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

FYS4260 2018

Microsystems, electronic packaging and interconnection technologies

Lecture 3 – Transfer to PCB, getting started with layout

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

First – File naming convensions

- All files shall be named as
  - uio username_Project name_version
  - Where version is
    - a number for the schematics, ex v1, and
    - for the pcb it is the corresponding number AND a charachter, ex v1A
  - Ex for me doing a FM Transmitter:
    - halvorst _FMTX _v1.sch AND halvorst_FMTX_v1A.pcb
    - If I then do updates to my pcb without changing the schematics next pcb version will be v1B. If I do updates to the schematics next version will be v2A.
- When delivering files always include the schematics as well. That is, if you deliver a pcb version 5x, send in the corresponding schematics version 5 as well.
- Do not zip files!

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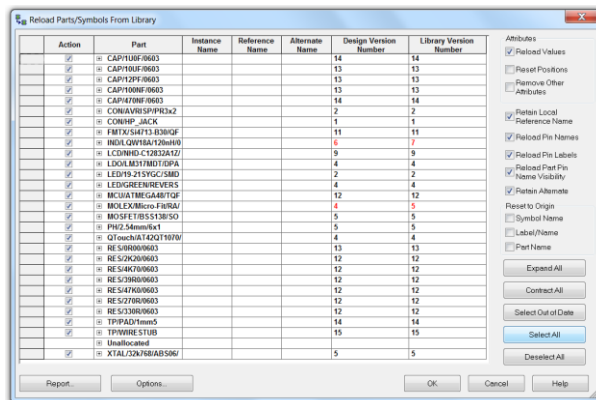University of Oslo

In this lecture - Agenda

- First, reload all parts to catch library updates.
  - See next slide
- Transfer design to pcb.
  - On Design tab -> choose «transfer to pcb»
  - Layers in a PCB design
- Mechanical layout
  - Do mechanical design and place components in CadSTAR
  - Decoupling capacitors
- Start routing the design using PREditor XR
  - On Tools tab -> PREditor XR
  - Very short on routing, more next week.

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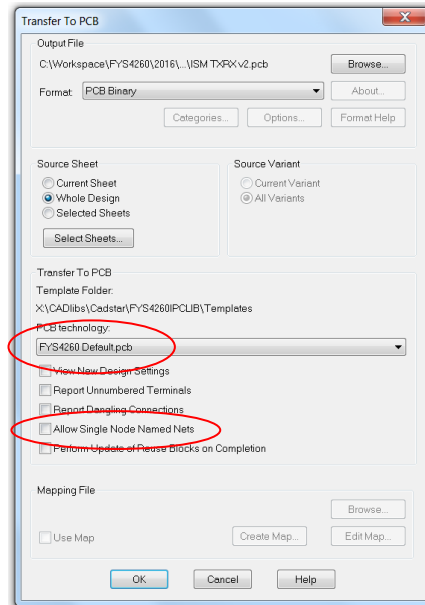University of Oslo

Reload Parts

- On Symbols tab, select Reload
- Select All -> OK, then Accept.

This will reload all parts from library, catching all updates to the parts since they were added to the design.

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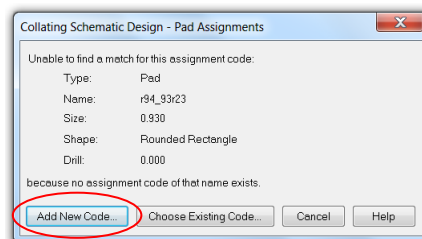University of Oslo

Transfer to PCB

- We need to move our design over to the layout editor where we will be doing the pcb layout design.
    - On Design tab -> choose «transfer to pcb»
- The window on the left opens.
    - Make sure «whole design» is selected
    - Use «FYS4260 4layer.pcb» as PCB template. (If this is not an option in the drop down list you have not set up file locations correct, see schematics foils.)
    - «Allow Single Node Named Nets» will force CadSTAR to not throw an error if you have named nets with only one single node. Can be handy for «unfinished» designs. Only check if you understand what it does!

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

Transfer to PCB cont.

- You will get a warning saying «unable to find a match for this assignment code». (Some of you have seen this kind of message before…)
- Make sure the type of this is PAD, then select Add New Code and answer YES to the next question to accept all other PAD assignment as well.
- If the type is not PAD then there are something missing in the libararies or templates. Choose Add new code but pay attention to what you are adding, and please let us know…!

Update!
If you get a warning about different design rules, select «use library rules»

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

Transfer finished – Master view

- The PCB design will start up with a black screen showing the board outline in the upper part of the screen, and all components in a pile in the bottom left corner. (ALT-F8 to View all).

- This is CadSTARs «Master View», where we manage our design. Here we can
    - do our «mechanical» layout, that is adjust the board outline and place components.
    - Export the design to other tools, f.ex to spice simulations, 3D model constraints viewer, and also to PREditor XR where we will do our routing.
    - Set up the layer stack, and the properties of each layer.
    - Define our design rules and design elements.
    - Run design verifications, reports, and a lot more…

- Notice that nominal connections have a white colour, GND connections are pink and Power connections are blue.

- The layers and design rules are set up for you, do not make changes unless you understand what you are doing and we have approved it!

UiO **:** **Department of Physics**
University of Oslo
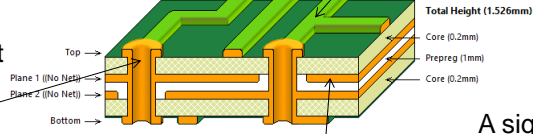
FYS4260 Project PCB «settings»

- If you used the correct template during transfer to pcb you will have all the correct settings.
    - 4 layer pcb, where the inner layers are used for GND and power, and the outer layers for signal routing.
        - TOP ELEC in RED
        - GND in PINK
        - Power in BLUE
        - BOT ELEC in GREEN
    - Board outline set to 5x7cm
    - Only through hole vias allowed
    - All correct design rules, track width, spacings, …
- All SMD components must be placed on top side
- Do not change anything on the assignments tab without checking with us first!

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

4 layer pcb basics

A signal moves
in a conductor

Conductor, trace,
track, route

Connections
between different
layers are done
through Vias

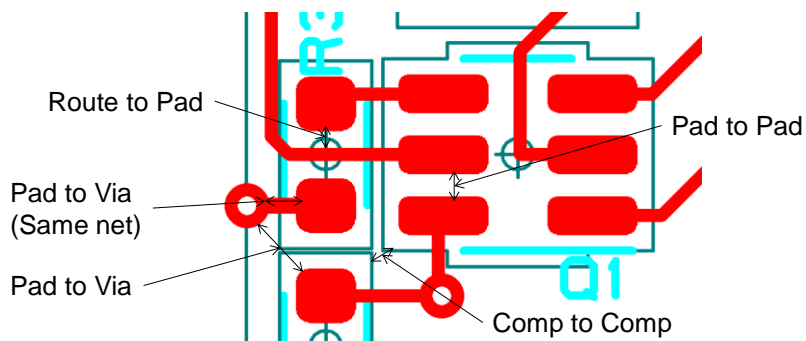Total Height (1.526mm)
Core (0.2mm)
Prepreg (1mm)
Core (0.2mm)

Top →
Plane 1 ((No Net)) →
Plane 2 ((No Net)) →
Bottom →

A signalpath is
the path a signal
takes in a plane
or conductor.

A layer filled with
copper are called
a Plane

1 mil =
1 thousandths of an inch =
25.4mm/1000 =
0.0254mm

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

What is a design rule?

Route to Pad

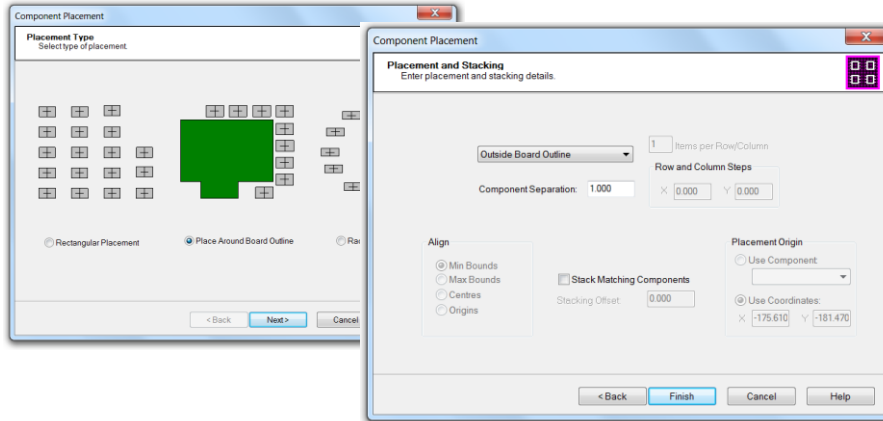Pad to Pad

Pad to Via
(Same net)

Pad to Via

Comp to Comp

Most routing rules in FYS4260 are in the order of 8-10 mils,
eg 0.2-0.25mm. You do not need to think about this when you
route, CadSTAR will tell if you violate any rules!

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

Mechanical layout

- Start with the mechanical constraints.
    - Define your board outline. (The starting point for your design is a 7x5cm rectangle, but you may change this slightly if you need to.)
    - Do you plan to mount the board in a box, placement of mounting holes etc…
    - Connector placement, think from a user perspective.
    - Other components with mechanical or user constraints (ex LEDs, jumpers, switches).
- Arrange components on the Component tab can be usefull in the beginning for spreading the components out. See next slide.

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

- Continue with component placement
    - Select components that belongs togheter in the schematics, cross-probing will highlight them as selected in the pcb as well. I usually start with grouping all components from one sheet in the schematics togheter, and do a rough placement based on this to get started. (CTRL-A to select all components on a sheet, activate the pcb, then move the group with F2).
- Time spent on component placement are always saved on simplified routing!

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

Arrange components

Component tab -> Arrange components. Experiment your way, this is only a suggestion to arrange componets around the board outline.

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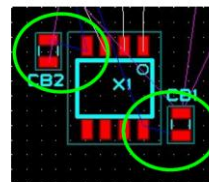University of Oslo

Grouping components

- Grouping components can be usefull when working with placement.
- A group of components will be rotated as a group, while a selection will rotate all components individually.
- To remove a group use the ungroup button.

- The center of a group is defined when you pick the group. Position your mouse on the component you want as your origin when moving a group, or make sure to select the comp you want as your origo last.

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

Decoupling / Bypass Capacitor

- As mentioned earlier, all active components shall have a decoupling capacitor on ALL its power pins.
- Power supplies are slow and can use several ms to deliver an increased current. In this time the voltage from the power supply will drop.
- Parasitic inductance on the power rail further limits the instantaneous current we can draw from a power source.
- Adding a small capacitor close to the IC will work as a battery holding up the voltage till the power supply can catch up and the current can suppress the inductance on the power rail.
- Rule of thumb is to use 100nF caps, but their exact value and number have to be analyzed in a critical design.
- Select small packages (you have 0402 and 0603 size in library)
  – They have less inductance and
  – Are easier to place close to chip terminals.

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

Decoupling capacitor placement

- Important to place bypass caps as close to the power/ground pins as possible to minimize impedance and current loops.
- If the ground and power pins are separated, one have to chose which pin to decouple best.
- Placing the capacitor close to the ground pin will help the board maintain a stable ground, thus reducing common mode noise.
- But this is at the cost of the power rail!
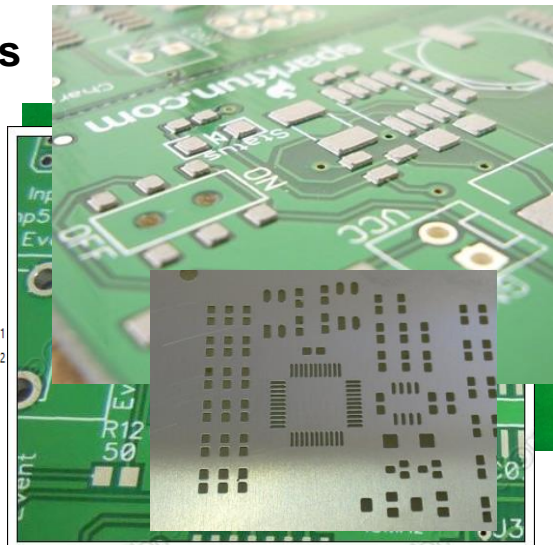- We will prioritize power pins!

- Larger bulk capacitors will also improve power distribution, but these have to be placed further away due to their physical size.

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

Layers in CadSTAR

- A PCB design is a «sandwich» of multiple layers.
  - The basic layers are the electrical layers we do our routing on, and the mechanical layers holding the board together.
  - But a pcb is also made up of protective, notation and production layers.
  - In addition we can have many documentation layers.
  - Some layers are sent to the pcb manufacturer, others to the assembly shop, and some are used only as help layers during design.

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

Physical layers

- Solder Paste
- Silk Screen
- Soldermask
- Top Elec
- Core
- Inner plane (GND)
- Prepreg
- Inner plane (POWER)
- Core
- Bot Elec
- Soldermask
- Silk Screen

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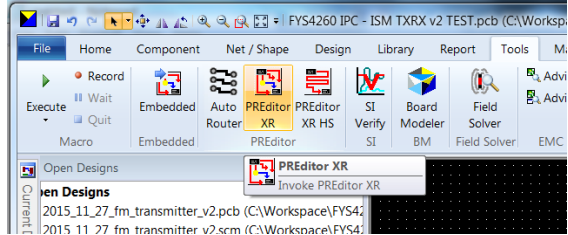University of Oslo

Other Layers

- Assembly
  - Simple outline with orientation, used by assembly shops or for manual mounting.
- Place
  - Oversized component outline, gives the placement area. Mechanical keepout areas can be defined here.
- Mech
  - Documentation layer for mechanical components.
- 3D Model Outline
  - Component outline used to create simple 3D shapes.
- Core/PrePreg
  - Construction layers.
- Doc 1-5
  - Misc documentation layers.

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

View layers in CadSTAR

- In CadSTAR, the board are viewed from the top side, looking through the board.
- Using the colour menu, one can select which layers are visible.
- You can also set different colours for each item on each net!
- And you can specify which items should be pickable.
- The dropdown menu shows you different predefined colour settings.
- Some views shows you a single layer, others are set up to simplify specific operations in CadSTAR. Some examples are:
  - PCB 4 layers   Standard view, shows all routing layers ++
  - PCB Normal   Shows all signal layers, that is no GND or Power.
  - Silkscreen Cleanup   View for cleaning up the silkscreen.
  - All   Shows all layers, used for finding view errors.
  - In addition all layers are set up as single views for inspection purposes.

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

Very short on Routing (We will continue this next week)
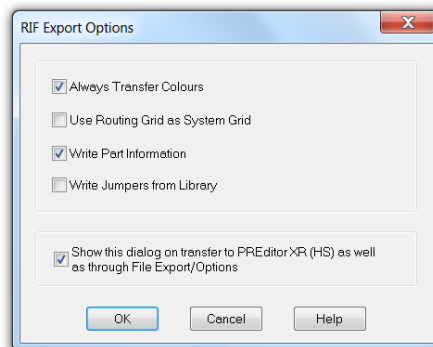
- It is possible to do the routing in the embedded router in CadSTAR, but we are going to use PREditor XR.
- PREditor is a much more powerfull tool, but has a slightly different user interface. You will get used to it…
- When you have placed most of your components and feel you have a starting point for your layout start PREditor from the Tools tab.

- You will go back and forth between CadSTAR and PREditor as you switch between working on routing and more «mechanical» layout and design changes.

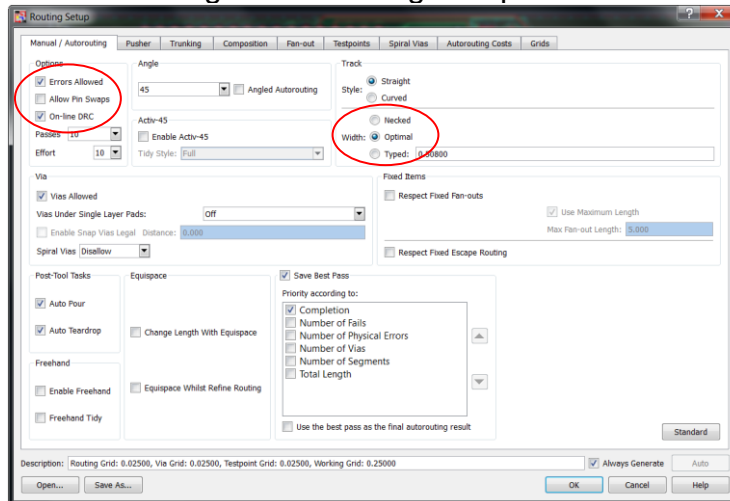UiO **: Department of Physics**
University of Oslo

PREditor - First use

- Use these settings when opening PREditor the first time. Make sure all layers are visible.

- 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.

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

Setup PREditor – First use

Select Configure -> Routing Setup

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

Recomended settings

- Manual / Autorouter tab
  - Errors allowed will allow you to make illegal routes
  - On Line DRC will mark illigal 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.
  - Recomend to enable springback if you use pusher.
- Grid tab
  - Change 0.0254mm to 0.025mm.

UiO **: Department of Physics**
University of Oslo
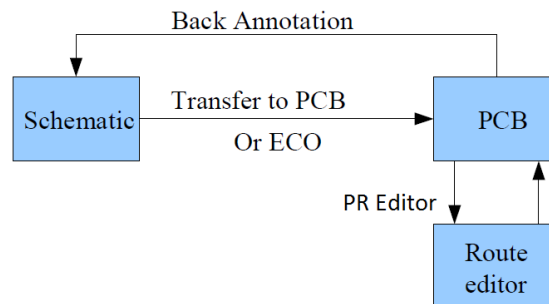
Tips on routing in PREditor

- Use Manual route tool to route signals.
- Start a route by clicking on a connection (not holdinge 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.
- Dobbleclick 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

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

- Route GND and Power on layers 2 and 3 as a start, signals are on layer 1 and 4.
- There are no rules as to how many vias one can use, but try to make the designs as neat as possible.
- Hold the pointer over an area to show all possible selections, TAB to loop through (shown in statusbar).
- Save and exit PREditor, changes are exported to your CadSTAR design.

- Play around to get to know the different functions.
- Try to keep the routing as tidy and neat as possible. This will not only lead to a nice looking design, but it will have better signal integrity, it is easier to debug and locate «bad» areas.
- More on routing next week!

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

PCB Layout workflow - Recap

- Schematics are transfered to PCB, to update an existing pcb from a schematic use ECO update. (see next slide)
- CadSTAR is our «main window», where we manage our design and do most of our placement.
- Routing are done in PREditor XR.

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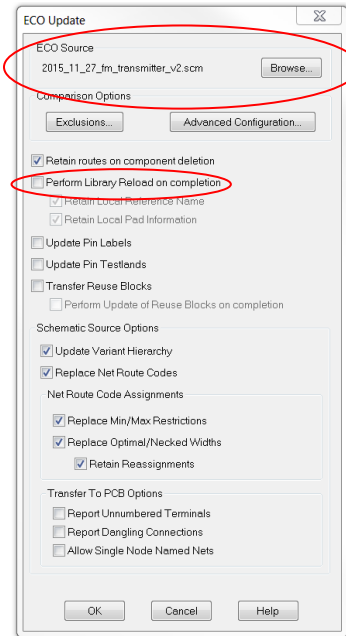University of Oslo

More on workflow

- The schematics is always the master!
- That is, if you want to do changes to your circuit do them in the schematics and use ECO to update those changes to the pcb.
- This is also the reason we have symbols in our schematics for pure mechanical parts such as mounting holes, testpoints and fiducial marks.
- If you for some reason must do changes within your pcb-file, use Back Annotation to export those changes to the schematics right away.

UiO **Department of Physics**
University of Oslo

Updating the design from 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
- Possible to perform library reload, same as the reload we did earlier.
- Use settings shown for a simple update
- Check for reload library if you haven't done that in a while.

UiO **Department of Physics**
University of Oslo

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!