder in your file directory.

Submit all three files by adding them to the corresponding folder in this Google Drive folder. Make sure both file names begin with your first name.

Then, email Prof. Boles with (a) A screenshot showing your PCB has passed the manufacturing check above; (b) How much dedicated time you spent on this lab (including the design review appointment); (c) If any other student(s) in the lab helped you, and who.

Check-off: The following will count as a check-off for this module: (a) Submitting all final files to the Google Drive folder on time; and (b) Emailing Prof. Boles with the required information on time. Late file submissions will not be accepted, and there will not be an opportunity to receive late check-off credit for this module.

Post-lab Assignment

Lab Results

1. Provide a PDF that includes your final schematic and PCB layout prints. This should have been generated as part of your final manufacturing files with the provided .outjob file.

Lab Takeaways

1. In your own words, describe what parasitic inductance is, what types of loops are most sensitive to it, how parasitic inductance in these loops affects power converter operation, and how parasitic inductance can be minimized.
2. In your own words, describe what parasitic capacitance is, what types of nodes are most sensitive to it, how parasitic capacitance in these loops affects power converter operation, and how parasitic capacitance can be minimized. Describe one strategy for protecting critical aspects of a power converter design from parasitic capacitance.
3. In your own words, describe important considerations for placing and routing power stage current paths.
4. In your own words, describe important considerations for placing and routing analog circuits.
5. Describe the benefits of using a ground plane, and considerations to be made when designing one.

Build Intuition

1. Describe a systematic strategy for power converter PCB layout. Which aspects would you prioritize, and in what order?
2. How would this PCB layout strategy change for a converter operating at a higher switching frequency?
3. Describe the role that component packages (i.e., package size, package shape, length of leads, etc.) play in the critical aspects of PCB layout. Are there certain aspects of your PCB layout that could have been improved with different component packages? If so, which components?
4. Describe how PCB layout affects the efficiency of your power converter.

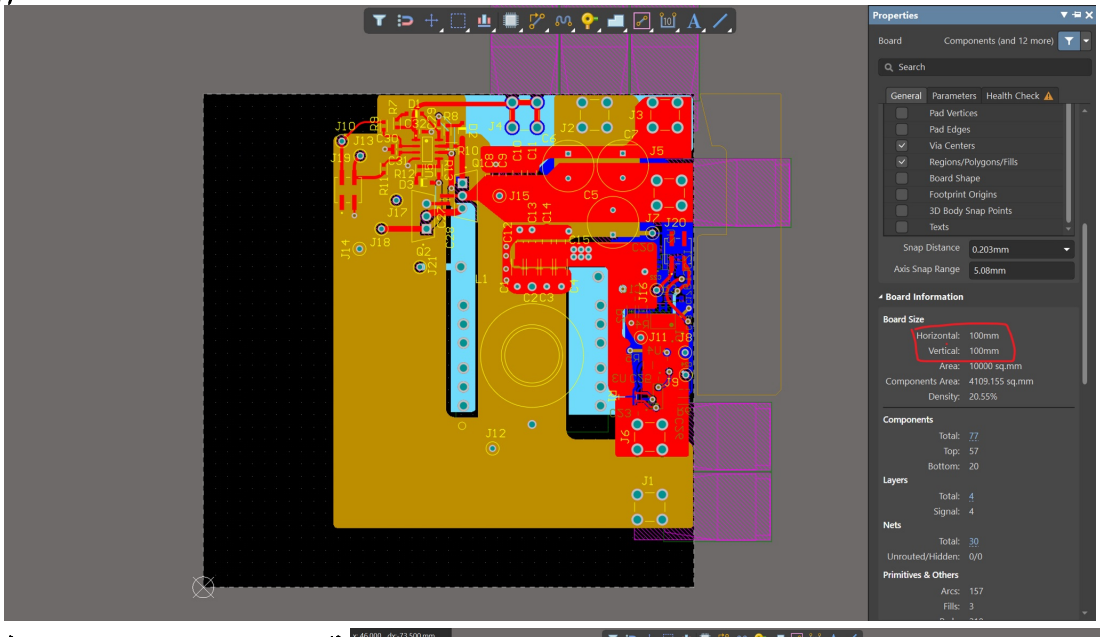
Assignment Feedback (Required)

How much dedicated time did you spend on this post-lab?

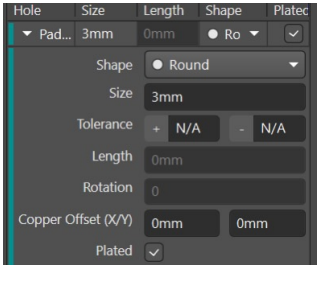
Once you have completed all routing, finalize* your PCB design and ensure the following:

1. Ensure your board is still 100x100 cm, even if you are not using all of the space.
2. Add four non-plated holes (typically in the corners) for standoffs. The standoffs in our parts library have size M3 screws.
3. Write your name in silkscreen somewhere on the top layer of your board. You are welcome to write other (class-appropriate) things in silkscreen, such as a title for your design, or additional labels.

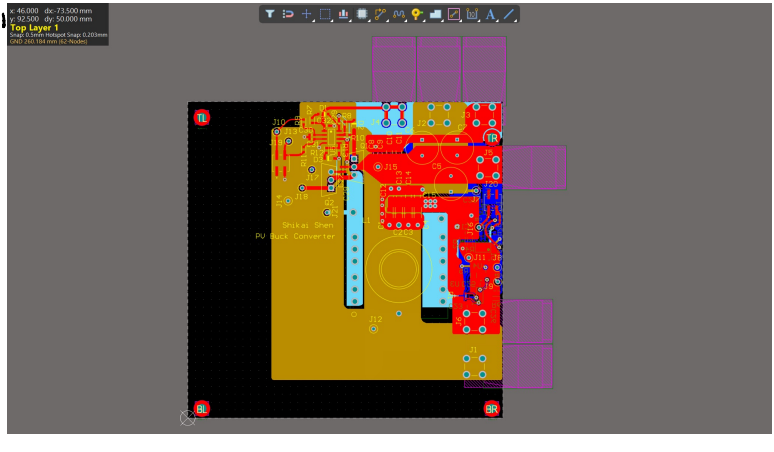
(1)



(2)

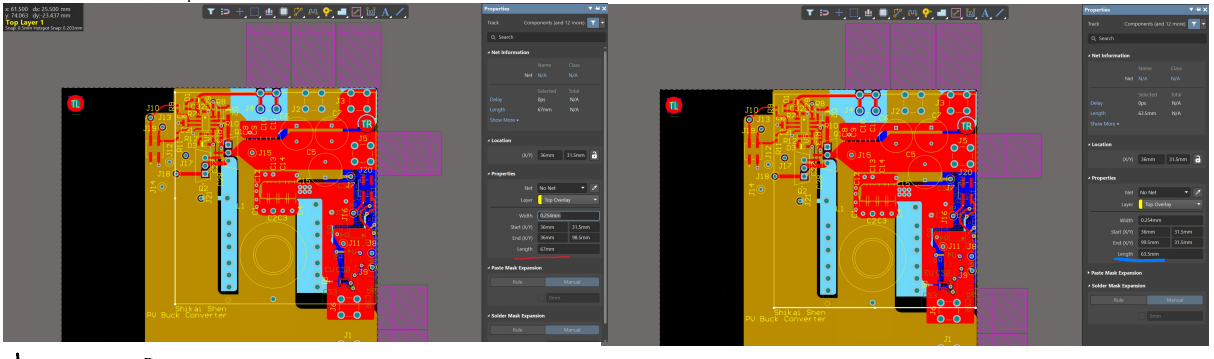


(3)



4. Draw a rectangular silkscreen border encircling all parts of your design that will be considered when evaluating your converter's power density. This should include how widely the eligible traces expand on internal layers, including planes. You are welcome to shrink planes to make this area smaller as long as you don't disrupt the performance of your power converter. As a reminder, your converter's power density will be measured based on its box volume (i.e., the volume of the smallest rectangular box that could enclose the whole design) considering all internal power traces, signal traces, planes, and all components besides connectors, test points, and unused parts of the PCB.

5. Ensure all other silkscreen labels are positioned as desired on the PCB. You are not required to keep unwanted component labels.



box area : 67mm x 62.5mm ≈ 4254.5mm²

Design Review Slide Deck

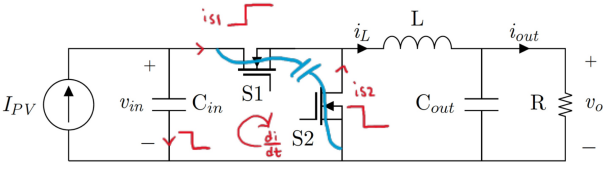
Prepare PowerPoint slides for a PCB design review appointment with course staff. The purpose of these slides is to clearly show important features of your PCB layout with screenshots. For each aspect, you are welcome to use multiple screenshots if necessary, especially if multiple layers are relevant. The slides themselves do not need to be fancy, but the feature of focus should be very clear. You should organize your slides as follows:

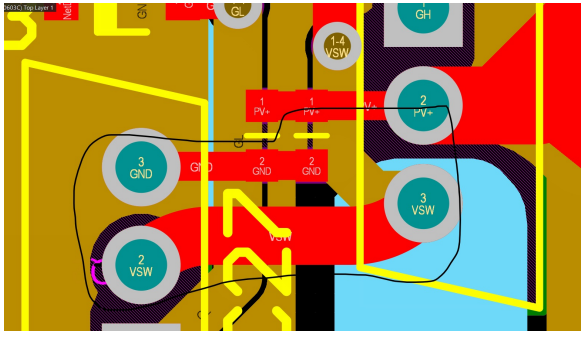
1. (213B only) Topology: Show a schematic of your converter topology.

2. Critical Loops: Clearly show with screenshots all critical loops, one by one. This includes all commutation loop(s), gate loop(s), and any other high di/dt loops. For clarity, 213B students should also show each loop of focus on the schematic.

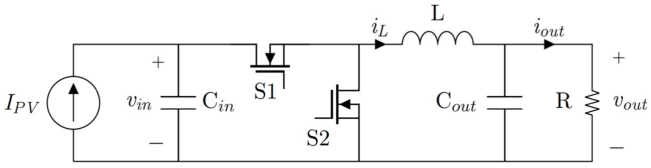
3. Critical Nodes: Clearly show with screenshots all critical nodes, one by one. This includes the switch node(s) and any other high dv/dt nodes. You should include the traces surrounding high dv/dt nodes in your screenshots to show the distance and/or shield between them. For clarity, 213B students should also show each node of focus on the schematic.

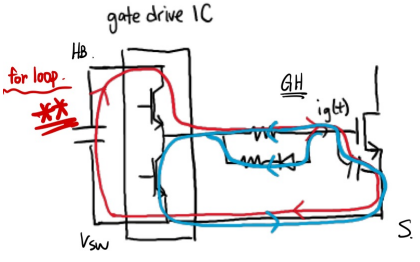
commutation loops: [high $\frac{di}{dt}$]

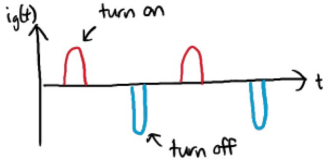




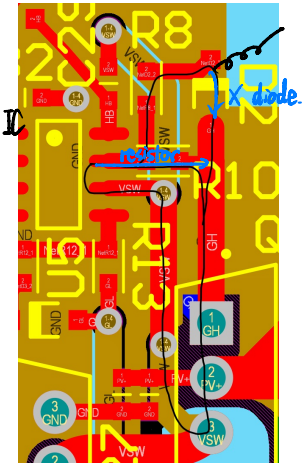
Gate Loops [high $\frac{di}{dt}$]







side view:

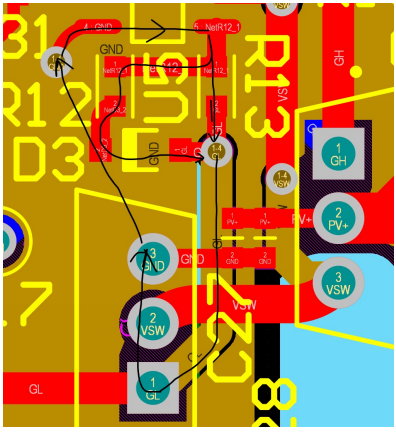


Gate High Loop: GH ⟶ VSW ⟹ BTI ⟶ Gate IC

Gate Ic

↓

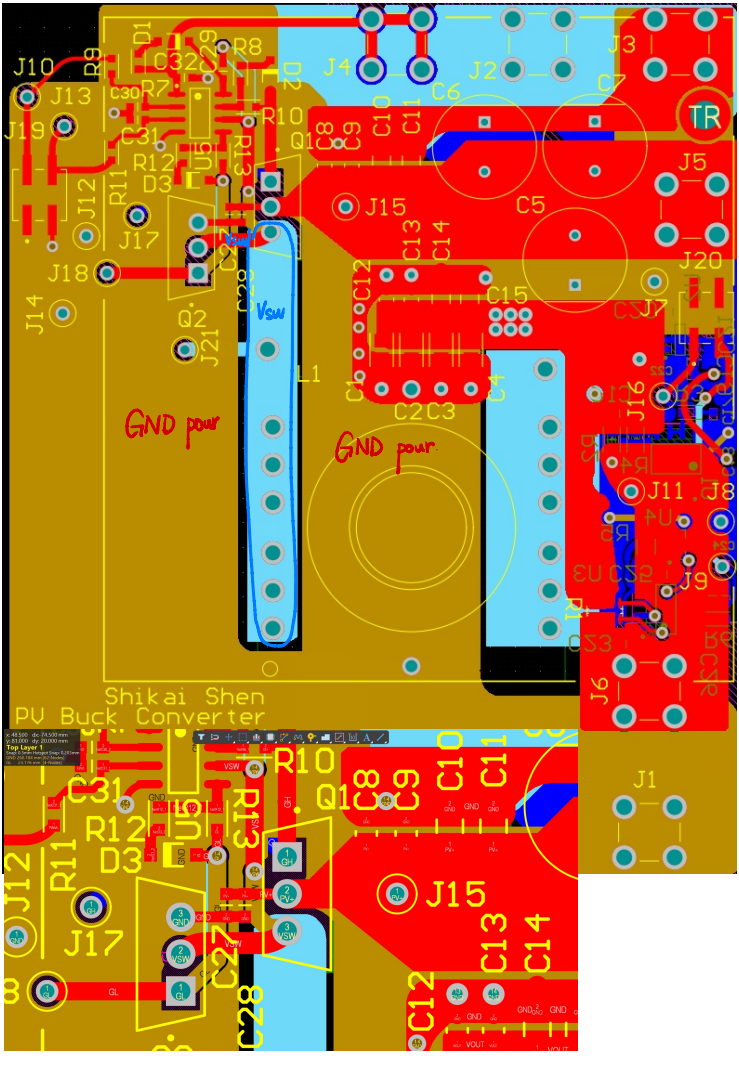
Gate Low Loop: LO → GL → GND



High $\frac{di}{dt}$ loops: commutation, gate high, gate low

3. Critical Nodes: Clearly show with screenshots all critical nodes, one by one. This includes the switch node(s) and any other high dv/dt nodes. You should include the traces surrounding high dv/dt nodes in your screenshots to show the distance and/or shield between them. For clarity, 213B students should also show each node of focus on the schematic.

high $\frac{dv}{dt}$: Vsw

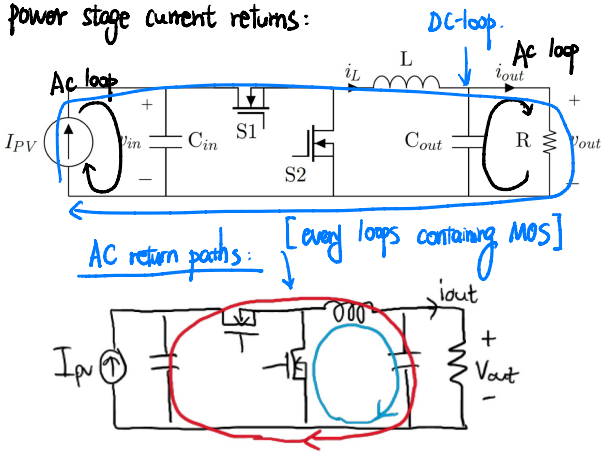


4. Power Stage Current Loops: Show screenshots of all power stage current loops, one by one. Show both the dc and ac current paths for each. For clarity, 213B students should also show each loop of focus on the schematic.

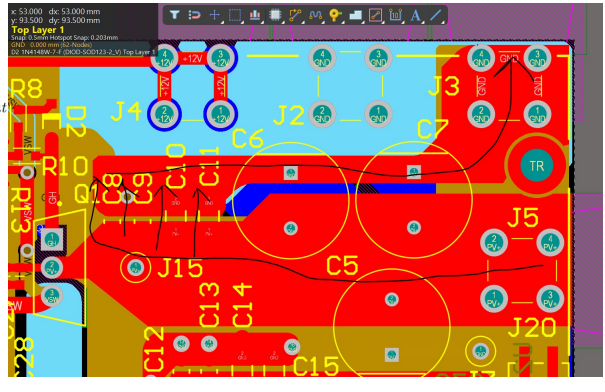
5. Analog Circuit Placement and Routing: Show screenshot(s) of how you have placed and routed analog circuits (e.g., sensing) to minimize interference and ensure minimal differences in ground potential. This includes showing the distance of such components from high di/dt loops and high dv/dt nodes, unless you are using shielding. This also includes showing the power rails and ground connections for these circuits.

6. Digital Circuit Placement and Routing: Show screenshot(s) of how you have placed and routed digital circuits (e.g., gate drive). This also includes showing the power rails and ground connections for these circuits.

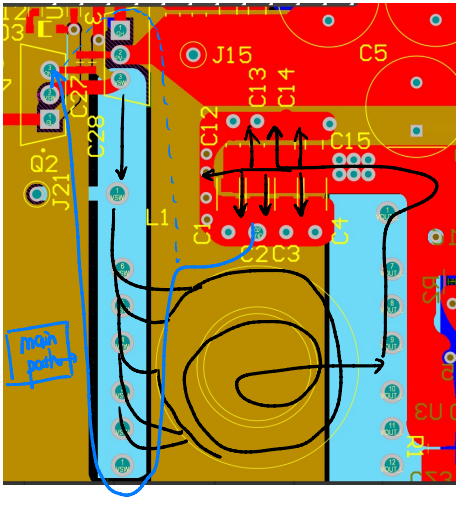
power stage current returns:



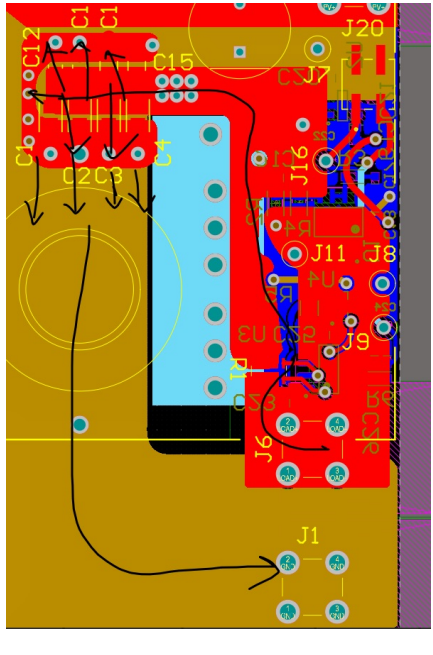
DC loops: (source) with large loop area ↓ minimize ESR



AC loop:



DC loop (load) ← minimize ESR



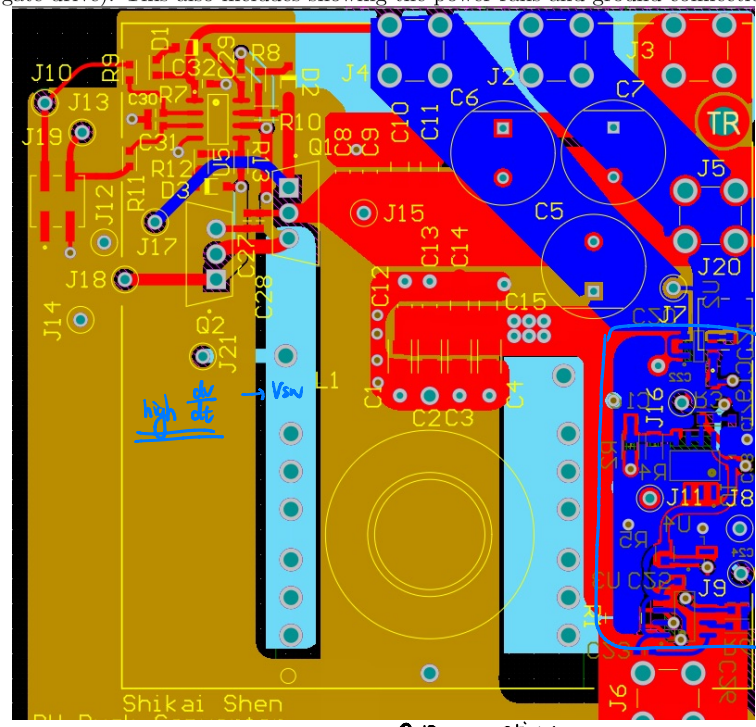
5. Analog Circuit Placement and Routing: Show screenshot(s) of how you have placed and routed analog circuits (e.g., sensing) to minimize interference and ensure minimal differences in ground potential. This includes showing the distance of such components from high di/dt loops and high dv/dt nodes, unless you are using shielding. This also includes showing the power rails and ground connections for these circuits.

6. Digital Circuit Placement and Routing: Show screenshot(s) of how you have placed and routed digital circuits (e.g., gate drive). This also includes showing the power rails and ground connections for these circuits.

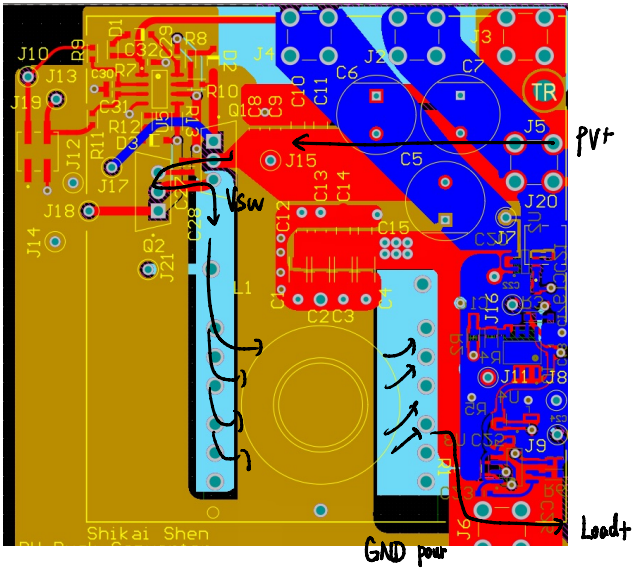
Sensing circuit:

Gate IC →

high $\frac{di}{dt}$

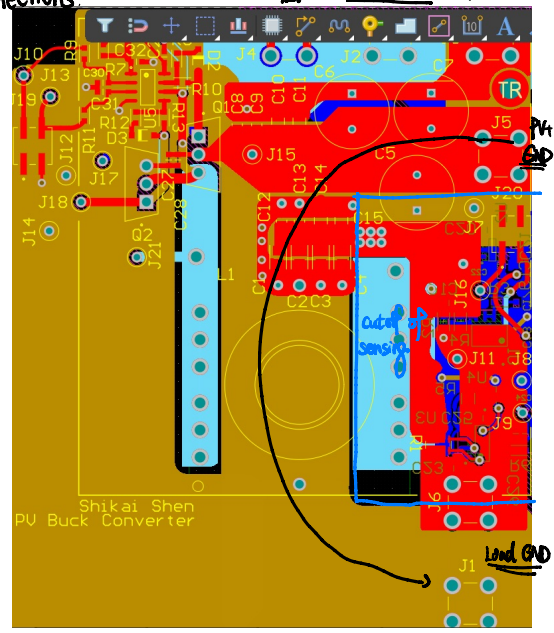


power rails:



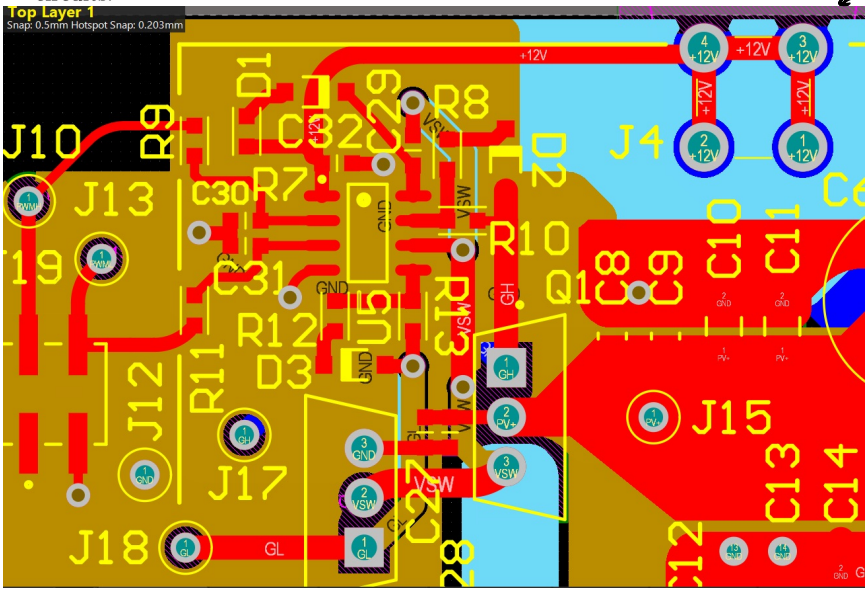
GND connections

12V 12V's GND PV+'s GND

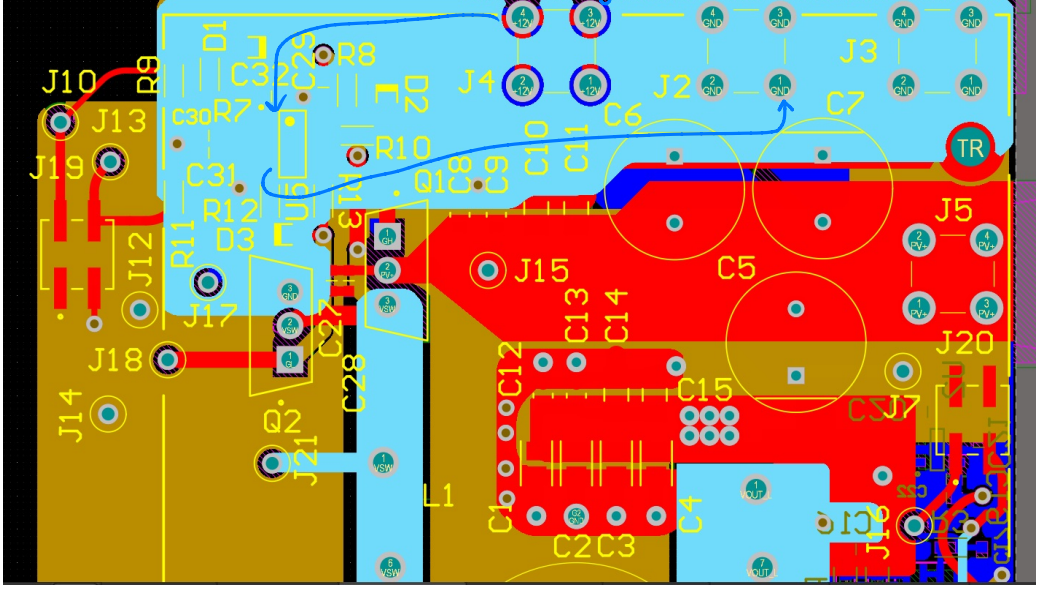


6. Digital Circuit Placement and Routing: Show screenshot(s) of how you have placed and routed digital circuits (e.g., gate drive). This also includes showing the power rails and ground connections for these circuits.

place position:



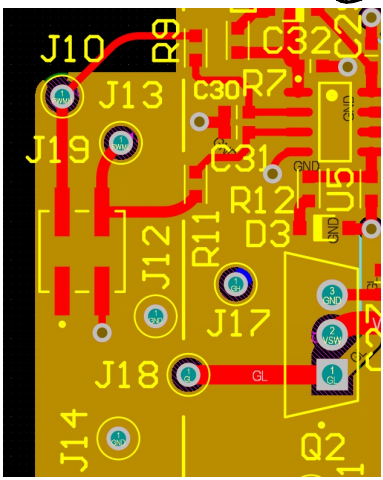
power rails and ground returns of Gate Driver.



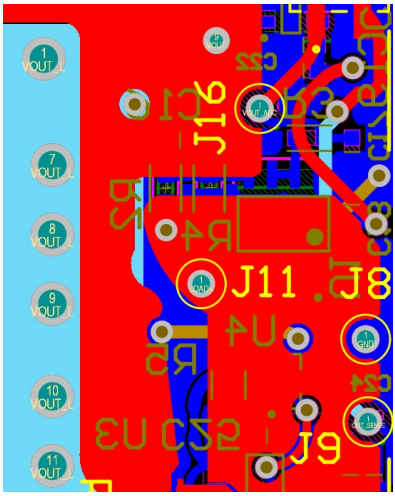
7. Test Points: Provide a list of all nets for which you have included test points for probing voltage and loops for probing current. For clarity, 213B students should also show these nets on the schematic.

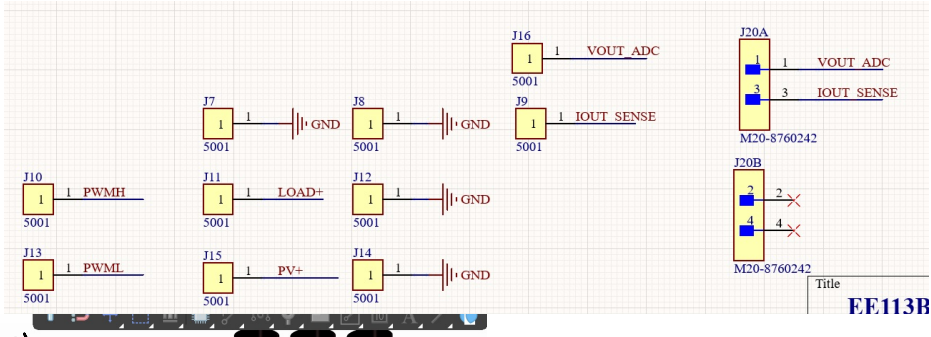
8. Silkscreen: Show a screenshot with your silkscreen layer and all four copper layers visible, to show that your silkscreen rectangle encloses all relevant traces and planes for power density. Also show that your name is written on the PCB in silkscreen.

(7). PWMH. PWML. GH. GL. VSW. GND

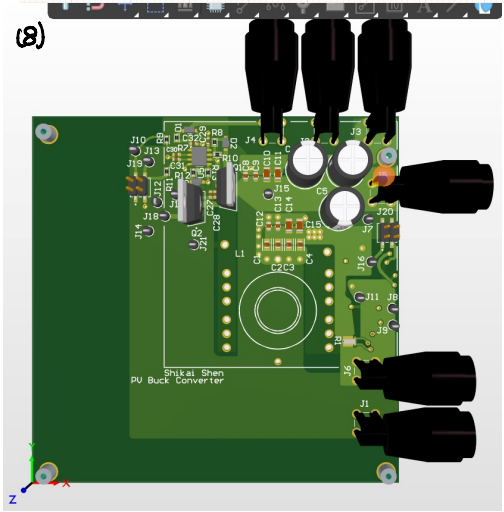


VOUT_ADC, IOUT_SENSE, LOAD+





(8)



Bill of Materials

Rather than providing a conventional Bill of Materials (BOM), open the BOM spreadsheet linked under this module on bCourses and locate the column with your name. In your column, enter the quantity of each part you will need for one build of your board (not including extra parts). For parts you do not plan to use, enter '0'. Make sure all parts on your PCB schematic are listed on this spreadsheet; this is the spreadsheet course staff will use to determine how many additional parts will need to be ordered. DO NOT modify any column but your own.

EE 213B: Enter parts not currently in our library under the "New Parts" heading at the bottom of the spreadsheet, and provide the part number, Digikey/Mouser link, and cost for a small quantity.

Assignment Feedback (Required)

How much dedicated time did you spend on this pre-lab? Add this at the end of your design review slides.

Submit a PDF of your design review slides to Gradescope.

10uF 0805: 4

470uF electrolytic: 3

22uF 0805: 4

4.7uF 1206: 4 → 10uF 1206

2.2uF 0603: 2+1=3

330pF 0603: 2

10uF 0603: 1+1=2

1uF 0603: 1+1+1+1=4

10000pF 0603: 1+1=2

0.1uF 0603: 1+1+1=3

10pF 0603: 1

Time: 2.5 hours.

2.5m 0612: 1

1R 0603: 1+2+2=5

10R 0603: 2

2.94KR 0603: 1

120R 0603: 1+1=2

470kR 0603: 1

47kR 0603: 1

67	4.7 ohm, 1/8 W	RNCP0603FTD4R70	https://www.digike	0.1	0	0
68	filter and gate, 10 ohm, 0603, 1/10 W	AGCR0603-QK-10R0FT-CT	https://www.digike	0.01	2	0.02
69	LED, 120 ohm 0603	ERA-3AEB121V	https://www.digike	0.13	2	0.26
70	LED, 200 ohm 0603	ERJ-PB3B2000V	https://www.digike	0.26	0	0
71	current sense resistors, 1m, 1%, 0612	CSSK0612FT1L00	https://www.digike	0.84	0	0
72	current sense resistors, 2.5m, 1%, 0612	D1FCP0612D2M50FF-T5	https://www.digike	0.57	1	0.57
73	current sense resistors, 5m, 1%, 0612	CSSK0612FT5L00	https://www.digike	0.84	0	0
74	current sense, 2.94kΩ, 1%, 0.1W, 100ppm/°C, 0603	CRCW06032K94FKEA	https://www.digike	0.1	1	0.1
75	resistor dividers, 1k, 0603	ERJ-PB3B1001V	https://www.digike	0.26	0	0
76	resistor dividers, 4.7k, 0603	ERJ-PB3B4701V	https://www.digike	0.26	0	0
77	resistor dividers, 10k, 0603	ERJ-PB3B1002V	https://www.digike	0.26	0	0
78	resistor dividers, 47k, 0603	ERJ-PB3B4702V	https://www.digike	0.26	1	0.26
79	resistor dividers, 100k, 0603	ERJ-PB3B1003V	https://www.digike	0.26	0	0
80	resistor dividers, 470k, 0603	RN73R1JTTD4703B25	https://www.digike	0.2	1	0.2

price: 64.78 $

	A	B	C	D	AR	AS
1	**Name**	**Part No**	**Link**	**Cost [$]**	Shen Shikai	
3	**PARTS: CONNECTORS LIBRARY**				Quantity	Cost [$/board]
4	Header connectors (10 pos)	61301011121	https://www.digike	0.95	0	0
5	Header connectors (4 pos)	M20-8760242	https://www.digike	0.39	2	0.78
6	Test point	5001	https://www.digike	0.38	13	4.94
7	Banana jack, blk	CT1151SP-0	https://www.digike	2.45	3	7.35
8	Banana jack, red	CT3151-2	https://www.mouse	2.45	3	7.35
9						0
10	**PARTS: IC LIBRARY**					0
11	Half bridge gate driver	LM5109BMAX/NOPB	https://www.digike	1.03	1	1.03
12	Op amp	LM358DR	https://www.digike	0.44	1	0.44
13	Current sense amplifier	LT1999CS8-50	https://www.digike	6.25	1	6.25
14	Voltage reference - 1.25 V	ADR1581ARTZ-REEL7	https://www.digike	2.26	1	2.26
15	Low side gate driver	UCC27517DBVR	https://www.digike	0.83	0	0
16	Digital isolator	ISO6720FBDR	https://www.digike	1.54	0	0
17	Voltage regulator - 3.3 V	LD1117ADT33TR	https://www.digike	0.71	0	0
18	LDO	LP2985-50DBVR	https://www.digike	0.55	1	0.55
19						0
20	**PARTS: SWITCH LIBRARY**					0
21	LED	TLMS1000-GS08	https://www.digike	0.45	0	0
22	Diode	1N4148W-7-F	https://www.digike	0.11	3	0.33
23	TO-220 55V 29A	IRFZ34NPBF	https://www.digike	0.92	0	0
24	TO-220 55V 47A	IRLZ44NPBF	https://www.digike	1.28	0	0
25	TO-220 55V 64A	IRFZ48NPBF	https://www.digike	1.27	0	0
26	TO-220 55V 89A	IRL3705NPBF	https://www.digike	1.85	2	3.7
27	TO-220 heat sink	507302B00000G	https://www.digike	0.3	2	0.6
28	Thermal pad	Q®-0.006-00-54	https://www.digike	0.43	2	0.86
29	MLPAK33 25 V 32A	PXNTR7-25QLJ	https://www.digike	0.58	0	0
30	MLPAK33 25 V 65A	PXN6R2-25QLJ	https://www.digike	0.58	0	0
31						0
32	**PARTS: INDUCTOR LIBRARY**					0
33	RM 8 cores (2 piece set)	RM8/I-3C95	https://www.digike	2.32	0	0
34	RM 10 cores (2 piece set)	RM10/I-3C95	https://www.digike	3.54	1	3.54
35	RM 12 cores (2 piece set)	RM12/I-3C95	https://www.digike	6.29	0	0
36	RM 8 bobbin	B65812C1512T001	https://www.digike	1.8	0	0
37	RM 10 bobbin	B65814D1512T001	https://www.digike	2.05	1	2.05
38	RM 12 bobbin	B65816C1512T001	https://www.digike	2.17	0	0
39	RM 8 clamp	CLI/P-RM8/I	https://www.digike	0.31	0	0
40	RM 10 clamp	CLI/P-RM10/I	https://www.digike	0.31	2	0.62
41	RM 12 clamp	CLI/P-RM12/I	https://www.digike	0.49	0	0
47	**PARTS: CAPACITOR LIBRARY**					0
48	Electrolytic, 330 uF	EL1V331MP51016U	https://www.digike	0.29	0	0
49	Electrolytic, 470 uF	50ZLJ470M10X20	https://www.digike	0.83	3	2.49
50	MLCC, 22 uF, 35V, X5R, 0805	C2012X5R1V226M125A	https://www.mouse	0.82	4	3.28
51	MLCC, 4.7uF, 35V, X7R, 0805	GRM21BZ7YA475KE15K	https://www.digike	0.3	0	0
52	MLCC, 22uF, 35V, X5R, 1206	C3216X5R1V226M160AC	https://www.digike	0.97	0	0
53	MLCC, 10uF, 35V, X7R, 1206	C1206X106K035T	https://www.digike	0.44	0	0
54	MLCC, 10uF, 50V, X5R, 0805	GRM21BR61H106KE43K	https://www.digike	0.34	4	1.36
55	MLCC, 10uF, 50 V, X7R, 1206	C1206X7R500-106KNE-CT	https://www.digike	0.26	4	1.04
56	2.2 uF, Commutation loop decoupling, 0603, 50V	GRM188R61H225KE11J	https://www.digike	0.21	3	0.63
57	10 uF, 25 V Decoupling for gate drive, LDO, lig reg,	GRM188R61E106KA73J	https://www.digike	0.26	2	0.52
58	1 uF, 50V	0603X105K500CT	https://www.digike	0.13	4	0.52
59	.1 uF, 25V Current sense, op amp, regulator input d	C0603Z5U250-104MNP-CT	https://www.digike	0.08	3	0.24
60	LDO bypass, 10nF, 0603, 3 V	C0603X7R250-103MNP-CT	https://www.digike	0.01	2	0.02
61	Filter cap, 330 pF, 50V, NPO, 0603	C0603C0G250-331KNE-CT	https://www.digike	0.01	2	0.02
62	Current sense SHDN cap, 10 pF, 5V max, 0603	C0603C100J3HAC7867	https://www.digike	0.1	1	0.1
63						0
64	**PARTS: RESISTOR LIBRARY**					0
65	.47 ohm, 0603, 1/4 W	73L2R47J	https://www.digike	0.31	0	0
66	boostrap and gate, 1 ohm, 0603, 1/5 W	AC0603FR-7W1RL	https://www.digike	0.1	5	0.5