

KiCad StepUp tools cheat sheet

<https://github.com/easyw/kicadStepUpMod>

1) What StepUp tools are for?

KiCad StepUp tools are a [FreeCAD Macro](#) and a [FreeCAD WorkBench](#) to help in **Mechanical Collaboration** between KiCad EDA and a Mechanical CAD.

With StepUp it is possible to:

- load kicad board and parts in FreeCAD and export it to STEP (or IGES) for a full ECAD MCAD collaboration
- load *kicad_mod* footprint in FreeCAD to easy and precisely align the mechanical model to kicad footprint
- convert the STEP 3D model of parts, board, enclosure to VRML with Materials properties for the best use in kicad
- check interference and collisions for enclosure and footprint design
- design a new pcb Edge with FreeCAD Sketcher and PUSH it to an existing kicad_pcb Board
- PULL a pcb Edge from a kicad_pcb Board, edit it in FC Sketcher and PUSH it back to kicad
- PUSH & PULL 3D models positions between FreeCAD and KiCAD
- ECAD / MCAD Collaboration and Synchronization (compressed 'stpZ' format allowed)
- footprint generation even for complex pads and shapes
- generate Blender compatible VRML files

2) Requirements

KiCad StepUp tools need with the following requirements:

- **KiCad Stable Release >= 4.0** or kicad **Nightly Development Builds**
- **FreeCAD** stable release >= **0.18** (FC daily is supported too)
- a library of STEP 3D models now available as default from [KiCad/packages3D](#)

3) How to install StepUp tools

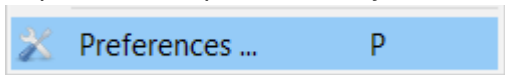
KiCad StepUp tools can be installed as a **FreeCAD Macro** but it is strongly suggested to install StepUp as a **FreeCAD WorkBench**.

Since KiCad StepUp tools have been added to [FC WorkBenches](#), so they can be installed through the FreeCAD [addons installer](#) or starting from FC version 0.17, through the **addons manager** in the FC Tools Menu. Then StepUp buttons will be available to be customized in FC Toolbars.

If KiCad StepUp tools are installed as a FC WorkBench, then **it will be possible to Open directly from the FC File Menu a *kicad_pcb* board file or a *kicad_mod* footprint file** and many useful features will be also available.

4) Configure StepUp tools

To use StepUp tools for converting a *kicad_pcb* Board to a mechanical STEP model you just need to **configure** your 3D prefix path(s) like your **KISYS3DMOD**. These settings are found the FreeCAD StepUp preferences page which only appear when using the **KiCadStepUp Workbench**. With that workbench activated, you can configure the paths via the preferences system of FreeCAD (Edit menu -> Preferences).



Just click the green icon:



Using **Snap** or **FlatPak** is requiring to 'bind mount' the 3D models folder
Some references at [GitHub Repo Issue](#)

5) Tips

Tips to use StepUp tools at its best

- never use a scale different from 1:1:1 in your 3D models
 - configure your [prefix3D] in the FreeCAD StepUp preference page to your KISYS3DMOD path
 - use STEP or STPZ or IGES or VRML or WRZ or mixed type of models in your board
 - use bounding boxes to reduce your STEP board file size if required
 - each 3D model is suggested to be be a single object (union of parts or compound in FC)
- note:** compound may be slower than union, because it needs to re-create a compound after loading the model

6) Useful Video Tutorials

Here some links of StepUp tutorial:

- StepUp: [Align Parts to Kicad footprint](#)
- StepUp: [converting a KiCad board and Parts to STEP](#)
- StepUp: [PUSH & PULL a PCB Edge using FC Sketcher](#)
- StepUp: [PUSH&PULL 3D models between KiCAD & FreeCAD \(ECAD MCAD Synchronization\)](#)

There is also a video tutorial made by a user:

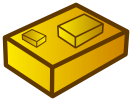
- StepUp: [Installing, Import 3D model, Exporting the Board](#)

Note: in the video the user is copying all demo files, when in fact it is better to install StepUp as a FreeCAD WorkBench.

7) Need Help?

KiCad info forum is a great resource:

<https://forum.kicad.info/search?q=step>



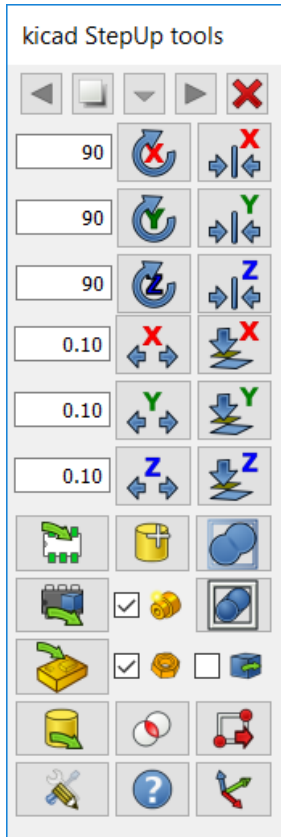
KiCad StepUp tools cheat sheet

<https://github.com/easyw/kicadStepUpMod>

The Main Gui

A brief recap on KiCad StepUp tools buttons.

Note: each button has a Tooltip



Load 'kicad_pcb' Board

Load a 'kicad_pcb' file into FreeCAD



Import 3D model to be Aligned

Import a 3D STEP model into FreeCAD



Load 'kicad_mod' Footprint

Load a 'kicad_mod' footprint into FreeCAD



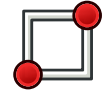
Export 3D model Aligned

Export a 3D STEP & VRML model back to KiCad



Export selected to STEP

Export selected objects or Board and Parts to hierarchical STEP file



Push & Pull PCB Edge

Read and Write pcb Edge from KiCad into FC Sketcher



Add Reference Axis

Add reference Axis to the FreeCAD design



Check Interferences and Collisions

Check Interference and Collisions in Board Design



Help

Mini Help inside StepUp tools



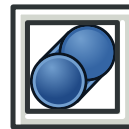
Make a Union

Make a Union of Parts



Preferences Config Page

showing the preferences Page



Make a Compound

Make a Compound of Parts

Option checkboxes



Materials properties

Adding Material to VRML when Exporting a 3D model



Virtual mechanical

Adding Virtual kicad Parts when Loading a 3D model of the PCB



export Board to STEP

Automatically export Board & Parts to STEP after Loading a 3D model of the PCB if checked

Useful Video Tutorials

Here some links of StepUp tutorial:

- StepUp: [Align Parts to Kicad footprint](#)
- StepUp: [converting a KiCad board and Parts to STEP](#)
- StepUp: [PUSH & PULL a PCB Edge using FC Sketcher](#)
- StepUp: [ECAD MCAD Synchronization & Collaboration](#)

There is also a video tutorial made by a user:

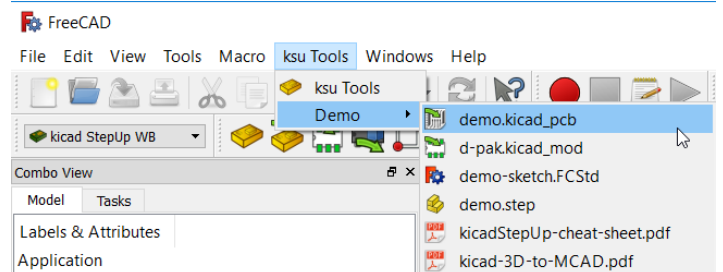
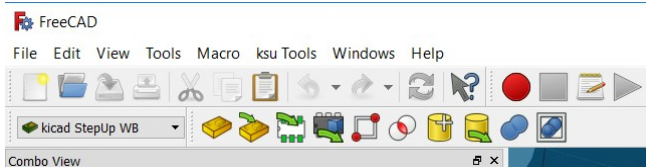
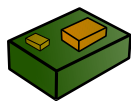
- StepUp: [Installing, Import 3D model, Exporting the Board](#)

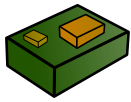
Note: in the video the user is copying all demo files, when in fact it is only needed *kicad-StepUp-tools.FCMacro* file.

The WorkBench

A screenshot on KiCad StepUp WB.

Demo and Manuals in the StepUp WB Menu





KiCad StepUp tools cheat sheet

<https://github.com/easyw/kicadStepUpMod>

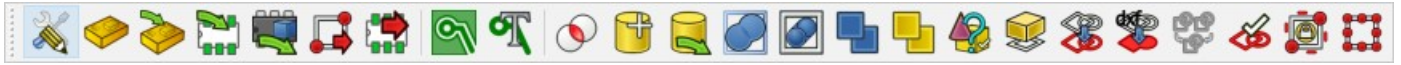
The WorkBench

A screenshot on KiCad StepUp WB.

Note: each button has a useful Tooltip

Demo and Manuals in the StepUp WB Menu

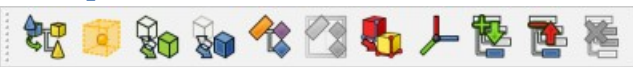
Main Tool bar



Push&Pull Tool bar



Helpers



Show tools



Useful Designing external workbenches



Two external workbenches:

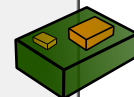
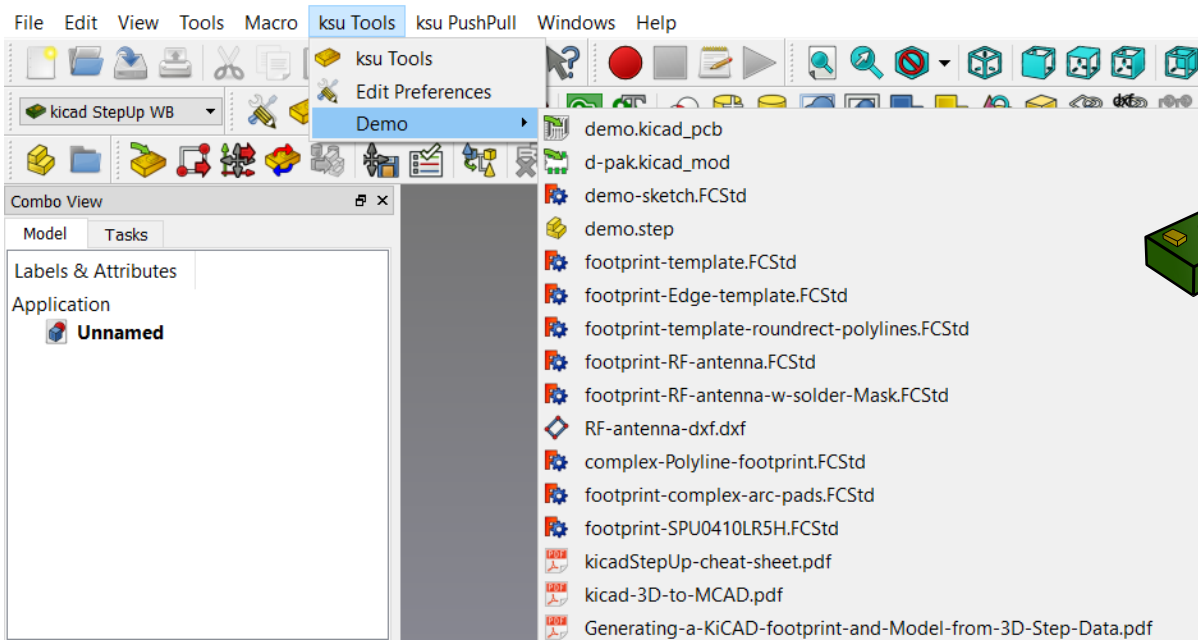
- [Manipulator workbench](#) useful to align and move assemblies and STEP models
Aligner Mover and Caliper are companions in 3D modelling

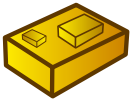


[Destructuring workbench](#) useful for editing STEP models, removing some features from the model; destructuring and repairing tools.



FreeCAD 0.18





KiCad StepUp tools cheat sheet

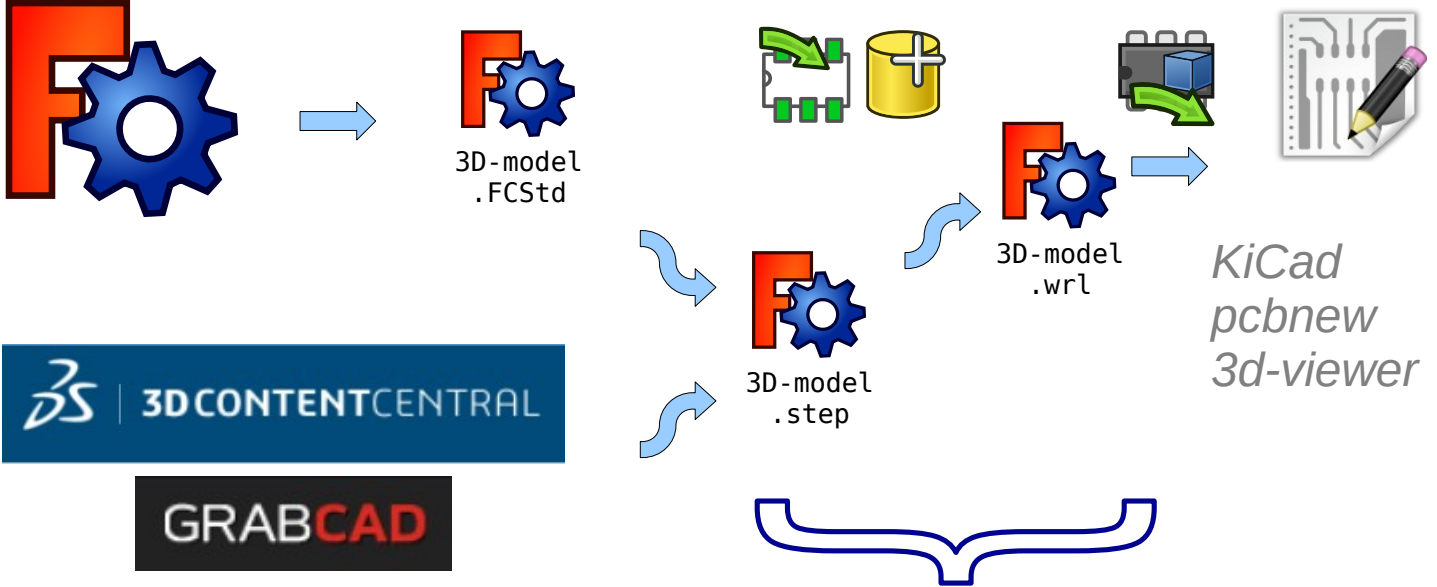
<https://github.com/easyw/kicadStepUpMod>

StepUp WorkFlow for 3D models

How to create a 3D model library for KiCad with StepUp tools

STEP compressed ['.stpZ']
VRML compressed ['.wrz']
formats are allowed

Use StepUp tools to Load the Footprint in FC and Align it



Use FreeCAD or any MCAD sw as 3D designer for a 3D model, or just download a 3D STEP model from on-line libraries

Place the exported models to the KISYS3DMOD folder

Use Manipulator WB

to align the STEP model to footprint
[Manipulator workbench](#)



Note:

when aligning a 3D model to a kicad footprint, StepUp takes care of:

- 2D footprint rotation of kicad for the footprint alignment
- vrml model z rotation

It is mandatory that the footprint has:

- x and y of the 3D model rotation set to 0
- x, y and z of the 3D model translation set to 0

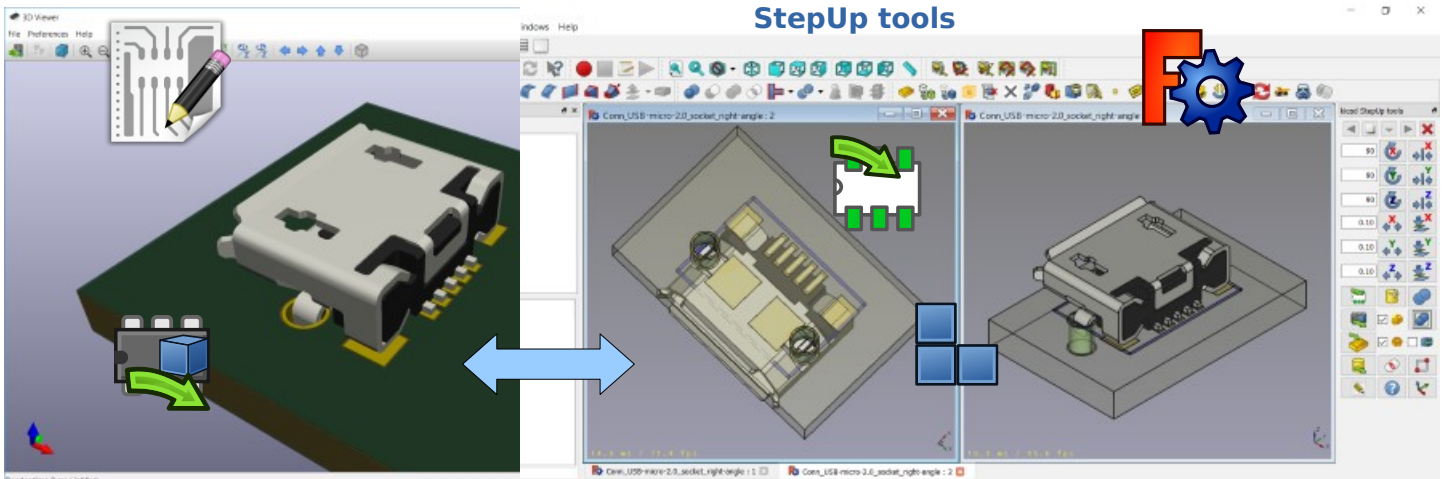
The user has to check/modify, if needed, the part of 3D vrml/step model in kicad as following

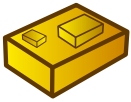
```
(model path/name.wrl
(at (xyz 0 0 0))
(scale (xyz 1 1 1))
(rotate (xyz 0 0 0))
```

at (xyz 0 0 0) is mandatory, as much as scale (1 1 1)
rotate (xyz 0 0 z_value) can have a z rotation value

Video Tutorials

- [Align Parts to Kicad footprint](#)
- [Installing, Import 3D model, Exporting the Board](#)





KiCad StepUp tools cheat sheet

<https://github.com/easyw/kicadStepUpMod>

Generating smaller 3D model with bounding boxes

Sometimes the need would be just a 3D MCAD model for analysis or simple space constraints, so a nice detailed component models in MCAD system may be not required or desired;

in that case it is possible to configure the exporter to:

- skip 3D models by name
- skip models with a volume less than an assigned value
- skip models with a height less than an assigned value
- skip DNP models (add DNP to blacklist)

And then convert the remaining parts, or all but edge connectors, to bounding boxes

The result 3D MCAD model will have the accuracy of the pcb and assemblies only when needed, maintaining the model light as required.

Configuration file: Blacklist & BoundingBox parameters



Preferences Page

Bounding Boxes

Black List

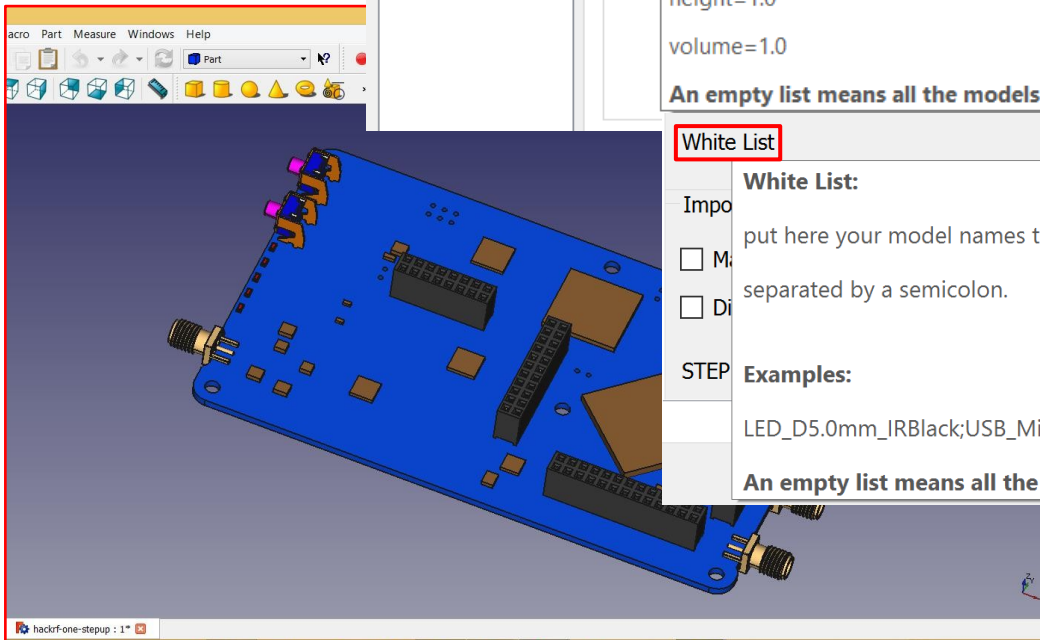
Bounding Box LIST:
 put here a list of 3D models to be converted to Bounding Box, separated by a comma
Examples:
ALL- -> all models will be converted to bounding boxes
 R_0603, C_0603 -> these two models will be converted to BBox
LIST DSUB-15-HD_FH, DSUB-9_FH -> these two models will NOT be converted to BBox

Make a...
 Directly...
 STEP exp...
 3D Loading...
 Assem...
 Enable...
 Start Turn...

KiCad StepUp: Using bounding boxes for all but connectors and skipping small parts

Black List

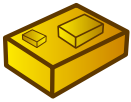
Black List:
 put here your model names that you don't want to load (e.g. smallest ones) separated by a comma.
 STEP volume and height are also configurable.
 A (volume=1 means all models with a volume < 1mm³ will not be included)
 E (height=1 means all models with a height < 1mm will not be included)
 S
Examples:
 r_0603,r_0402,c_0402,c_0603
 height=1.0
 volume=1.0
An empty list means all the models will be parsed.



White List

White List:
 put here your model names that you must load (e.g. smallest ones) separated by a semicolon.
 STEP **Examples:**
 LED_D5.0mm_IRBlack;USB_Mini-B_Lumberg_2486_01_Horizontal
An empty list means all the models will behave as usual.

WhiteList: for must load models



KiCad StepUp tools cheat sheet

<https://github.com/easyw/kicadStepUpMod>

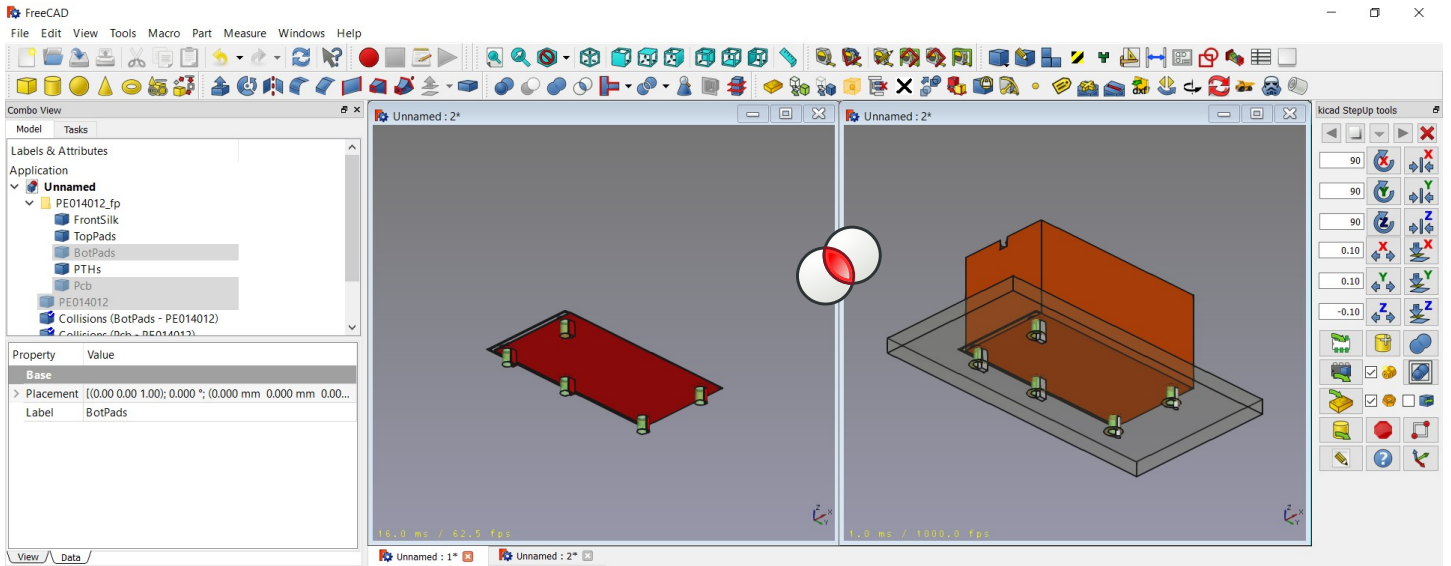
Check for Interference and mechanical constrains

With kicad-SteUp-tools it is also possible to detect collisions and check mechanical constrains:

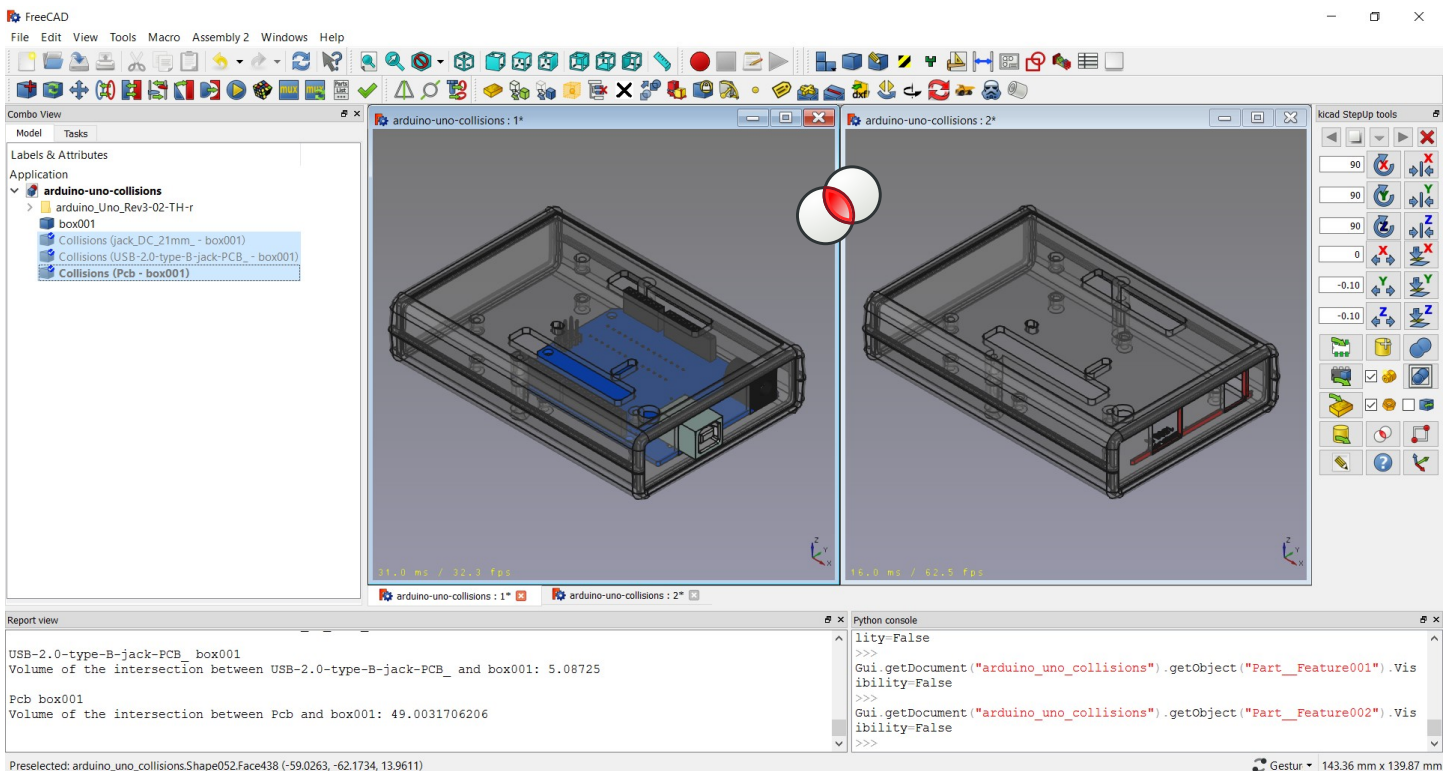
- detect collisions among part pins and drills for footprints
- detect collisions for enclosure clearance (between pcb with parts/connectors and enclosure)

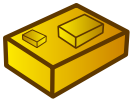


Interference checking for Footprints



Interference checking for PCB & Enclosure





KiCad StepUp tools cheat sheet

<https://github.com/easyw/kicadStepUpMod>

StepUp: ECAD MCAD Synchronization

KiCAD StepUp goes to a new level of **ECAD / MCAD Collaboration and Synchronization**: Push/Pull 3D model placement from/to KiCAD board to/from FreeCAD mechanical design.

It is possible to move 3D packages around on the 3D PCB mechanical sw, via both the X and Y axis.

The syncing process can be done even if the board is (fully) routed (i.e. when a new release requires some mechanical reviews).

ECAD MCAD integration is now fully implemented.



kicad StepUp: ECAD MCAD Synchronization



The ECAD MCAD
collaboration tutorial
ECAD MCAD Synchronization



Tips

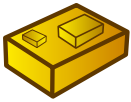
It is suggested to configure *the preferences Page* to use **grid origin** and **place a grid origin** to `kicad_pcb` file

PCB Placement Grid Origin ▾

KiCad StepUp Workbench



STEP compressed ['.stpZ']
VRML compressed ['.wrz']
formats are allowed



KiCad StepUp tools cheat sheet

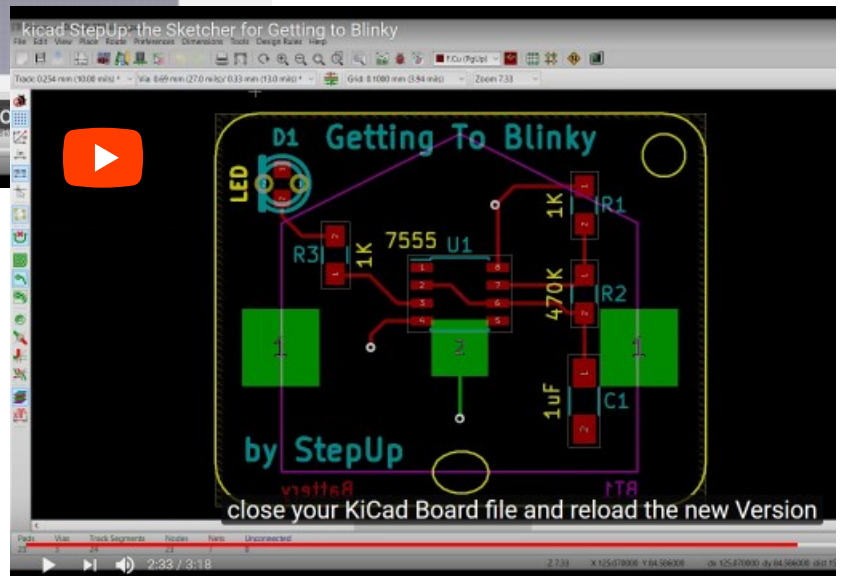
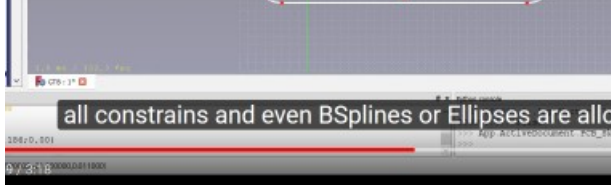
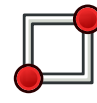
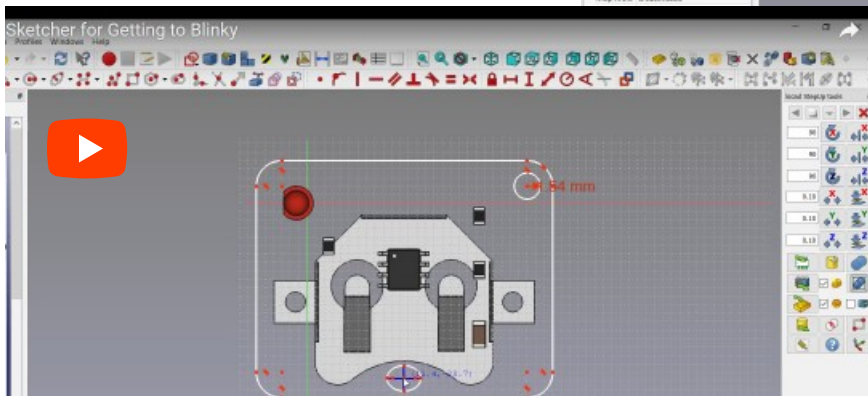
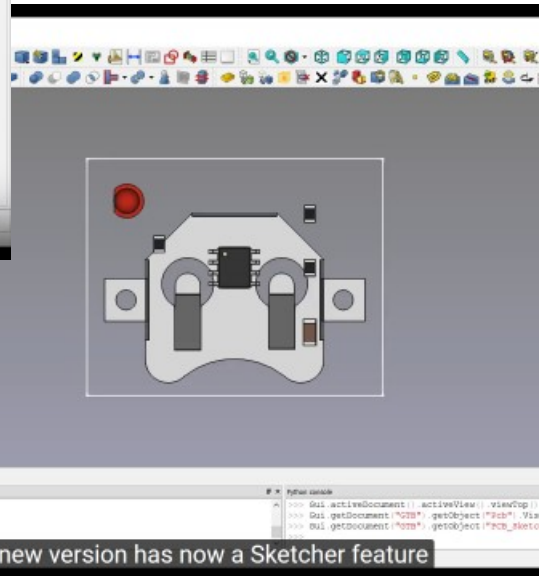
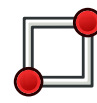
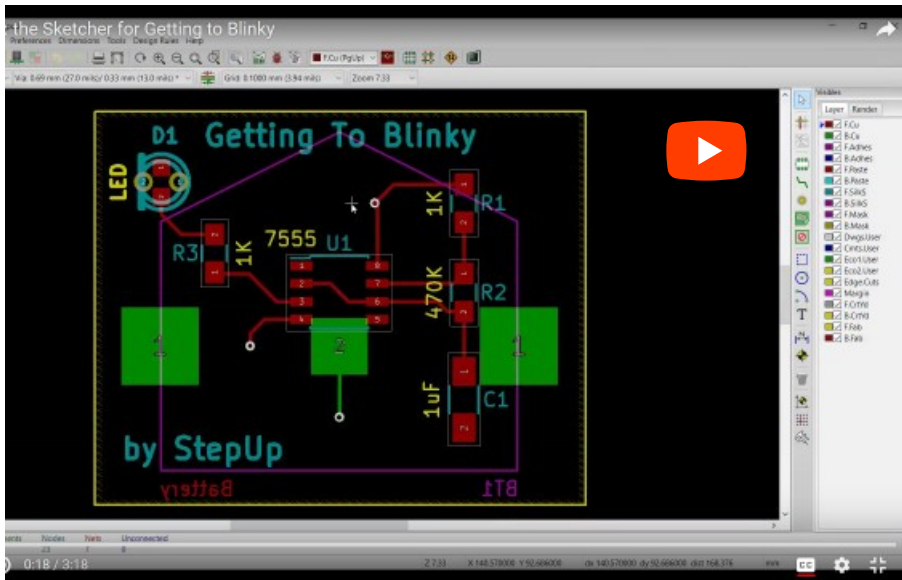
<https://github.com/easyw/kicadStepUpMod>

StepUp: The Sketcher

With kicad-SteUp-tools it is also possible to use FreeCAD Sketcher to create or modify a PCB Edge.

- create a new PCB Edge in FreeCAD Sketcher and PUSH it to kicad_pcb file
- read a PCB Edge from an existing kicad_pcb file and PULL it to FreeCAD Sketcher
- modify a PCB Edge in FreeCAD Sketcher and PUSH it to KiCad Board

Line, Circles, Arcs are supported and also **Bsplines or Ellipses** are supported and converted to KiCad compatible format



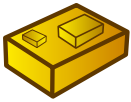
The Sketcher tutorial 
[PUSH & PULL a PCB Edge using FC Sketcher](#)

Tips

It is suggested to configure the preferences Page to use **grid origin** and **place a grid origin** to kicad_pcb file

PCB Placement

Grid Origin



KiCad StepUp tools cheat sheet

<https://github.com/easyw/kicadStepUpMod>

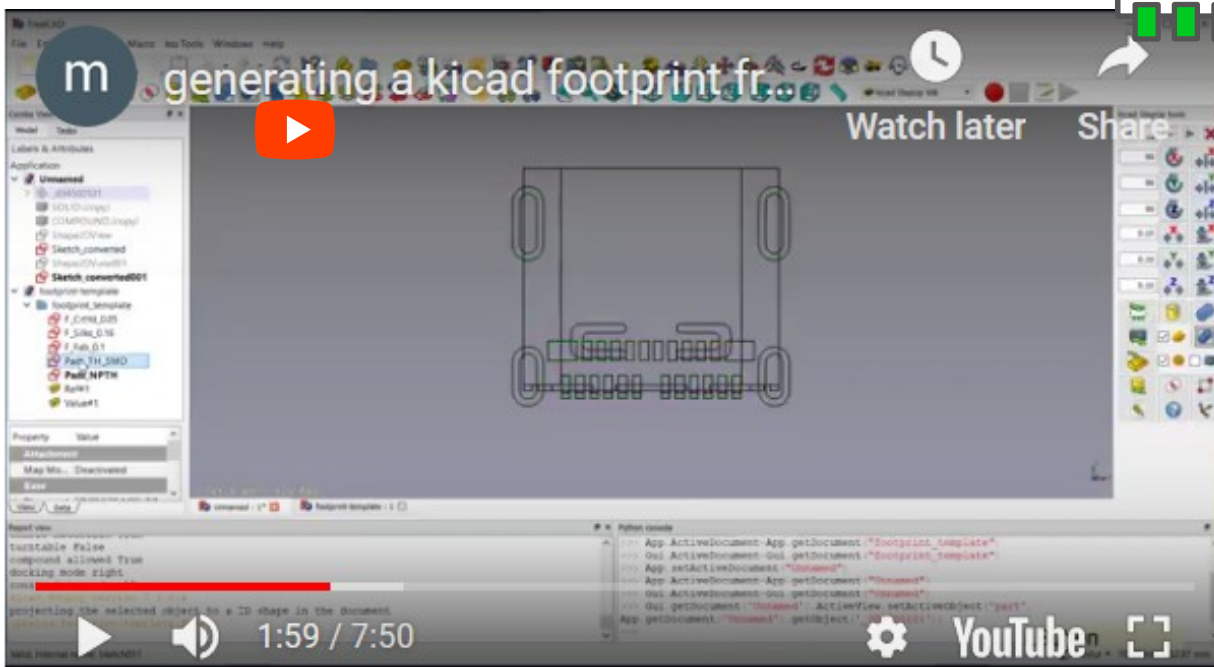
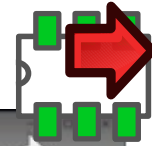
StepUp: The Sketcher for footprint generation

With kicad-SteUp-tools it is also possible to use FreeCAD Sketcher to create or modify a Kicad footprint.

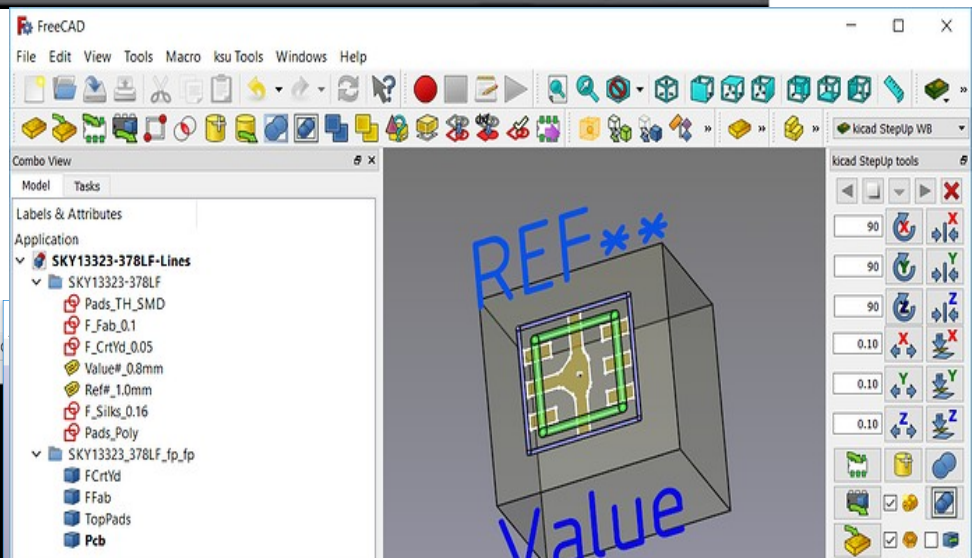
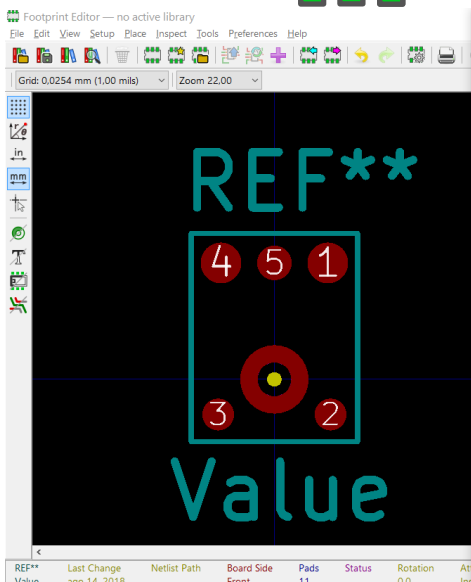
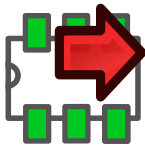
- create a new footprint in FreeCAD Sketcher and PUSH it to kicad_mod file
- modify an existing kicad footprint in FreeCAD Sketcher and PUSH it back to 'kicad_mod'

Line, Circles, Arcs are supported and also Bsplines or Ellipses are supported and converted to KiCad compatible format

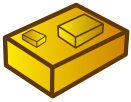
generating a kicad footprint from a 3D STEP model



The Sketcher for footprint: Tutorial



The Sketcher for footprint
 @ **KiCad**INFO

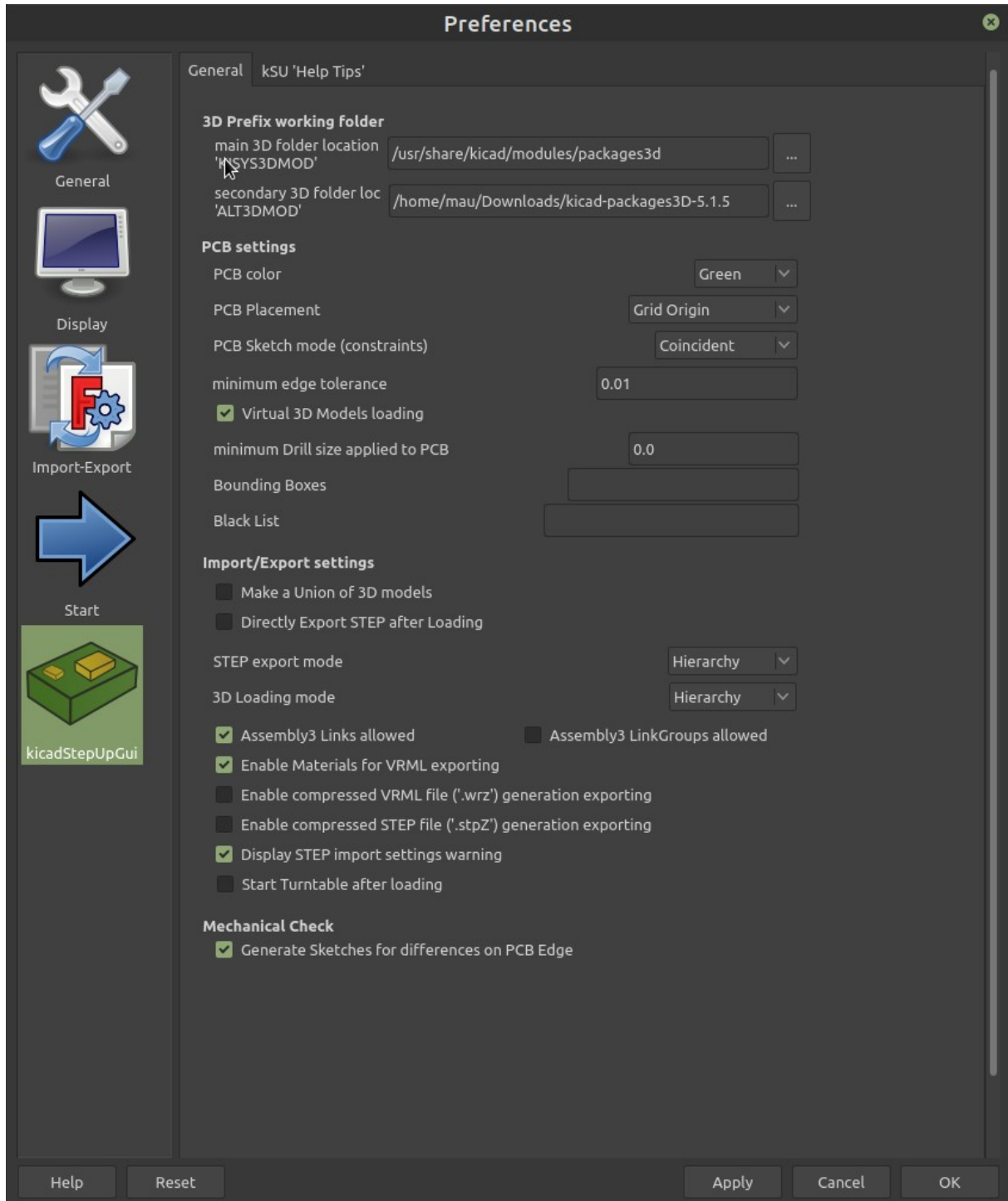


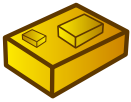
KiCad StepUp tools cheat sheet

<https://github.com/easyw/kicadStepUpMod>

Preferences Page for configuring main parameters

All buttons have Tooltips





KiCad StepUp tools cheat sheet

<https://github.com/easyw/kicadStepUpMod>

Preferences Page for configuring STEP Import Export (FC0.19)

All buttons have Tooltips

The screenshot shows the 'Preferences' dialog box with the 'STEP' tab selected. The left sidebar contains icons for 'General', 'Display', 'Import-Export', 'Start', 'kicadStepUpGui', and 'Part design'. The 'Import-Export' icon is highlighted. The main area is divided into 'Export' and 'Import' sections. The 'Export' section includes a units dropdown set to 'Millimeter', and checkboxes for 'Write out curves in parametric space of surface', 'Export invisible objects', 'Export single object placement', and 'Use legacy exporter'. The 'Scheme' section has radio buttons for 'AP 203' and 'AP 214'. The 'Import' section includes checkboxes for 'Enable STEP Compound merge', 'Use legacy importer', 'Use Part container', 'Import invisible objects', 'Reduce number of objects', 'Expand compound shape', 'Show progress bar when importing', and 'Ignore instance names'. The 'Mode' dropdown is set to 'Single document'. The 'Header' section has text boxes for 'Company', 'Author', and 'Product' (pre-filled with 'Open CASCADE STEP translator 7.3'). At the bottom are 'Help', 'Reset', 'Apply', 'Cancel', and 'OK' buttons.

Preferences

IFC DAE DXF DWG SVG OCA IGES STEP

Export

Units for export of STEP Millimeter ▾

- Write out curves in parametric space of surface
- Export invisible objects
- Export single object placement
- Use legacy exporter

Scheme

- AP 203
- AP 214

Import

- Enable STEP Compound merge
- Use legacy importer
- Use Part container
- Import invisible objects
- Reduce number of objects
- Expand compound shape
- Show progress bar when importing
- Ignore instance names

Mode Single document ▾

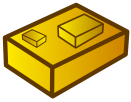
Header

Company

Author

Product

Help Reset Apply Cancel OK

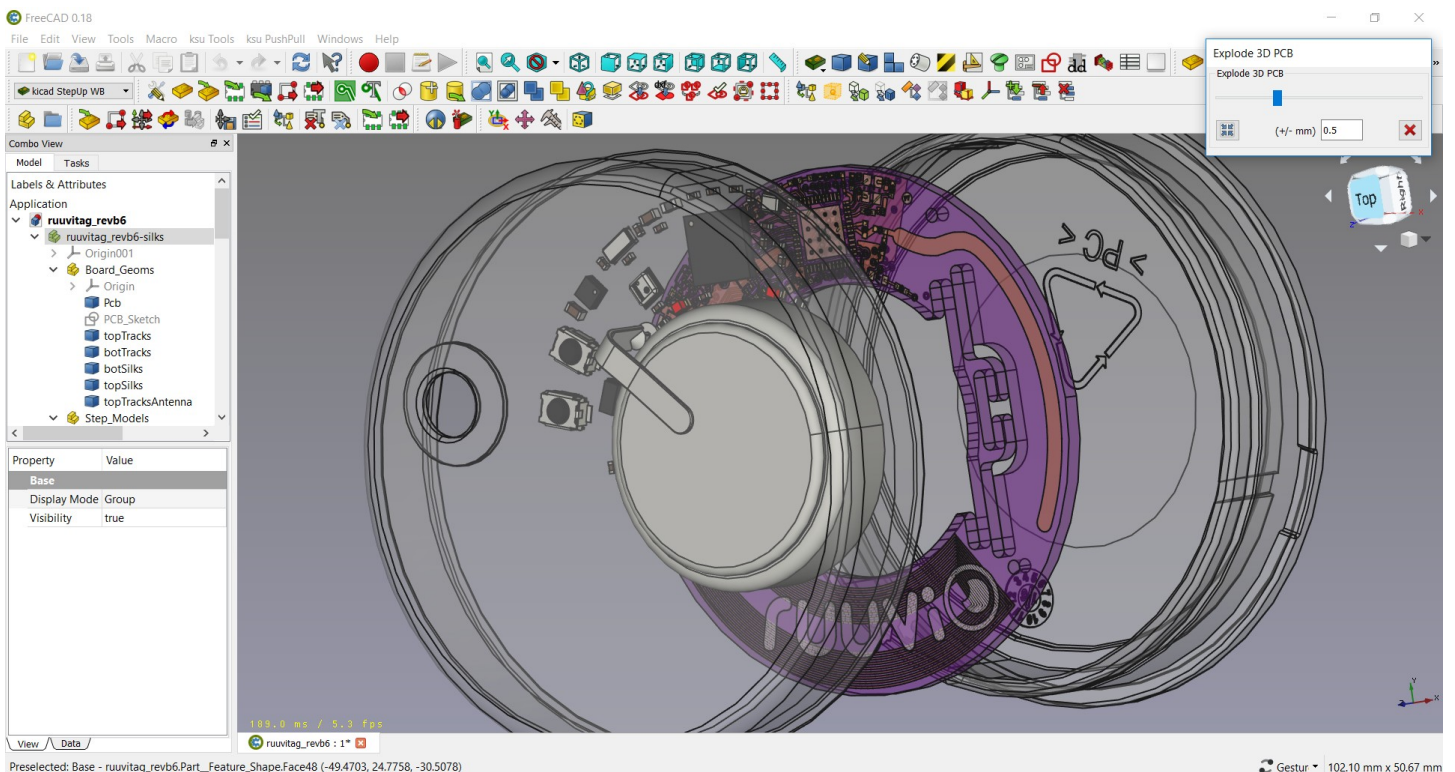
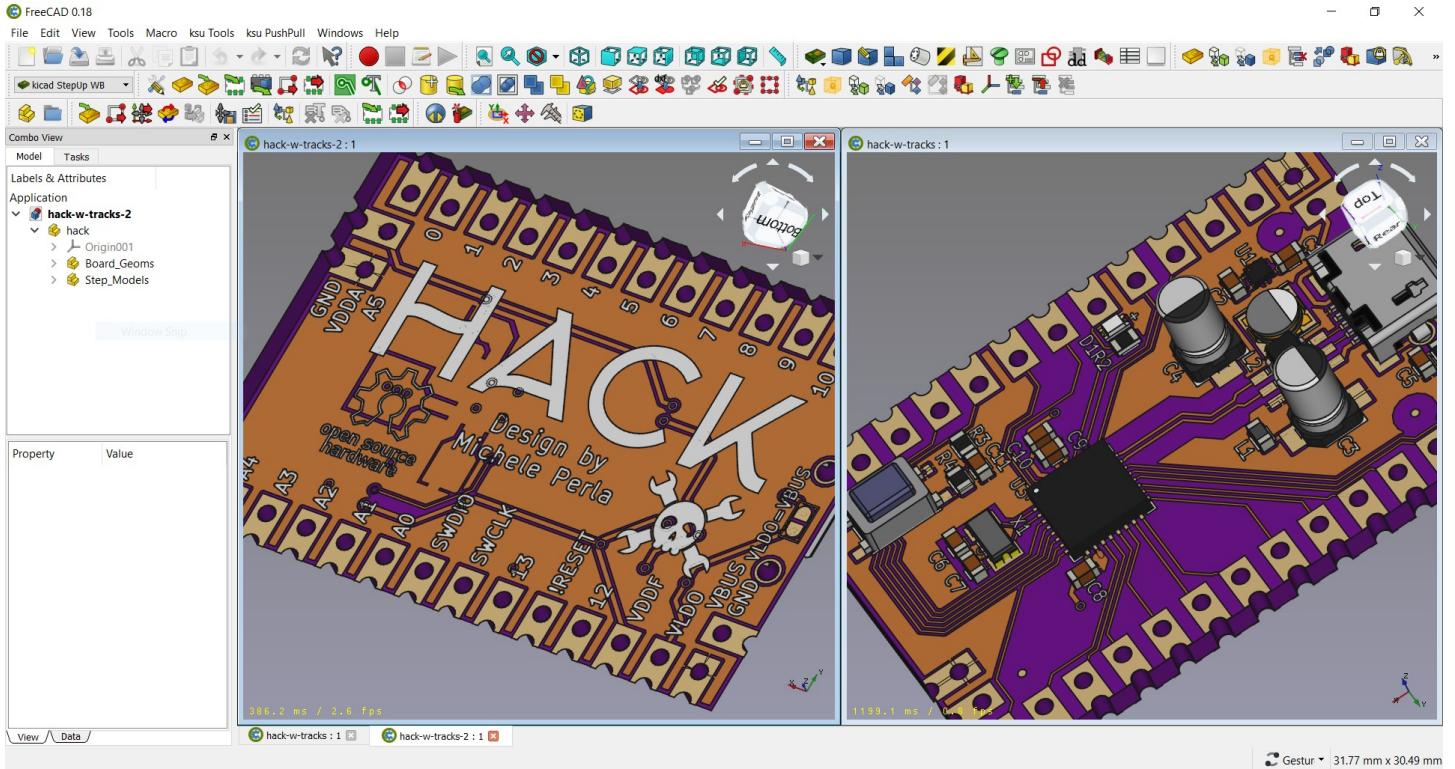


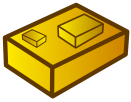
KiCad StepUp tools cheat sheet

<https://github.com/easyw/kicadStepUpMod>

Tracks and SilkScreen MCAD integration

New ability to import Top and Bottom tracks and SilkScreen layers





KiCad StepUp tools cheat sheet

<https://github.com/easyw/kicadStepUpMod>

Tracks MCAD integration

New ability to import Top and Bottom tracks and SilkScreen layers

(for PCB signal integrity and EMC simulation design please refer to [fcad_pcb repo](#))



Top and Bottom tracks are imported directly from 'kicad_pcb' source file into FreeCAD designing document.



Top and Bottom SilkScreens are imported from Top and Bottom DXF files, exported from KiCAD source file.

KiCAD export configuration

The screenshot shows the 'Plot' dialog box in KiCAD. The 'Plot format' is set to 'DXF'. The 'Output directory' is empty. Under 'Included Layers', 'F.SilkS' and 'B.SilkS' are checked. In the 'General Options' section, 'Plot footprint references', 'Exclude PCB edge layer from other layers', 'Exclude pads from silkscreen', and 'Check zone fills before plotting' are checked. In the 'DXF Options' section, 'Plot graphic items using their contours' and 'Use Pcbnew font to plot texts' are checked. The 'Export units' dropdown is highlighted with a red box and set to 'Millimeters'. At the bottom, the 'Plot' button is highlighted with a blue box.

Plot

Plot format: **DXF** Output directory:

Included Layers

- F.Cu
- B.Cu
- F.Adhes
- B.Adhes
- F.Paste
- B.Paste
- F.SilkS
- B.SilkS
- F.Mask
- B.Mask
- Dwgs.User
- Cmts.User
- Eco1.User
- Eco2.User

General Options

- Plot border and title
- Plot footprint values
- Plot footprint references
- Force plotting of invisible values / refs
- Exclude PCB edge layer from other layers
- Exclude pads from silkscreen
- Do not tent vias
- Use auxiliary axis as origin

Drill marks: **None**

Scaling: **1:1**

Plot mode: **Filled**

Default line width: **0.15** mm

- Mirrored plot
- Negative plot
- Check zone fills before plotting

DXF Options

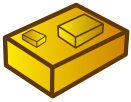
- Plot graphic items using their contours
- Use Pcbnew font to plot texts

Export units: **Millimeters**

Output Messages

Show: All Errors Warnings Actions Infos

Run DRC... Plot Close Generate Drill Files... Save...



KiCad StepUp tools cheat sheet

<https://github.com/easyw/kicadStepUpMod>

Tracks MCAD integration

New ability to import Top and Bottom tracks and SilkScreen layers



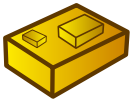
Top and Bottom SilkScreens are imported from Top and Bottom DXF files, exported from KiCAD source file.

FreeCAD import configuration

The screenshot shows the FreeCAD Preferences dialog box with the DXF tab selected. The dialog is titled "Preferences" and has a sidebar on the left with icons for General, Display, Import-Export, Part design, kicadStepUp..., and Sketcher. The main area contains the following settings:

- IFC | DAE | **DXF** | DWG | SVG | OCA | IGES | STEP
- Show this dialog when importing and exporting
- Use legacy python importer
- Automatic update (legacy importer only)
 - Allow FreeCAD to automatically download and update the DXF libraries
- Import options
 - Note: Not all the options below are used by the new importer yet
 - Import texts and dimensions points layouts *blocks
 - Create simple Part shapes Draft objects Sketches
 - Scale factor to apply to imported files: 1.0000
 - Get original colors from the DXF file
 - Join geometry
 - Group layers into blocks
 - Use standard font size for texts
 - Use VisGroups
 - Import hatch boundaries as wires
 - Render polylines with width
 - Treat ellipses and splines as polylines
 - Max Spline Segment: 10.00mm
- Export options
 - Export 3D objects as polyface meshes
 - Export Drawing Views as blocks
 - Project exported objects along current view direction

Buttons at the bottom: Reset, OK, Cancel, Apply, Help.



KiCad StepUp tools cheat sheet

<https://github.com/easyw/kicadStepUpMod>

StepUp Credits

kicad StepUp tools author is Maurice <https://github.com/easyw/kicadStepUpMod>

IDF import for FreeCAD - Milos Koutny (milos.koutny@gmail.com)

CadQuery module - CadQuery FreeCAD module <https://github.com/jmwright/cadquery-freecad-module/>

hyOzd freecad macros - <https://bitbucket.org/hyOzd/freecad-macros>

FreeCAD-PCB - marmni <marmni@onet.eu26>

Kicad semantic parser - "Zheng, Lei" https://github.com/realthunder/fcad_pcb

Copyrights

this document and kicad StepUp tools are Copyrighted © 2015 2016 2017 by Maurice.

Kicad STEPUP™ is a TradeMark and cannot be freely useable.

This program is free software; you can redistribute it and/or modify it under the terms of the GNU Affero General Public License as published by the Free Software Foundation to ensure cooperation with the community in the case of network server software; for detail see the LICENCE text file.

<http://www.gnu.org/licenses/agpl-3.0.en.html>

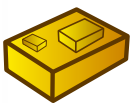
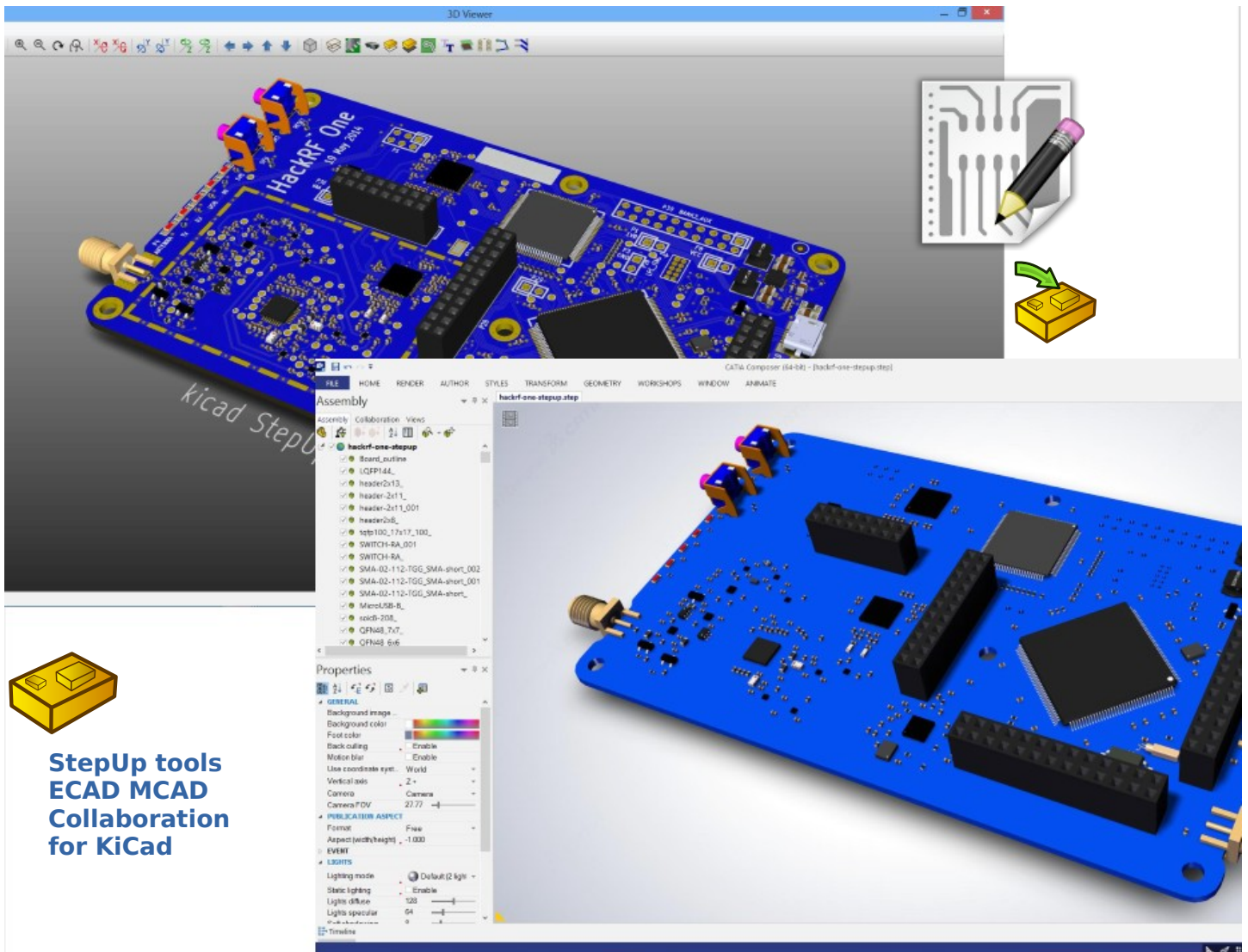
Moreover you have to include the original author copyright.

All trademarks within this guide belong to their legitimate owners.

Risk disclaimer

USE 3D CAD DATA AT YOUR OWN RISK

DO NOT RELY UPON ANY INFORMATION FOUND HERE WITHOUT INDEPENDENT VERIFICATION



**StepUp tools
ECAD MCAD
Collaboration
for KiCad**