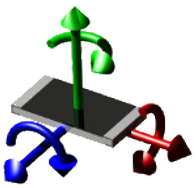

Printed Circuit Board Workbench for FreeCAD (PCB)
Flexible Printed Circuit Board Workbench for FreeCAD (FPCB)

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http://sourceforge.net/projects/eaglepcb2freecad/

CONTENTS

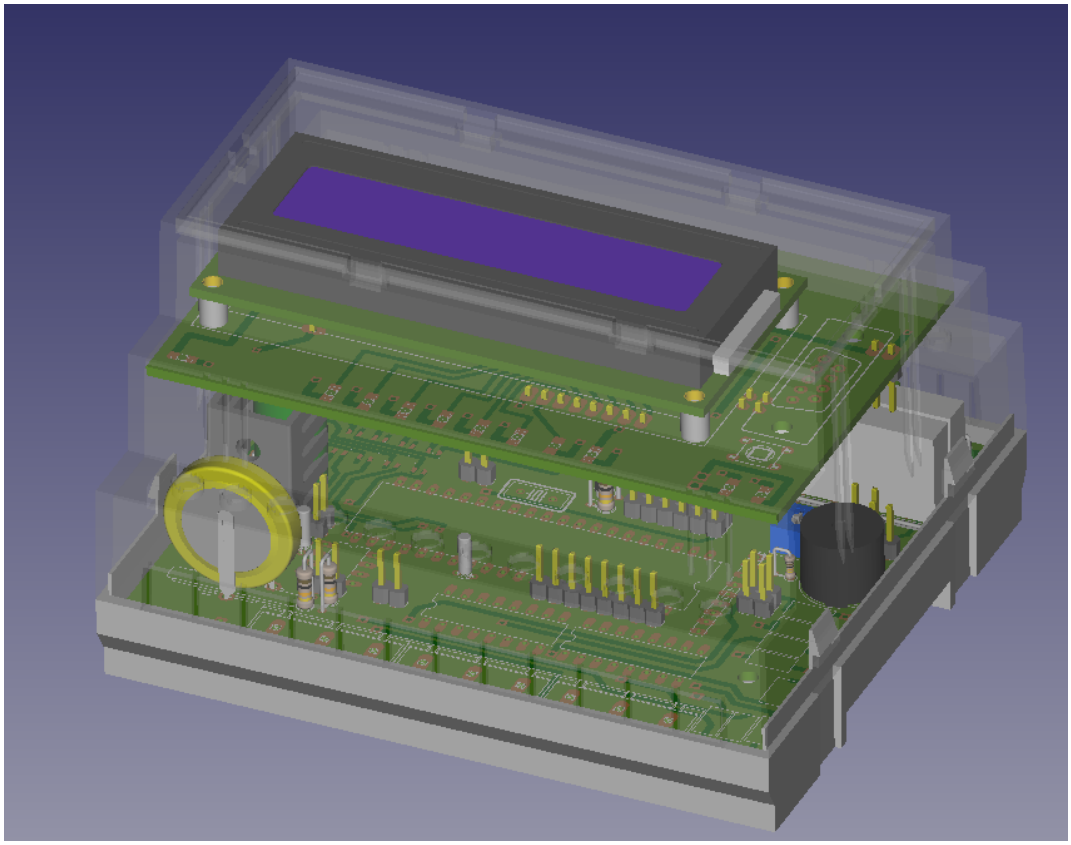
INTRODUCTION

[ENG]
Mod allow you to import/create PCB boards in FreeCAD. Scope of mod:

- support for many different layers,
- possible to choose colours, transparency and names for each layer,
- mod allows you to import IGES/STP models with colours,
- possible to show holes/vias independent.

[PL]
Moduł pozwala na importowanie/tworzenie płytek PCB w programie FreeCAD. Możliwości modułu:

- wsparcie dla wielu różnych warstw,
- wyświetlanie otworów, przelotek niezależnie od siebie,
- możliwość wyboru koloru, przeźroczystości oraz nazwy dla poszczególnych warstw,
- importowanie modeli zapisanych w formacie IGS/STP wraz z kolorami.



Requirements

FreeCAD-PCB require FreeCAD in version 14.0 or never. Module was tested on Windows and GNU/Linux.

Supported files

- Eagle (*.brd),
- Razen (*.rzp),
- FreePCB (*.fpc),
- gEDA (*.pcb),
- FidoCadJ (*.fcd),
- KiCad (*.kicad_pcb),
- IDF v2/v3,
- HyperLynx (*.HYP).

INSTALLATION

Unpack downloaded zip file and copy extracted folder to direction where FreeCAD is installed (subfolder Mod).

GNU/Linux

Example:

FreeCAD path:

```
/Programs/FreeCAD
```

So copy mod to folder

```
/Programs/FreeCAD/Mod
```

You can also copy files to folder /.FreeCAD/Mod.

Next change read/write permission to 777. Please don't forget about parameter -R!
Example:

```
chmod 777 -R PCB
```

Windows

Example:

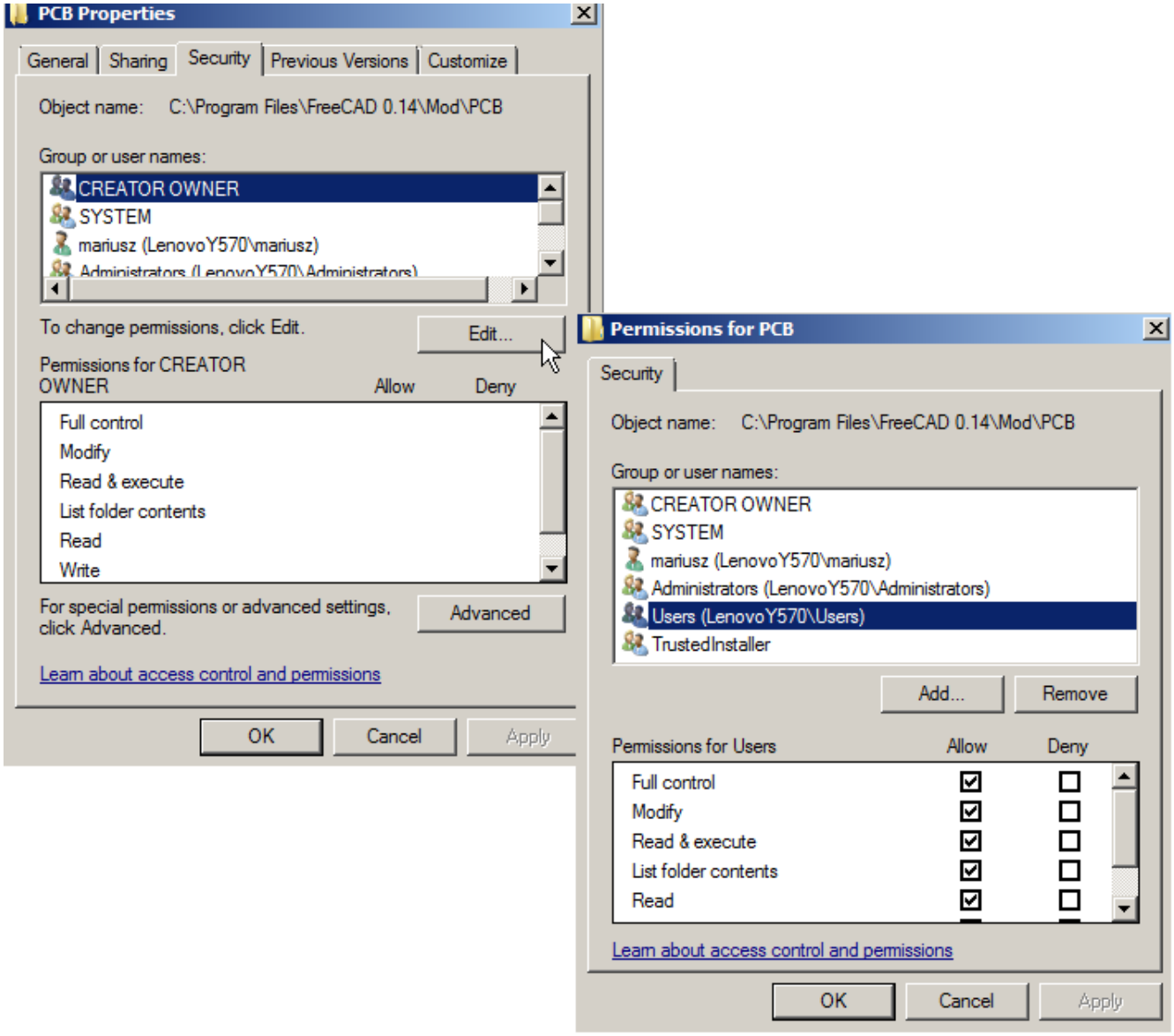
FreeCAD path:

```
C:/Program Files/FreeCAD-0.14
```

So copy mod to folder

```
C:/Program Files/FreeCAD-0.14/Mod
```

Next change read/write permission for all users. Click right button on folder PCB and choose Properties → Security → Edit → Users and mark all checkboxes under 'Allow' option.



CONFIGURATION

At this moment some settings need to be configured in file PCBconf.py. You can open this file in any text editor (please avoid Notepad).

STP file format colors definition

During loading board, You can meet with error connected with missing STP color definition. To fix that problem just add new color definition in PCBconf.py file in spisKolorowSTP variable.

For example:

Missing color name:
red

Actual situation:

```
spisKolorowSTP = {
    "white": (1.0, 1.0, 1.0),
    "black": (0.0, 0.0 ,0.0)
}
```

Write to file:

```
spisKolorowSTP = {
    "red": (1.0, 0.0 ,0.0),
    "white": (1.0, 1.0, 1.0),
    "black": (0.0, 0.0 ,0.0)
}
```

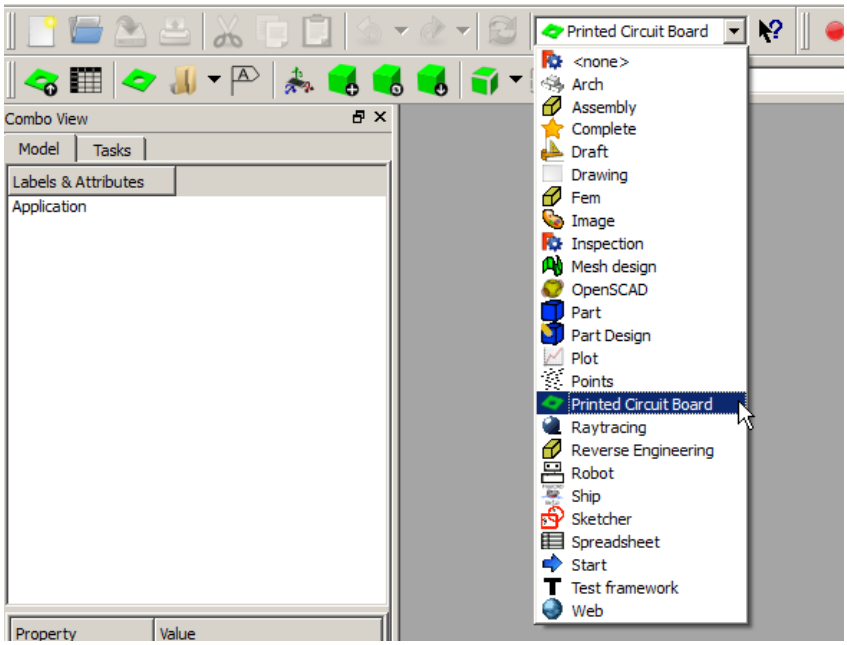
Where:
"red": (1.0, 0.0 ,0.0), == "colorName": (R / 255, G / 255, B / 255)

Please do not change anything else in the PCBconf.py file! More configuration options You can find in Customizing Workbench section.

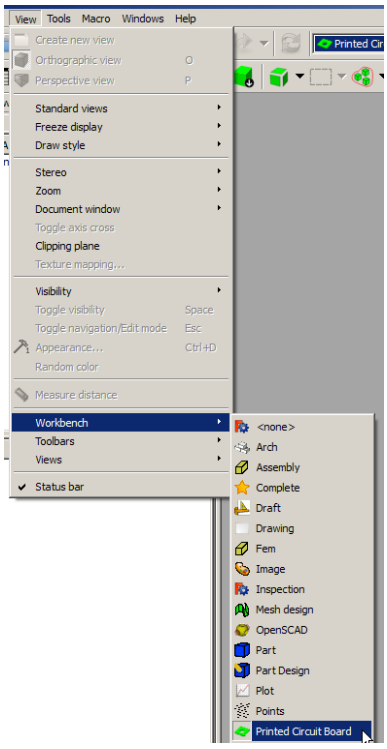
ACCESSING THE WORKBENCH

There are two methods to access to the PCB workbench:

1. In toolbar 'File' locate drop down list and choose 'Printed Circuit Board'.

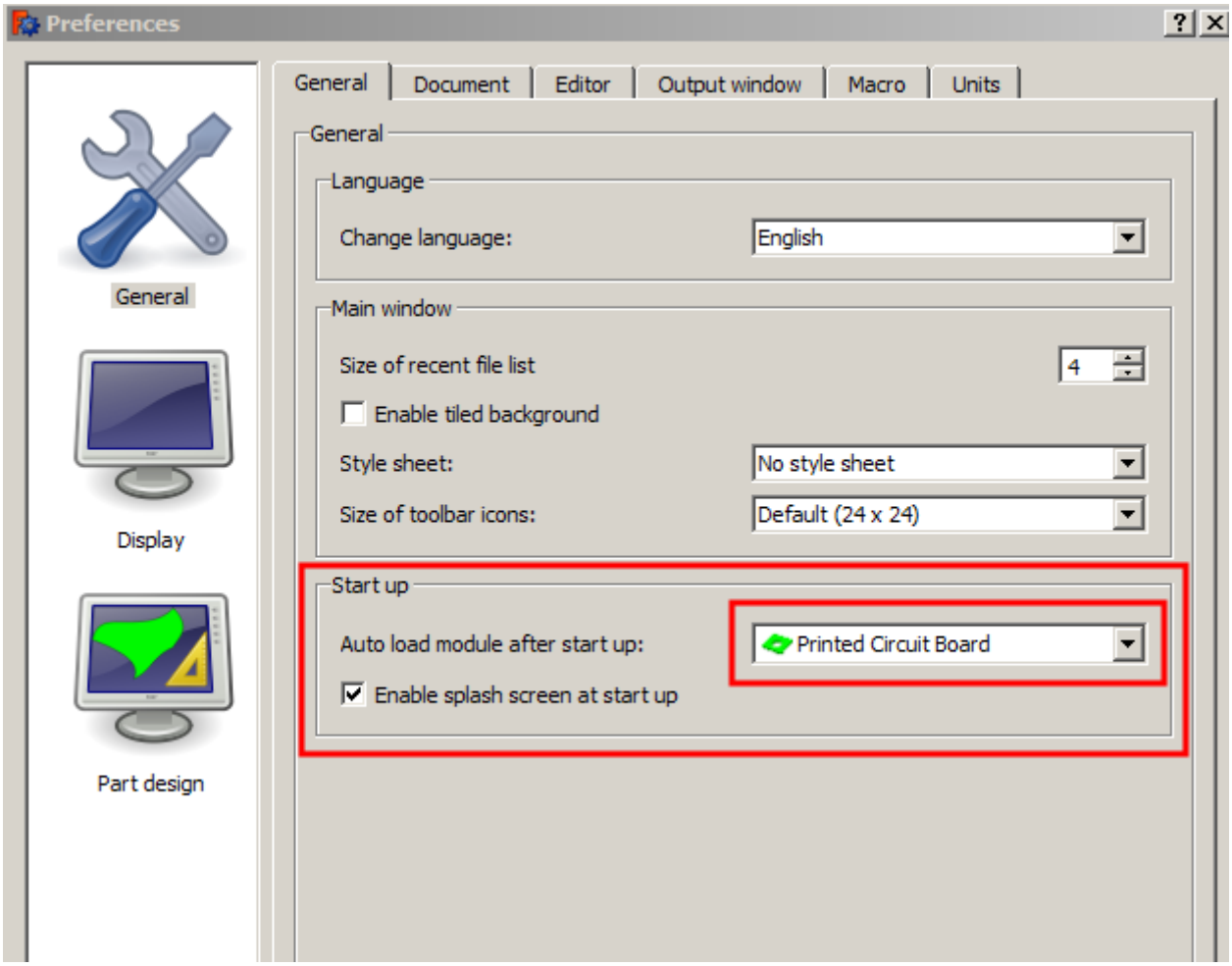


2. From top menu bar choose View → Workbench → Printed Circuit Board.



Set PCB module as main workbench

There is possibility to set PCB module as main workbench. To do this choose from top menu bar Edit → Preferences, in settings window choose General and tab General. In displayed tab You should find 'Start up' section, where You can set which workbench should be loaded after FreeCAD start.



MENU BAR

Menu bars are not available.

TOOLBARS

Three toolbars are available in PCB workbench:

1. PCB View.
2. PCB Settings.
3. Sketcher.

This section describes the various icons available in mentioned toolbars.

PCB View toolbar

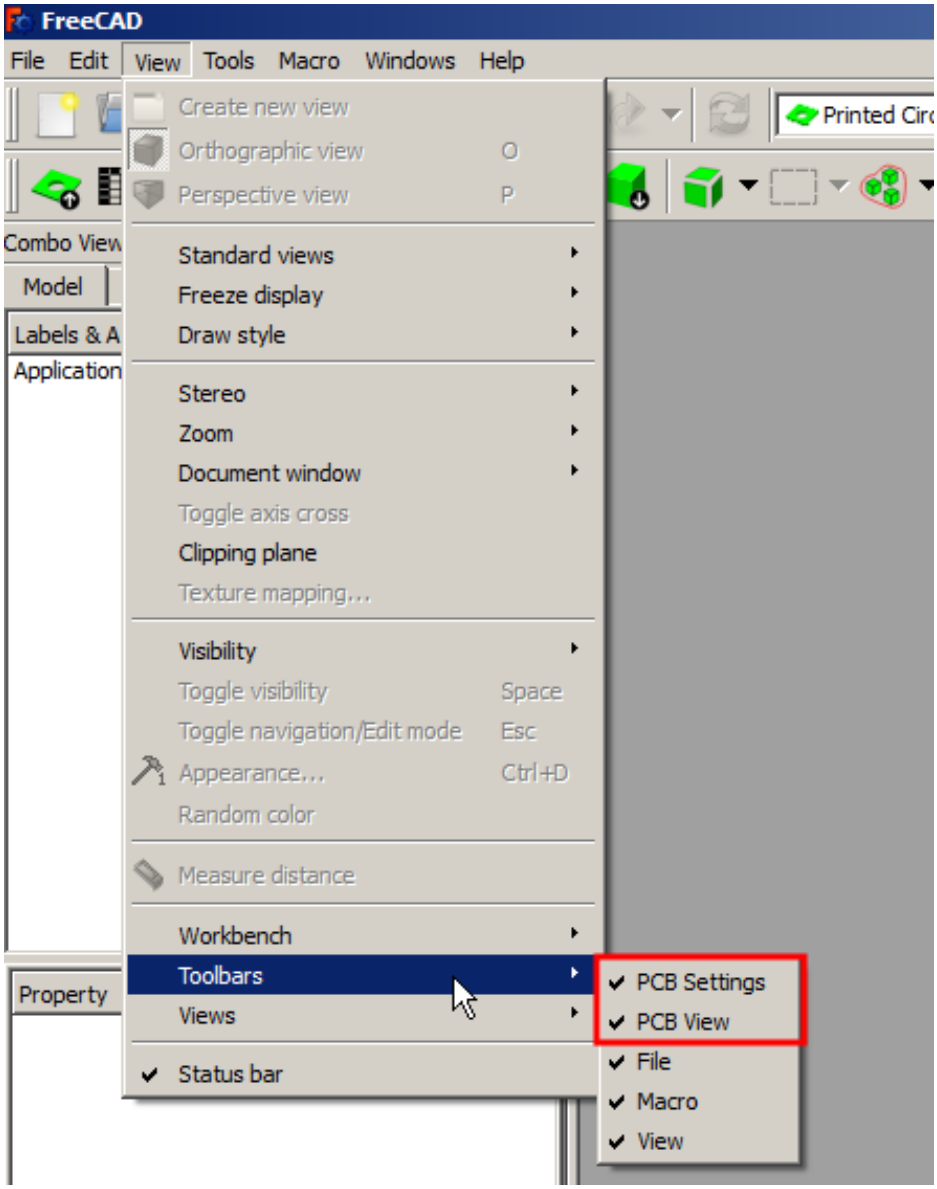
	Change display mode to Shaded	See Display modes section
	Change display mode to Flat Lines	See Display modes section
	Change display mode to Wireframe	See Display modes section
	Change display mode to Internal View	See Display modes section
	Layers settings	See Layers section
	Cut to board outline	See Cut to board outline section
	Ungroup models in 'Parts' folder	See Grouping parts section
	Group models in 'Parts' folder	See Grouping parts section
	3D rendering: export to Kerkythea	See Kerkythea section
	Load file as assembly	See Add assembly section
	Update selected assemblies	See Update assembly section

PCB Settings toolbar

Export PCB	See Export board section
Export BOM	See Export Bill Of Materials (BOM) section
Export hole locations	See Export hole locations section
Export hole locations report	See Export hole locations report section
Create drilling map	See Create drilling map section
Create drill center	See Create drill center section
Create new project	See Create new project section
Create PCB	See Create PCB section
Create glue path	See Create glue path section
Add annotation	See Add annotation section
Assign models folder	See Assign models section
Add model	See Add model section
Update models	See Update models section
Download models	See Download models section
Explode	See Explode section
Create constraint area	See Create constraint area section
Bounding box	See Bounding box section

Displaying toolbars

When mentioned toolbars are not displaying after choosing PCB workbench in main FreeCAD window, You need to do it manually. From top menu bar choose View → Toolbars and mark toolbars from Printed Circuit Board workbench.



SPECIFICATION TREE

There are few object types directly connected with PCB workbench. They can be identified in the 'Combo view' by the specific icons.


Board


Constraint area


Explode


Layer


Part model found in database

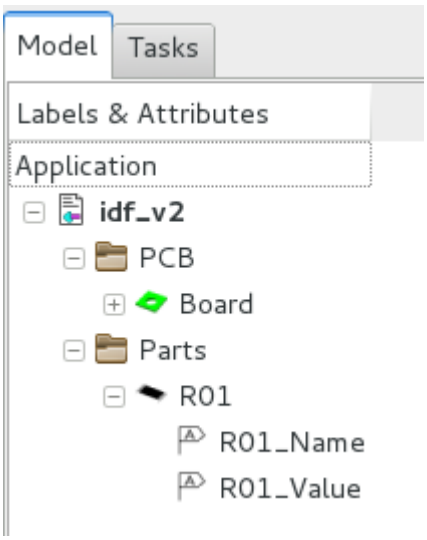

Part model not found in database


Annotation/Object Name/Object Value


Glue path


Main assembly object


Main assembly subcomponent

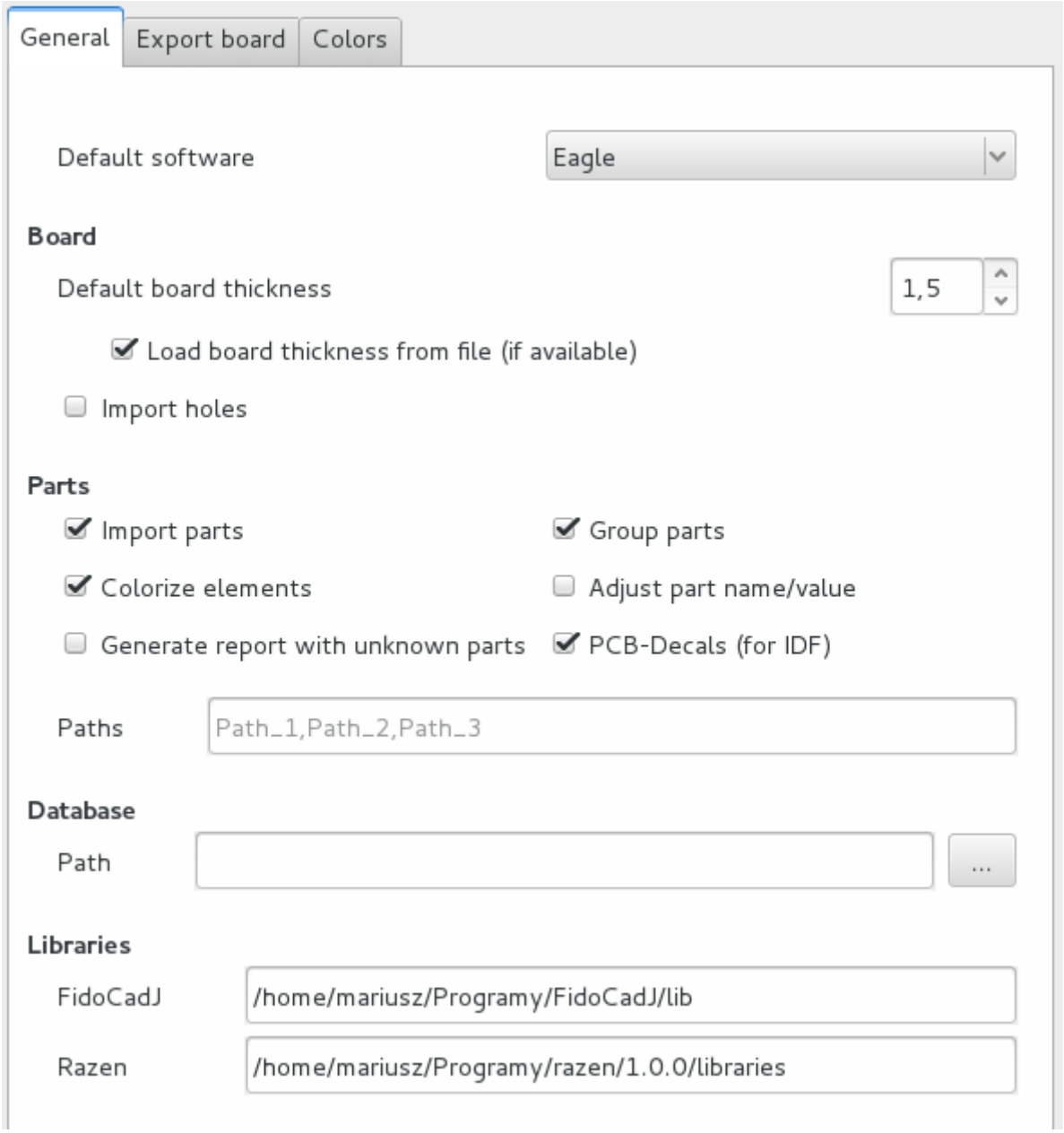


More info about mentioned objects You can find in Objects properties section.

CUSTOMIZING WORKBENCH

To access to the PCB workbench settings You need to choose from top menu Edit → Preferences: section PCB. Preference tab for module contain three groups:

1. General



This section contains default settings for import process:

- Default software: this field allow You to set default used by you software,
- Board thickness: default value is 1.5mm,
- Paths to: database, 3D models, extra libraries,
- Checkboxes associated with importing parts/colors/holes,

- Checkbox associated with generating report with unknown parts.

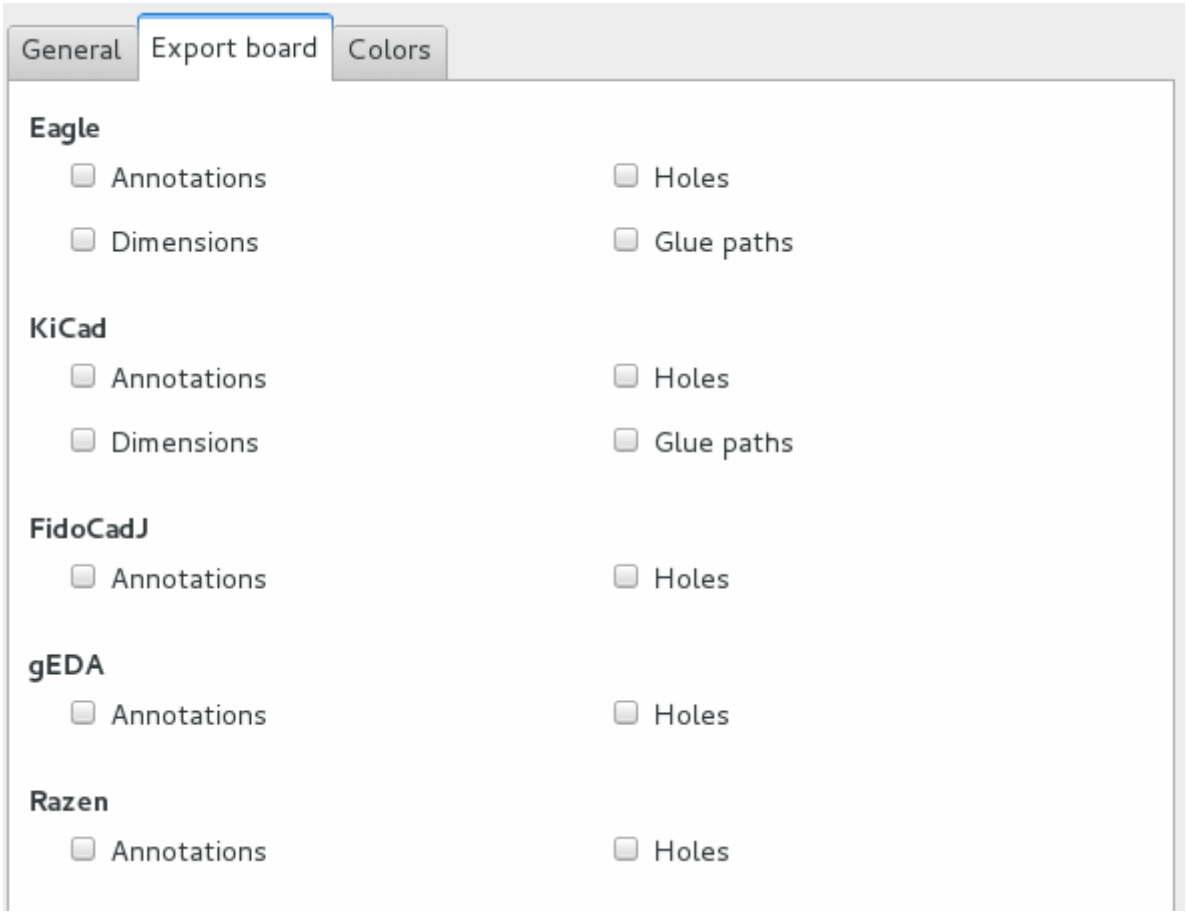
If checkbox 'Group parts' is checked, imported parts will be splitted to groups according to Category they belong.

For more information about grouping parts see 'Grouping parts' section.

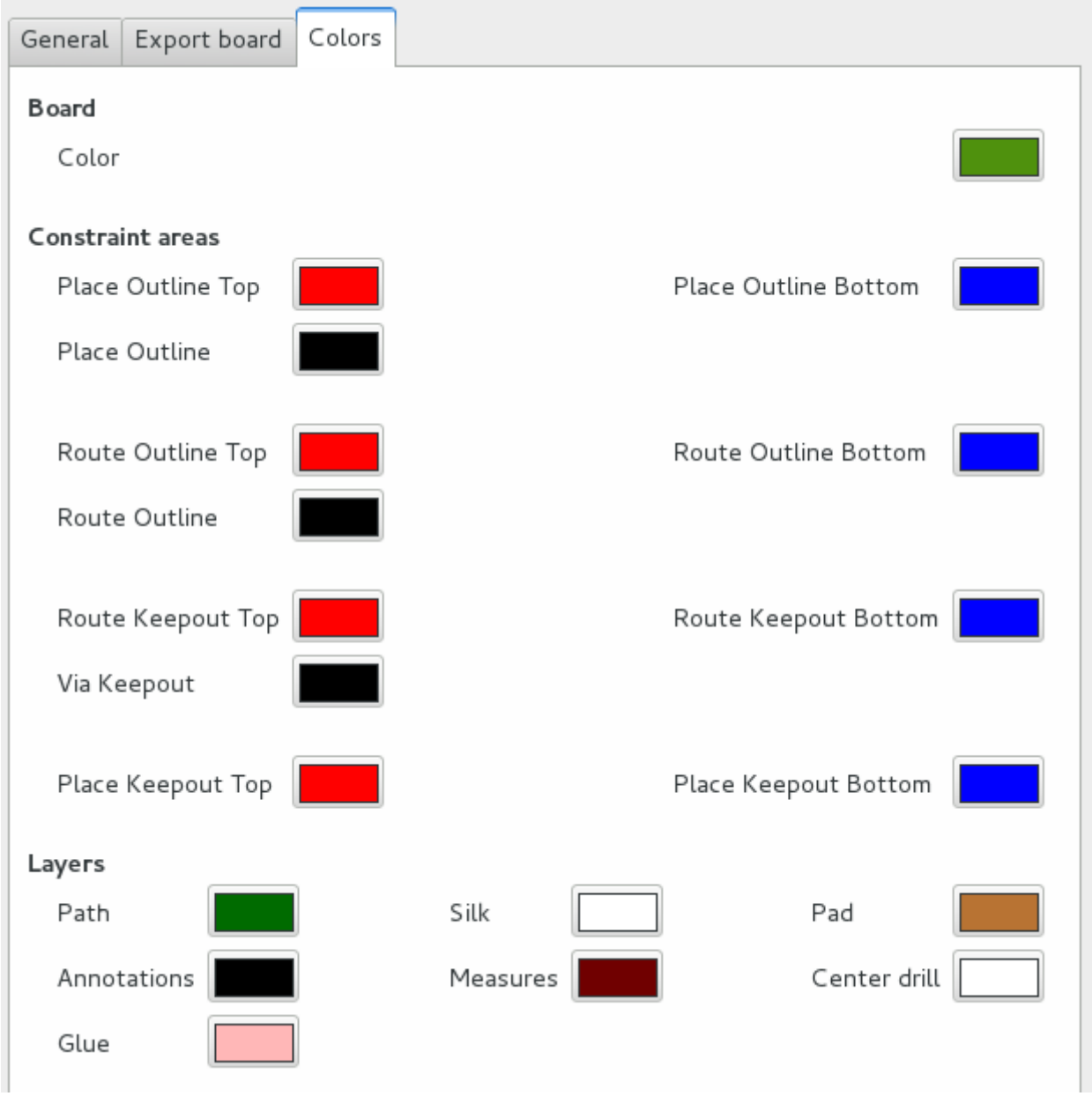
It is recommended to keep parts and database.cfg outside PCB folder.

To set libraries for FidoCadJ you can indicate folder or main jar file.

2. Export board Default settings associated with exporting board to one of supported formats can be set in this tab.

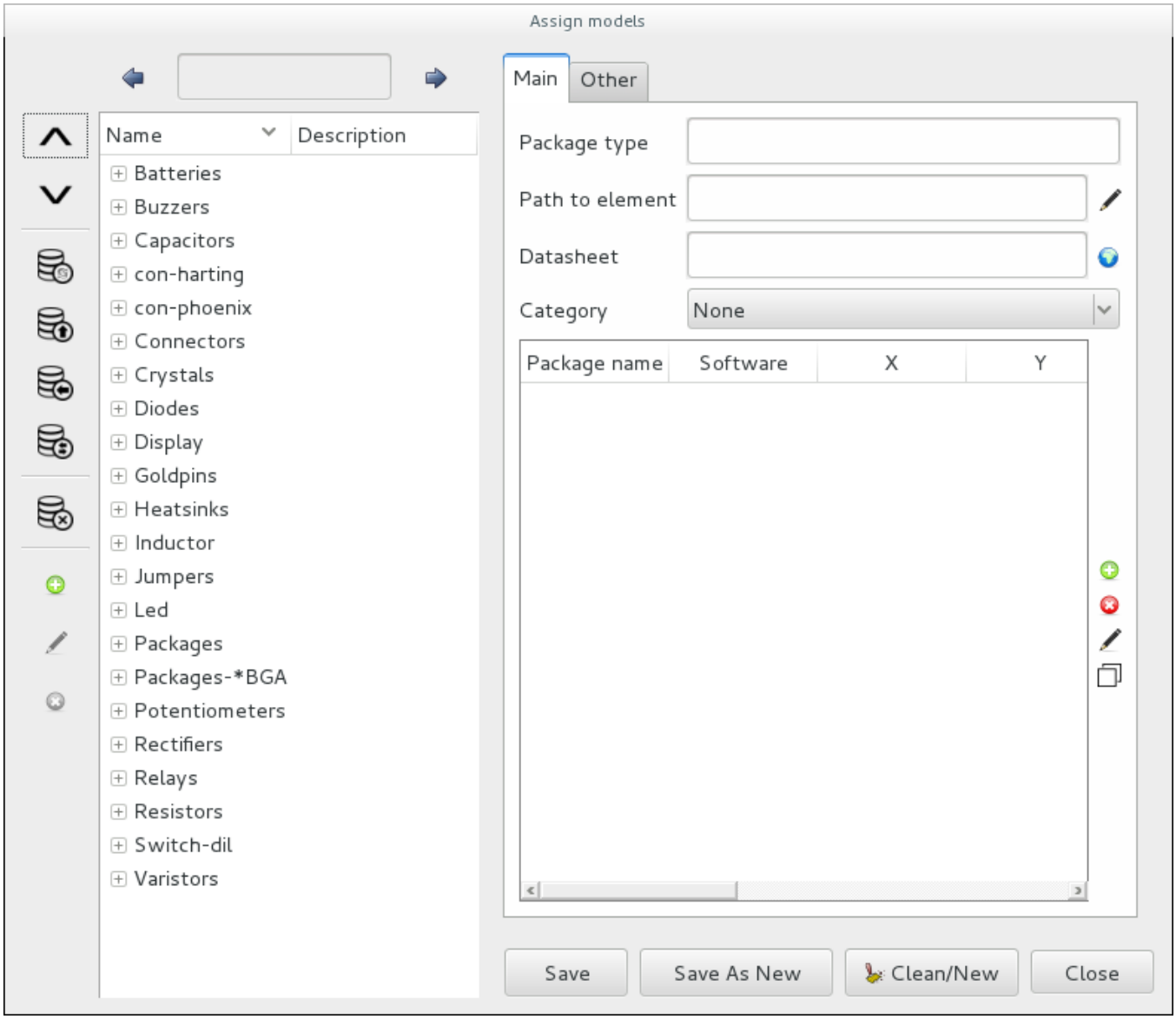


3. Colors All default color can be set in 'Colors' section.


General
Export board
Colors
Board
Color
Constraint areas
Place Outline Top
Place Outline Bottom
Place Outline
Route Outline Top
Route Outline Bottom
Route Outline
Route Keepout Top
Route Keepout Bottom
Via Keepout
Place Keepout Top
Place Keepout Bottom
Layers
Path
Silk
Pad
Annotations
Measures
Center drill
Glue

ASSING MODELS

Window 'Assign models' allow for assigning 3D models to corresponding part from one of supported software.



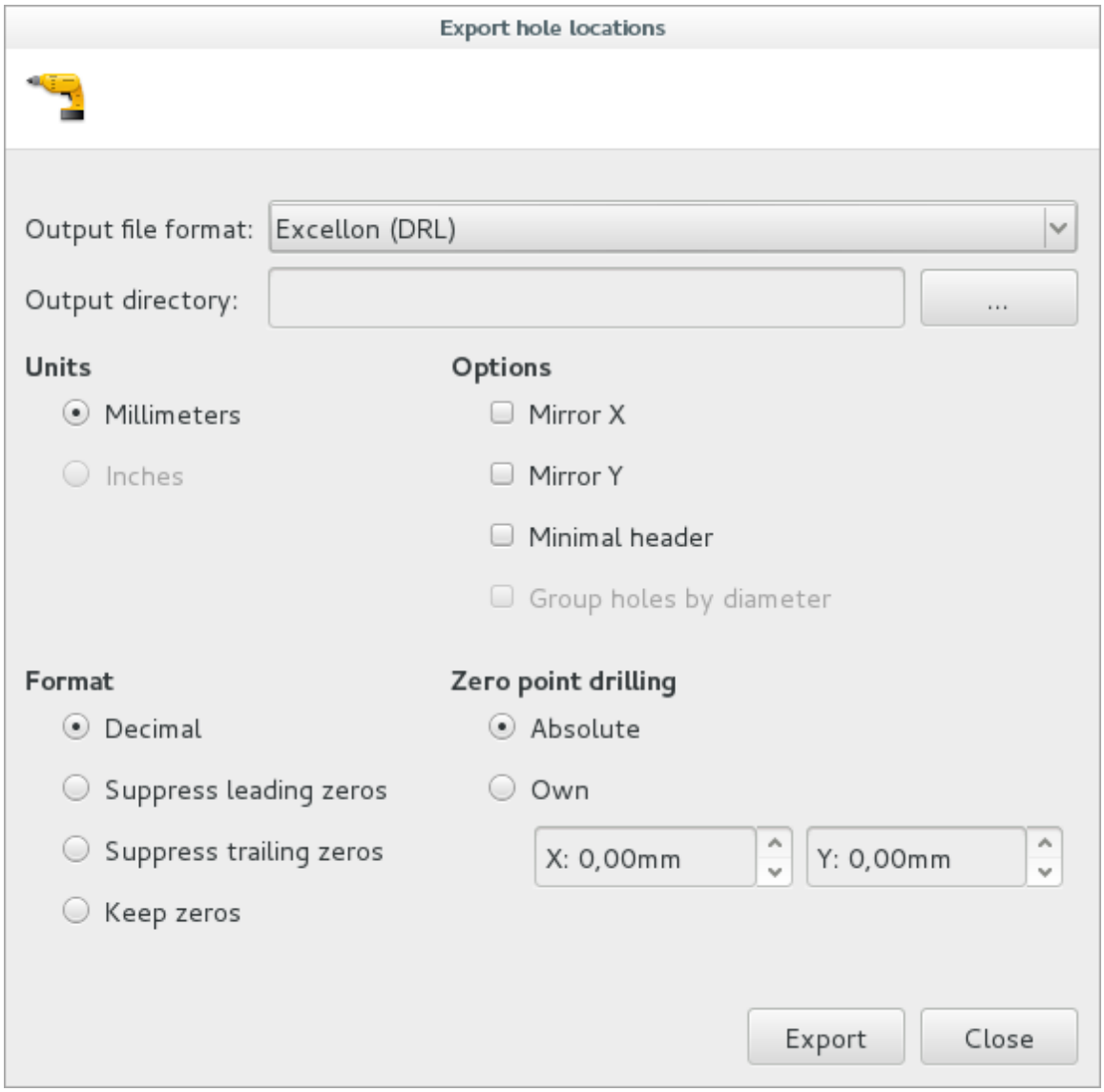
Window comprises two main columns. Left column comprises function necessary to manage parts in database. While right side comprises form where You can set (or edit) package data.

Categories are stored directly in FreeCAD.
Tools → Edit parameters =¿ Preferences → Mod → PCB and variable partsCategories

After deleting model from database it is not possible to undo this operation!

EXPORT HOLE LOCATIONS

Option Export hole locations allow You to export holes list to one of supported file formats.



Export hole location contains a number of settings that allow you to obtain the desired output file format:

1. Output file format:
   - Comma Separated Values (*.csv)
   - Text File (*.txt)
   - HyperText Markup Language (*.html)
   - Excellon (DRL)

2. Output directory: set path where file will be saved
3. Units:
   - Millimeters: measure Everything in Metric , default value
   - Inches: measure Everything in Inches, disabled option
4. Format: choose format, in which values will be saved in file
   Base value: 12.5[mm]
   - Decimal: without changes, value = 12.5
   - Suppress leading zeros: value = 12500
   - Suppress trailing zeros: value = 00125
   - Keep zeros: value = 0012500
5. Zero point drilling
   - Absolute: base point for drilling is set in global 0, 0
   - Own: set new base point for drilling
     - X: X value for new base point for drilling
     - Y: Y value for new base point for drilling
6. Extra options
   - Mirror X: multiply X value by -1
   - Mirror Y: multiply Y value by -1
   - Minimal header: set whether extra data (project name, date, format) will be saved in to output file

```
Drill file
Project: sterownik
Date: 2015-04-25 16:02:37.990862
Unit: mm
Format: Decimal
Zero point drilling: Absolute (0 x 0)
```

   - Group holes by diameter: some output formats support grouping for holes by diameter

```
Diameter        X               Y
0.5             34.3            35.6
0.5             22.5            29.5
0.5             31.7            35.6
1.0             14.9            2.6
1.0             85.2            11.3
1.0             94.31           70.25
1.0             98.09           70.25
1.0             94.31           64.45
1.0             98.09           64.45
1.0             65.61           70.25
1.0             69.39           70.25
1.0             65.51           64.35
1.0             69.29           64.35
3.0             10.0            18.5
3.0             90.0            18.5
```

Diameter	X	Y
0.5		
	34.3	35.6
	22.5	29.5
	31.7	35.6
1.0		
	14.9	2.6
	85.2	11.3
	94.31	70.25
	98.09	70.25
	94.31	64.45
	98.09	64.45
	65.61	70.25
	69.39	70.25
	65.51	64.35
	69.29	64.35
3.0		
	10.0	18.5
	90.0	18.5
0.8		
	97.4	27.6
	97.5	23.2

Python

To export holes list by Python You need to make few basic steps:

1. Import PCBexportHoles module.
2. Set basic export parameters.
   Available settings:
   - fileFormat:
     - Comma Separated Values → csv
     - Text File → txt
     - HyperText Markup Language → html
     - Excellon → drl, def. value
   - filePath = def. home directory
   - fileName = def. value 'untitled'
   - units = mm/inch
   - saveFormat:
     - -2: Decimal, def. value
     - -3: Suppress leading zeros
     - -4: Suppress trailing zeros
     - -5: Keep zeros
   - zeroPointDrilling:
     - -2: Absolute, def. value
     - -3: Own
       * `zeroPointDrilling_X` = def. value 0

        * `zeroPointDrilling_Y` = def. value 0
    - `mirror_X` = True/False, def. value False
    - `mirror_Y` = True/False, def. value False
    - minimalHeader = True/False, def. value False
    - groupHoles = True/False, def. value False
3. Call function export().

Example:

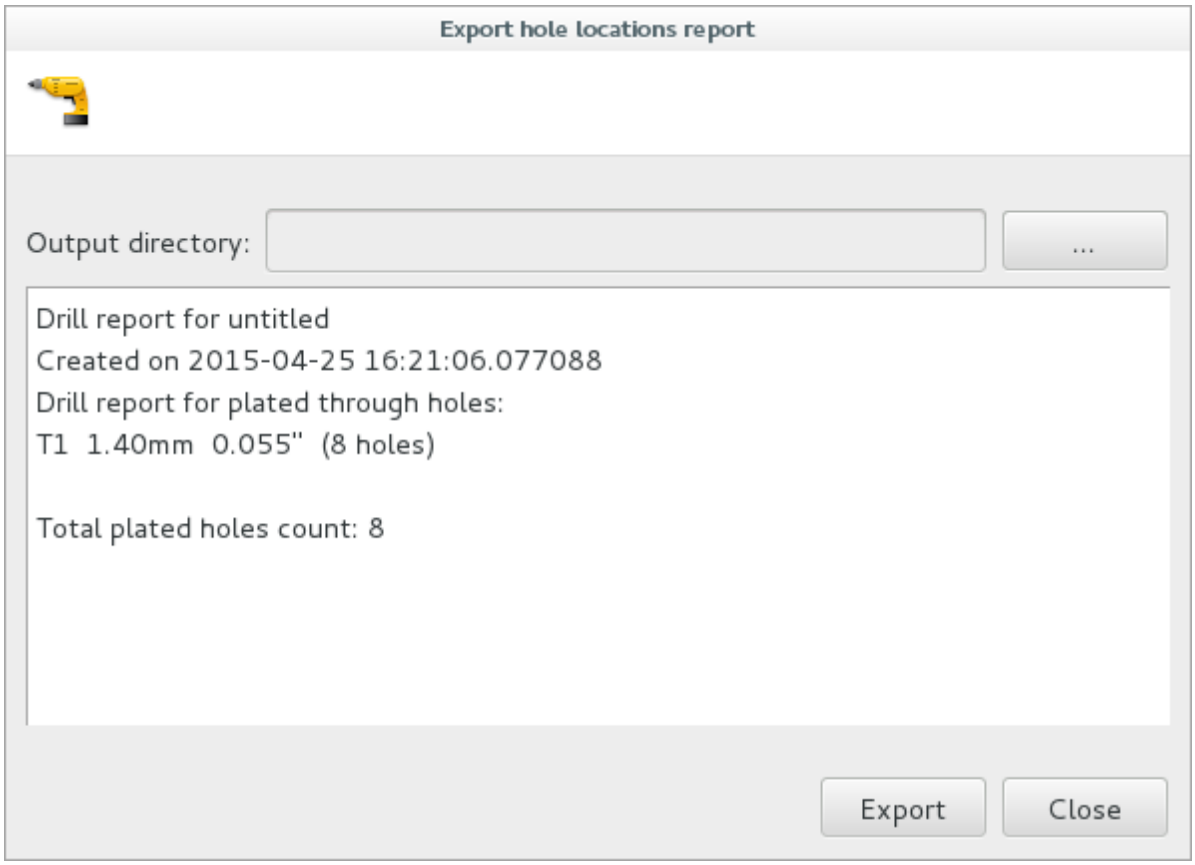
```
from PCBexportHoles import exportHoles
export = exportHoles()
export.fileFormat = 'html'
export.filePath = '/home/mariusz'
export.fileName = 'test'
export.units = 'mm'
export.saveFormat = -2
export.zeroPointDrilling = -2
export.groupHoles= True
export.export()
```

Where:

- /home/mariusz: path for output file,
- test: output file name (file extension is not required).

EXPORT HOLE LOCATIONS REPORT

Option Export hole locations report allow You to export report about needed, for drill process, tools.



Output file have 'rpt' extension.

Python

To export board by Python You need to make few basic steps:

1. Import PCBexportHoles module.
2. Set basic export parameters. Available settings:
   - filePath = def. home directory
   - fileName = def. value 'untitled'
3. Call function export().

Example:

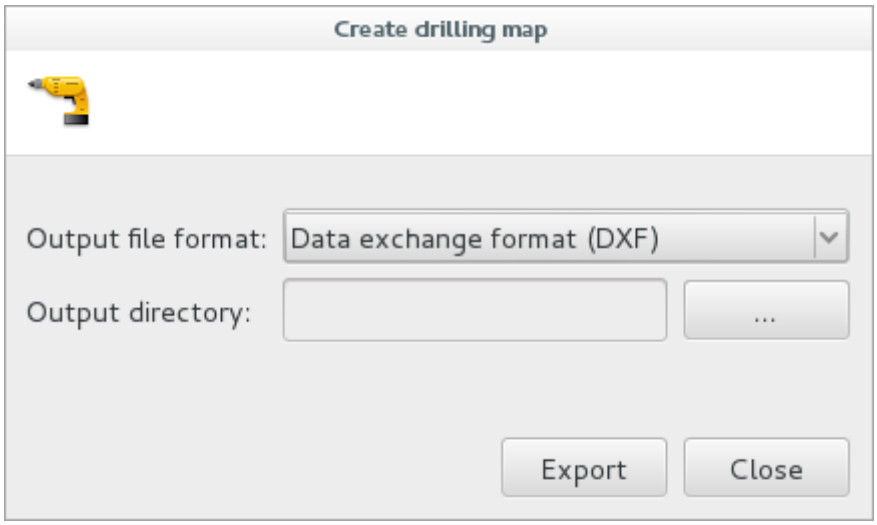
```
from PCBexportHoles import exportHolesReport
export = exportHolesReport()
export.filePath = '/home/mariusz'
export.fileName = 'test'
export.export()
```

Where:

- /home/mariusz: path for output file,
- test: output file name (file extension is not required).

CREATE DRILLING MAP

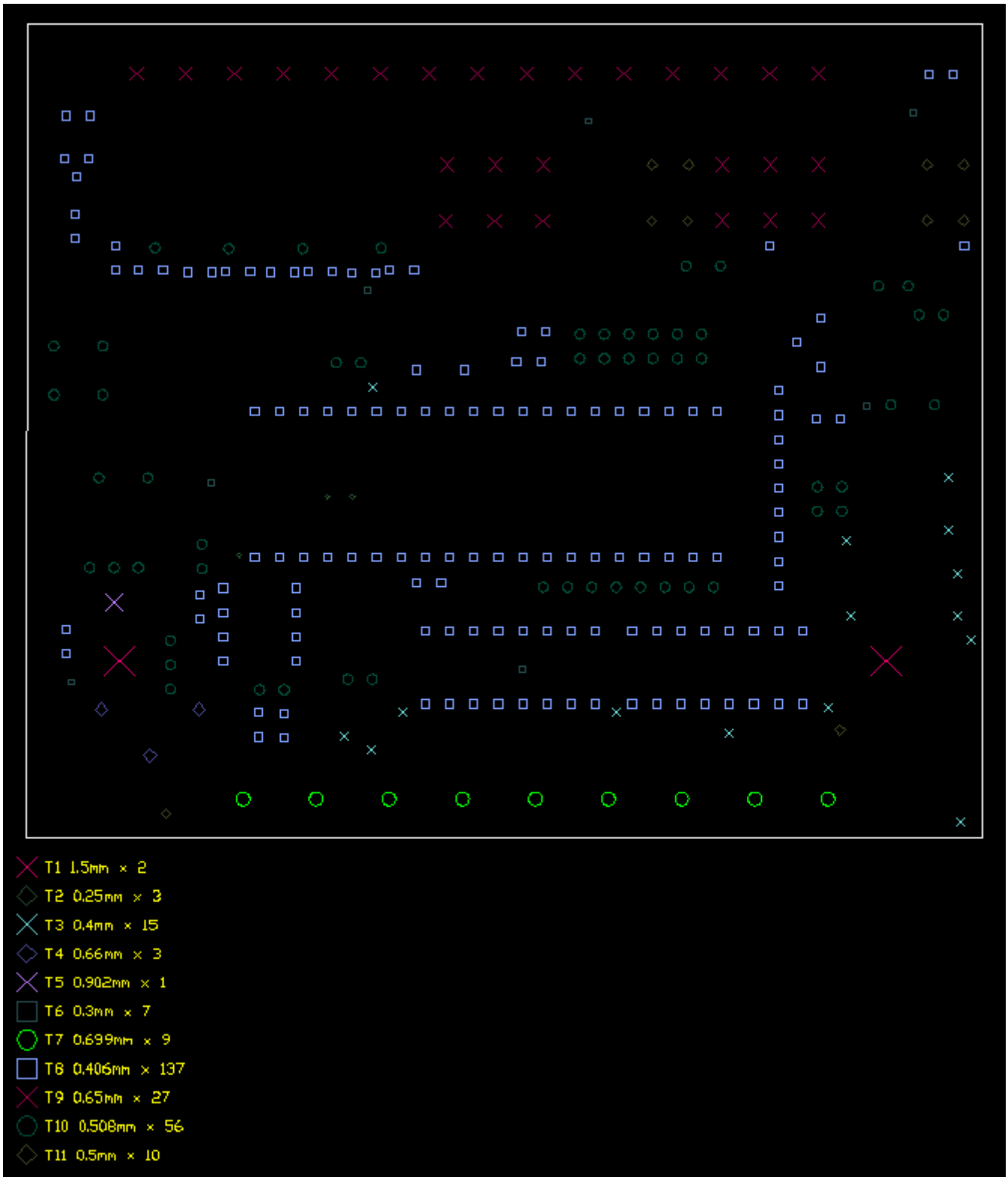
Option 'Create drilling map' allow You to create 2D representation of board with marked drilling points. Holes are splitted by diameter – each diameter value is represented by different symbol and color.



Supported formats:

- DXF: Data exchange format (*.dxf),
- SVG: Scalable Vector Graphics (*.svg).

Output file name is the same as project in FreeCAD.
File extension depends from selected output format.



Format in the legend: T1 1.5mm x 2

- T1: tool number,
- 1.5mm: hole dimater in [mm],
- 2: number of holes with same diameter.

Python

To create drilling map by Python You need to make few basic steps:

1. Import PCBexportDrillingMap module.
2. Set basic export parameters.
   Available settings:
   - fileFormat
   - Comma Separated Values → csv,
   - Data exchange format → dxf,
   - filePath = def. home directory
3. Call function export().

Example:

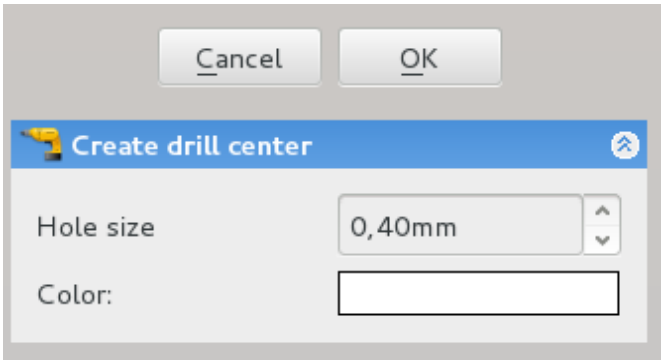
```
from PCBexportDrillingMap import exportDrillingMap

export = exportDrillingMap()

export.filePath = '/home/mariusz'

export.fileFormat = 'dxf'

export.export()
```

Where:

- /home/mariusz: path for output file,
- dxf: output file format.

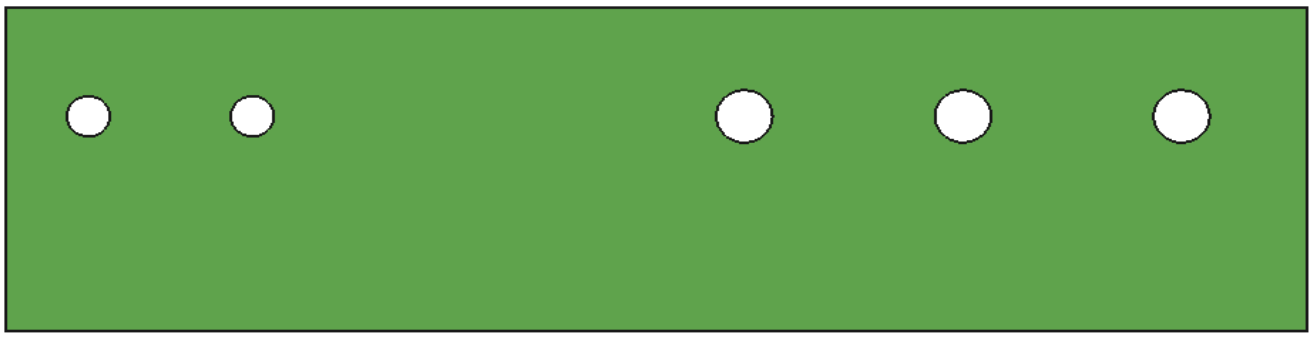
CREATE DRILL CENTER

Option 'Create drill center' is useful for persons which will drill holes in PCB manually. This function allow to decrease holes sizes for better drill position during work.

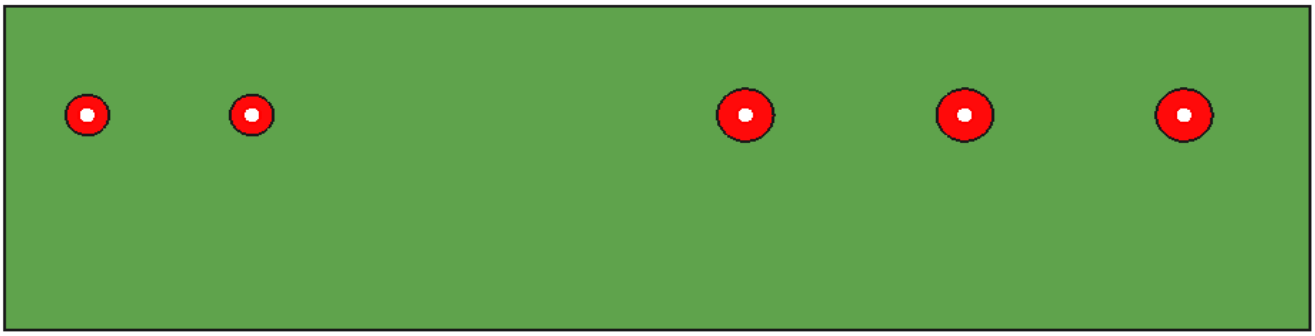


To update/change drill size, use same function. Script will automatically update layer.

Before:



After:



Python

To create drill center by Python You need to make only two steps:

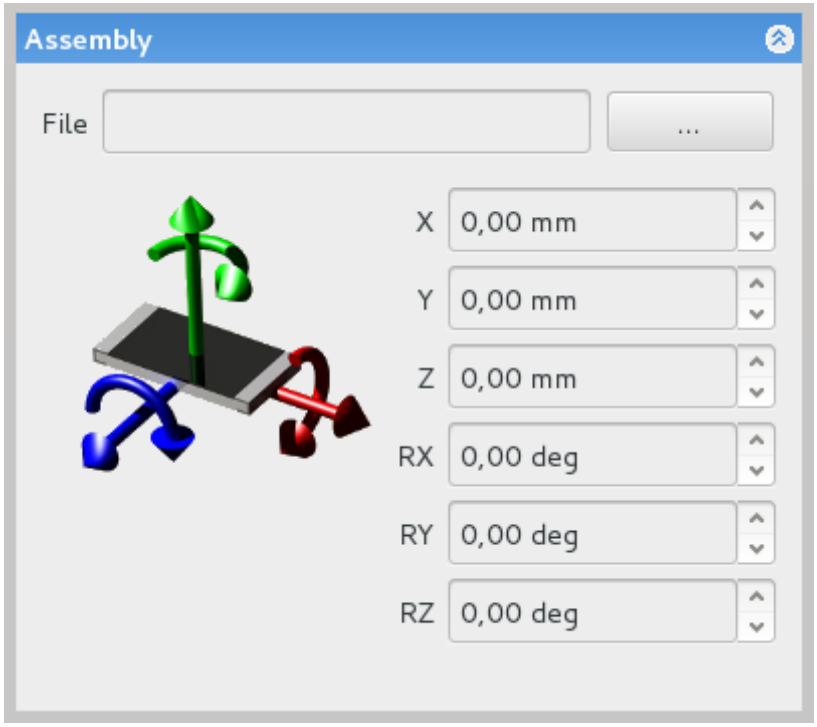
1. Import PCBdrill module.
2. Call function createDrillcenter(size, color).
   Where:
   - size: new hole size in [mm],
   - color: (R / 255, G / 255, b / 255).

Example:

```
from PCBdrill import createDrillcenter

createDrillcenter(0.4, (1, 0.5, 0))
```

ADD ASSEMBLY

dfdgfd



Python

To add assembly object by Python You need to import PCBassembly:

1. Import PCBassembly module.
2. Set basic export parameters.
   Available settings:
   - fileName = full path: destination path + filename,
   - x = position in X direction,
   - y = position in Y direction,
   - z = position in Z direction,
   - rx = rotation value around X axis,
   - ry = rotation value around Y axis,
   - rz = rotation value around Z axis,
3. Call function create().

Example:

```
from PCBassembly import createAssembly

asm = createAssembly()

asm.fileName = '/home/mariusz/dol.fcstd'

asm.create()
```

Where:

- /home/mariusz/dol.fcstd: path and file.

UPDATE ASSEMBLY

These option allow you to update loaded assemblies. Script can recognize if selected to update assembly is currently opened in FreeCAD or not.

Auto update after file loaded does not work at the moment.

During update process, script will keep only main assembly placement in 3D space – all deleted previously objects are reloaded.

At one time you can select and update more than one assembly.

Python

To update assembly object by Python You need to import PCBassembly:

1. Import PCBassembly module.
2. Select assemblies to update.
3. Call function updateAssembly().

Example:

```
from PCBassembly import updateAssembly
asm1 = FreeCAD.ActiveDocument.dol
FreeCADGui.Selection.addSelection(asm1)
updateAssembly()
```

Where:

- asm1: assembly to update.

EXPORT TO KERKYTHEA

OBJECTS PROPERTIES

Each object created in PCB workbench has unique parameters that can be set in Property View (View or Data tab). This task explains meaning each parameter.

Part model not found in database

Part Name: reference to part name object

Part Value: reference to part value object

Keep Position: part will ignore changes in correction values if this value will be set to True

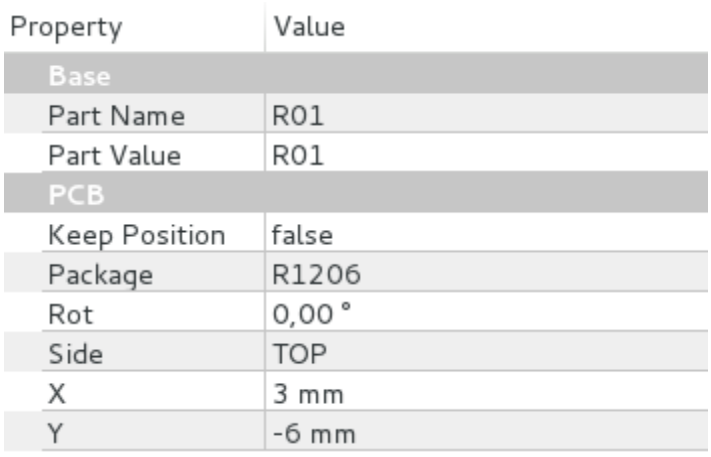
Package: 3D model name, parameter disabled for editing

Rot: rotation value around Z axis, parameter disabled for editing

Side: part position on board (top/bottom side), parameter disabled for editing

X: model position in X direction, parameter disabled for editing

Y: model position in Y direction, parameter disabled for editing.



Property	Value
Base	
Part Name	R01
Part Value	R01
PCB	
Keep Position	false
Package	R1206
Rot	0,00 °
Side	TOP
X	3 mm
Y	-6 mm

Context menu PCB model:

- Assign model: assign 3D model to part,
- Update model: implement new model/correction values for selected part,
- Find model on-line: find 3D model in internet.

Part model found in database

Part Name: reference to part name object

Part Value: reference to part value object

Keep Position: part will ignore changes in correction values if this value will be set to True

Package: 3D model name, parameter disabled for editing

Rot: rotation value around Z axis

Side: part position on board (top/bottom side)

X: model position in X direction

Y: model position in Y direction

Property	Value
Base	
Part Name	R01
Part Value	R01
PCB	
Keep Position	false
Package	R1206
Rot	0,00 °
Side	TOP
X	3 mm
Y	-6 mm

Context menu PCB model:

- Placement model: change correction values for model in 'real time',
- Update model: implement new model/correction values for selected part.

Glue path

Base: reference to sketch that containing glue path shape

Flat: if this parameter is set to True, object will ignore Width/Height parameters

Height: glue path height

Width: glue path width

Property	Value
Base	
⊞ Placement	[(0,00 0,00 1,00); 0 °; (0 mm 0 mm 1,...
Label	Glue_0
Base	Sketch
Flat	false
Height	0,2 mm
Width	1,2 mm

Main assembly object

File: path to *.fcstd file

Placement: position of whole assembly in 3D space

Property	Value
Base	
Label	qora
File	/home/mariusz/Pulpit/qora.fcstd
Placement	[(0,00 0,00 1,00); 0 °; (0 mm 0 mm 0 ...
Angle	0 °
Axis	[0,00 0,00 1,00]
Position	[0 mm 0 mm 0 mm]
x	0 mm
y	0 mm
z	0 mm

Annotation/Object Name/Object Value

Text: text displayed by annotation object

Align: text position according to X, Y values

Mirror: mirror text

Spin: if parameter set to True text will keep rotation, parameter works for angle value ¿= 90deg

Font: font name, parameter disabled

Size: font size

Rot: rotation value around Z axis

Side: text position on board (top/bottom side)

X: text position in X direction

Y: text position in Y direction

Property	Value
Base	
Text	[asdasdsa]
Visibility	true
Display	
Align	botton-left
Mirror	None
Spin	true
Font	
Color	[0, 0, 0]
Font	Hursheys
Size	4,27 mm

Property	Value
Base	
Label	PCBannotation_0000002
Placement	
Rot	0,00 °
Side	TOP
X	0 mm
Y	1,5 mm

Board

Display Holes: turn on/off holes

Holes: reference to sketch that containing holes

Border: reference to sketch that containing board outline

Thickness: board thickness

Property	Value
Base	
Label	Board
Auto Update	true
Holes	
Display	true
Holes	Board
PCB	
Border	Board
Thickness	1,6 mm

Explode

Active: turn of/off explode effect

Bottom Step Size: distance between parts placed on bottom side of board

Inverse: switch exploded parts from top to bottom and conversely

Top Step Size: distance between parts placed on top side of board.

Property	Value
Base	
Part Name	R01
Part Value	R01
PCB	
Keep Position	false
Package	R1206
Rot	0,00 °
Side	TOP
X	3 mm
Y	-6 mm

Context menu Explode:

- Edit: edit list of models which will be exploded.

Constraint area

Height: area height, parameter available only for some constraints areas type

Base: reference to sketch that containing area outline

Property	Value
Base	
Label	tPlaceOutline_0
Height	0,5 mm
Draft	
Base	tPlaceOutline_0

FILE FORMAT

This task explains database.cfg file format.
Each connection between 3D model and component used in ECAD software is stored in mentioned database.cfg file. All parameters can be set by Assign model window and manualy by editing database.cfg file (not recomendet).

Example 3D model setting:

```
[hibhb_8788937480]
socket = [False, 0.0]
description =
add_socket = [False, None]
datasheet =
path = connectors/goldpin/1x08
soft = [[u'1X08', u'Eagle', 0.0, 0.0, 2.77, 90.0, 0.0, 0.0]]
name = 1X08
category = 10
```

Where:

`[unique ID]`

unique ID = String

socket = `[modelIsSocket, socketHeight]`

modelIsSocket = True / False

socketHeight = Float

description = modelDescription

odelDescription = String

add_socket = `[addSocket, socketID]`

addSocket = True / False

socketID = ID / None

datasheet = pathToDatasheet

pathToDatasheet = String

path = pathTo3DModel

pathTo3DModel = String

soft = `[[componentName, softName, X, Y, Z, RX, RY, RZ]]`

componentName = String

softName = String

X = Float

Y = Float

Z = Float

RX = Float

RY = Float

RZ = Float

name = modelName

modelName = String

category = categoryID

categoryID = Integer

SCRIPTS

There are available few scripts which are helping exporting the boards to FreeCAD.

Eagle

Directly exporting boards from Eagle to FreeCAD [path: scripts/eagle]

- scripts/eagle/ulp/freecad.ulp – copy file to $EAGLEDIR/ulp/
- scripts/eagle/scr/freecad.scr – copy file to $EAGLEDIR/scr/
- scripts/eagle/bin/freecad.png – copy file to $EAGLEDIR/bin/

In Eagle choose File → Execute Script → freecad.

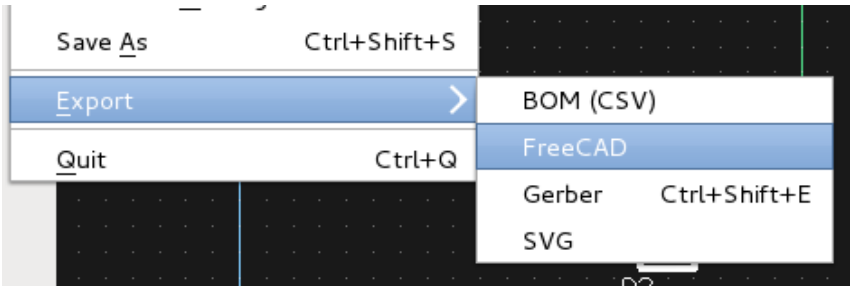
On Linux to set path to FreeCAD change value of var 'programPath_LIN' in file freecad.ulp.
On Windows to set path to FreeCAD change value of var 'programPath_WIN' in file freecad.ulp.

Razen

Directly exporting boards from Razen to FreeCAD [path: scripts/razen]

- scripts/razen/freecad – copy folder 'freecad' to $RAZENDIR/plugin/export/

In Razen choose File → Export → FreeCAD.



On Linux to set path to FreeCAD change value of var 'programPath_LIN' in file conf.cfg.
On Windows to set path to FreeCAD change value of var 'programPath_WIN' in file conf.cfg.

ERRORS CODE

Code	Description	File
1	Function getColorFromIGS()	PCBpartManaging
2	Function partExist()	PCBpartManaging
3	Function reloadList()	PCBaddModel
4	Function deletePackage()	PCBassignModel
5	Function convertDatabase()	PCBassignModel
6	Function reloadList()	PCBassignModel

LICENCE

#***

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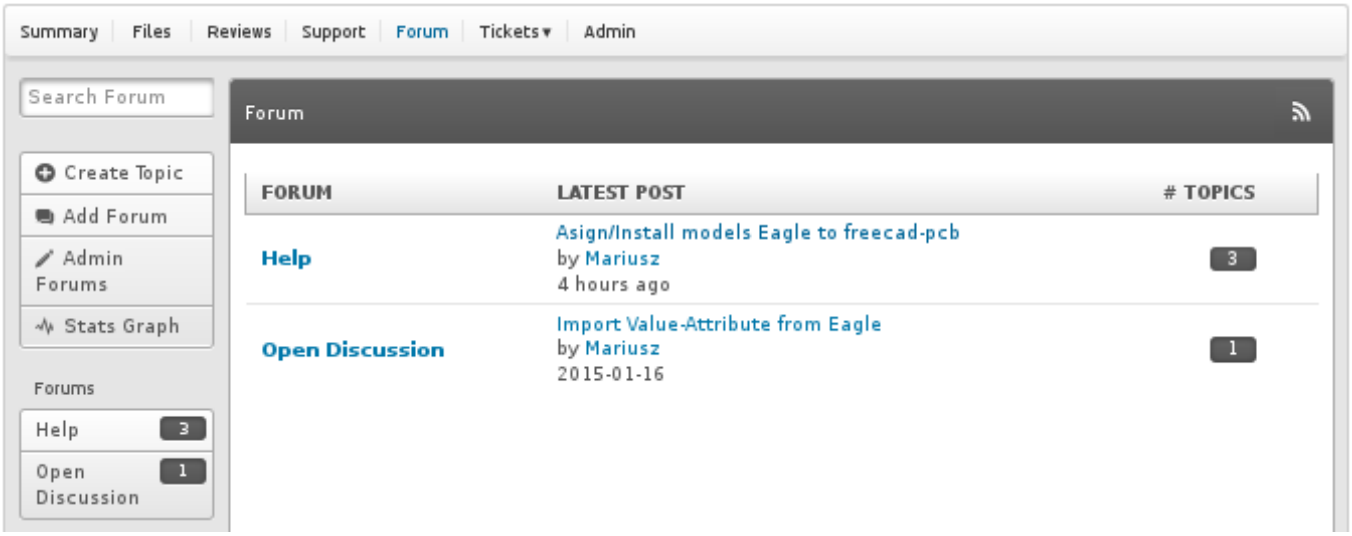
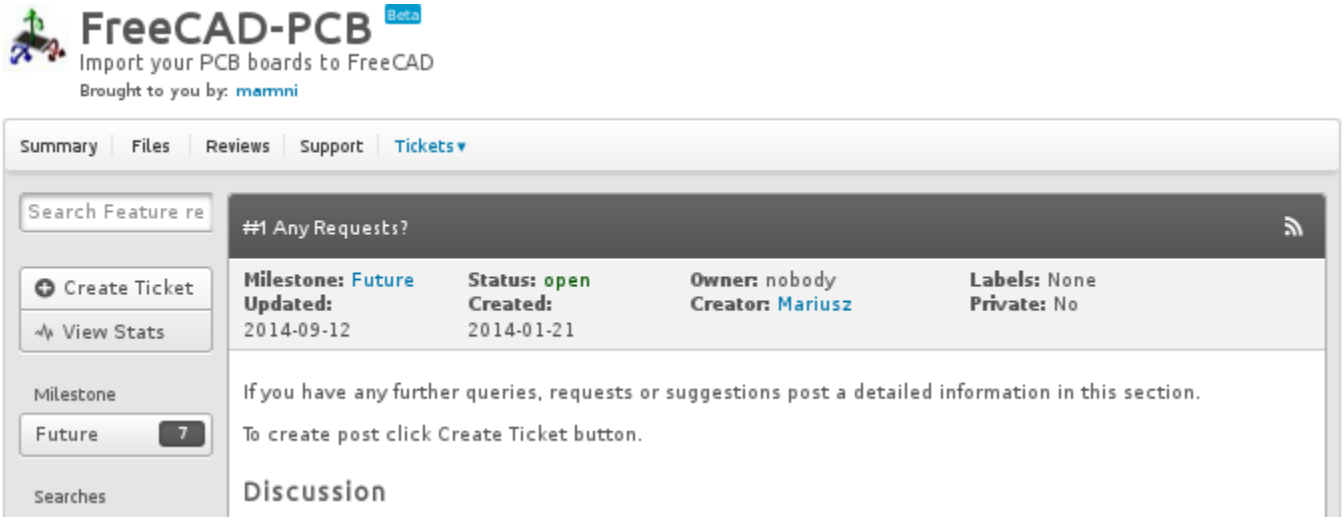
TODO LIST

If you have any further queries, requests or suggestions post a detailed information on project site http://sourceforge.net/p/eaglepcb2freecad/feat-req/ or on forum https://sourceforge.net/p/eaglepcb2freecad/forum/.

To create post on forum click Create Topic button.



To create ticket click Create Ticket button.



ERRORS

Found a bug? Post a detailed information on project page:

http://sourceforge.net/p/eaglepcb2freecad/bugs/

To create post click Create Ticket button.