

KiCad StepUp tools cheat sheet

<https://sourceforge.net/projects/kicadstepup/>

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
- 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 **0.15 4671** or release **0.16 >=6712** or even better **0.17 >=11707**
- 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** following this [How to install FC Macros](#). Just copy the **kicad-StepUp-tools.FCMacro** file and the **kicad-StepUp-icon.svg** to the FreeCAD Macro folder. The suggested use is through [Customize Toolbars](#) and placing the Macro in the **Global Toolbar** of FreeCAD.

KiCad StepUp tools can be also be installed as a **FreeCAD WorkBench** following these instructions: Simple copy the **kicadStepUpMod** folder to the FreeCAD Mod folder.

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.**

Recently 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.

4) configure StepUp tools

To use StepUp tools for converting a *kicad_pcb* Board to a mechanical STEP model you just need to edit 'ksu-config.ini' 3D prefix path assigning it to your **KISYS3DMOD** value, 'ksu-config.ini' file is in %Homepath% (Win) or \$HOME (OSX & Linux) folder.

```
[prefix3D]
prefix3d_1 = C:\Program Files\KiCad\share\kicad\modules\packages3d\
```

In case of any problem, just delete ksu-config.ini file and restart kicad StepUp tools... a new ini file will be generated; edit then your 3D prefix and re-run the tools

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 ksu-config.ini to your KISYS3DMOD path
- use STEP or IGES or VRML or mixed type of models in your board
- use bounding boxes to reduce your STEP board file size if required
- each 3D model must 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](#)

There is also a video tutorial made by a user:

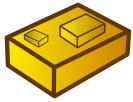
- StepUp: [Installing, Import 3D model, Exporting the Board](#)

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

7) need Help?

KiCad info forum is a great resource:

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



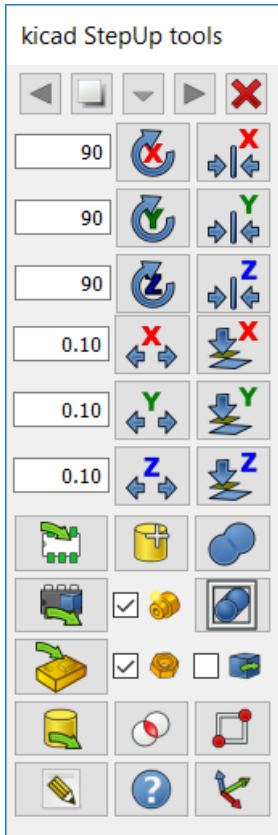
KiCad StepUp tools cheat sheet

<https://sourceforge.net/projects/kicadstepup/>

The 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



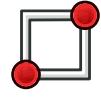
Export 3D model Aligned

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



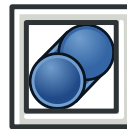
Config Editor

showing the 'ksu-config.ini' content



Make a Union

Make a Union of Parts



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](#)

There is also a video tutorial made by a user:

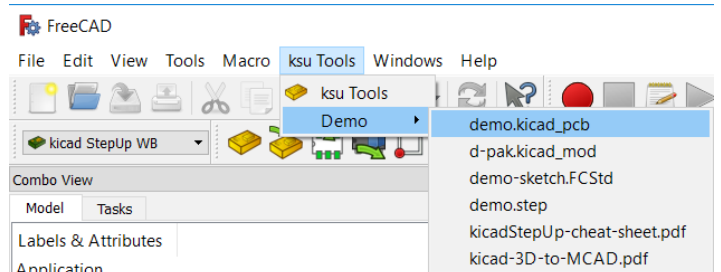
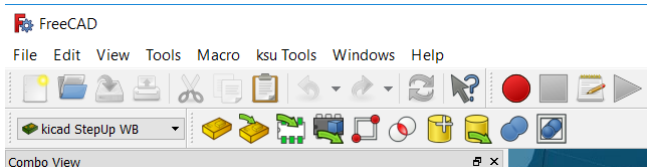
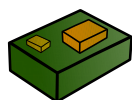
- StepUp: [Installing, Import 3D model, Exporting the Board](#)

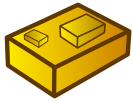
Note: in the video the user is copying all demo files, when in fact it is only needed *kicad-StepUp-tools.FCMacro* file.

The WorkBench

A screenshot on KiCad StepUp WB.

Demo and Manuals in the StepUp WB Menu





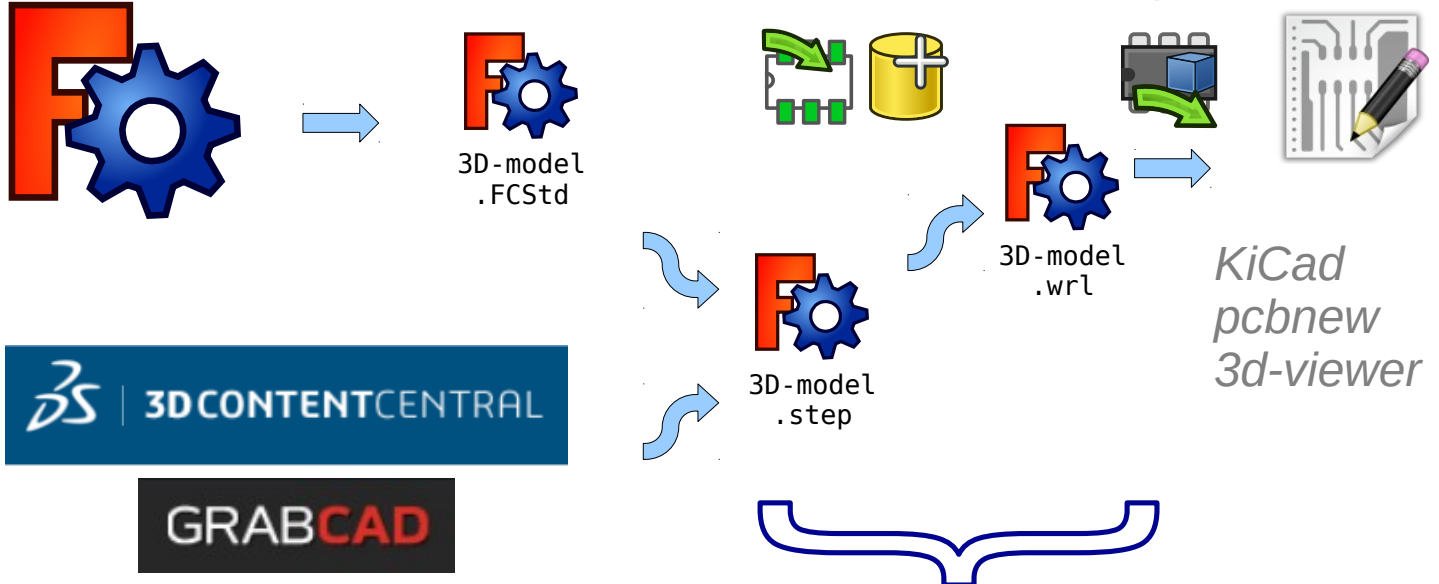
KiCad StepUp tools cheat sheet

<https://sourceforge.net/projects/kicadstepup/>

StepUp WorkFlow for 3D models

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

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

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

use Assembly2 WB

to align the STEP model to footprint
[Assembly2 WB](#)



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

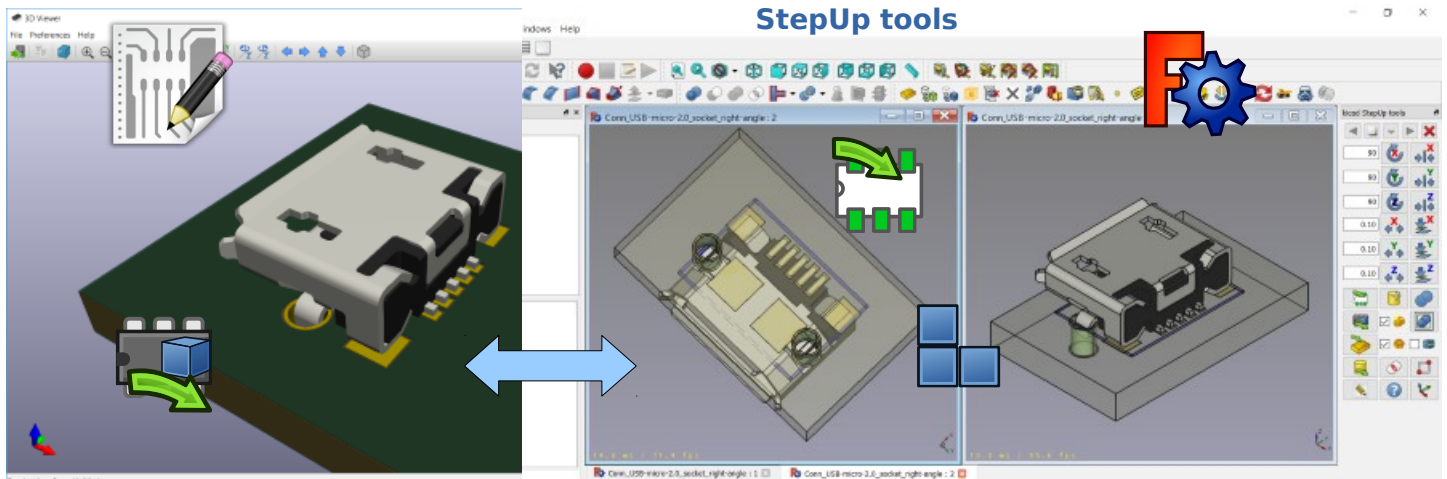
```
(model path/name.wrl
(at (xyz 0 0 0))
(scale (xyz 1 1 1))
(rotate (xyz 0 0 0))
```

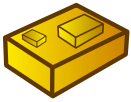
at (xyz 0 0 0) is mandatory, as much as scale (1 1 1)
rotate (xyz 0 0 z_value) can have a z rotation value

Video Tutorials

[Align Parts to Kicad footprint](#)

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





KiCad StepUp tools cheat sheet

<https://sourceforge.net/projects/kicadstepup/>

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

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 *ksu-config.ini*



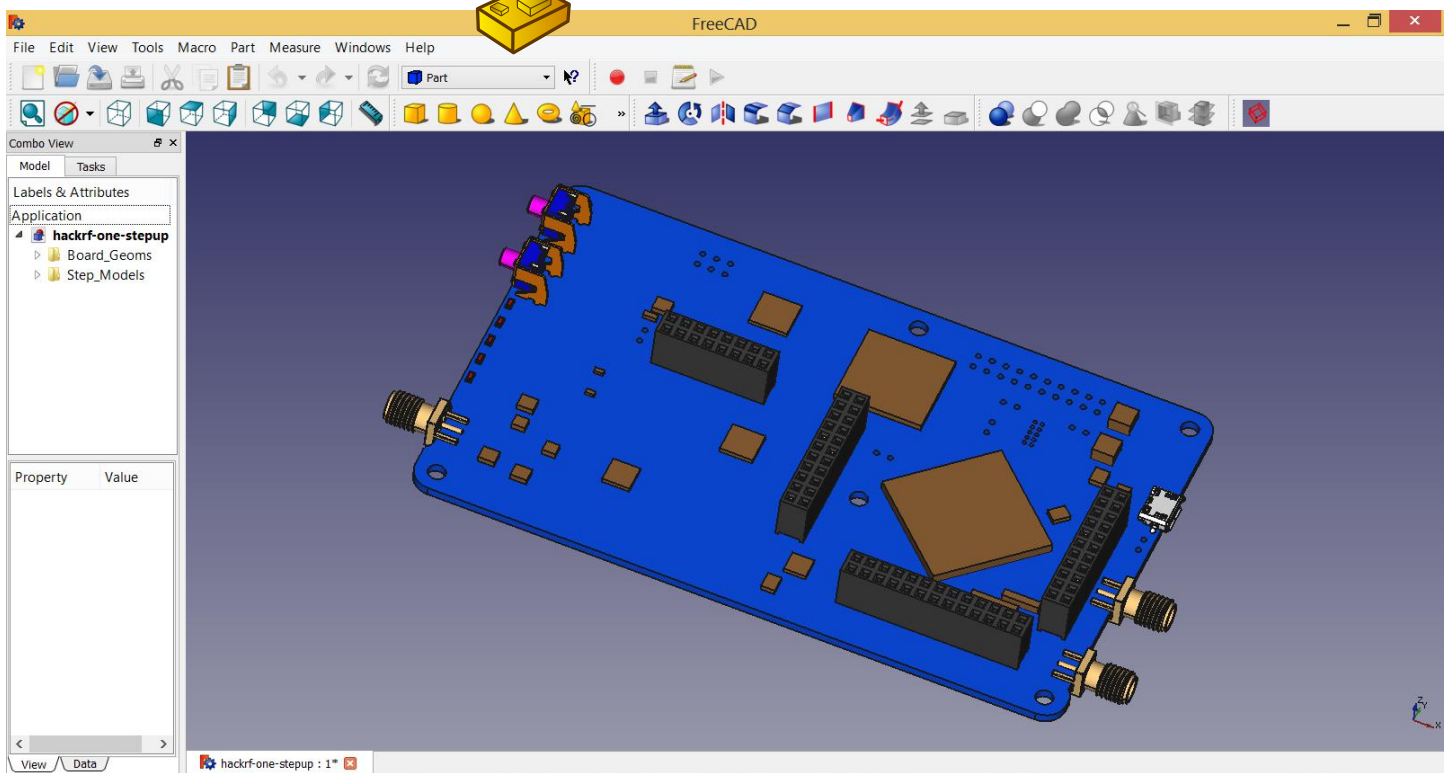
[Blacklist]

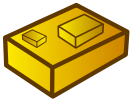
```
;; put here your model names that you don't want to load (e.g. smallest ones)
;; separated by a comma (none means all the models will be parsed)
;; (volume=1 means all models with a volume < 1mm3 will not be included)
;; (height=1 means all models with a height < 1mm will not be included)
;bklist = r_0603,r_0402,c_0402,c_0603
;bklist = height=1.0
;bklist = volume=1.0
;bklist = none
bklist = volume=1.0
```

[BoundingBox]

```
;; bounding box option list=>whitelist (not converted to bbox)
;bbox = list dpak-to252,sod80
;bbox = all
;bbox = off default
bbox = list SMA,USB,PushBtn,Header
```

kicad StepUp: using bounding boxes for all but connectors and skipping small parts





KiCad StepUp tools cheat sheet

<https://sourceforge.net/projects/kicadstepup/>

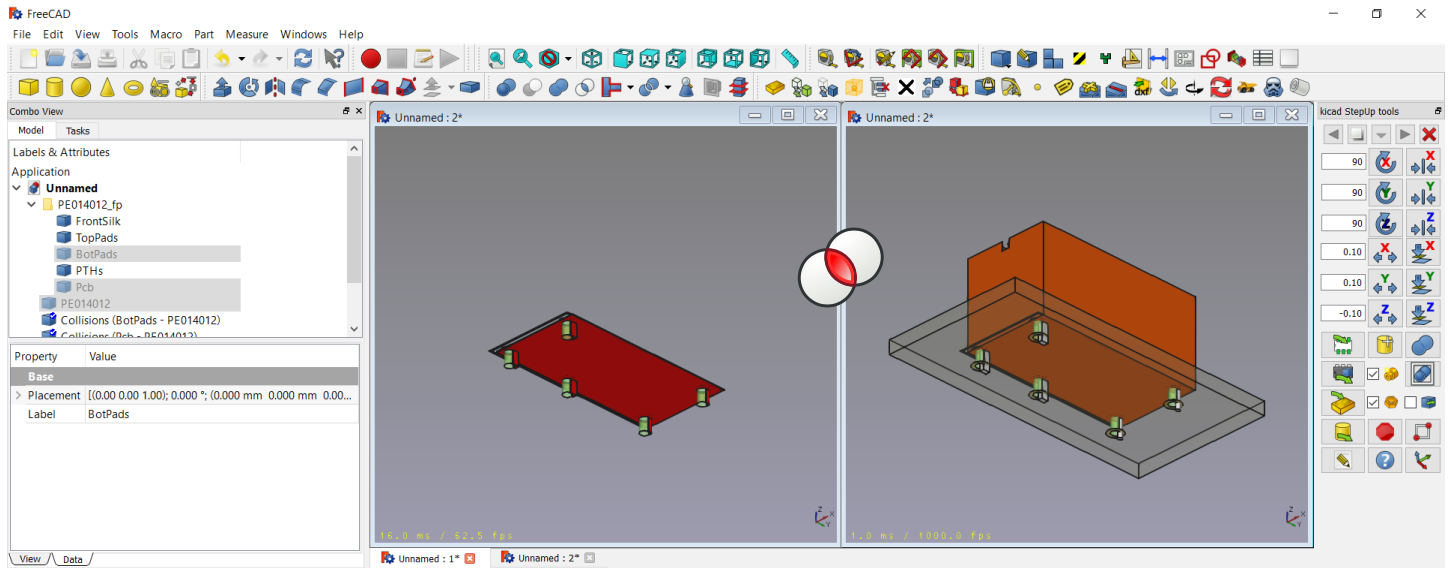
Check for Interferences and mechanical constrains

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

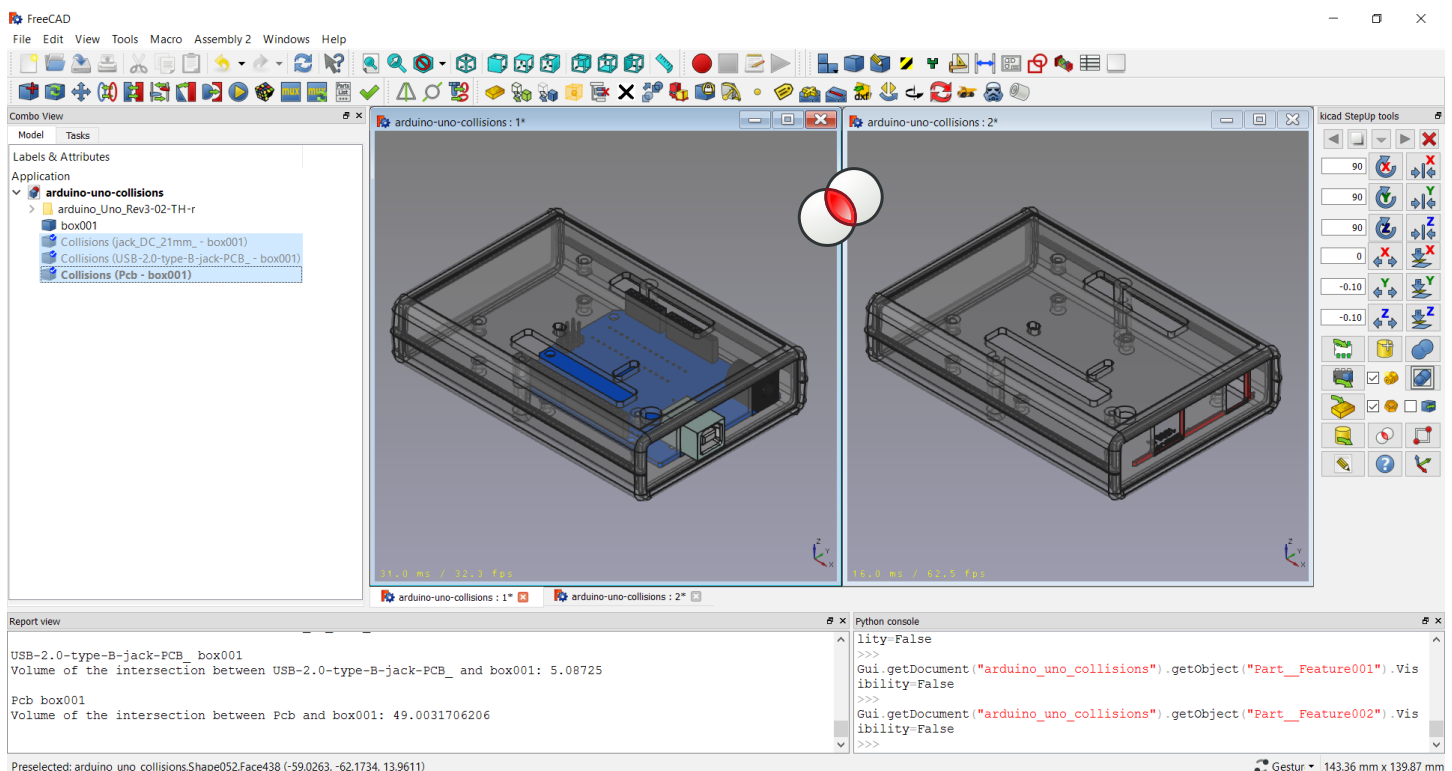
- detect collisions among part pins and drills for footprints
- detect collisions for enclosure clearance (between pcb with parts/connectors and enclosure)

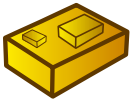


Interference checking for Footprints



Interference checking for PCB & Enclosure





KiCad StepUp tools cheat sheet

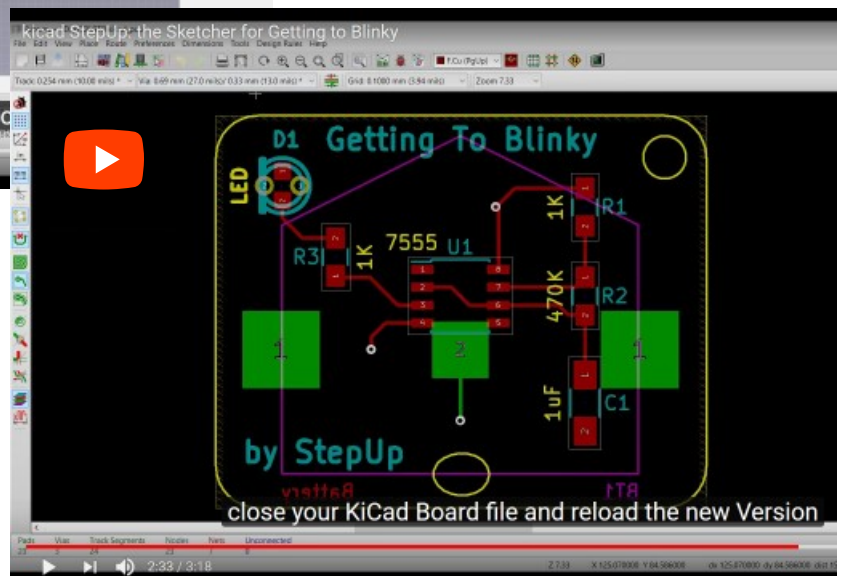
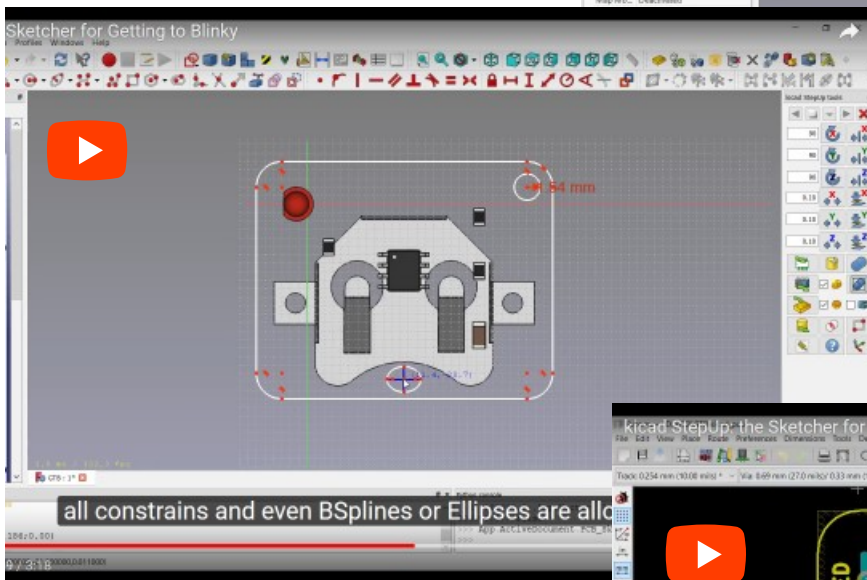
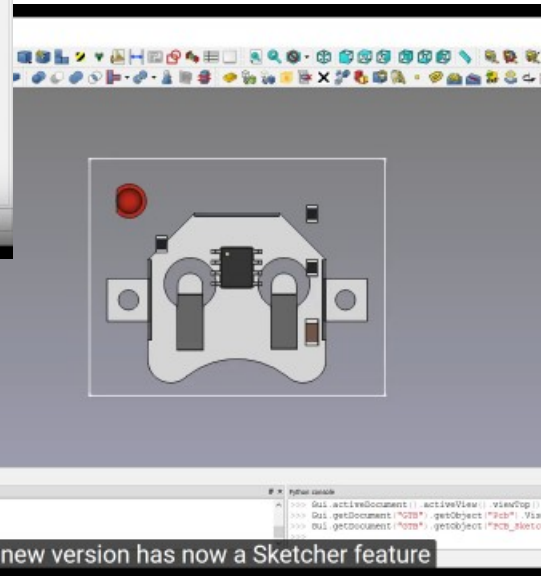
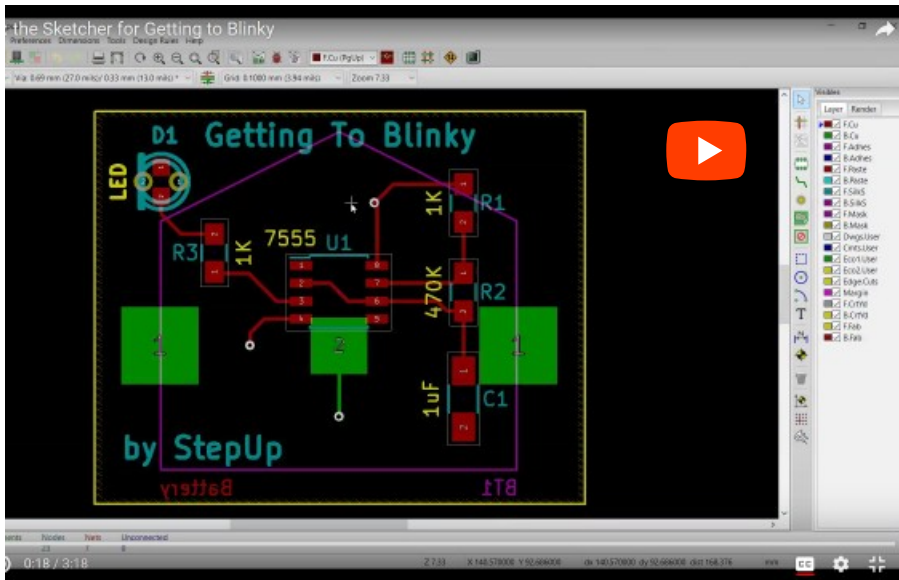
<https://sourceforge.net/projects/kicadstepup/>

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



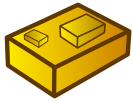
The Sketcher tutorial

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

tips

It is suggested to configure '*ksu-config.ini*' file to use **grid origin** and **place a grid origin** to kicad_pcb file

[Placement]
placement = usegridorigin



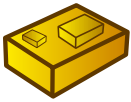
KiCad StepUp tools cheat sheet

<https://sourceforge.net/projects/kicadstepup/>

Configuration file main parameters

ksu-config.ini

```
;utf-8 coded: do not edit this line
[info]
;; kicad stepup tools config file
;; each line starting with a semicolon is a comment
[prefix3D]
;; put here your kisys3dmod path or 3d model prefix path or 3d alias
;; only two prefixes are allowed; must finish with slash or backslash
prefix3d_1 = C:\Program Files\KiCad\share\kicad\modules\packages3d\
prefix3d_2 = D:\extra3Dpackages
[PcbColor]
;; pcb color r,g,b e.g. 0.0,0.5,0.0,light green
pcb_color = 0.0,0.5,0.0,light green
[Blacklist]
;; put here your model names that you don't want to load (e.g. smallest ones)
;; separated by a comma (none means all the models will be parsed)
;; (volume=1 means all models with a volume < 1mm³ will not be included)
;; (height=1 means all models with a height < 1mm will not be included)
bklist = none
[BoundingBox]
;; bounding box option list=>whitelist (not converted to bbox)
;bbox = list dpak-to252,sod80
;bbox = all
;bbox = off default
bbox = off default
[Placement]
;; placement options
;placement options: useauxorigin, usebaseorigin, usebasepoint;x;y, usedefault, +autoadjust
placement = usegridorigin
[Virtual]
;; virtual modules to be or not added to board
virt = addVirtual
[ExportFuse]
;; fuse modules to board
;; be careful ... fusion can be heavy or generate fc crash with a lot of objects
;; please consider to use bbox or blacklist small objs
exportfusing = nofuse #default
[minimum_drill_size]
;; minimum drill size to be processed in mm
;; set 0.0 to process all sizes
min_drill_size = 0.0
[export]
;; export to step
export_to_step = no
[Materials]
;; VRML models to be or not exported with material properties
mat = enablematerials
[compound]
;;allow compound for STEP models
compound = allowed
[sketch_constraints]
;;constraints generated for pcb sketch
constraints = all
```



KiCad StepUp tools cheat sheet

<https://sourceforge.net/projects/kicadstepup/>

StepUp Credits

kicad StepUp tools author is Maurice <https://sourceforge.net/projects/kicadstepup/>

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

