# KiCad Install and Use Notes

## Installation/upgrade issues

Download KiCad from
http://kicad-pcb.org/download/osx/
    stable release
    eg.: kicad-4.0.7.dmg
    and
    eg.: kicad-extras-4.0.7.dmg

Remove all existing KiCad programs,
    but NOT designs :-)
Install latest KiCad package
    eg.: kicad-4.0.7.dmg
IMPORTANT: Open and close KiCad
Install latest KiCad extras package
    eg.: kicad-extras-4.0.7.dmg
Say yes to "merge/flet"
(Arrange apps and icons in AppStarter, if you have it)

Open "eeschema"
    open some existing drawing
    see if it all works

Download Freerouter from
https://github.com/freerouting/freerouting
    freerouting-master.zip to desktop
unzip
From freerouting-master/binaries copy FreeRouting.jar to desktop
    rename to "freeroute.jar"
Copy freeroute.jar to
    /Applications/Kicad/kicad.app/Contents/MacOS/
    (remember to select view content in kicad.app)

You maybe have to install JDK to use the Java tool
This can be tricky, since sometimes you have to use an older version!
Don't use JRE !
From here
https://support.apple.com/kb/DL1572?locale=da_DK
Download the limited "**javaforosx.dmg"** and install
This will make autoroute start in "gui.MainApplication" later when you call from "pcbnew"

# Using KiCad  issues

**A new project**
Create a new folder in your own “KiCad_projects” directory
    eg.: “Test_KiCad”
Open “eeschema”
    Select “Create new project”
    IMPORTANT: First select the right folder “Test_KiCad”
    (little tricky because no  good browse option. Go one step up)
    Write “Save as” name: “test_kicad_1.sch
    Select “Save”
Close “eeschema” and select “Save and exit”
We now have the files

- test_kicad_1-cache.lib
- test_kicad_1.pro
- test_kicad_1.sch

Double-click on the file “test_kicad_1.sch”
“eeschema” opens and we are ready to create a new schematic
Do something
(component libraries should be present now)
Close and open again
Do something
Close
We now have the files

- test_kicad_1-cache.lib
- test_kicad_1.bak
- test_kicad_1.pro
- test_kicad_1.sch

## When drawing the diagram in “eeschema” is finished

Open “Annotate schematic components”
    Select “Use entire schematic”
    optional Select “Keep existing annotation”
Run Annotate

Run “Electrical Rules Checker”
    Check if markers are relevant
    Repair any faults
    Remove markers

Print a .pdf copy of the schematic (free name)
(remember to select same folder)

***remember to SAVE***

Now we have the files

- test_kicad_1-cache.lib
- test_kicad_1.bak
- test_kicad_1.pdf (the name is free, not auto generated)
- test_kicad_1.pro (still empty footprint association list)
- test_kicad_1.sch

Run "Cvpcb", and in the list that comes up

Select YES to attempt converting

Allocate footprints to all components

***remember to SAVE***

(this can maybe also be done prior, in the "eeschema" for each component?)

Open "Generate netlist"

Select "Pcbnew" tab

Select Default format

Select "Use default netname"

Run "Generate"

MUST run without any errors

***remember to SAVE and EXIT "eeschema"***

Now we have the files

- test_kicad_1-cache.lib
- test_kicad_1.bak
- test_kicad_1.net (the list describing all connections. And that it is not a .net link!)
- test_kicad_1.pdf (free name)
- test_kicad_1.pro (with all footprints)
- test_kicad_1.sch (the schematic drawing)

## When ready for making the PCB

Open "Pcbnew"

Use default settings or select "Design Rules"

Select "Layer Setup" (eg.: 2 layers, thickness....)

Select "Design Rules" opens window "Design Rules Editor"

Select "Net Classes Editor" eg.:

Set Clearance to 0,5 mm

Set Track Width to 0,5 mm

Select "Grid: 2,5400 mm (100,00 mils)" in dropdown list

(if too small grid is not visible)

Start “Read netlist”
Browse to find the setlist, eg.: “test_kicad_1.net”
(keep all selector in top)
Run “Read Current Netlist”
Hopefully no faults
Close read-window
Spread out all the footprints. Point and use M +mouse

Place and orient (point + R => ccw) components
For text and lines on pcb select “F.SilkS” in dropdown list
***At some point SAVE for the first time to .kicad_pcb***
(mane is free, and we choose “test_kicad_1.kicad_pcb”)
Now we have the files

- test_kicad_1-cache.lib
- test_kicad_1.bak
- test_kicad_1.kicad_pcb
- test_kicad_1.kicad_pcb-bak
- test_kicad_1.net
- test_kicad_1.pdf
- test_kicad_1.pro
- test_kicad_1.sch

For edge cuts on pcb select “Edge.Cuts” in dropdown list, and draw
For view in 3D: **alt +3** turn off by closing tab with **X**

## When ready for Auto-routing

Start “Fast access to FreeROUTE….”
Select “Export a Spectra Design and Launch FreeRoute”
Save the default .dsn file (in the same folder as all the rest)
“Board Layout” window comes up
Start “Autorouter”, and watch the magic
NOTE: Wait until activity stops (NO indicator!)
Select “File” and “Export Specctra Sessions File”
Select YES
Close “Board Layout” window
In the still open “Fast access to FreeROUTE….” window
Select “Back import Spectra Design (.ses) File”
Find and select the newly saved .ses file, and open
Select YES to confirm rebuild
OK will close “Fast access to FreeROUTE….” window

We now have a layout with the tracks and footprints
Rearrange and edit whatever is necessary
***SAVE***

Now we have the files

- test_kicad_1-cache.lib
- test_kicad_1.bak
- test_kicad_1.dsn (the inputfile to the autorouter)
- test_kicad_1.kicad_pcb
- test_kicad_1.kicad_pcb-bak
- test_kicad_1.net
- test_kicad_1.pdf
- test_kicad_1.pro
- test_kicad_1.rules (output from autorouter…?)
- test_kicad_1.sch
- test_kicad_1.ses (for the back-import to PCBNew)

## Creating the output files to pcb-manufacturer

Still in "PCBNew"
View the result with "alt+3"
Select "File" and "Fabrication Output"

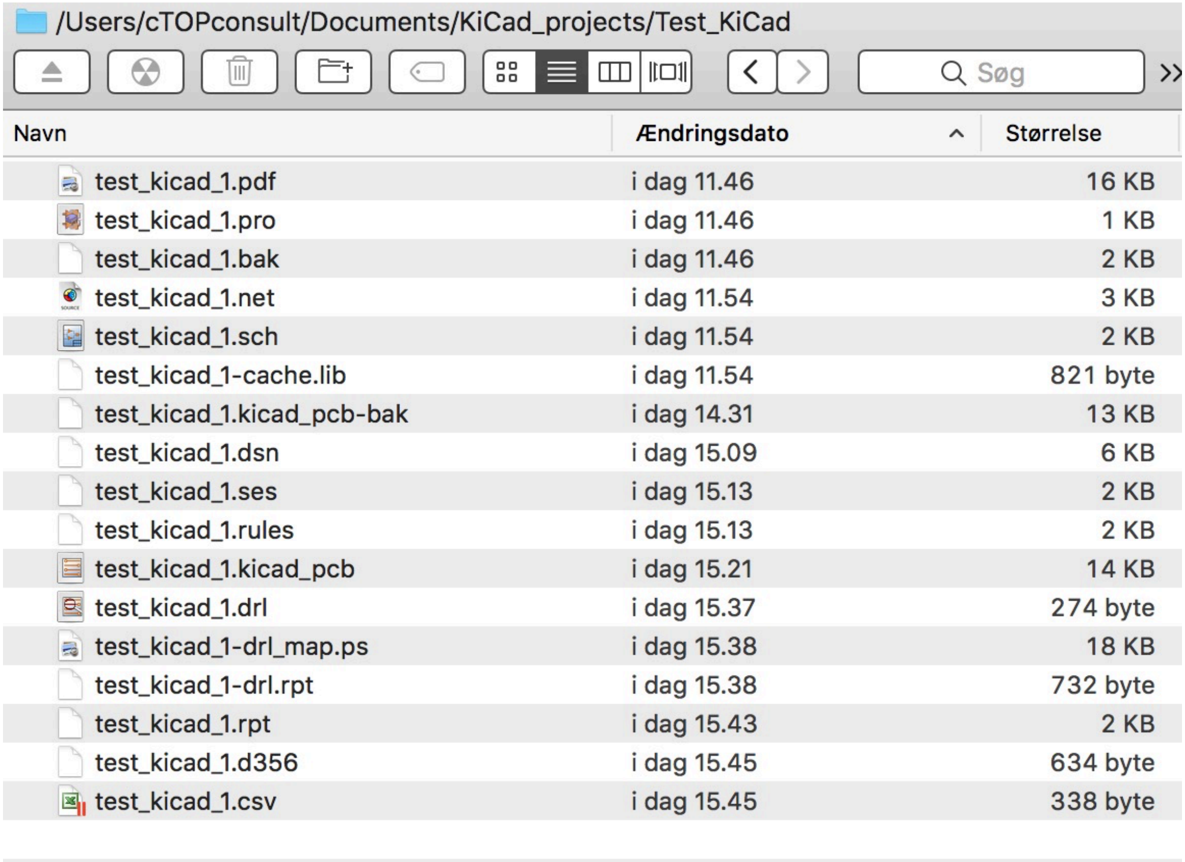
- Select "Footprint Position (.pos) File
    - Browse to the folder we use
    - Select no to relative
    - Msg.: No Footprint for automatic insertion…
    - Cancel
- Select "Drill (.drl) File"
    - (Browse)
    - Run "Drill File"
    - Run "Map File"
    - Run "Report File"
- Select "Footprint Report (.rpt) File"
    - Browse and open
    - (finished at once)
- Select "IPC-D-356 Netlist File"
    - Browse and open
    - (finished at once)
- Select "BOM File"
    - Browse and open
    - (finished at once)

Now we have the files

- test_kicad_1-cache.lib
- test_kicad_1-drl_map.ps (this one did not succeed)
- test_kicad_1-drl.rpt
- test_kicad_1.bak
- test_kicad_1.csv
- test_kicad_1.d356

- test_kicad_1.drl
- test_kicad_1.dsn
- test_kicad_1.kicad_pcb
- test_kicad_1.kicad_pcb-bak
- test_kicad_1.net
- test_kicad_1.pdf
- test_kicad_1.pro
- test_kicad_1.rpt
- test_kicad_1.rules
- test_kicad_1.sch
- test_kicad_1.ses

Chronologically:




| Navn | Ændringsdato | Størrelse |
|---|---|---|
| test_kicad_1.pdf | i dag 11.46 | 16 KB |
| test_kicad_1.pro | i dag 11.46 | 1 KB |
| test_kicad_1.bak | i dag 11.46 | 2 KB |
| test_kicad_1.net | i dag 11.54 | 3 KB |
| test_kicad_1.sch | i dag 11.54 | 2 KB |
| test_kicad_1-cache.lib | i dag 11.54 | 821 byte |
| test_kicad_1.kicad_pcb-bak | i dag 14.31 | 13 KB |
| test_kicad_1.dsn | i dag 15.09 | 6 KB |
| test_kicad_1.ses | i dag 15.13 | 2 KB |
| test_kicad_1.rules | i dag 15.13 | 2 KB |
| test_kicad_1.kicad_pcb | i dag 15.21 | 14 KB |
| test_kicad_1.drl | i dag 15.37 | 274 byte |
| test_kicad_1-drl_map.ps | i dag 15.38 | 18 KB |
| test_kicad_1-drl.rpt | i dag 15.38 | 732 byte |
| test_kicad_1.rpt | i dag 15.43 | 2 KB |
| test_kicad_1.d356 | i dag 15.45 | 634 byte |
| test_kicad_1.csv | i dag 15.45 | 338 byte |