EET-1190: PRINTED CIRCUIT LAYOUT

Cuyahoga Community College

Viewing: EET-1190 : Printed Circuit Layout

Board of Trustees:
January 2023

Academic Term:
Fall 2023

Subject Code
EET - Electrical/Electronic Engineer

Course Number:
1190

Title:
Printed Circuit Layout

Catalog Description:
Examines use of contemporary program(s) to lay out printed circuit board in single and multiple layers. Design rules, current return paths, cross talk, creation of custom parts and other details are explored.

Credit Hour(s):
2

Lecture Hour(s):
1

Lab Hour(s):
2

Requisites

Prerequisite and Corequisite
EET-1161 Direct Current Circuits or concurrent enrollment.

Outcomes

Course Outcome(s):
Design and/or build a circuit using a program with schematic capture.

Objective(s):
1. Using the parts tool bars, select and place components on the schematic work sheet.
2. Wire components that represent the circuit.
3. If a component is not in the database, select a part with the same pin out or, after building the part in the printed circuit layout portion of the program, back annotate the part into the schematic capture program.
4. If the part exists in the User Database, access the User Database and place the part.
5. If components in the schematic design has defined models, simulate the circuit to verify operation. This is optional and is not required for printed circuit board layout.
6. Save the circuit layout into a student-created folder in a college directory that is not removed upon boot up.
7. Backup the circuit layout onto removable storage.
8. Create title blocks for the circuit on the schematic diagram.
9. Set the paper size as determined by the size (number of components) needed in the circuit.
10. Determine the size of the component that meets power, current or voltage rating(s).
11. Find real, not virtual, components that have a defined foot print and select the component.
12. Place holes of the proper diameter in the circuit board where needed.

Course Outcome(s):

Transfer design from the schematic capture program to the circuit layout program.

Objective(s):

1. Follow the correct procedure to transfer the schematic design to the circuit layout problem and save the transported program.
2. Set the size of the desired circuit board and analyze the cost/size relationship and cost/number of layers relationship.
3. Drag the components onto the active layout area and place components according to external interface access, closeness to edge connectors, close proximity of parts where applicable and set trace width as determined by component current specifications.
4. Demonstrate by the layout the primary object is to produce a circuit board that works and secondary objectives are board size and number of layers.
5. Identify critical ground paths and the need to make traces as wide as possible, avoid sensitive components sharing ground paths with power components.
6. Design star-out grounds where necessary.
7. Design ground and/or power planes where necessary.
8. Design dual ground systems when the circuit uses analog and digital components.

Course Outcome(s):

Route the circuit.

Objective(s):

1. After dragging the components to the circuit board, arrange the parts such that critical trace distances, locate and route sensitive traces away from digital and/or noisy traces and grounds.
2. Route sensitive traces such that they cross digital or noisy traces at a 90 degree angle.
3. Set the desired trace width that meets design rules.
4. Select the side of the circuit board to place traces.
5. Set the constraints to permit shoving parts.
6. Use the trace tool to set traces, switching between layers when necessary.
7. Place vias where necessary.
8. Place Keep-outs where necessary.
9. If desired, change the setting to rename parts as ordered top left to bottom right or similar.

Course Outcome(s):

Use silk screen layers for descriptive purposes.

Objective(s):

1. Select the silkscreen top and/or bottom layer.
2. Label parts with additional comments, name/purpose of the circuit board, describe adjustments, version number, company name, etc.

Course Outcome(s):

Build a custom component.

Objective(s):

1. Build a custom component using the parts wizard.
2. Use a micrometer and/or datasheet to determine the physical properties of the new part.
3. Assign the name, value, and related in the parts wizard.
4. Place the part on the layout and back annotate to the schematic capture program to set and wire the part() or use the net-list editor in the layout program to connect the part into the circuit.
5. Save the part into the user database and copy the user database to a safe directory.
6. If resuming work on a layout, copy the user database from a safe directory to the layout program.

Course Outcome(s):

Run Design Rule Check (DRC).

Objective(s):

1. While laying out the circuit, note any design rule error flags and correct the problem.
2. If DRC flags an error that is understood and acceptable, mask that error.
3. As necessary, clear the history of DRC errors.

Course Outcome(s):

Examine the Bill of Material (BOM).

Objective(s):

1. Select the BOM tab and examine the BOM to be sure that component designations match the component.
2. Print the BOM.

Course Outcome(s):

Export the Printed Circuit Board (PCB) design.

Objective(s):

1. Select the export tab.
2. When the export worksheet appears, select the components and PCB characteristics that are desired for exporting including the type of export.
3. Export the PCB layout that creates a Gerber file and if errors occur, cancel the export, correct the errors, and re-export the PCB layout.
4. Print the PCB layout and check for completeness (silk screen designations not located under parts, part labels close to parts, etc.).

Methods of Evaluation:

1. Design printed circuit board(s)
2. Tests & quizzes
3. Final Project

Course Content Outline:

a. Concepts, General
   i. Direct and alternating current laws as they apply to printed circuit traces
   ii. Sources of cross talk
   iii. Mechanical constraints
   iv. Electrical constraints
   v. Design principals
   vi. Ground loops
b. Designing circuit boards
   i. Custom Component Footprint creation
   ii. Building circuits in Multisim (schematic capture program)
   iii. Exporting capture to Ultiboard (PCB layout program)
   iv. Rats nest
   v. Sizing the circuit board
   vi. Placing components
   vii. Routing the circuit
   viii. Net list editor
   ix. Placing traces on the top and bottom of the circuit board
   x. Silkscreen
   xi. Design rule check
   xii. Selecting different attributes

xiii. creating custom foot print
xiv. Saving the User file

c. Creating a Gerber file
   i. Setting parameters to be exported
   ii. printing the circuit in various sizes
   iii. selecting the 274X format
   iv. Exporting
   v. Examine the export for errors

d. Skills
   i. Use of a caliper
   ii. Reading technical documents and applying the concepts
   iii. Evaluate, analyze, and troubleshoot circuit boards
   iv. Compare and contrast different kinds of circuit boards (flex, rigid, etc.)

e. Issues
   i. Safety concerns
   ii. Environmental concerns
   iii. Procedural concerns
   iv. Regulation concerns

Resources

National Instruments. *Ultiboard User's Manual*. Changes with software updates. National Instruments, Current version.

Scarpino. *Designing Circuit Boards with Eagle*. 1st Ed. Prentice Hall, 2014.

Eric Bogstin. *Signal and Power Integrity*. 3rd ed. Prentice Hall, 2018.

Brooks. *PCB Currents: How They Flow*. 1st ed. Prentice Hall, 2013.

Fabrian Ndagijimana. *Signal Integrity: From High Speed to Radiofrequency Applications*. 1st ed. Wiley, 2022.

National Instuments. *National Instruments Circuit Design Technical Library*. 2019.

Simon Monk. *Make Your Own PCBs with Eagle*. 2022.

Resources Other

1. Instructor provided lab documents, PowerPoint presentation, lecture video, and handouts
2. *Printed Circuit Board Designer's Reference; Basics*, Robertson, 1st ed., Prentice Hall 2004, ISBN-13: 9780130674814
3. *Signal Integrity Issues and Printed Circuit Board Design*, Brooks, 1st ed., Prentice Hall, 2003, ISBN-13: 9780131418844

Top of page
Key: 1619