



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

FYS4260 – Spring 2018
Microsystems and electronic packaging and interconnection technologies

Workshop 2

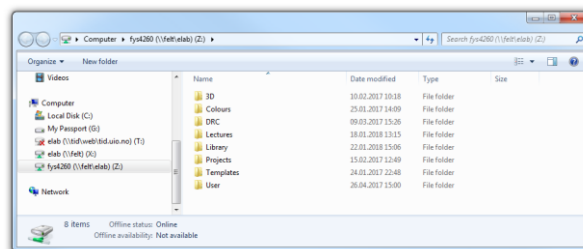
CadSTAR Schematics



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

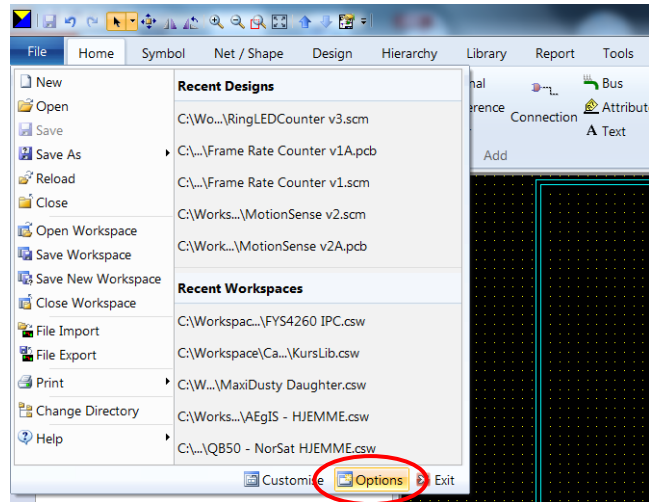
Set up CadStar

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



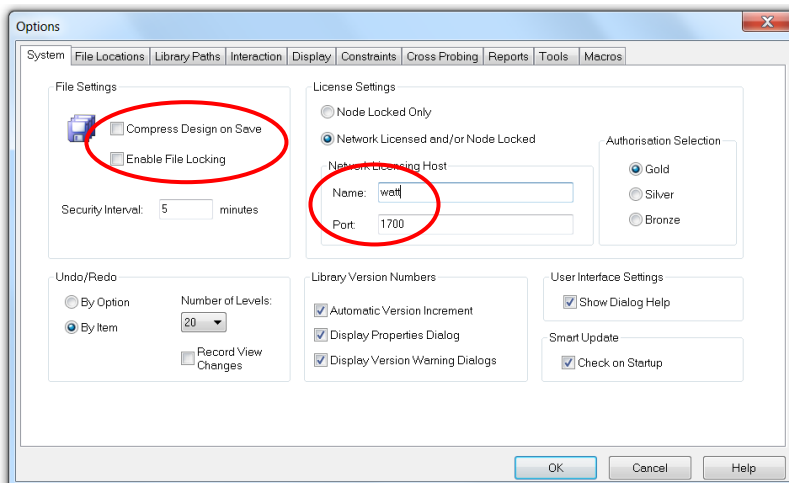
- Start CadStar
 - All Programs -> CadStar 17 -> Design Editor

Select File menu -> Options



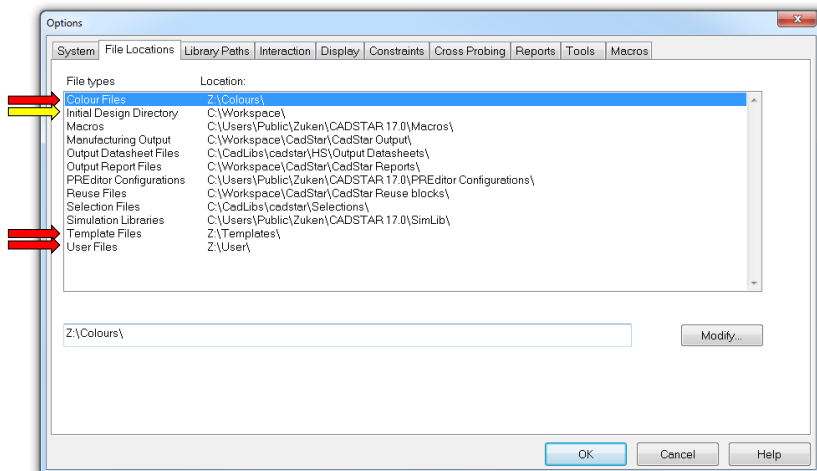
System tab

- Network license: Name: watt Port: 1700



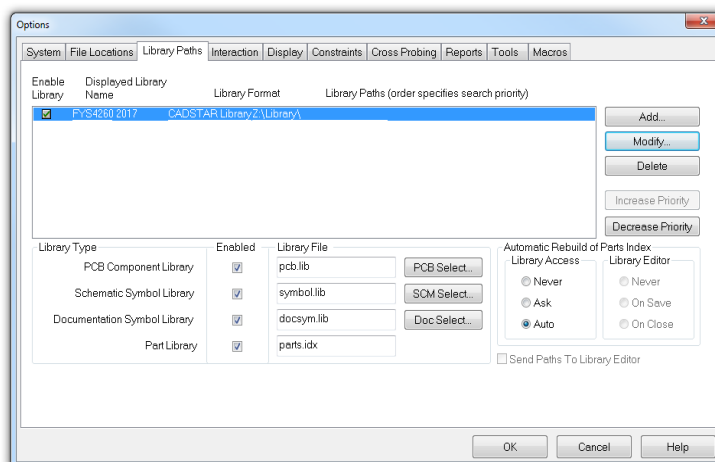
File Locations tab

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



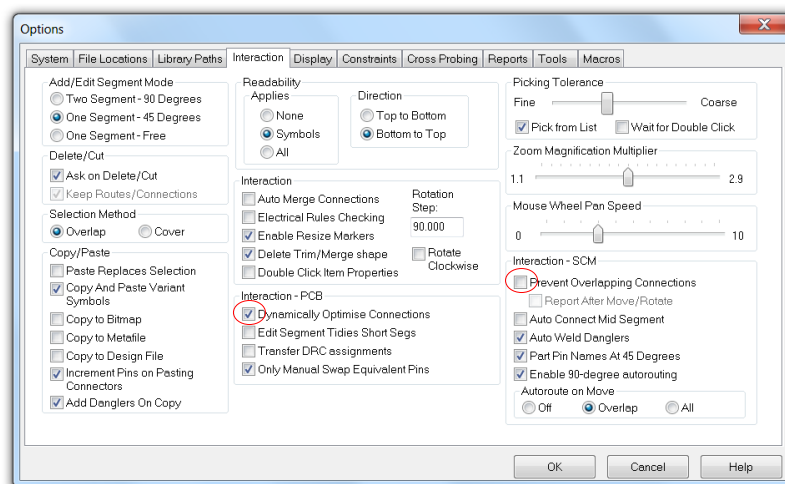
Libraries tab

- 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

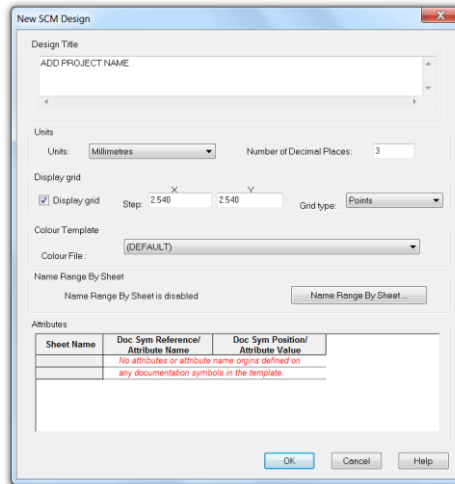
-> Save design,
Use filename format:

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.

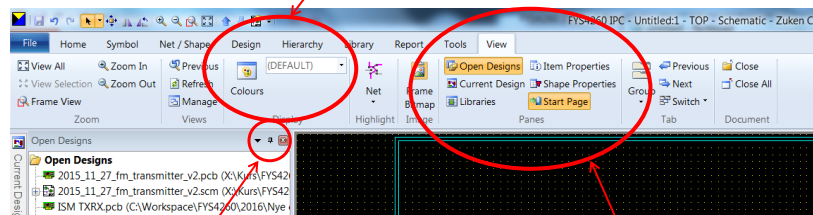
- No version control in CadSTAR, use save as.

- Use kurs-fys4260@fys.uio.no for all deliveries.



Views

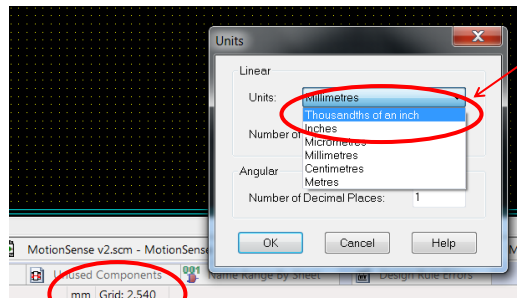
Color Settings



Activate Views

Use pin symbol to select visible or auto hide

Grid

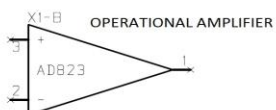


Thousandths of an inch
= mils
1 mil = 25.4/1000
= 0.0254mm
2 mil ~ 0.05mm
4 mil ~ 0.1mm
100 mil = 2.54mm

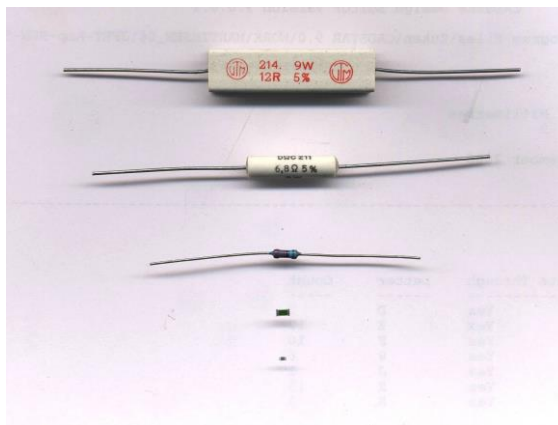
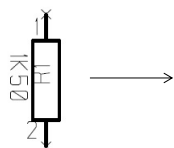
Left / Right click to change grid settings
ALWAYS use 2.54mm (100mil) in schematics

Symbols

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

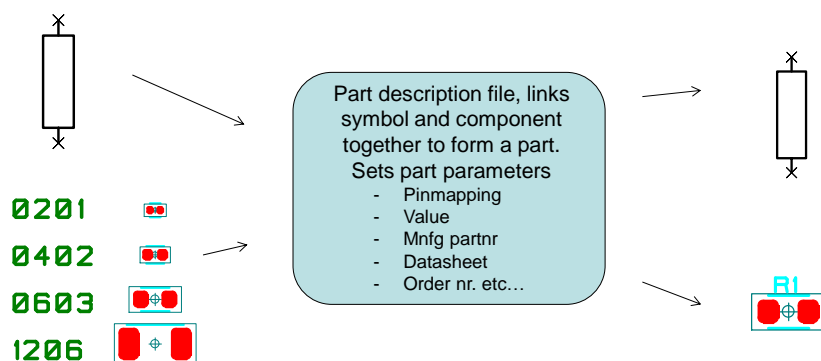


One symbol can be different parts



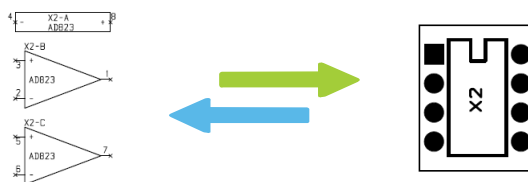
PCB Library

- 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.



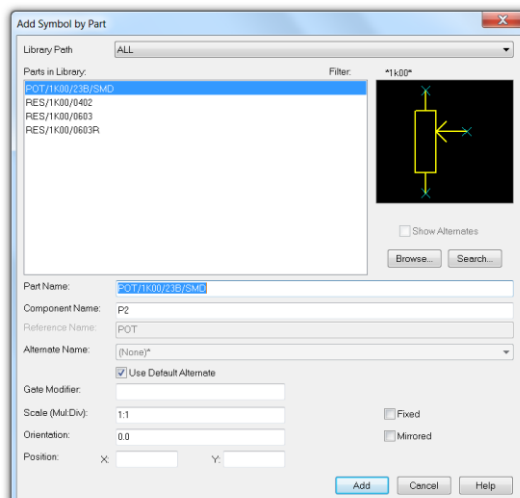
Multiple symbols, One part

- 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.

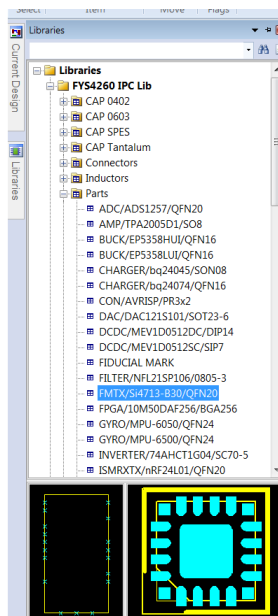


Adding parts Method Two

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

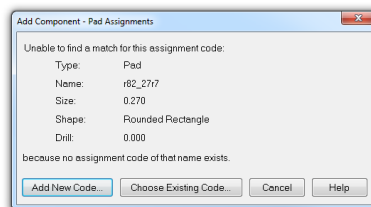
Part naming conventions

- 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

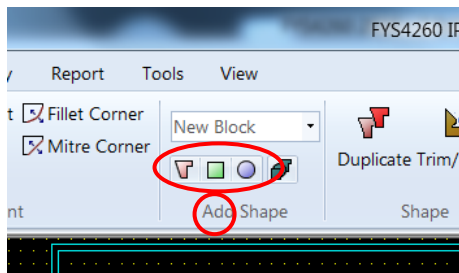
Increase the font

- 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

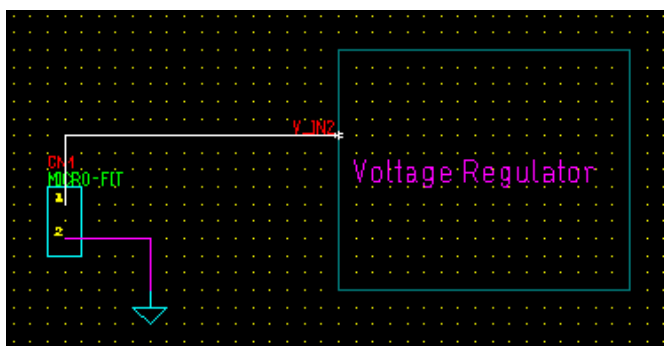
- Home -> Connection -> left click - left click
- All nets are identified by their net name.
 - Nets are named automatically as \$xxx.
 - Can be changed by selecting the net -> right click -> item properties -> Signal name
- Two nets with identical names are merged, do not need to be physically connected 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!

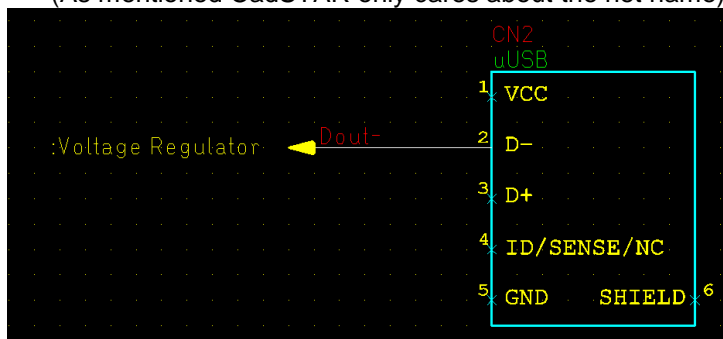
Block terminals

- 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

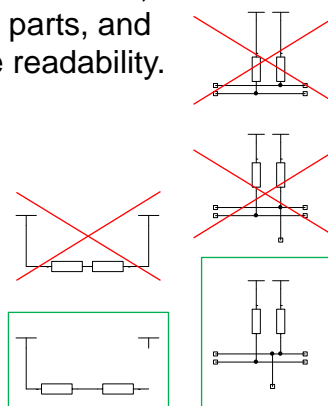
- Draw connections -> Right click -> Signal ref
- Select library -> smallin/out/bi
- Give net name
- Block terminals and signal references are just two methods to organize the design and increase readability. (As mentioned CadSTAR only cares about the net name)



Junction or overlapping connection

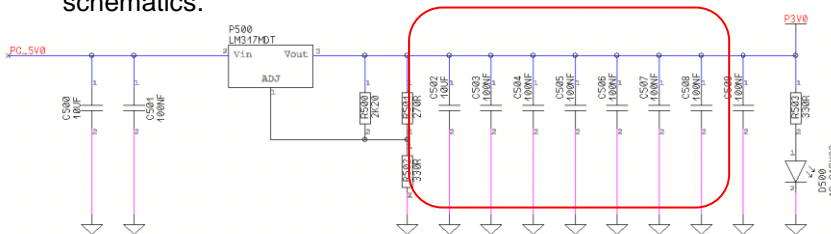
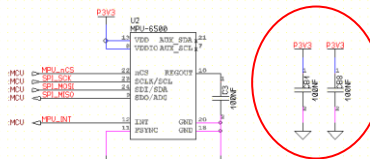
To prevent unwanted or missing connections, use spacing in your schematics. Space parts, and add short wire between to increase readability.

- 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.



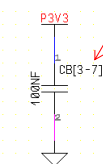
Decoupling / Bypass Capacitors

- 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.



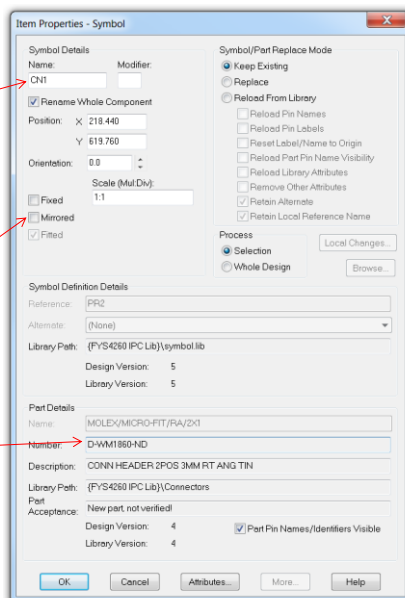
Tips n' Tricks...

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

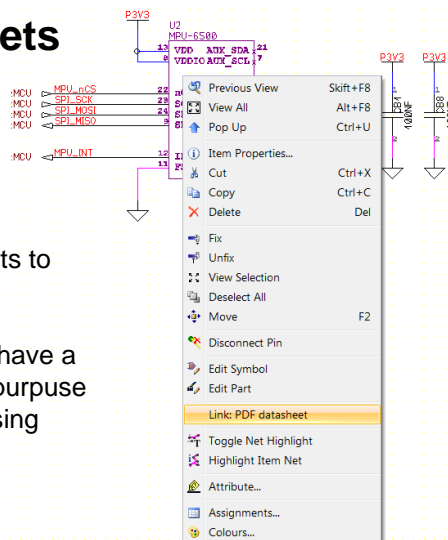


Mirror part
F3 to rotate

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

