



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

## **FYS4260 2019**

Microsystems, electronic packaging and  
interconnection technologies

Lecture 4 – Routing and Copper planes



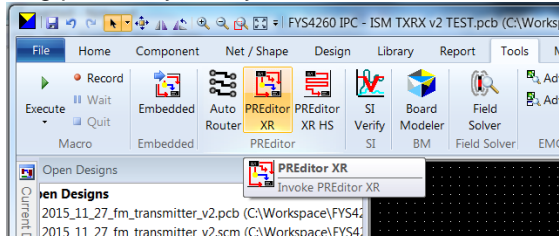
UiO : **Department of Physics**  
University of Oslo

## **Agenda**

- PR Editor setup and basic routing
  - Recap from last week
- Powerplanes and Templates
  - How to set up the GND and Power layers.
- FYS4260 Do's and Don'ts
  - Routing rules and best practice.

## Routing

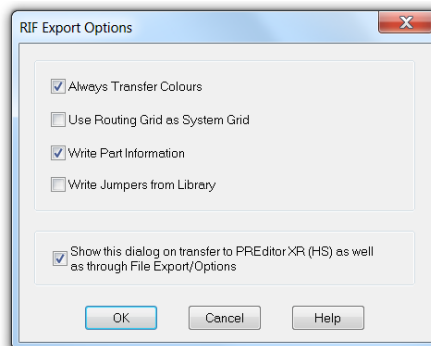
- It is possible to do the routing in the embedded router in CadSTAR, but we are going to use PREditor XR.
- PREditor is a much more powerful tool, but has a slightly different user interface. You will get used to it...
- When you have placed most of your components and feel you have a starting point 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.

## PREditor - First use

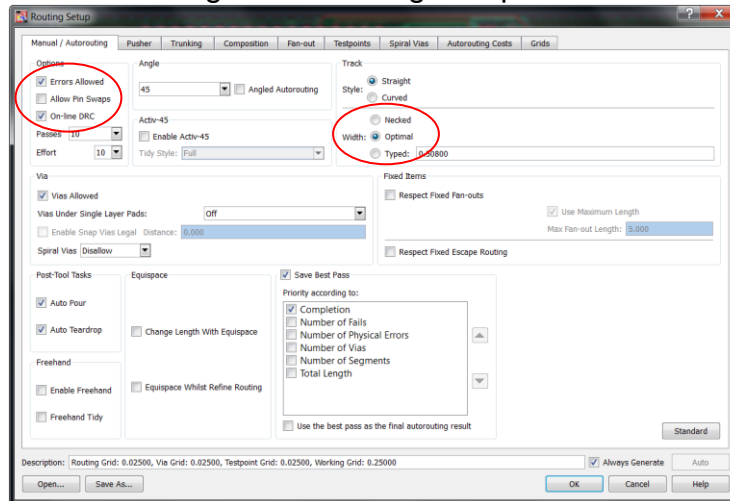
Use these settings  
when opening  
PREditor the first  
time.



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 PReEditor – First use

Select Configure -> Routing Setup



## Recomended 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.
  - Recomend to enable springback if you use pusher.
- Grid tab
  - Change 0.0254mm to 0.025mm all over.

## Short on routing in PReDitor

- 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.
- Use optimal track width as much as possible.
- 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

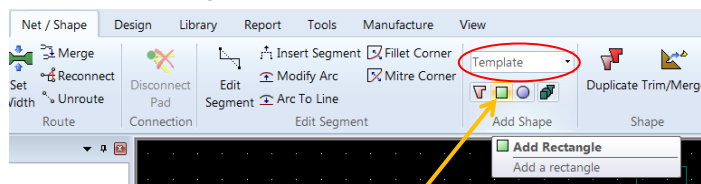
- Signals are routed on layers 1 and 4, while GND and Power use layers 2 and 3 respectively.
- There are no rules as to how many vias one can use, but keep in mind that a via will degrade signal integrity. Think about which signals you need to prioritize.
- Hold the pointer over an area to show all possible selections, TAB to loop through (shown in statusbar).
- Save and exit PReDitor, changes are exported back 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.

## Power and GND Layers

- As mentioned, we are going to use dedicated layers for GND and Power; layer 2 for GND and layer 3 for power.
- The ground layer should be one full copper layer, the power layer may be split into different areas if needed. For instance VPOS and VNEG for the amplifier project.
- This means that every GND connections will just be a via connecting down to the GND plane. Same for power, although the power plane might be split up.
- Some components may need special routing for GND and/or power on the top/bot layers, eg switchers, heatsinks etc.
- Copper layers are defined in CadSTAR using templates, and then filled in PREditor.

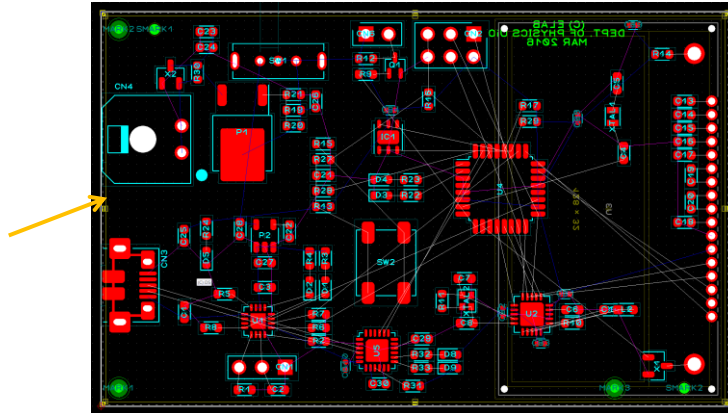
## Create template

- On the Net/Shape tab, make sure Template is selected in the drop down menu and select add rectangle.



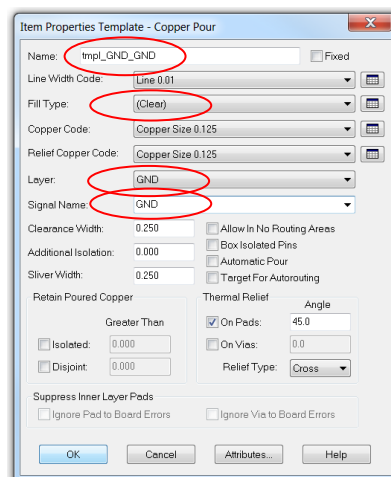
## Draw template

- Draw the template, pull the shape 0.5-1mm back from the board edge.



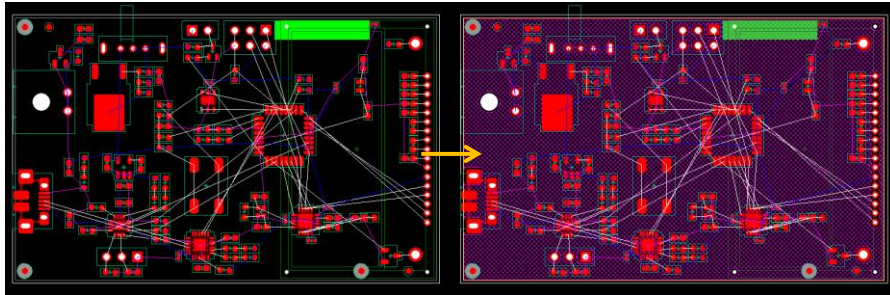
## Define the template

- Select the new template, right click -> item properties.
- Give it a name
- Select which layer to use.
- Assign signal name.
- Leave the rest as is.
- **DO NOT FILL THE TEMPLATE HERE!**



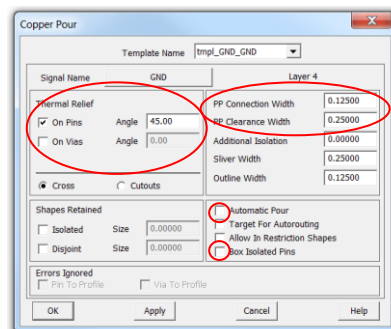
## Fill the shape in PReEditor.

- Move to PReEditor.
- Select template, right click -> Repour.
- You will get a report after the fill, read it and make sure you have no errors.



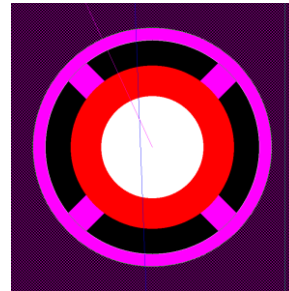
## Refine the template

- How we want the copper to fill the template is set up in template properties.
- Select the template, right click -> Template properties.
- See next slide for thermal relief, connection widths and clearance.
- Automatic Pour will repour the copper when needed (eg when you insert a via, move a through hole component, ...)
- Test the other options until you get a copper fill you are satisfied with.



## Thermal reliefs

- Why use thermal relief?
- In template properties,
  - PP Connection width is the width of the stubs connecting to the plane. For a power plane I recommend using 0.25mm. Times four this equals our 1mm track width for power routes.
  - PP Clearance is the spacing between the pin and the copper. Leave it at 0.25mm.
  - Pins not connected to the plane will use the copper spacing defined in the design rules.
- Angle can be selected 0 or 45.
- All options apply to all pins in the design, if you want to modify one pin this can be done using pin attributes.




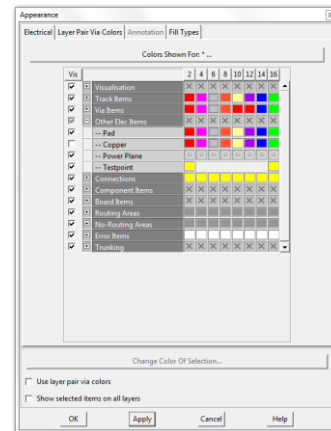
## Using copper templates

- Once filled, this copper can be used as a target for routing. That is, to route a track from the top layer to GND you can just insert a via and it will terminate on this copper.
- If you clear the copper, the route will not finish on the template, as there is no copper there. The shape has to be filled to behave as an electrical copper plane.
- Create the copper plane for the power signals the same way. If you need multiple power rails, add more templates and split the plane between them.



## Hiding copper layers

- Adding copper planes might make CadSTAR less responsive, hiding these layers will improve performance.
- To hide a copper plane, first make sure «Automatic repour» is selected for the template under «Template properties».
- Then select the Appearance button on the Display menu. 
- Uncheck Copper under «Other Elec Items».
- This will make all copper invisible, but it is still possible to route to these areas, and as long as automatic repour is on, cadstar will make sure to update the copper pour on any change.



## FYS4260 Routing «Best Practice»

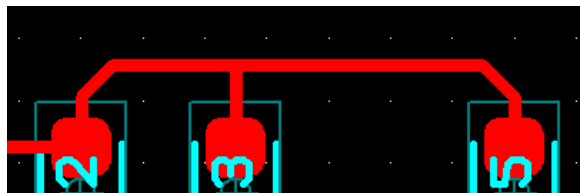
- The following slides shows some «best practice» routing tips for FYS4260.
- Some are specific rules, others are to be considered as guidelines for creating a board that is easy to manufacture, debug and modify.

## Track width

- There are two predefined widths for each net route code, Optimal and Necked.
- These are predefined for you in the design rules.
- In addition you can use a Typed width, that is a custom width you set with the «c» hotkey.
- Use item properties to set other track widths if needed.
- Switch between route widths with the 'o'(optimal), 'n'(necked), 't'(typed) and 'c'(change) hotkeys.
- Always use optimal to start with. Change to typed when you need to go down in width. You set the typed width with the «c» hotkey.
- Only use necked track width when absolutely needed, or when ending a track on a pad.
- Try to avoid using multiple widths on a single route segment. (CadSTAR does not handle this well)

## Mitre corners

- Smooth out the design by using 45 degree angles on all corners. Referred to as octilinear routing.



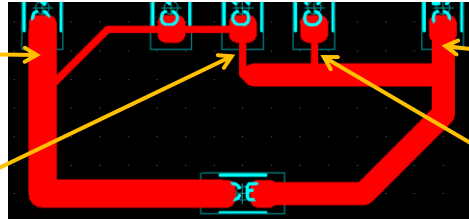
- Looks better and uses less space.
- Don't be fanatic, use 90 degree corners when you feel it makes more sense. As in the T junction in the picture. But you may use a Y junction if you think that looks better.

## Entering pads

- Do not enter or exit pads with a track wider than the pad!
- Use typed width to set a width that fits the pad size, or use necked width.

NO. Optimal track width twider than pad.

NO! Necked track continuing through pad. (See later slide)



Track width set to 0.8mm to fit pad size. OK

OK. Necked track when ending the track on a pad.

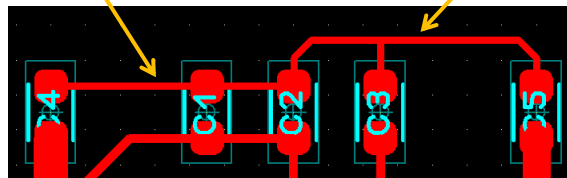
- Looks better
- Better pad-to-pad copper balancing (reduces chance of tombstoning).

## Connecting Pads

- Try to avoid running tracks through a pad

Track running through pad.

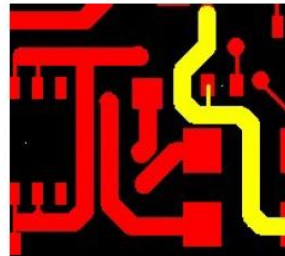
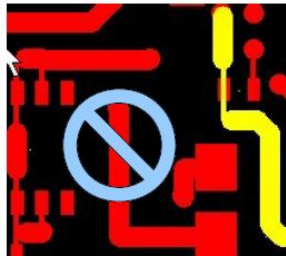
Track pulled away from pads, with stubs connecting each pad.



- Makes modifications easier.
- Creates pads with equal thermal mass (reduces tombstoning).

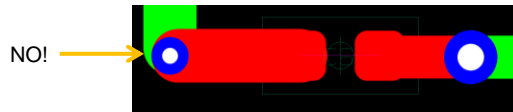
## Power Route Width

- Power nets should be defined in the schematics as having a power net route code (shown by blue and pink colour), indicating they need a fatter track width.
- Only use necked/typed width on these tracks when ending the route on a component pad, if needed.
- Continuing a necked power route will in effect create a one-time fuse!



## Vias

- You have two via codes defined for you
  - 1/0.5 1mm pad, 0.5mm drillhole
  - 0.7/0.3 0.7mm pad, 0.3mm drill (default)
- Always use a via that has an outer diameter the same as or larger than the route width used.
- That is, use 0.7/0.3 vias for tracks maximum 0.7mm wide, 1/0.5 vias for tracks up to 1mm.



- CadSTAR handles these situations bad.
- Insert a via by double clicking when you are routing.
- Change via by right click -> item properties.

## Acid Traps / Peelables

- Gaps smaller than the route-to-route spacing between tracks or pads on the same net will not by default throw an error (it is the same net!), but can create difficulties in the manufacturing etching process.
- Same for angles below 90 degrees.
- Avoid if possible.

