

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

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

## Handout 2

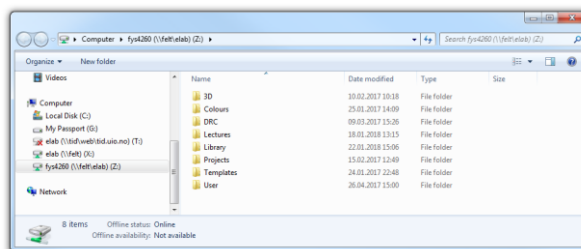
### CadSTAR Schematics



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

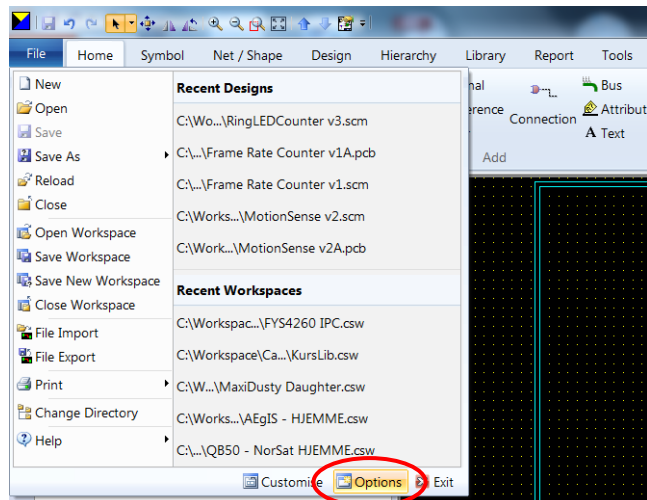
## Set up CadStar

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



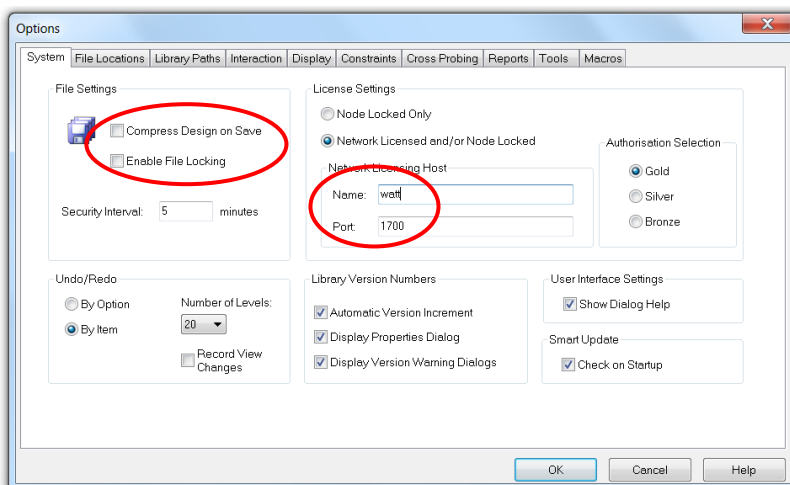
- Start CadStar
  - All Programs -> CadStar 18 -> Design Editor

## Select File menu -> Options



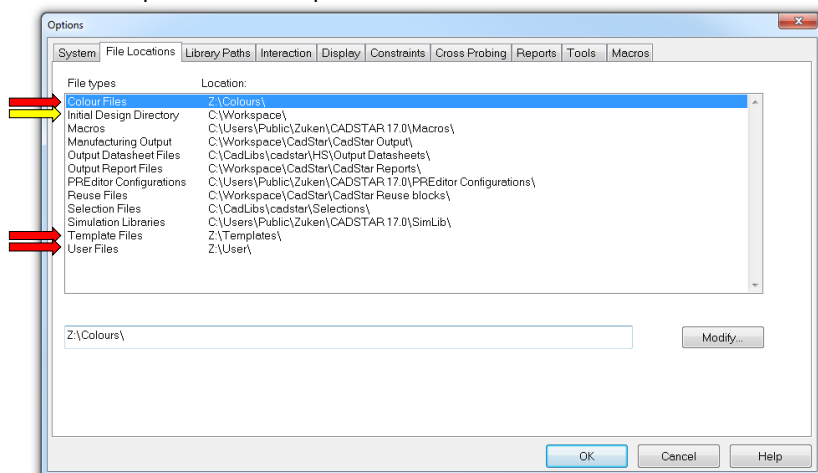
## System tab

- Network license: Name: watt Port: 1700



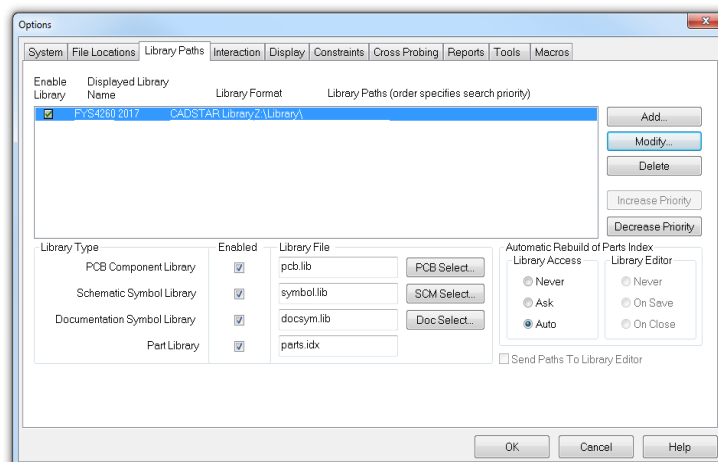
## File Locations tab

- 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.



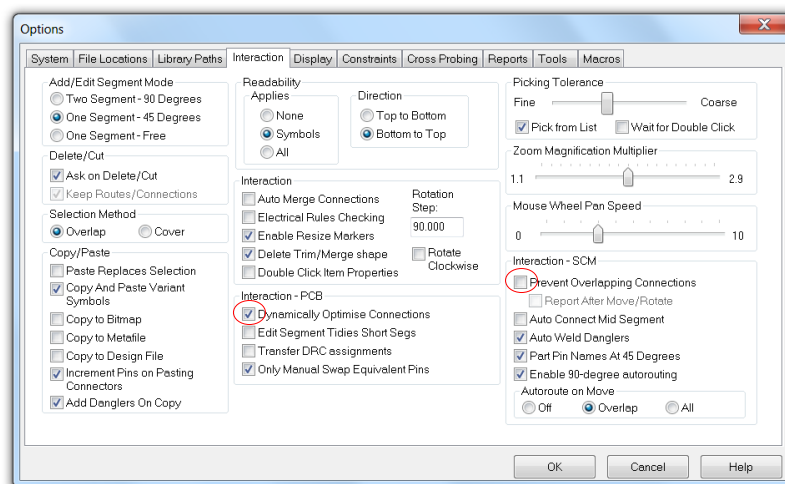
## Libraries tab

- 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!

## Start new design

File -> New -> Schematic Design  
Select FYS4260 template

-> Add project name -> OK

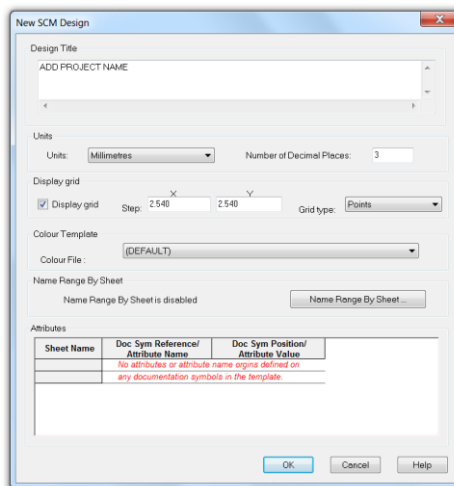
-> Save design,  
Use filename format:

**username\_projectname\_version.sch**  
eg halvorst\_ISM\_v1.sch

- Always use a version number for the schematics, we will use letters for the pcb versioning.

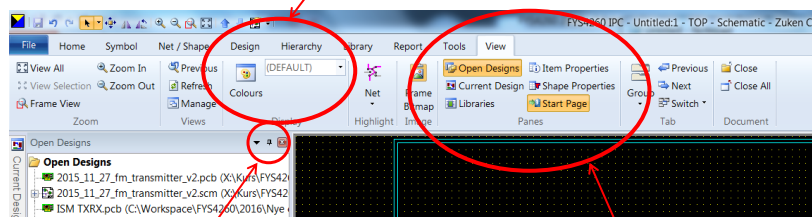
- No version control in CadSTAR, use save as.

- Use [kurs-fys4260@fys.uio.no](mailto:kurs-fys4260@fys.uio.no) for all deliveries.



## Views

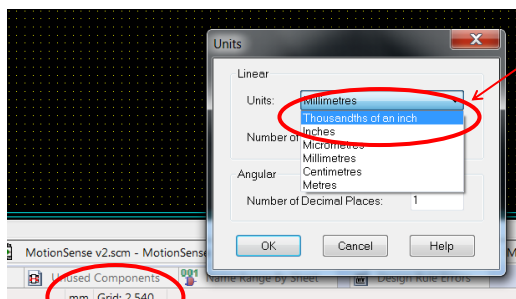
Color Settings



Activate Views

Use pin symbol to select visible or auto hide

## Grid



Thousandths of an inch  
= mils

1 mil =  $25.4/1000$   
= 0.0254mm

2 mil ~ 0.05mm

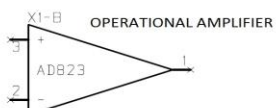
4 mil ~ 0.1mm

100 mil = 2.54mm

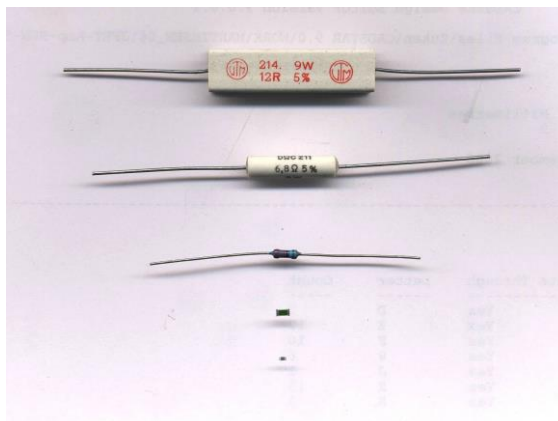
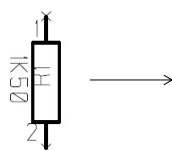
Left / Right click to change grid settings  
ALWAYS use 2.54mm (100mil) in schematics

## Symbols

- Symbols are graphical representations, pictograms, of physical parts.

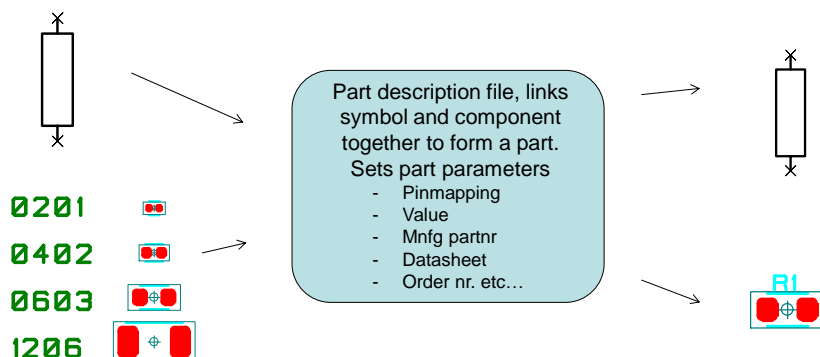


## One symbol can be different parts



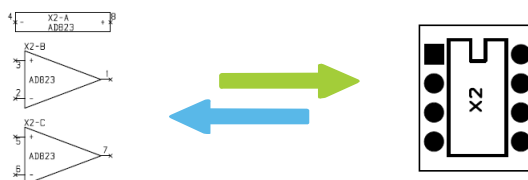
## PCB Library

- We need a library that maps the symbols to the real functional chip.
- A PCB library is a collection of parts, where each part has its own graphical symbol and a "footprint" that the part is going to be soldered to.



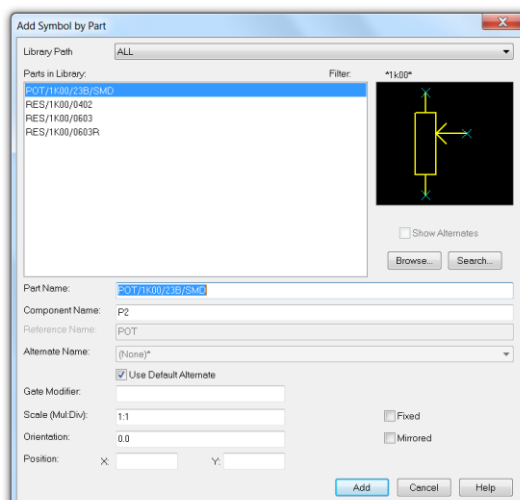
## Multiple symbols, One part

- Components with multiple functions can be divided into separate symbols, called gates.
- Ground all *inputs* on unconnected gates!



### Adding parts Method One

- Home meny  
-> Part or Symbol
- Search, using \*xxx\*
- Left click to place, esc to abort
- Be carefull not to add just an empty symbol, but a full part.



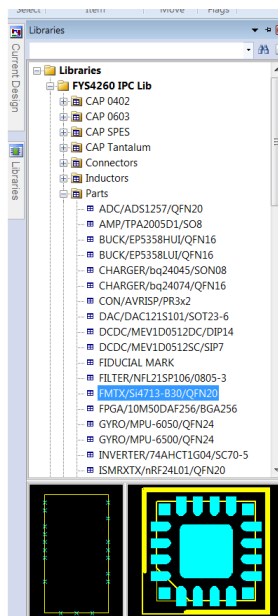


## Adding parts Method Two

- Libraries View  
-> Drag and drop
- Search, using \*xxxx\*

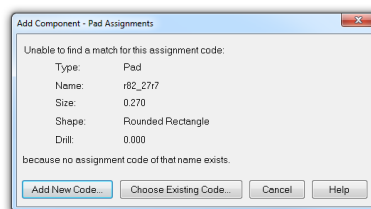
## Part naming conventions

- Specific parts:  
Function/Partname/Case
- Generic parts  
Function/Value/Case



## Unable to find a match for...

- If you get a warning saying «unable to find a match for...» this is not an error! It is to warn you that the part you are adding to your design have one or more setting/property which are not found in the current design. You can
  - Add the new setting/property to the desing or
  - Override the setting used in the part and choose a property already used in the design.



As long as you are using the correct libraries it is always safe to add the new setting!

## Hierarchical Design

Create a new block

- Net/Shape menu -> New Block
- Select and draw rectangle

Give the block a name

- Select the new block
- Right click -> Item Properties

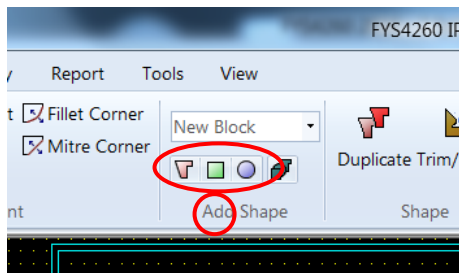
Increase the font

- Select the block name -> Item Properties
- Change Text Code

Create the new level

- Double click block name -> Create New Sheet

Create the design as a top-down design, with a top schematic describing the overall function, and with details in the lower sheets. Look at the schematics provided in the course projects.



## Add connections

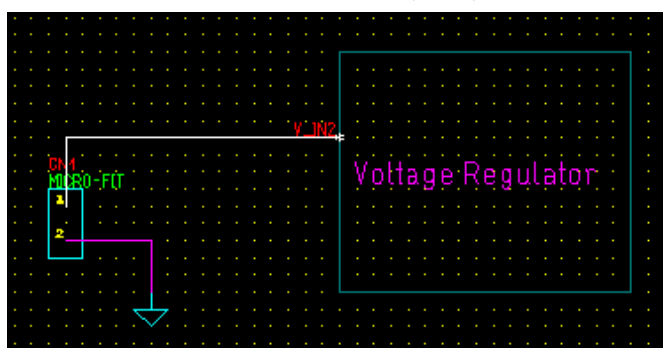
- Home -> Connection -> left click - left click
- All nets are identified by their net name.
  - Nets are named automatically as \$xxx.
  - Can be changed by selecting the net -> right click -> item properties -> Signal name
- Two nets with identical names are merged, do not need to be physically connected with wire!
- Be aware of named nets in the project schematics!
- Use this to make it easier to identify nets when we move to pcb later -> name all important nets!
- Makes the schematics more readable.

## Global signals and Net Route Code

- A global signal is a predefined signal used for important nets.
  - To add: draw connection, right click -> global signal
  - Defined in the library
- A Net Route Code is used to assign the optimal and necked route width on a net.
  - Sets max/min and optimal track widths.
  - select net -> right click -> item properties -> Net -> Net Route Code
  - Must be used for **GND** and **Power** nets!
  - Remember other nets that *might* conduct higher currents as well, eg traces to power regulators etc.
- Multiple nets may use the same net route code!

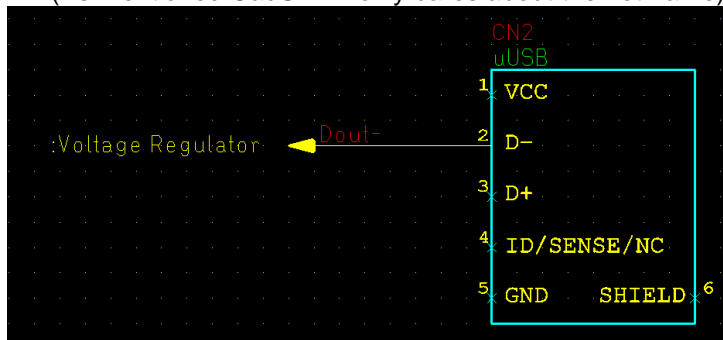
## Block terminals

- Draw connection -> Right click -> Block Terminal
- Or draw connection to edge of existing block
- Cannot create a block terminal on an unnamed net  
(You will be asked to fill in net name if you try)



## Signal References

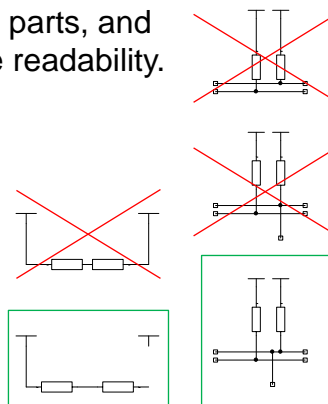
- Draw connections -> Right click -> Signal ref
- Select library -> smallin/out/bi
- Give net name
- Block terminals and signal references are just two methods to organize the design and increase readability.  
(As mentioned CadSTAR only cares about the net name)



## Junction or overlapping connection

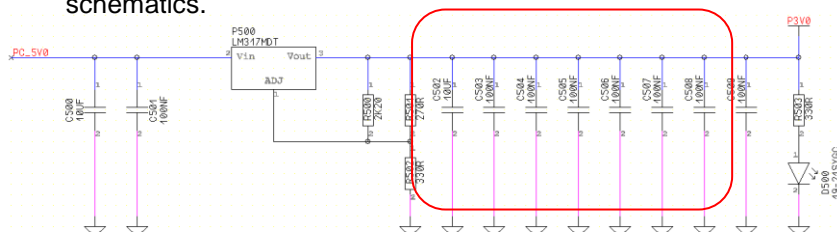
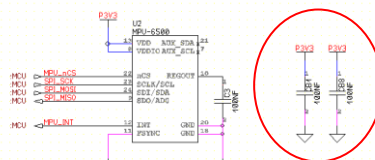
To prevent unwanted or missing connections, use spacing in your schematics. Space parts, and add short wire between to increase readability.

- Do not connect symbol to symbol without wire, or symbol directly on a crossing wire.
- Do not connect more than 3 traces together in one junction point. At least be very careful!
- In both cases it is difficult to visually see if a connection is correct, or if the symbol is just overlapping the wire or another symbol.



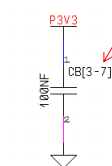
## Decoupling / Bypass Capacitors

- Every active power pin shall be decoupled with a 100nF 0603 or 0402 bypass capacitor.
- Use CAP/BYPASS/0402 (0603) part in library.
- More on this later, for now just remember to include them in the schematics.



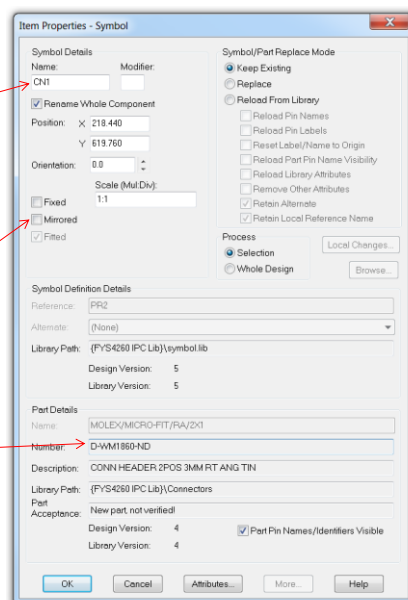
## Tips n' Tricks...

Use brackets in the symbol name to add multiple instances of the same part.  
(Usefull for bypass capacitors)

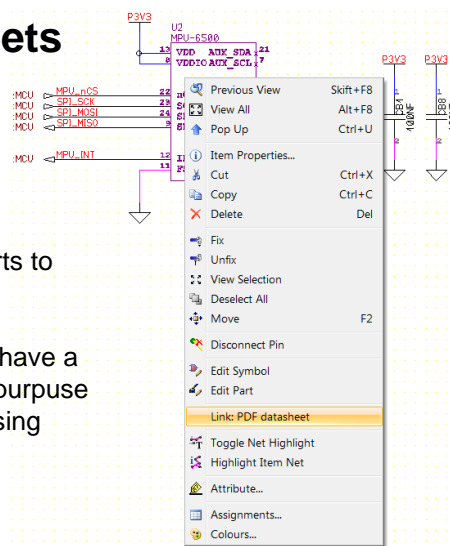


Mirror part  
F3 to rotate

Supplier part nr  
D – Digikey  
F – Farnell  
E – Elfa  
Look the part up if you  
dont know what it is!



## Link to Datasheets



- Select and right click parts to get a link to the part's datasheet.
- All specific parts should have a valid link, some special purpose or generic parts are missing datasheets

## Mechanical

Add Mechanical parts to the schematic, not directly in the pcb.

**The schematic is always the master!**

- Fiducial Marks
- Mechanical Holes
- Testpoints

