

UiO : **Department of Physics**  
University of Oslo

**FYS4260 – Spring 2019**  
Microsystems and electronic packaging and interconnection technologies

## **OBLIG 1**

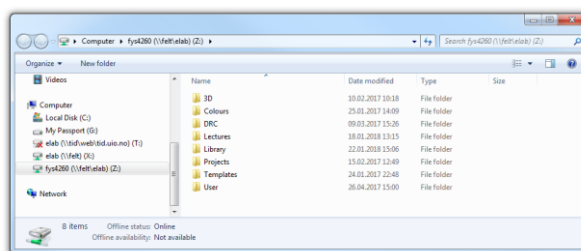
### **Run-thru project; processes and tools**



UiO : **Department of Physics**  
University of Oslo

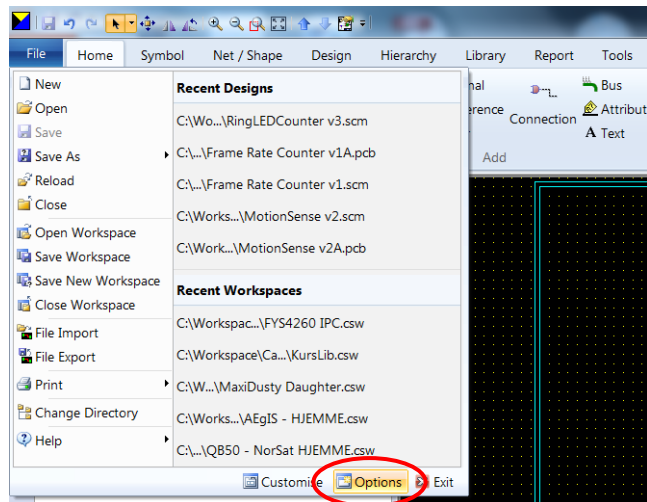
## **Set up CadStar**

- Map Drive <\\Felt\\Elab\\FYS4260>



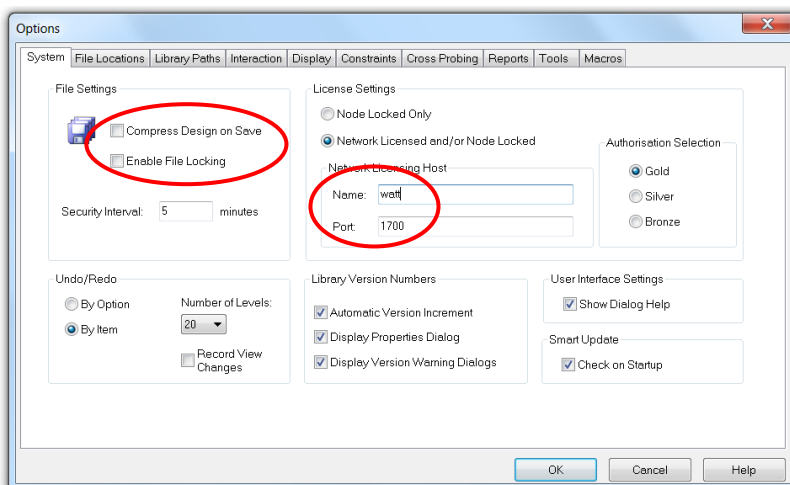
- Start CadStar
  - All Programs -> CadStar 18 -> Design Editor

## Select File menu -> Options



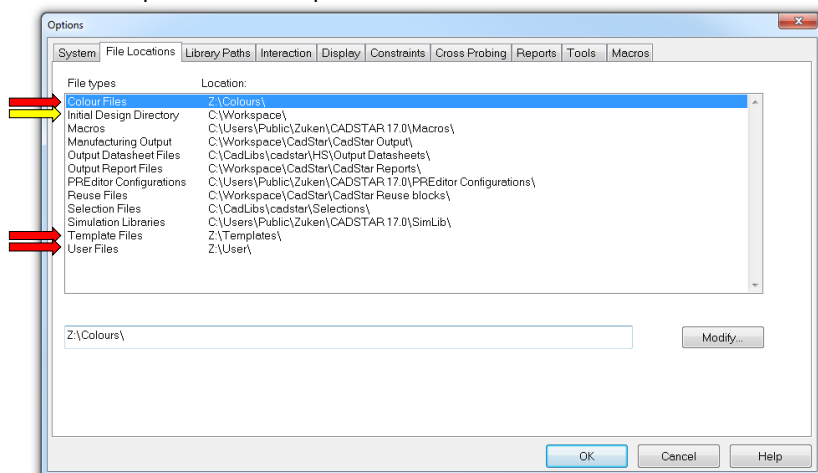
## System tab

- Network license: Name: watt Port: 1700



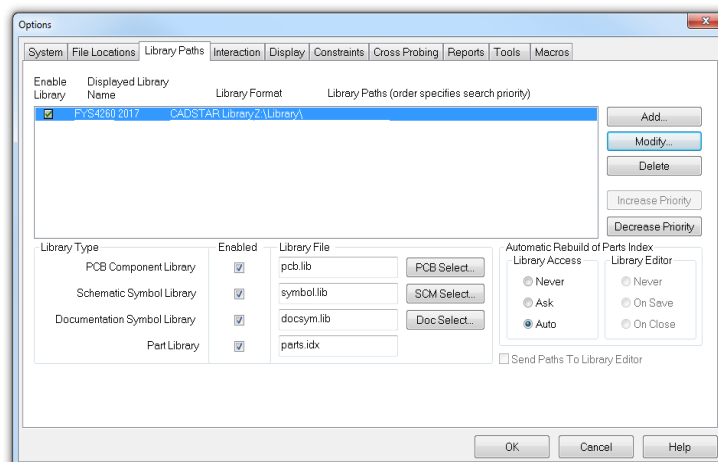
## File Locations tab

- Z mapped to \\Felt\\Elab\\FYS4260 • User Files: User folder
- Color Files: Colours folder
- Template Files: Templates
- Recommend to set *Initial Design Directory* to your work folder.



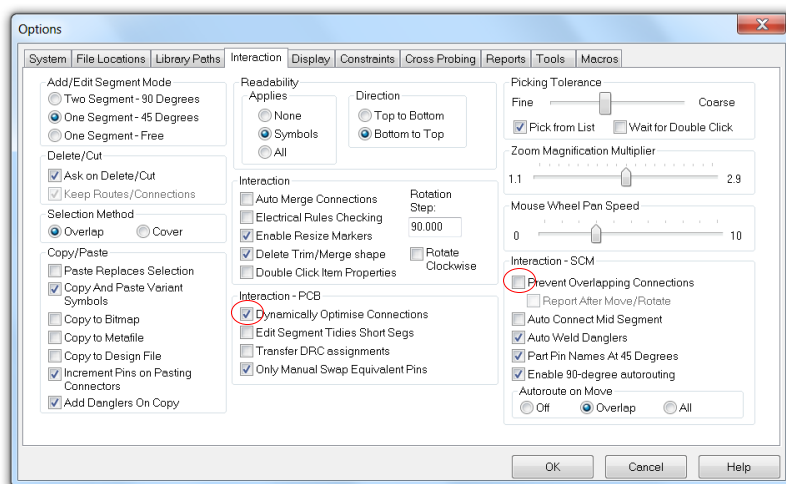
## Libraries tab

- Remove Default cadstar library if enabled
- Add Z:\Library folder (from \\Felt\\Elab\\FYS4260)



## Interactions tab

- To your preferences, see settings below
- Recommend to uncheck "Prevent Overlapping Connections"



# Save Workspace!

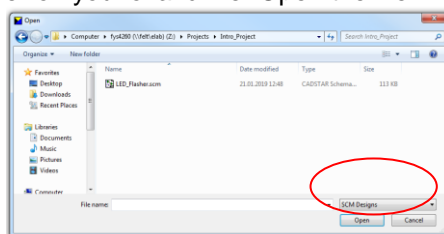
- File -> Save New Workspace
- Save in your home folder! Then its available on all computers the next time you come to the lab.
- Use this workspace on your main project as well.

## About oblig 1

- In this lab we will design a simple circuit that uses a 555 timer IC to flash a LED.
- You will start with an unfinished schematic, where you will add a connector and some tracks to make a complete circuit.
- Then we will move this over to the PCB routing tool. Some of the parts have been placed for you, you shall complete the placement and routing.
- Then you will go back and make a change in the schematics, update the pcb and redo the routing.
- At the end you will use Boardmodeller to create a 3D representation of your board.

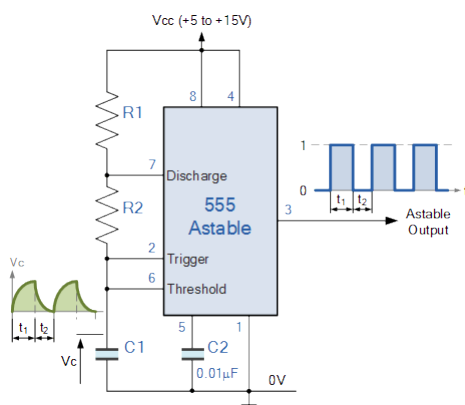
## Open schematic

- Open the schematic
  - Copy the file  
[\\Felt\\Elab\\FYS4260\\Projects\\Intro\\_Project\\Led\\_Flasher.scm](\\Felt\\Elab\\FYS4260\\Projects\\Intro_Project\\Led_Flasher.scm)  
to a folder on you loval drive. Open the file in CadSTAR.



- If you don't see the correct file, make sure the dropdown menu right above the open button says «SCM Designs» or «All Design Files».
- Save the file on your home drive!!!

## Intro Project



The circuit consist of one 7555 (low voltage version of a 555) timer chip connected in the astable mode, making the output toggle high-low. Frequency and duty cycle is set by resistors R1- R2, and capacitor C1.

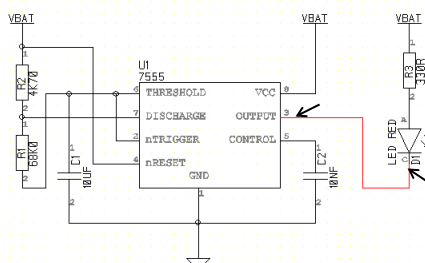
The output is directly connected to a high side LED, making the LED light up when the 7555 output goes low.

## Add LED connection

- On the Home tab find the Connection tool

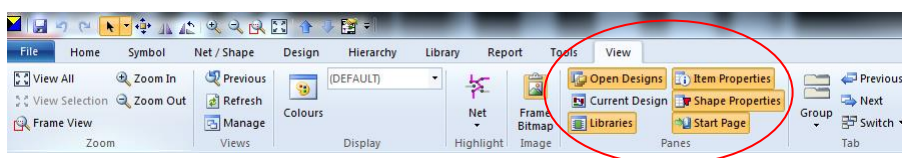


- One track missing to complete the LED connection. Click on terminal 3 on the 7555 chip, click to insert corners, and end on the cathode of the LED to complete the connection.



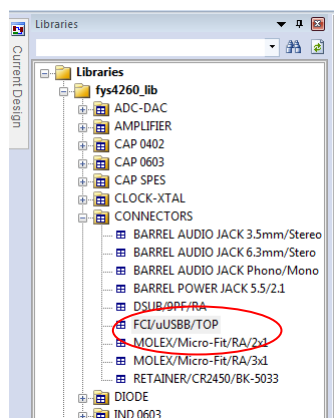
## Library Pane

- Make sure the library pane is visible by clicking the «View» tab and select «Libraries»



- The library will show up as a pane, usually on the left side of your screen.
- Use the pin symbol to select auto hide or always on.

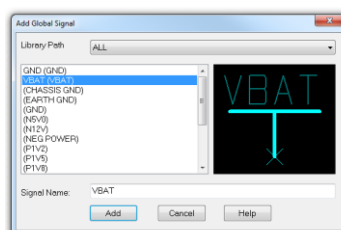
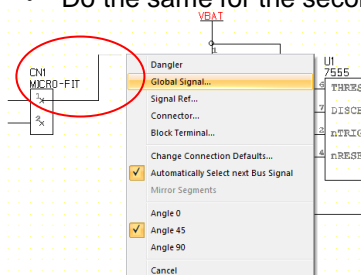
## Add connector



- Expand the FYS4260 library and find the Connectors folder.
- Select either a micro USB B-connector (uUSBB) or the polarized 2-pin Micro-Fit battery connector. (Don't matter, design choice...)
- Drag and drop onto schematics. Place connector on the left side of the circuit.
- ESC to abort.

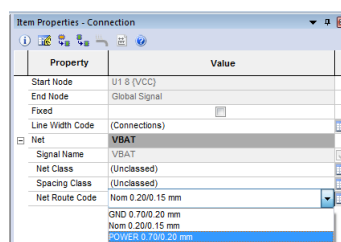
## Add connections

- Start on pin 1 of the connector to add a power signal.
- A power signal is often added as a global signal, that is a named net which is connected through its name and doesn't have a visible connection.
- To add a global signal start adding a connection, then right click and select «Global Signal». Select VBAT in the meny, and place.
- Do the same for the second terminal, but select signal GND.



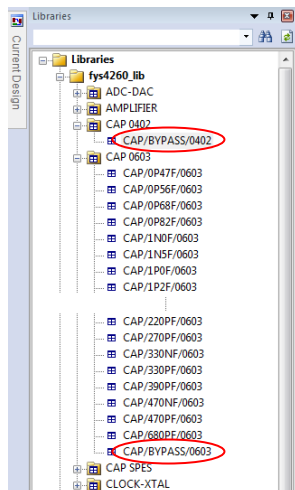
## Set route codes.

- Different nets have different functionality and requirements.
- A power net might need a thicker route to handle more current, a high speed net might need length matching or meet special spacing requirements. In this course we will only add requirements on route width, and separate between power nets and «nominal» nets.
- To define VBAT as a power net and set correct requirements to track width, select the VBAT net somewhere in the design such that it is highlighted.
- If you have the «Item Properties» pane visible, you will find the Net route code setting under «Net», expand and change to «POWER 0.70/0.20mm».
- If not you find the item properties by right clicking when you have selected a net. Click «item properties», then «Net» and select «Net Route Code» to change.
- See the colour of the connection change!
- Do the same for the GND net, choosing «GND 0.70/0.20mm»



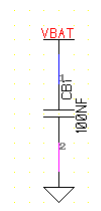


## Add Decoupling/Bypass capacitor



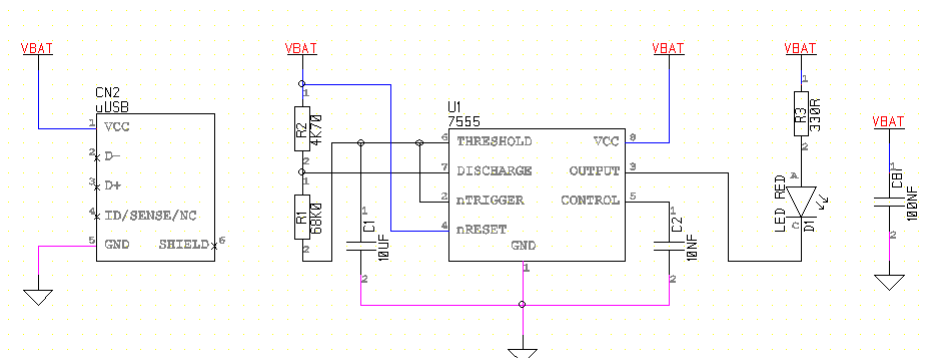
- Find the CAP0402 or CAP0603 folders in the library. (0402 is physically smaller than 0603, takes less board space, but harder to place)
- Add a BYPASS capacitor to the design.
- Connect to VBAT and GND

A bypass is just a normal ceramic 100nF capacitor placed across the power domains. Think of it as a small battery for all active components.



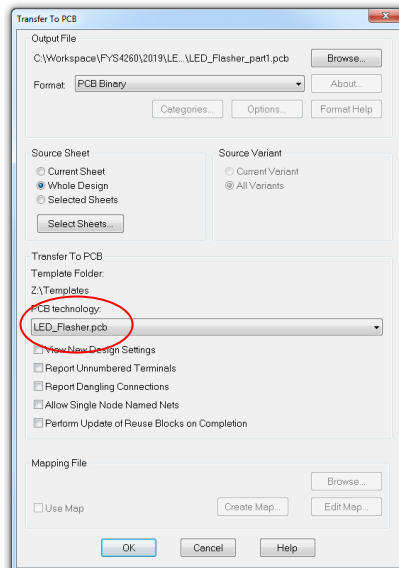
## Final Schematics

- Your schematics should now look something like this



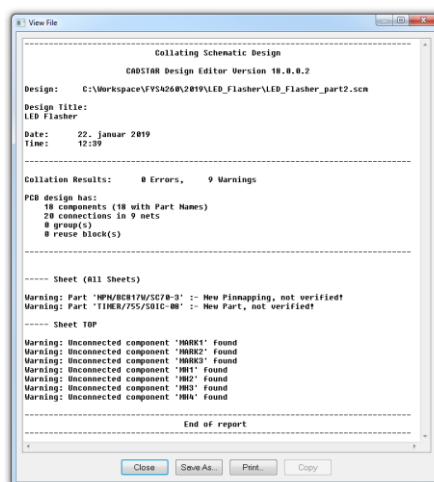
## Transfer to PCB

- Locate «Transfer to PCB» button on the «Design» tab.
- In the next window select «LED Flasher» under «PCB Technology».
- All other settings can be left as is for now.
- OK



## Transfer to PCB

- Read the report. It gives a short summary of the design. You should have no errors, but a few warnings is normal.
- Warning on a part is usually a notification that is added to a part in the library, to give information to the user.
- Unconnected component is just that, a component that is not connected to anything. May or may not be an error, only you as the designer knows that!
- Hit «Close» when you are ready.



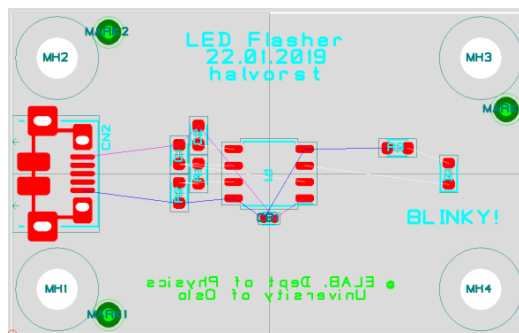
## PCB Editor

- On a successful transfer you enter the PCB editor part of CadSTAR.
- The main window is your design canvas. You will see a frame that is the board outline of your pcb. Inside some of the components have been placed for you.
- Unplaced components are always placed at coordinates X0Y0 (bottom left).
- Lines showing the different net connections to be made in white, blue and pink.
- Zoom out to, select the unplaced components and move them closer to the board outline.

## Place connector

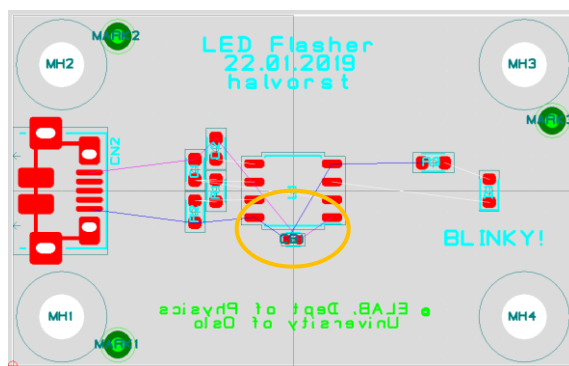
- Place the connector on the left side of the board. Make sure to orient it correct with the mating side facing out. This is shown with small arrows on the component.

- Hotkeys
  - F2 to move
  - F3 to rotate
  - Space to place
  - Mouse gestures



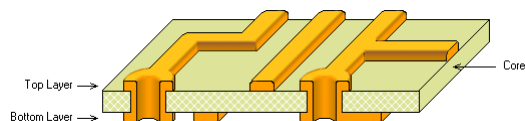
## Place bypass capacitor

- Place the capacitor close to the power pins of the 7555 timer chip to minimize impedance.



## Short on PCBs

A PCB is a «sandwich» of multiple layers. The frame is an insulating material, often made of glass reinforced epoxy. On one or both sides is a copper layer making out the actual conductive traces and planes. To connect two or more layers vias are used, a via is a hole with conductive walls.



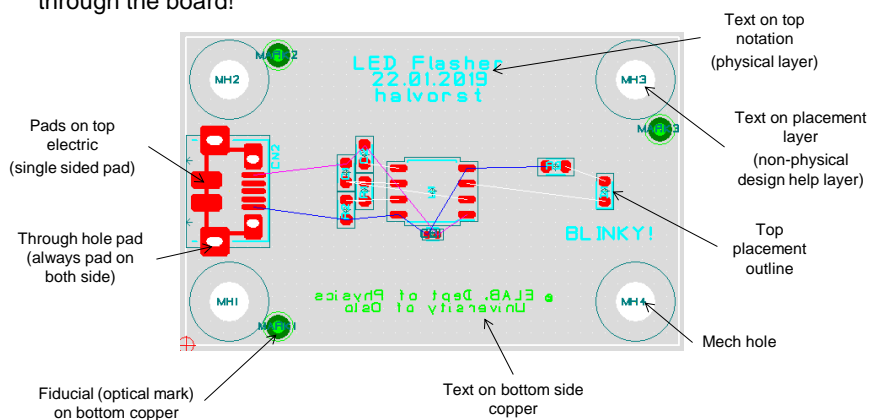
These are the basic layers of a PCB, in addition most PCBs have at least one protective layer on top of the copper, and a notational layer.

In the CAD software we have non physical layers as well, to aid in the design process. This can be placement layers to aid in component placement, assembly layers or documentation layers.

## PCBs in CadSTAR

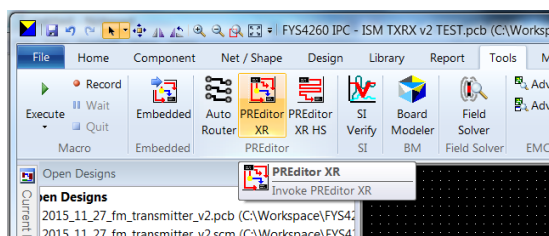
In CadSTAR colours denote the different layers. By default we always view the board looking from the top through the board!

Red	Top elec layer
Green	Bot elec layer
White	Mech hole/board outline
Bright blue	Top notation
Deep green	Placement outline top side



## Routing tool

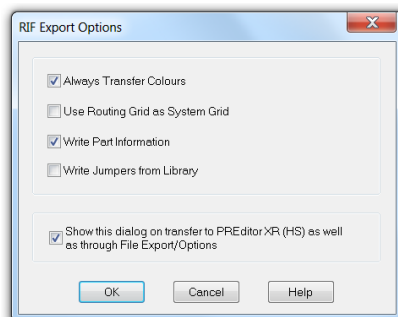
- We have now placed all components, but no physical connections has been made. To do the actual routing we are going to use «PREditor XR», find it on the «Tools» tab.



- This will transfer the design to the routing tool, closing PREditor will import the design back to cadstar.

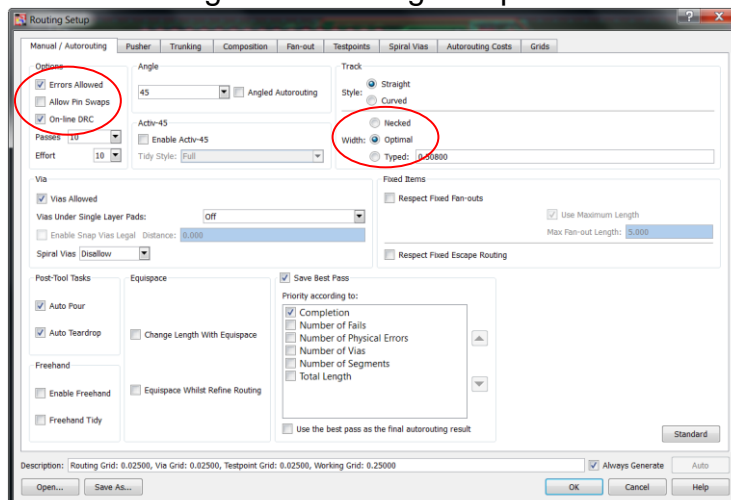
## PREditor - First use

- Make sure «Always Transfer Colours» are checked the first time you transfer a design, all other settings can be left as is/unchecked.
- If you change colour settings in PREditor later, uncheck «Always Transfer Colours» the next time you start to not overwrite the changes you have done.



## Setup PREditor – First use

Select Configure -> Routing Setup

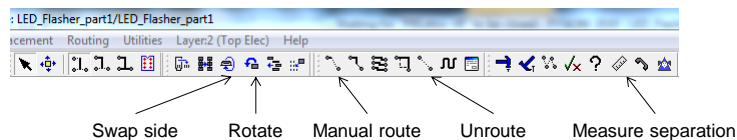


## Recommended settings

- Manual / Autorouter tab
  - Errors allowed will allow you to make illegal routes
  - On Line DRC will mark illegal routes in white colour.
  - Use optimal track width (Required)
  - No vias in pads (Required)
  - 45 degree routing
  - Active 45 -> Test and see if you like it.
- Pusher tab
  - Test it, use if you like.
  - Recommend to enable springback if you use pusher.
- Grid tab => **DO NOT CHANGE, BUG IN CADSTAR18**
  - Change 0.0254mm to 0.025mm.

## Routing in PReitor

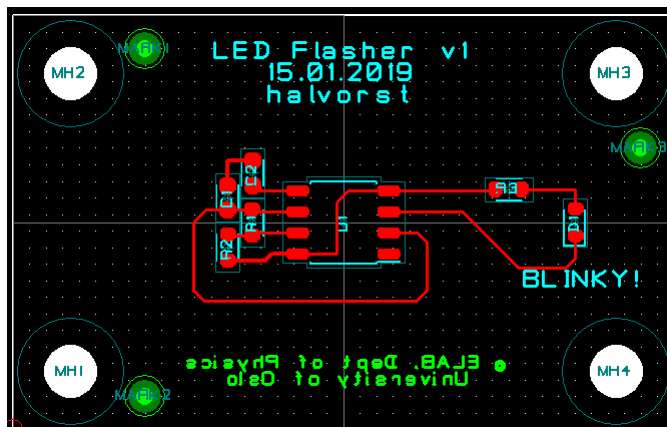
- Use Manual route tool to route signals.



- Start a route by clicking on a connection (not holding down). Depending on your active-45 setting, routing are done by moving the mouse in the direction you want, or by clicking for each segment you want to add.
- Doubleclick to insert via and continue routing on another layer.
- Unroute tool to delete route segments. Do not use DEL key, this will delete the net in CadSTAR!
- Change active layer with F5 / F6

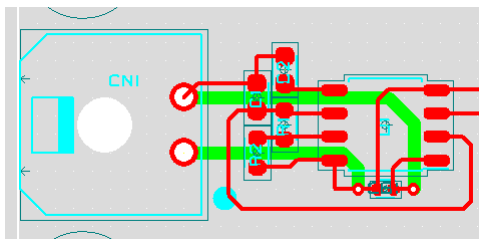
## Route signal tracks

- Example signal track routing



## Route Power tracks – Battery connector

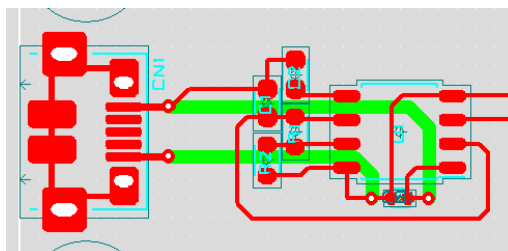
- Route the power tracks on the bottom layer, using vias to switch back to the top layer and connect to the bypass capacitor.
- Use 'o' and 'n' keys to change between necked and optimal route width, using necked only to enter pads that are smaller than the optimal width.
- If you are using the 2 pin battery connector, the pins are through hole and you can route directly from the pads on both sides.
- If using the uUSB connector, see next slide.





## Route Power tracks – micro USB connector

- If using the uUSB connector, this is single sided and you need route a little bit out from the pad and use vias to switch to the bottom layer.
- The pads on the connector are too narrow for the optimal route width, but we would prefer not to use a necked width for the power entry route. Using the 'c' key one can set a typed track width, and select it with the 't' key.
- Start a route from one of the pads on the connector. Hit 'c' and set route width to 0.4mm. Doubleclick to insert a via, then hit 'o' to change back to optimal width on the bottom side.

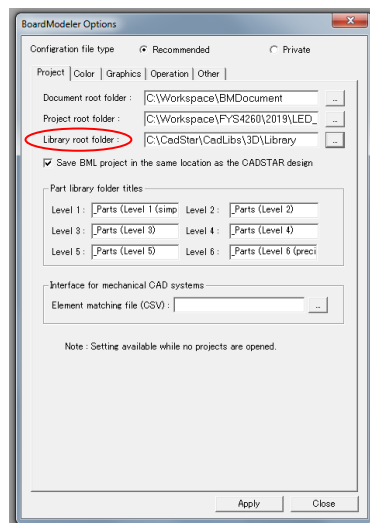


## Design rule check

- Hit save button. Do not give design a new name!
- File -> Close to close PReDitor, when prompted in Cadstar accept to rebuild results.
- To verify the design run a Design Rule Check, found on the «Report» tab.
- RO – Route offset can be accepted
- AA – placement to placement can be accepted if you are sure there is no conflict.
- All other errors must be sorted out before moving on.

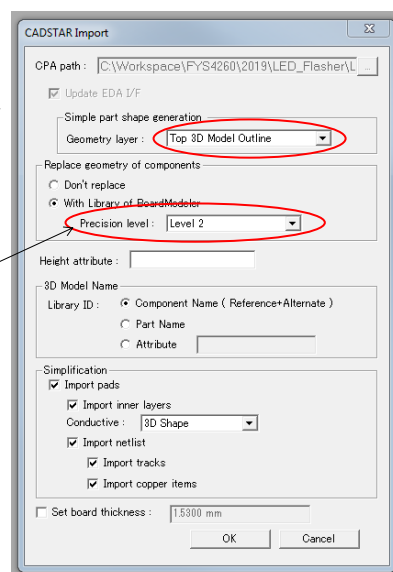
### 3D model – Setup Boardmodeller

- Before we can import the design to the 3D model tool, we need to set up the model libraries.
- Open Boardmodeller from Windows Start menu
- Make sure no project is open, then open Tools->Options
- Under Library Root Folder point to [\\Felt\\Elab\\FYS4260\\3D](#)
- Close Boardmodeller



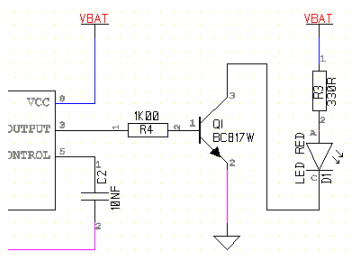
### Generate 3D model

- In Cadstar, open Boardmodeller from the Tools menu.
- Answer Yes to the next three(?) pop-ups
- Replace geometry with library precision level 2.
- To make the model more readable, only show the top and bot elec layers, and the board figure and hole layers. (using checkboxes on the left)
- Take a screen shot/Export to pdf



## Update schematics design

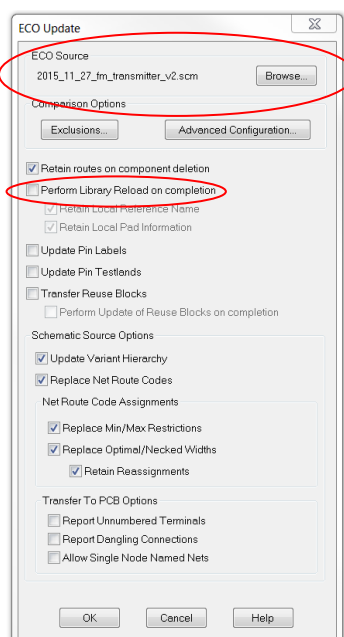
- Go back to the schematics and modify the circuit to use a transistor to drive the LED.



- Save the new schematics, and go to the pcb design.

## Update pcb with new schematics

- On Design tab -> ECO Update
- Updates changes in schematics to pcb.
- Make sure you have selected the right schematics!
- Many of the same settings as for transfer to pcb
- Use settings shown for a simple update
- Place the new parts, update routing, run DRC and export to Boardmodeller.



## Comments, Questions, Notes?

- ??

## Hotkeys

All general Windows keys work (CTRL-X, CTRL-Z, etc)

### CadSTAR

- View All ALT-F8
- Move F2
- Rotate F3
- Redraw F8
- Reconnect F11
- Place Space
- Find Component:
  - F «Comp name» + ENTER
- Change Grid:
  - G «x.x» + ENTER
- Zoom In/Out F9/F10

### PREditor

- Step selection TAB
- Change active layer F5/F6
- Change single active layer CTRL+J/K
- Optimal/Necked/Typed route O/N/T
- Change typed route C
- Change layer with via L
- 0 degree routing 0
- 45 degree routing 4
- Active 45 degree 5

See Help file for mouse gestures, can be very usefull!