# Fundamentals of Design, Layout and Fabrication of PCB

**Software – Altium Designer**

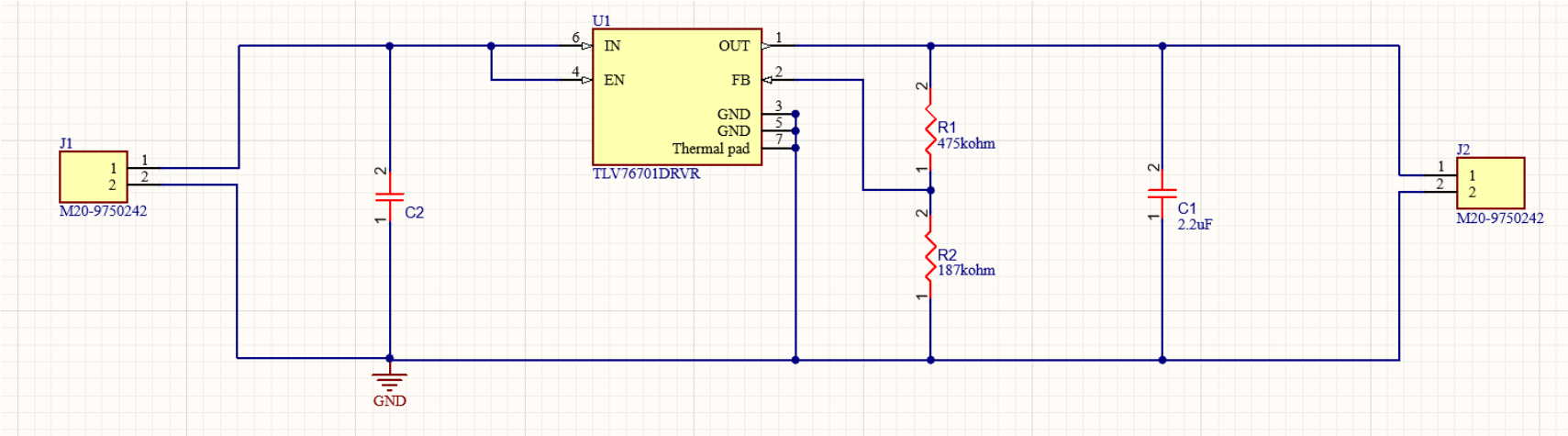
Prof. Kurian Polachan, IIIT-B

# Schematic

# Things to Learn

- Manufacturing part search, placing and wiring of components. View/hide component parameters (e.g., make and value), Component positioning (mirror image, rotation), Grid size adjustments, auto designators
- Adding components to the schematic by downloading ecad models via Samacsys
- Managing Samacsys schematic and PCB library locally for supporting sharing of projects
- Concept of thermal pad, connector packages (positions, rows, pitch size etc.,)
- Creating active BOM. Prioritizing vendor. Fixing the currency.
- Schematic no-connection symbol, schematic checks etc.,
- Generating PDF report comprising the schematic and BOM.
- Introduction to hierarchical schematic design. Concept of sheet symbols
- Exercises: regulator designs, battery charging modules, controller designs (RP2400, PSoC), sensor designs, etc.,

# E.g., Schematic



# Checks

- Use generic components with generic footprints for passives (enter the passive value in the value or comment section)
- Ensure the component libraries resides in the local project
- Ensure that the component designators are assigned
- Ensure that the text associated with the components are readable. Ensure that only required texts are made visible.
- When wiring, ensure that the joints are present only at the junctions
- Ensure no errors when validating the schematic

# Regulator Designs

Objective: Learn schematic entry of regulators using Altium Designer

- Exercise#1: 3.3V and 1.8V voltage regulator using TLV76701.
- Exercise#2: 3.3V and 1.8V buck voltage regulators using TPS62A01A
- Exercise#3: 3V regulator using RT6150BGQW
- Exercise#4: 3.3V regulator using RT6150B-33GQW

Note – Ensure that SamacSys library loader for Altium is installed.

# SoC Designs

Objective: Learn schematic entry of SoCs e.g., PSoC-5 and RP2040

- Exercise#1: Design a stamp board for PSoC-5
- Exercise#2: Design a stamp board for RP2040

Learn –

- Stamp Board – What's it
- Different power domains
- Decoupling capacitors

# Layout

# Things to Learn

- Introduction to different PCB layers and its stacking (top, bottom, top solder, bottom solder, top overlay, bottom overlay, drill)

- Creating a board shape
  - Draw the board shape in Keep-Out layer
  - Design – Board Shape – Define Board Shape from Selected Objects
  - Toggle Units (View – Toggle Units)
  - Set Origin (Edit – Origin – Set)

- Load the components from schematic (Design – Import Changes From …) *Untick loading of classes.

- Placement of the components (refer component datasheet for reference placements and layout)
  - For rotation: Select Component + Key “Space Bar”
  - Push component to bottom layer: Select Component + Key “L”
  - View 3D: Key “3”, View 2D: Key “2”
  - View 3D: Zoom: Control Key + Mouse, Rotate: Shift Key + Mouse, View Control: View – 3D View Control

- Routing of the Components
  - Design Rules Import/Export: Design – Rules - Right Click - Import/Export. *Select all rules to export/import *Use rule wizard if design-rules not visible
  - Autoroute: Route – Autoroute – All
  - Manual Interactive Routing, Adding via and go to a different layer – Ctrl + Shift + Scroll button of the mouse
  - Changing the size of text in top/bottom overlay – Refer to Altium PCB Board Layout - Rapid Tutorial in YouTube

- Design Rule Check
  - Ensure design rules are set (imported) correctly
  - DRC Run: Tools – Design Rule Checker – Run Design Rule Check *Ensure no errors/warnings

# Layout Guidelines

- Design the shape of the board to fit the planned hardware enclosure
- Fix the board design rules for ease of fabrication; minimize cost and improve yield.
- Position components for ease of routing; minimize trace width and vias. Minimize crossing of nets.
- Co-locate components sharing connections → Helps avoid long traces spanning the length and breadth of the board.
- Fix and lock the positions of connectors, interfaces (e.g., USB), switches, large ICs etc., prior to placing any other component on the board.
- For SMD designs, place all surface mount devices on one side of the board for ease of soldering.
- Try different orientation of components to ease the routing

reference - https://resources.altium.com/p/pcb-layout-guidelines

- Add ground planes (e.g., polygon pour ground on top and bottom layer for a 2-layer board) reduces the loop area formed by the signal paths and their return currents. Helps reduce EMI. Ground planes also act as heat sink, dissipating heat more effectively from the components to ambient. Reduces cross talk between signal traces. Allows easy connection of component ground pads to ground.

- Separate analog and digital grounds and then connecting them at a single point; referred to as a "star connection" or "star grounding" for minimizing interference of digital systems on analog circuits. Minimizes ground bounce.

reference - https://resources.altium.com/p/pcb-layout-guidelines

# Fabrication

21 February 2024

Prof. Kurian Polachan, IIIT-B

# Things to Learn

- Fabrication steps of 1-layer and 2-layer boards. Fabrication/printing of different layers – signal layer, solder layer, overlay layer, drill files etc.,

- Generating fabrication files
  - Generating Gerber, drill and assembly files
  - Online tools to view fabrication files

- Board Specification
  - Selection of dielectric, surface finish, copper thickness, stencil etc.,
  - Cost Estimation

- List of PCB Fabs
  - PCB Power https://www.pcbpower.com/ (Gujarat, India)
  - PC Process https://www.pcprocess.in/ (Bangalore, India)
  - PCB Gogo https://www.pcbgogo.com/ (China)

# Fabrication Steps

- What is a PCB? https://www.youtube.com/watch?v=YJr-kHy6STg
- 1-layer PCB Manufacturing Process , PCB making https://www.youtube.com/watch?v=t-zxq6aNg7Y&t=108s
- PCBWay - How to make a 2-layer PCB (full version) https://www.youtube.com/watch?v=aIiKpdY68qM
- Other references
  https://www.4pcb.com/media/presentation-how-to-build-pcb.pdf
  https://www.raypcb.com/pcb-board-manufacturing/
- Learn –

  Signal Layer Printing
  Solder Mask Printing
  Legend (Overlay) Printing
  Drill Holes & Plating

**Layer Stack of 2-Layer PCB**

| # | Name | Material | Type | Weight | Thickness | Dk |
|---|---|---|---|---|---|---|
| | Top Overlay | | Overlay | | | |
| | Top Solder | Solder Resist | Solder Mask | | 0.4mil | 3.5 |
| 1 | Top Layer | | Signal | 1oz | 1.4mil | |
| | Dielectric 1 | FR-4 | Dielectric | | 12.6mil | 4.8 |
| 2 | Bottom Layer | | Signal | 1oz | 1.4mil | |
| | Bottom Solder | Solder Resist | Solder Mask | | 0.4mil | 3.5 |
| | Bottom Overlay | | Overlay | | | |

# Generate Board Files

- Generating Gerber Files
  Select the .PcbDoc File, File → Fabrication Outputs → Gerber Files
  The above step creates a .Cam file.
  Select the .Cam file, File → Export → Gerber. Do not alter any default value setting.
  The above step generates gerber files. Save these files to folder: fab_folder.

  Select the .PcbDoc File, File → Fabrication Outputs → NC Drill Files
  The above step creates a .Cam file.
  Select the .Cam file, File → Export → Gerber. Do not alter any default value setting.
  The above step generates drill files. Save these files to the folder: fab_folder.

- Generating Assembly Files (if you want the PCB manufacturer to assemble components on the PCB)
  Select the .PcbDoc File, File → Assembly Outputs → Generate Pick and Place Files
  The above step creates new files in 'projects output' folder (inside the Altium project folder)
  Copy these files to fab_folder.

- Generating BOM
  Select the .BomDoc file, → Reports Bill of Materials
  The above step exports the BOM in CSV format.

**For fabrication and assembly – Send the fab_folder (zip) and BOM file in CSV format to the PCB manufacturer.**
For viewing the gerber files, upload the fab_folder(zip) in https://www.pcbgogo.com/GerberViewer.html

# Fabrication Specs

- Estimating Cost of PCB Fabrication https://www.pcbgogo.com/pcb-fabrication-quote.html

- Specs

  Material Type: FR4, Metal Base, Rogers
  Layers: 2, 4, …
  Min Track/Spacing:
  Min Hole Sizing:
  Gold Fingers: https://www.pcbgogo.com/Blog/WHAT_ARE_GOLD_FINGERS_.html
  Surface Finish: https://www.pcbgogo.com/knowledge-center/Surface_finish.html
  Finished Copper: https://pcbprime.com/pcb-tips/how-thick-is-1oz-copper/
  PCB Assembly: Should the PCB be fabricated and assembled. ?
  SMD Stencil: https://www.youtube.com/watch?v=sOZ_Bry957k

Objective – Learn to determine the cost of fabricating and assembling a PCB.

# Exercise

Objective: Estimate the price of fabricating and assembling 10 numbers of PICO board. Size of the dev board should be comparable to PICO board (https://www.raspberrypi.com/products/raspberry-pi-pico/)

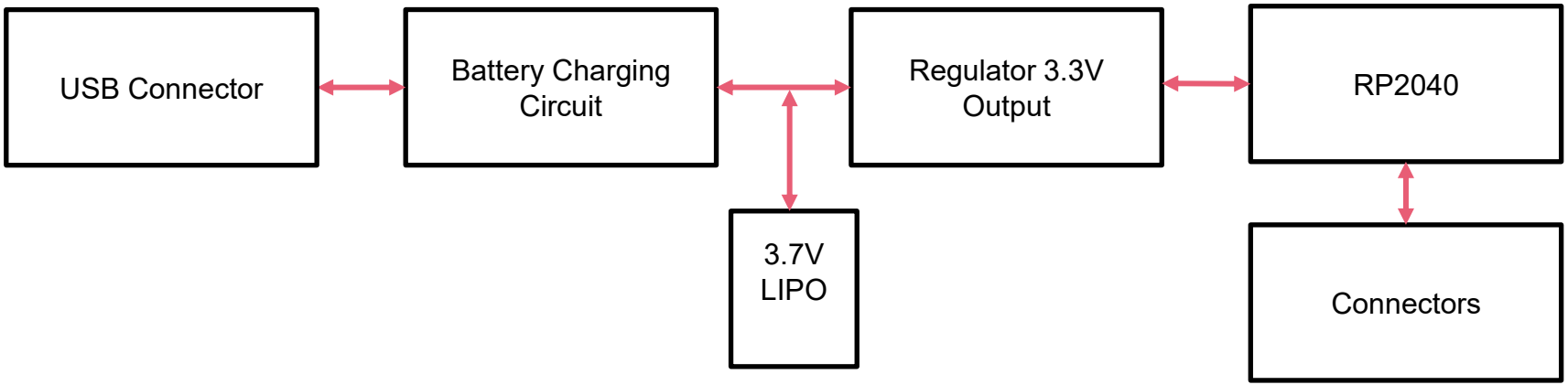
Use https://www.pcbgogo.com/pcb-fabrication-quote.html for estimating the price.

# PCB Design Exercise

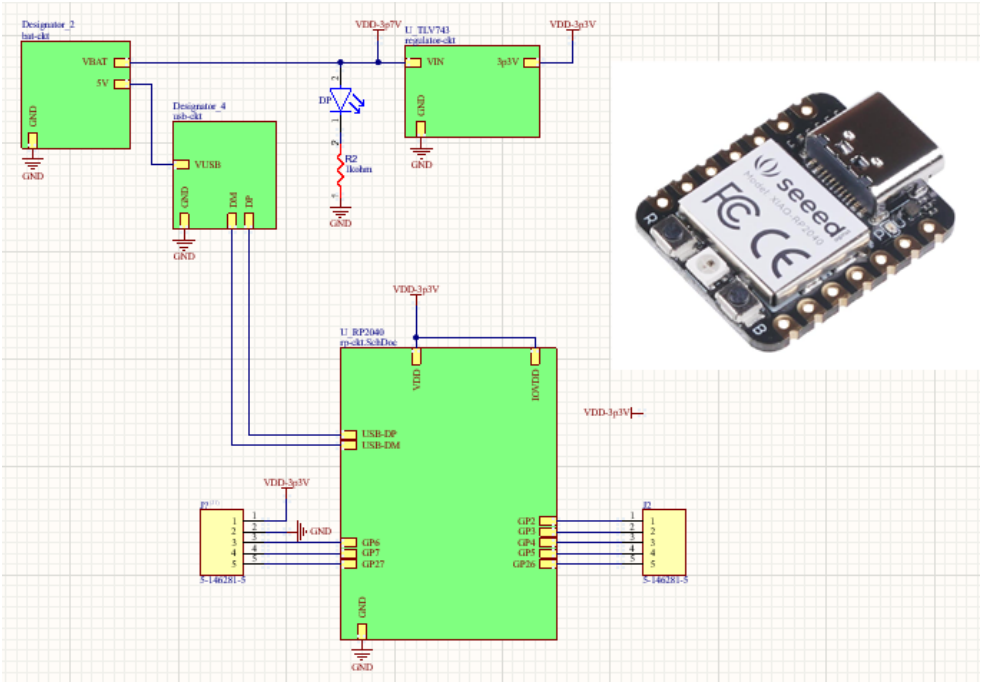
Prof. Kurian Polachan, IIIT-B

# Draft Schematic

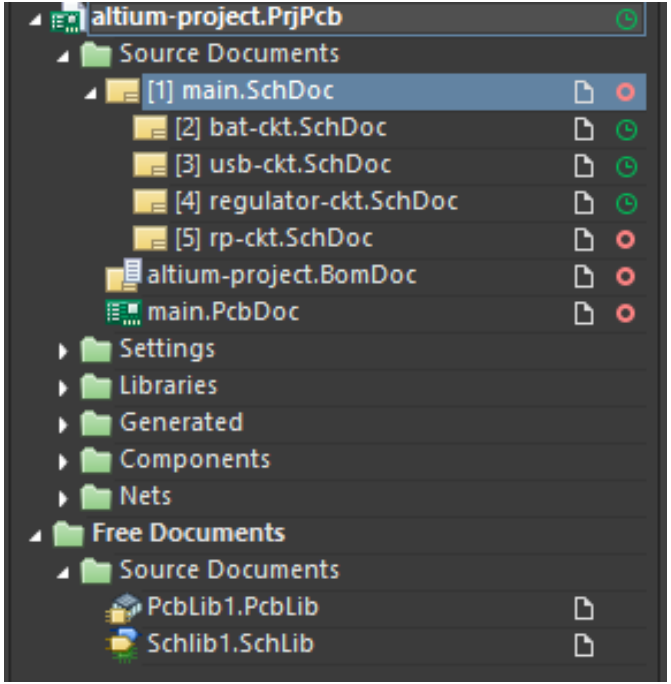
Design Requirement – A dev board based on RP2040 supporting on-board battery and QWIIC connector interface.



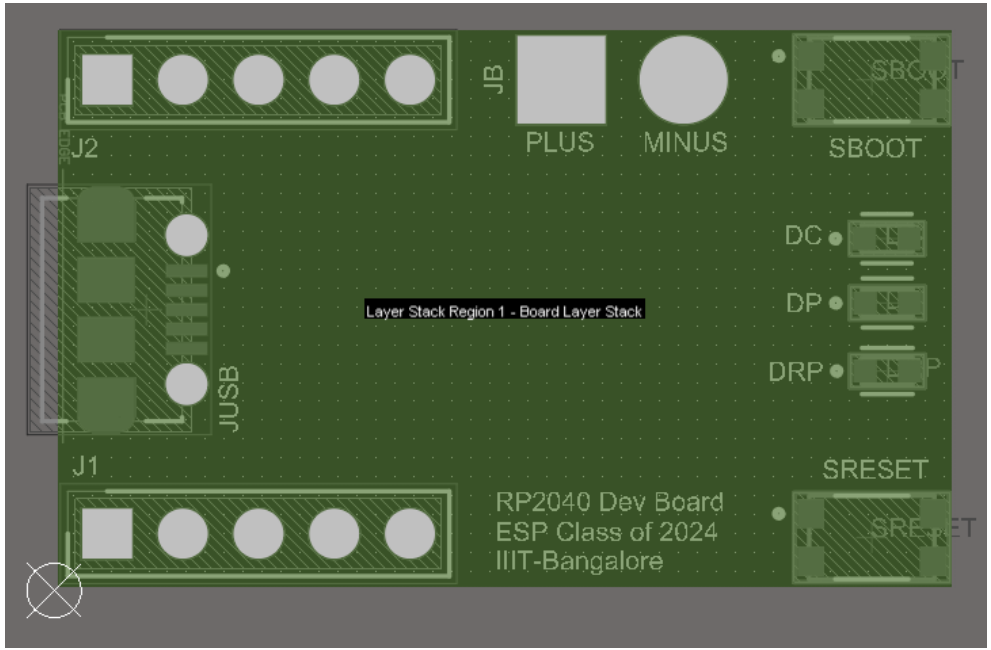
# Schematic


Designator_2
bat-ckt
VBAT
5V
GND
Designator_4
usb-ckt
VUSB
GND
DM
DP
VDD-3p7V
U_TLV743
regulator-ckt
VIN
3p3V
VDD-3p3V
DP
R2
1kohm
U_RP2040
rp-ckt.SchDoc
VDD
IOVDD
USB-DP
USB-DM
GP6
GP7
GP27
GP2
GP3
GP4
GP5
GP26
J2
5-1462281-5


Seeed
FC CE


altium-project.PrjPcb
Source Documents
[1] main.SchDoc
[2] bat-ckt.SchDoc
[3] usb-ckt.SchDoc
[4] regulator-ckt.SchDoc
[5] rp-ckt.SchDoc
altium-project.BomDoc
main.PcbDoc
Settings
Libraries
Generated
Components
Nets
Free Documents
Source Documents
PcbLib1.PcbLib
Schlib1.SchLib

# Layout


J2
JB
PLUS
MINUS
SBOOT
DC
DP
DRP
JUSB
Layer Stack Region 1 - Board Layer Stack
J1
SRESET
RP2040 Dev Board
ESP Class of 2024
IIIT-Bangalore