

Getting started with Altium

ENGR100-950

Log into CAEN

- Open up Google Chrome
- A tab will open titled “S2 Hub” with all the Virtual apps
- Search for “Altium 15.1.14”
- Launch the app
- Launch on Cloudpaging Player
- Wait 20 min for Altium to open up...

Starting your project

- If you have a “Storage Manager” window that opens, feel free to close it
- File -> New -> Project
- Project Types: PCB
- Project Templates: <Default>
- Name: Whatever you want to call it
- Click “OK”
- Project should appear in the “Project” taskbar on the left

Adding relevant documents

- Right click on your project in the left taskbar
- Add New to Project -> Schematic
- Add New to Project -> PCB
- Download the libraries that are provided on Canvas, and drag them onto the the left taskbar, ensure they are a part of your project
 - You may have to drag them to the taskbar and they will be under “Free Documents”
 - Drag again to ensure they are a part of your project and save your project immediately after
 - Right click on project title in left taskbar -> Save Project

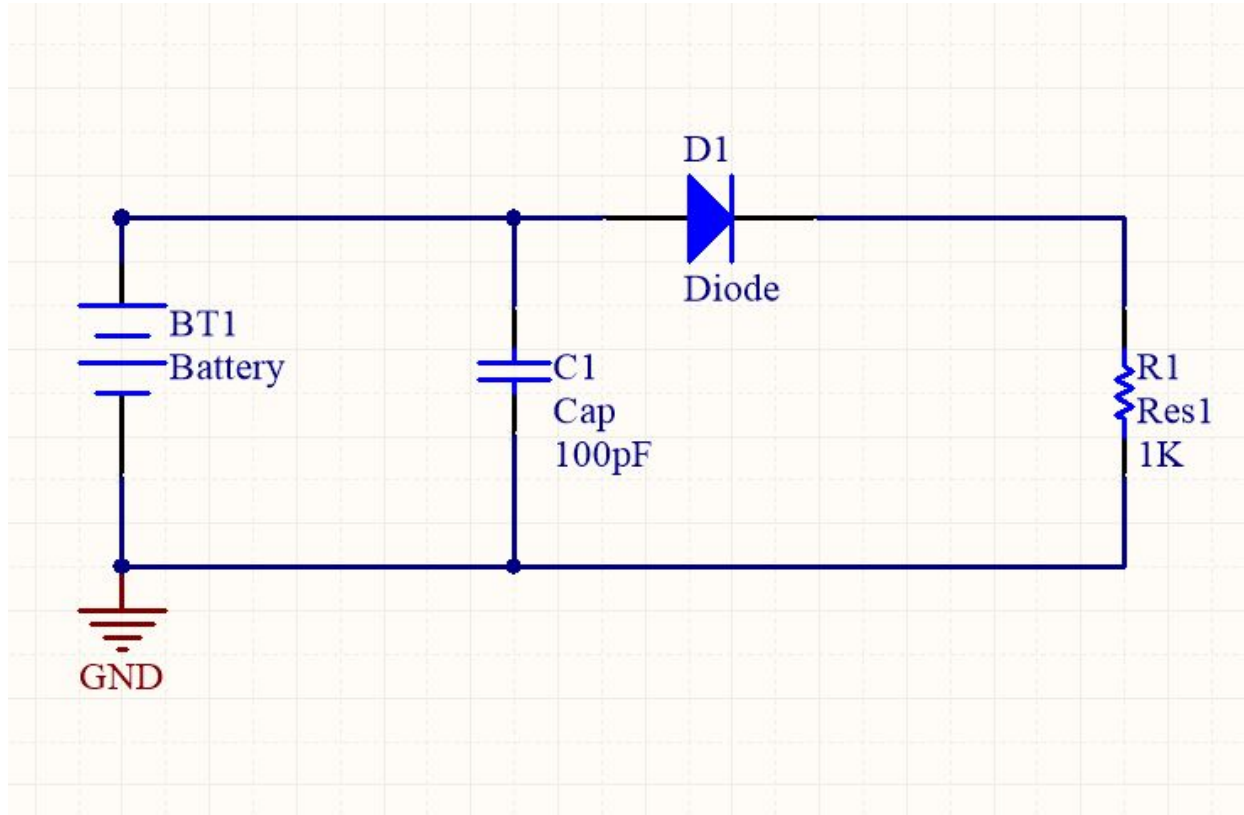
Using key shortcuts

- The letter underlined on the different options on the top taskbar can be used as key shortcuts to save you some time
- For example, on Altium schematic let's say you want to do the following :
Place -> Part
- Instead of hovering your mouse over to “Place” and then to “Part”, you could instead just press “p” and “p” on your keyboard and you will end up with the same result
- Will save you a lot of time when placing components, wires and traces

Some common shortcuts

- Schematic
 - p+p - Place a part
 - p+w - Place a wire
- PCB
 - p+t - place a trace
 - p+g - place a polygon pour
 - t+g+a - refill a polygon pour (after adding traces that cut through the polygon pour)

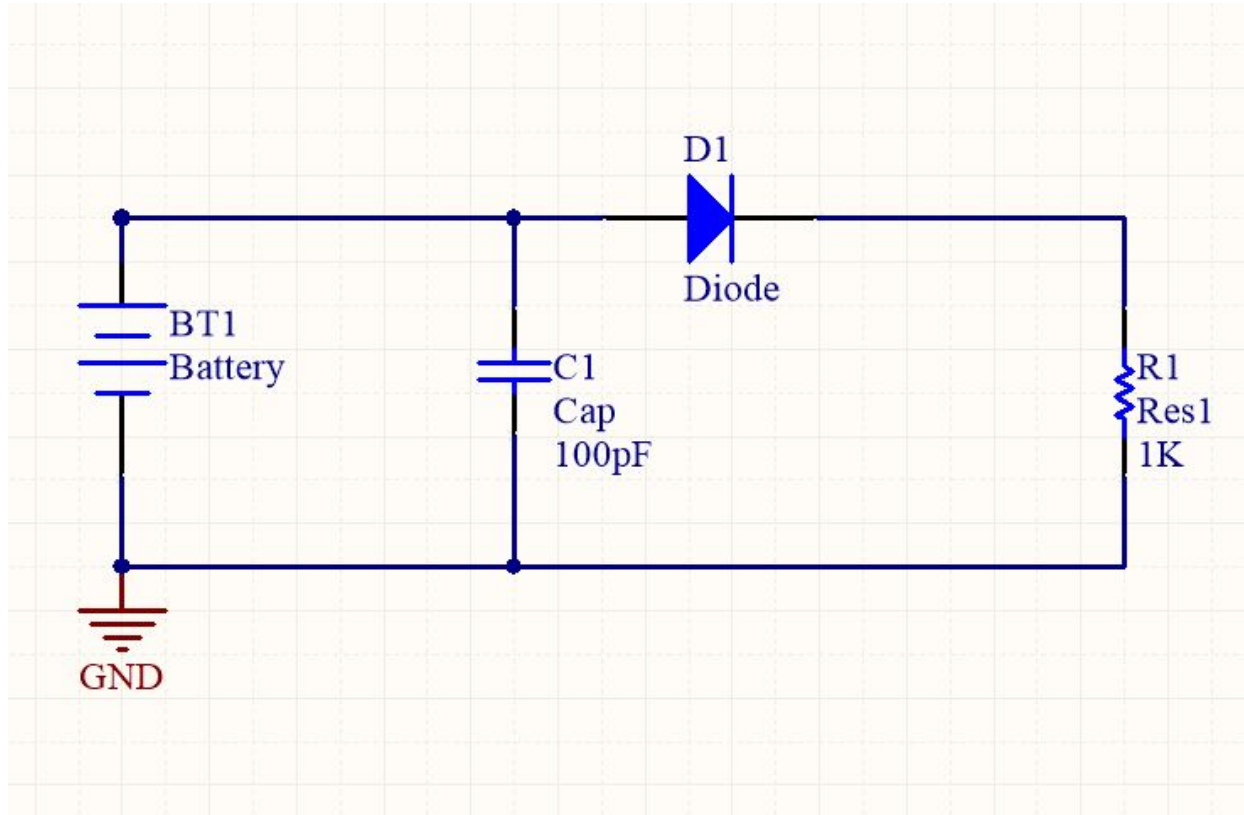
Circuit to Build



Back to building your circuit

- Go to your schematic document
- Place components by going to Place -> Part
- Click on “Choose”
- You will have 3 schematic libraries you will have parts from
 - Miscellaneous Devices
 - Miscellaneous Connectors
 - ENGR100Sch
- Find the component, click “OK” and “OK” again
- Place the component on the schematic, and press “Esc” + “Esc” if you do not need more copies of that component
- You can press the spacebar while placing the component to rotate it

Circuit to Build



Connecting components

- Once components have been placed, there are a couple of ways to “connect” them
- Wire:
 - Place -> Wire
 - Allows you to drag a “wire” that connects two components on the schematic
 - Your net/wire will have a cross mark at your pointer, if that cross mark becomes large and red you know you are at another pin
 - If you connect across a wire a dot will appear that confirms electrical connection
- Net:
 - Place -> Net Label
 - Allows you to place a net that can be labelled a certain thing (5V, ARD_D1, HUM_OUT,etc.)
 - If two pins are connected to the same net label then they will be electrically connected
 - Will reduce the number of wires and make it easier to understand what components are connected in your schematic visually

Some schematic pointers

- GND has a separate net: Place -> Power Port
 - Depending on your preferences, this may not be the case
 - ^If above holds true, can either edit preferences or have another net that is labelled GND
- Schematic layout is only concerned with electrical connections, not mechanical footprints
 - That is for the PCB
- Make sure you label your components (Your resistors should not be “R?” but “R1”, “R2”, etc.) or you will get errors when compiling

Finishing up schematic design

- Double click on each component
- Under “Models”, click on “Edit”
- Under “PCB Library”, select “Any” if not already selected
- Under “Footprint Model”, select “Browse”
- You will have libraries for:
 - Miscellaneous Devices
 - Miscellaneous Connectors
 - ENGR100Pcb
- Select the correct footprint corresponding the spreadsheet provided (Canvas > Lab Files > Altium Resources) to ensure we have the correctly sized component for you when soldering
- Click “OK” (x3)

Saving and Compiling Schematic

- Right click on your project -> Add New to Project -> Output Job File
- Under Validation Outputs -> Right click on Add New Validation Output -> Electrical Rules Check -> Your schematic document
- On the left taskbar, right click your Output Job File to save
- Right click on the .SchDoc document under your project on the left Taskbar and click “Save”
- Right click on the .SchDoc document under your project on the left Taskbar and click “Compile document”
- On the bottom right, find and click the button System -> Messages
 - If you find any errors or warnings, correct them
 - If not, you should not have electrical issues
- This does not mean your connections are correct wrt your circuit, so please quadruple check your connections

Onwards and Upwards (Upwards = to PCB design)

- Save your PCB (it will ask you to rename the file)
- Go to your PCB document (.PcbDoc)
- Design -> Import Changes from (..)
- Under “Add Rooms”, untick the Add box
- Click Execute Changes
- Click Close
- Component footprints should appear on the screen
 - They will appear to the left of the board, you can drag across the screen by holding down on right click and moving your mouse
- Now the fun begins!

PCB General View and Sizing

- Pressing 1 will show the board size and shape
 - Can use to adjust the dimensions of the board
 - Design -> Edit Board Shape
- Pressing 2 will show the board and components as a 2-D model
 - Ideal for actually building your PCB
- Pressing 3 will show the board and components as a 3-D model
 - Ideal for getting a visual idea of how your PCB will look
 - Not ideal for actually moving parts and traces
 - Can use a combination of Ctrl and Shift keys along with the mouse to investigate the board further

Editing part placement on PCB

- Click on part to drag it
- You can rotate the part by pressing the spacebar while clicking/dragging the component
- You can switch the component between top and bottom layers by toggling the L key while selecting the component
- HIGHLY HIGHLY suggest placing all the parts before you start connecting them
- Don't worry if the gray lines are intersecting; traces can be drawn around them if needed
- You can move parts around after you start connecting them if needed, just have a general idea of where they will all be on the board

Traces

- Route -> Interactive Routing to place the trace
- Make sure you know which layer of the board you are on
 - We are using a two layer so the only two layers traces should be on are “Top Layer” and “Bottom Layer”
- Can adjust layer of trace after placing on board by double clicking on the trace
-> Layer -> Select Layer
- Once you see a larger circle around your pointer you know you’ve established an electrical connection
- Trace Guidelines are provided at end of presentation

Layers you need to worry about

- Top Layer - Top of the board electrical connections
- Bottom Layer - Bottom of the board for electrical connections
- Top Overlay - Your silkscreen for the top layer, can edit labels and add any text
- Bottom Overlay - Your silkscreen for the bottom layer, can edit labels and add any text
- You will use Mechanical 1, Keep Out and a couple of other layers later, but don't worry about them right now

Polygon Pour

- Can be used to remove the need for too many traces all over the board
- A pour that is usually used is a GND or 5V over the entire top or bottom layer
- For example, if you add a Polygon pour for the GND net for the entire bottom layer, then all components that have GND pins in the bottom layer will all be connected and you do not need traces to connect them
- Place -> Polygon pour
- Under Net Options, select the net you'd like to connect to
 - The net should have been defined in your schematic
- Under Properties, select the layer you'd like this pour to be on
- Click okay, and click around until you are satisfied with the shape, then click "Esc" to have your pour
 - You can adjust dimensions of the pour later if need be

Design Rule Check (Actual rules are in the last slide)

- Ensure you are achieving mechanical specs set by the manufacturer based on the fidelity of their equipment
- To edit a Design Rule, click Design -> Rules
- On the left bar, search the different sections till you find the specific rule you need to change (most rules are under Manufacturing)
 - Can double click to edit
- Enter the value (and units) that is specified and click “Apply” followed by “OK”
 - Values to edit are above the diagram in blue
- Once you have all rules edited per the guidelines, click on Tools -> Design Rule Check to run the DRC
 - Click on Run Design Rule Check on bottom left
- You need ZERO errors before you can manufacture the design, we will require it as part of your final submission

Trace Width Guidelines

- Power lines to the input of the LDO's = 30 mil
- Output of LDO's to electrolytic capacitors and Status LED's = 25 mil
- All other traces = 15 mil
- NOTE: The other traces can be thinner if needed, but keep the power lines as thick as possible to ensure excessive current doesn't burn them off

Design Rules

- All below are minimum values
- 6 mil trace clearance (minimum solder mask sliver)
- 6 mil trace width
 - Under Routing
- 13 mil drill size (hole size)
- 7 mil annular ring
 - You might need to create a New Rule for this one