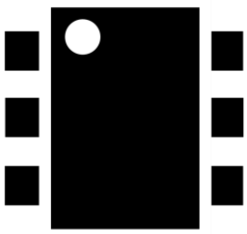


Printed Circuit Board Workbench for FreeCAD (PCB)

marmni (marmni@onet.eu)

Copyright 2013-2021

https://github.com/marmni/FreeCAD-PCB

https://sourceforge.net/projects/eaglepcb2freecad/

https://www.freecadweb.org/

Spis treści

GENERAL INFORMATIONS

LICENCE

#***

#*

#* This program is free software; you can redistribute it and/or modify

#* it under the terms of the GNU Lesser General Public License (LGPL)

#* as published by the Free Software Foundation; either version 2 of

#* the License, or (at your option) any later version.

#* for detail see the LICENCE text file.

#*

#* This program is distributed in the hope that it will be useful,

#* but WITHOUT ANY WARRANTY; without even the implied warranty of

#* MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the

#* GNU Library General Public License for more details.

#*

#* You should have received a copy of the GNU Library General Public

#* License along with this program; if not, write to the Free Software

#* Foundation, Inc., 59 Temple Place, Suite 330, Boston, MA 02111-1307

#* USA

#*

#***

INTRODUCTION

Printed Circuit Board Workbench for FreeCAD. Workbench allows you to:

1. Importing boards created in various dedicated PCB softwares. Layers/colors are supported. Supported softwares:
   - Eagle (*.brd),
   - FreePCB (*.fpc),
   - gEDA (*.pcb),
   - KiCad (*.kicad_pcb),
   - IDF v2/v3.

2. Creating and exporting boards to various formats. Supported formats:
   - Eagle (*.brd),
   - FreePCB (*.fpc),
   - gEDA (*.pcb),
   - KiCad (*.kicad_pcb),
   - IDF v2/v3.

Workbench supports 3D models saved in one of the following formats: STP/IGS

Requirements

FreeCAD-PCB require FreeCAD in version 0.18 (or newer) and Python **2.7** (or newer). Module was tested on Windows and GNU/Linux.

Supported softwares

Soft name	PCB										
		Holes/Vias	Parts	Border	Measures	Soldermask	Keepout layers	Paths	Pads	Soldermask ARC	PCB round corners	Annotations
Eagle	brd	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes
gEDA	pcb	Yes	Never	Yes	Never	Yes	Never	Yes	Yes	Yes	Never	Yes
FreePCB	fpc	Yes	Yes	Yes	Never	Yes	Never	Yes	Yes	Yes	Yes	Yes
KiCad	kicad_pcb	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes
FidoCadJ	fcd	Future	Future	Future	Future	Future	Future	Future	Future	Future	Future	Future
Razen	rzp	Future	Future	Future	Future	Future	Future	Future	Future	Future	Future	Future
IDF v2	idf	Yes	Yes	Yes	Never	Never	Yes	Never	Never	Never	Yes	Never
IDF v3	idf	Yes	Yes	Yes	Never	Never	Yes	Never	Never	Never	Yes	Yes
IDF v4		Future	Future	Future	Future	Future	Future	Future	Future	Future	Future	Future
HyperLynx	HYP	Future	Future	Future	Future	Future	Future	Future	Future	Future	Future	Future

Yes
No
Never
In progress
Future

INSTALLATION

There are two method to install workbench: manually and by FreeCAD-addons manager. Second solution is recommended.

Manual installation

Unpack downloaded zip file from github/sourceforge and copy extracted folder to:

- **GNU/Linux**

  On GNU/Linux distributions better do not keep PCB workbench folder under standard FreeCAD installation path (often under /sys path). This is connected with root permissions. Better idea is to keep it under /home directory.

  Go to your user directory '/home/**userName**' and display all hidden folders. Search for folder ".FreeCAD". Under this directory you should find subfolder 'Mod'

  ---

  */home/**userName**/.FreeCAD/Mod*

  Replace **username** with our user name

  Next change read/write permissions to 777 (also for subfolers).

  ---

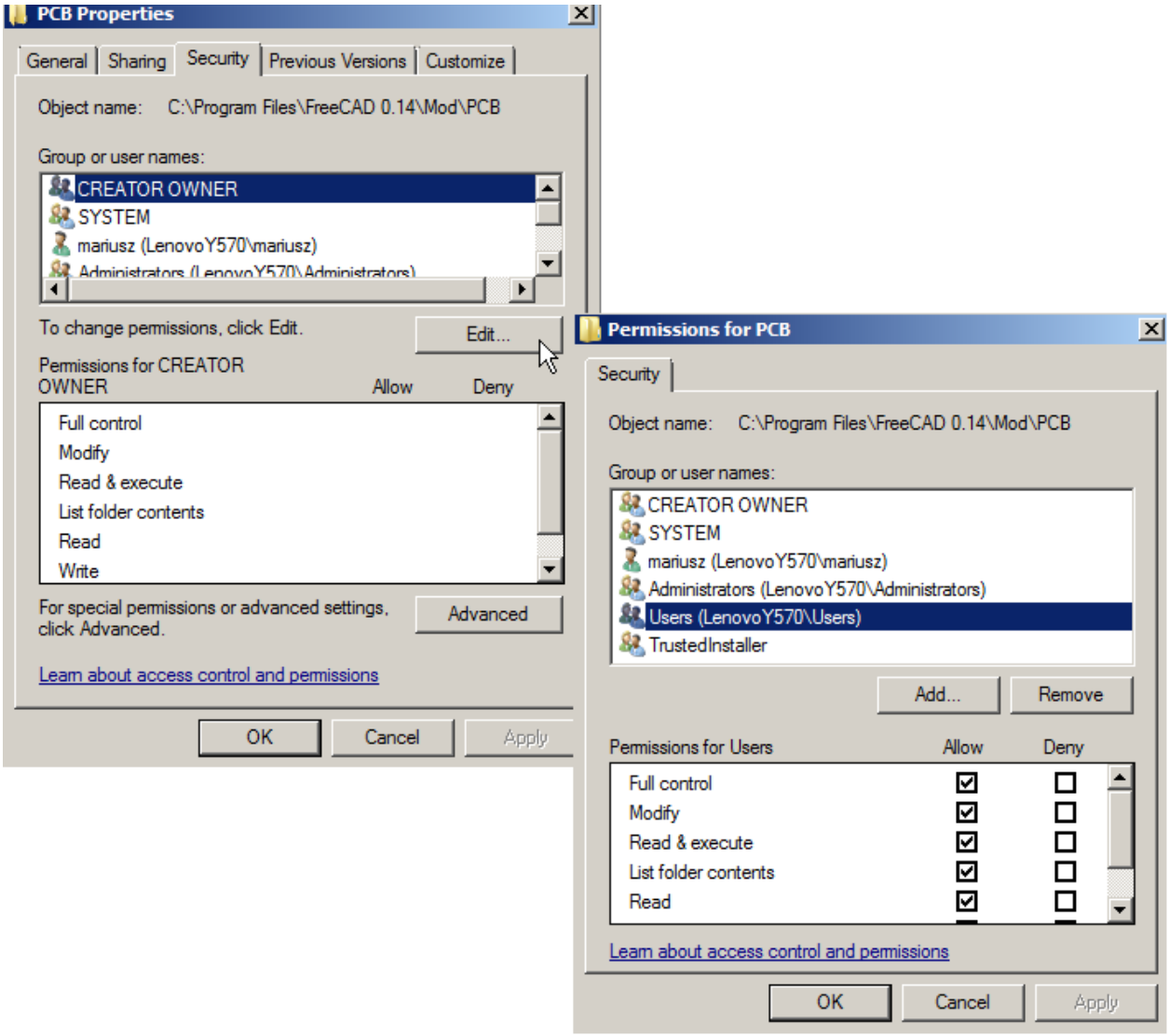
  chmod 777 -R PCB

- **Windows**

*C:\Users**userName**\AppData\Roaming\FreeCAD\Mod\PCB*

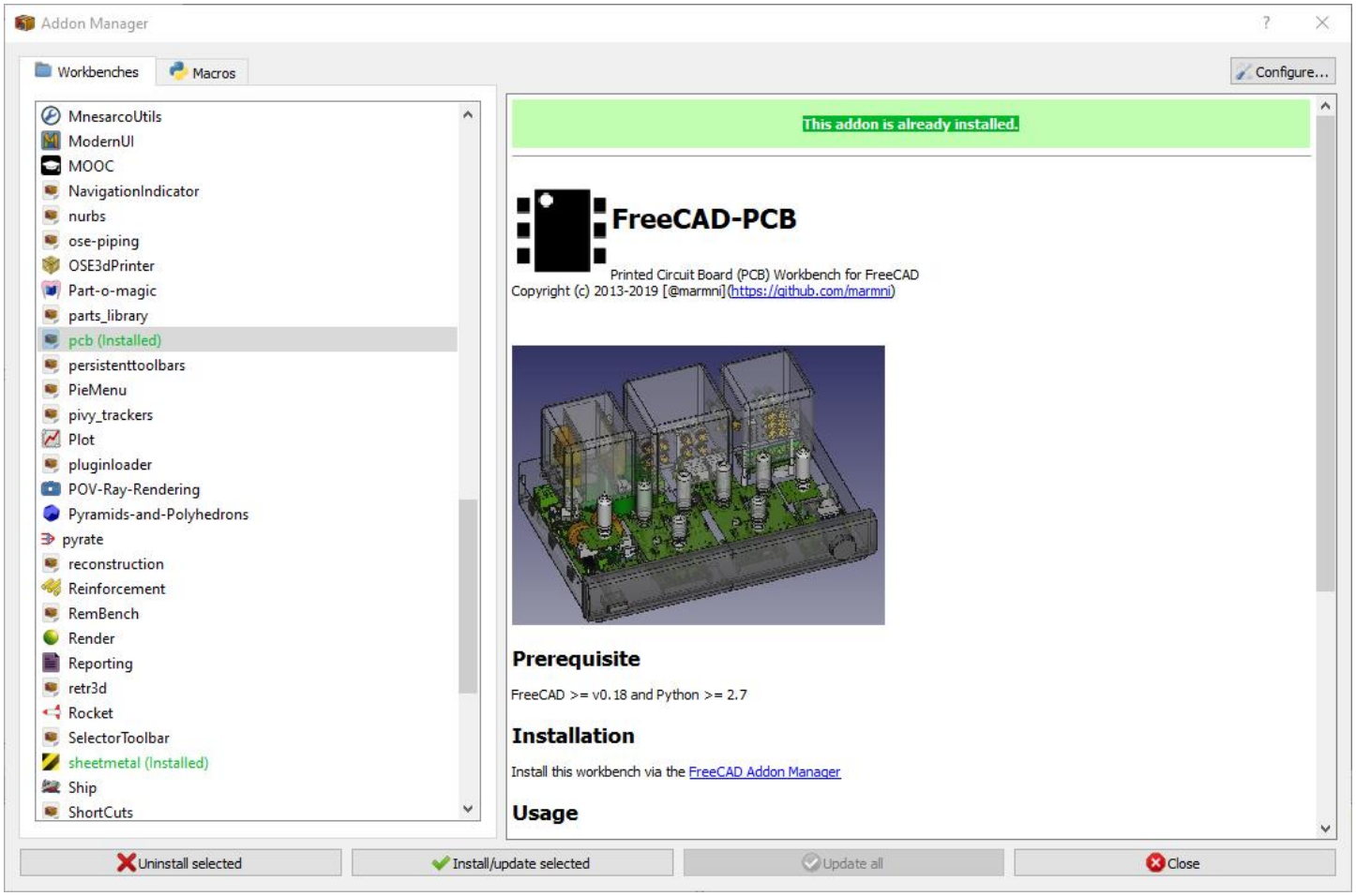
Replace **username** with our user name.

It is recommended also to change read/write permission for all users. Click right button on PCB folder and select Properties → Security → Edit → Users and mark all checkboxes under 'Allow' option.



Addon manager

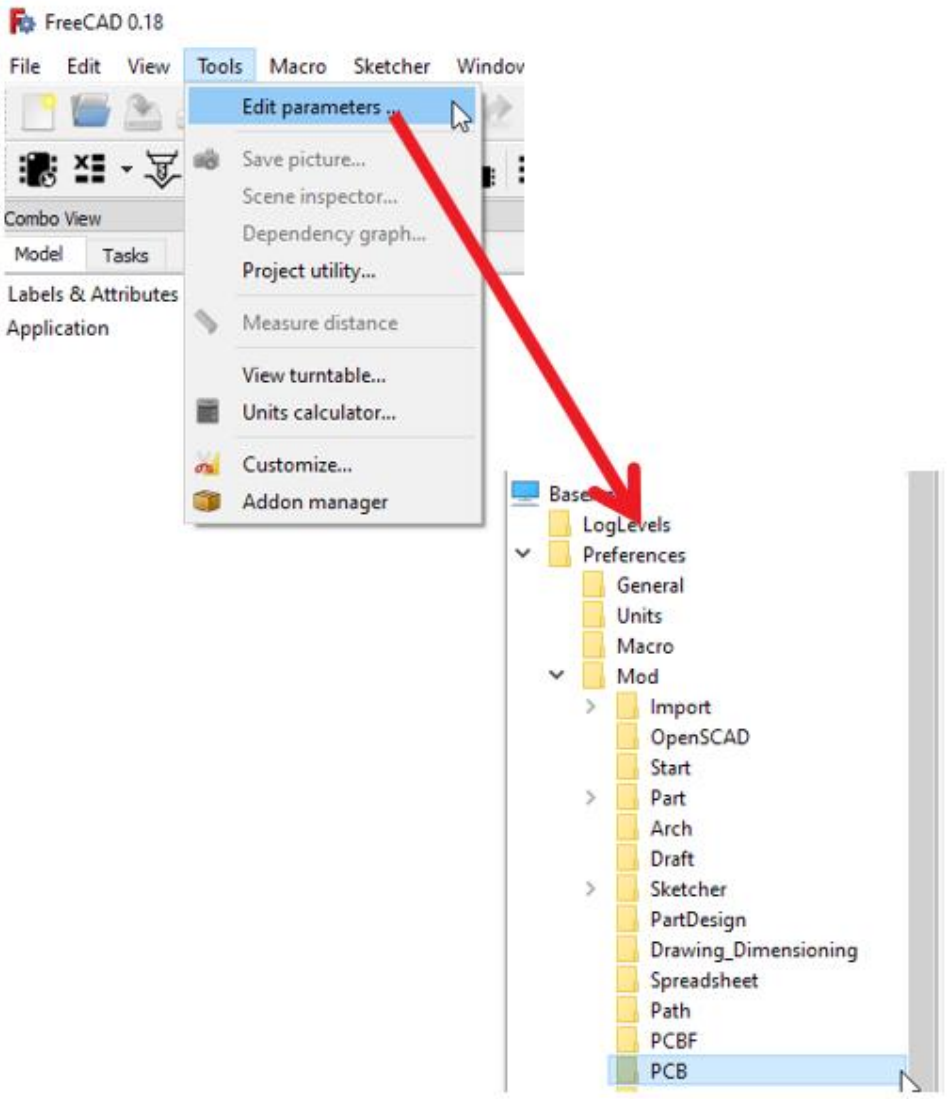
FreeCAD-addons is a standard part of FreeCAD which allows you to automatically install new workbenches/macros for FreeCAD. You will find it in the main menu Tools -> Addon manager



It is recommended to use the add-ons manager instead of manual installation. The manager also allows you to update installed modules.

CONFIGURATION

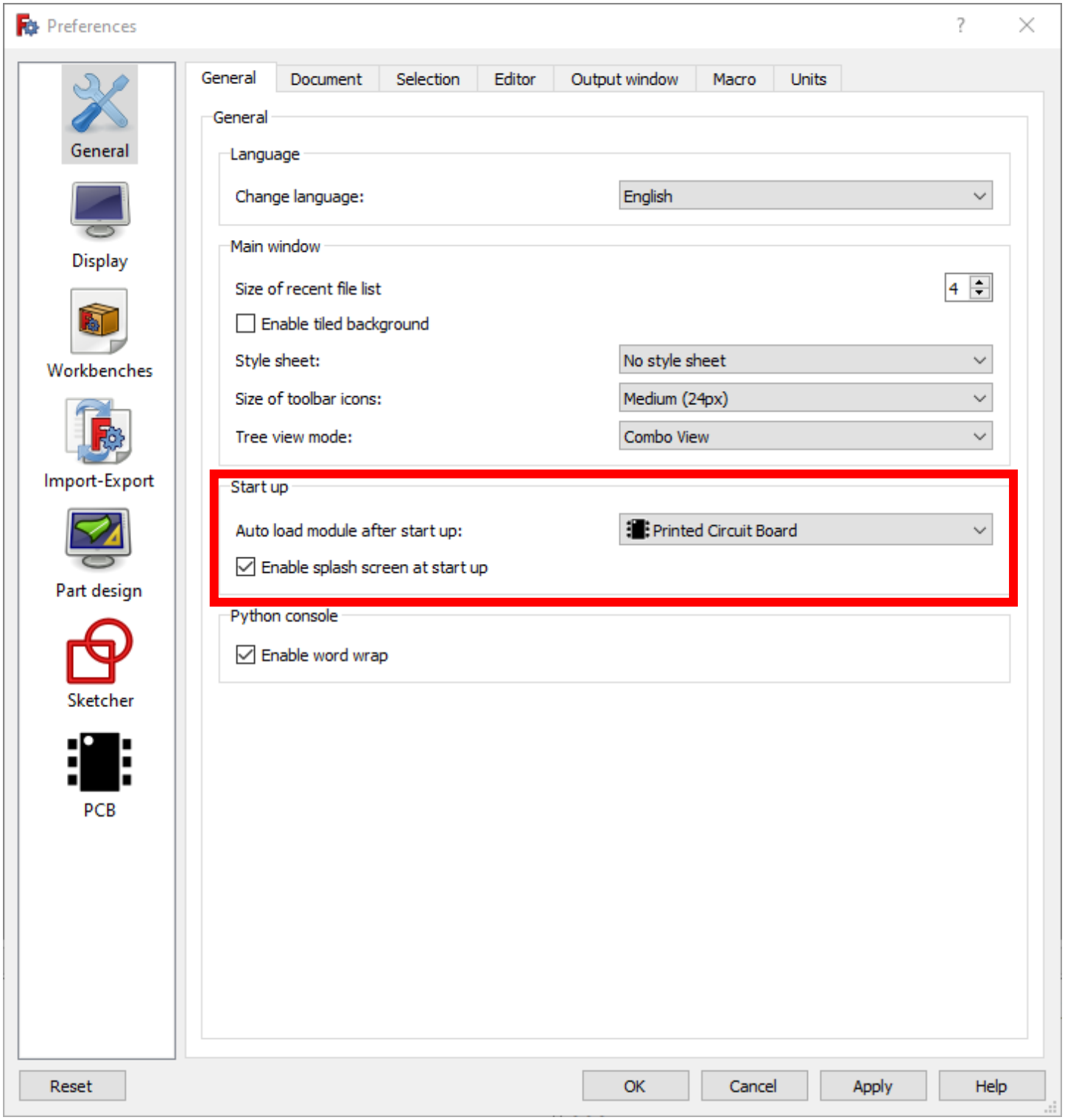
Most of variables/parameters are stored in FreeCAD configure files. You can find a parameter editor in the main menu.



Do not change anything in file 'PCBconf.py'!

Setting PCB module as main workbench

There is a possibility to set PCB module as main workbench. To do this choose General tab (Edit -> Preferences). Under this tab you should find 'Start up' section, where you can set which workbench should be loaded after FreeCAD start.



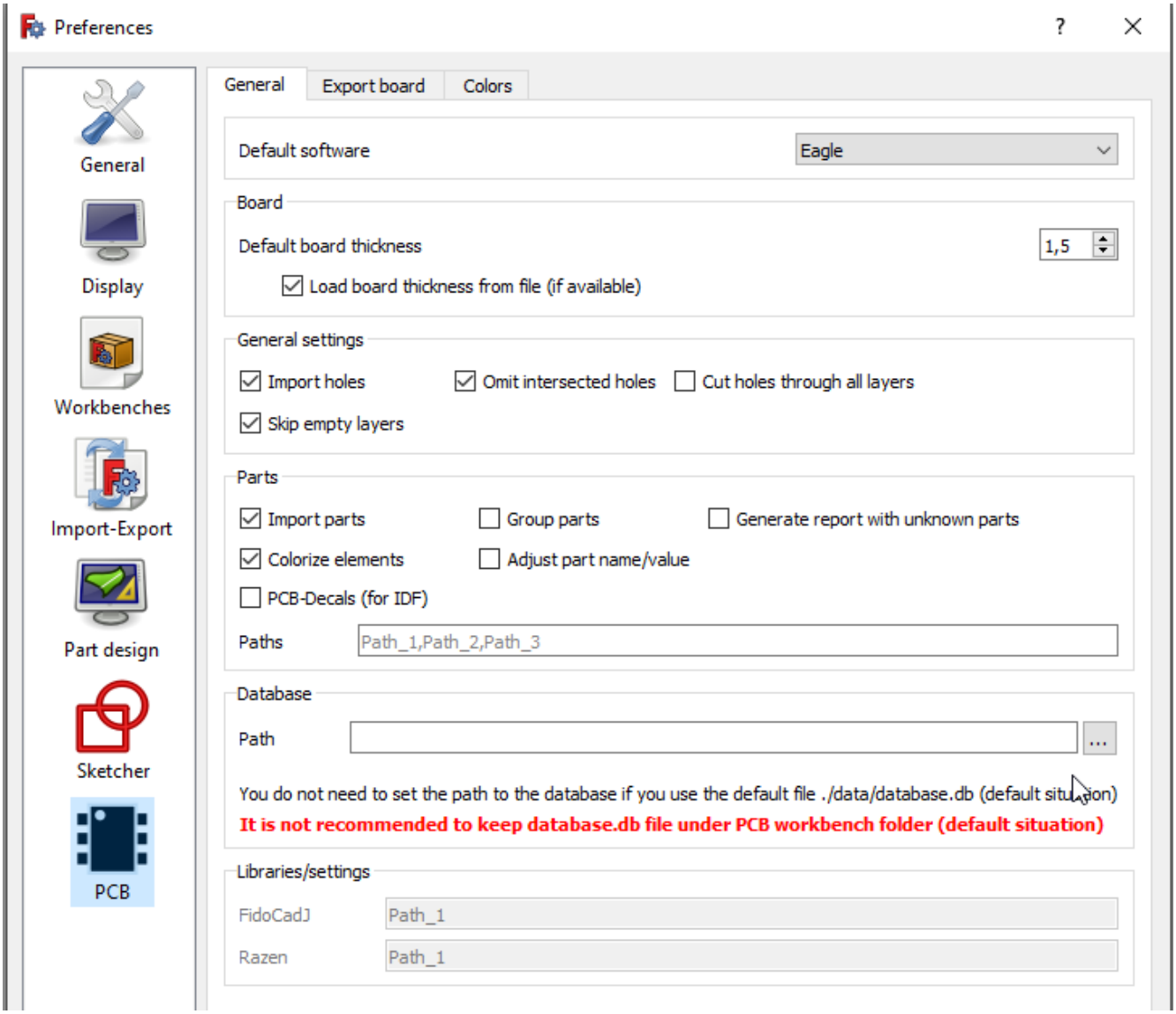
CUSTOMIZING WORKBENCH

Workbench is fully configurable - you can set various variables which are directly connected with importing/exporting PCB boards. All setting are automatically stored in FreeCAD so you need to set them only once (of course you can change them also whenever you want).

In main menu choose Edit → Preferences - > PCB.

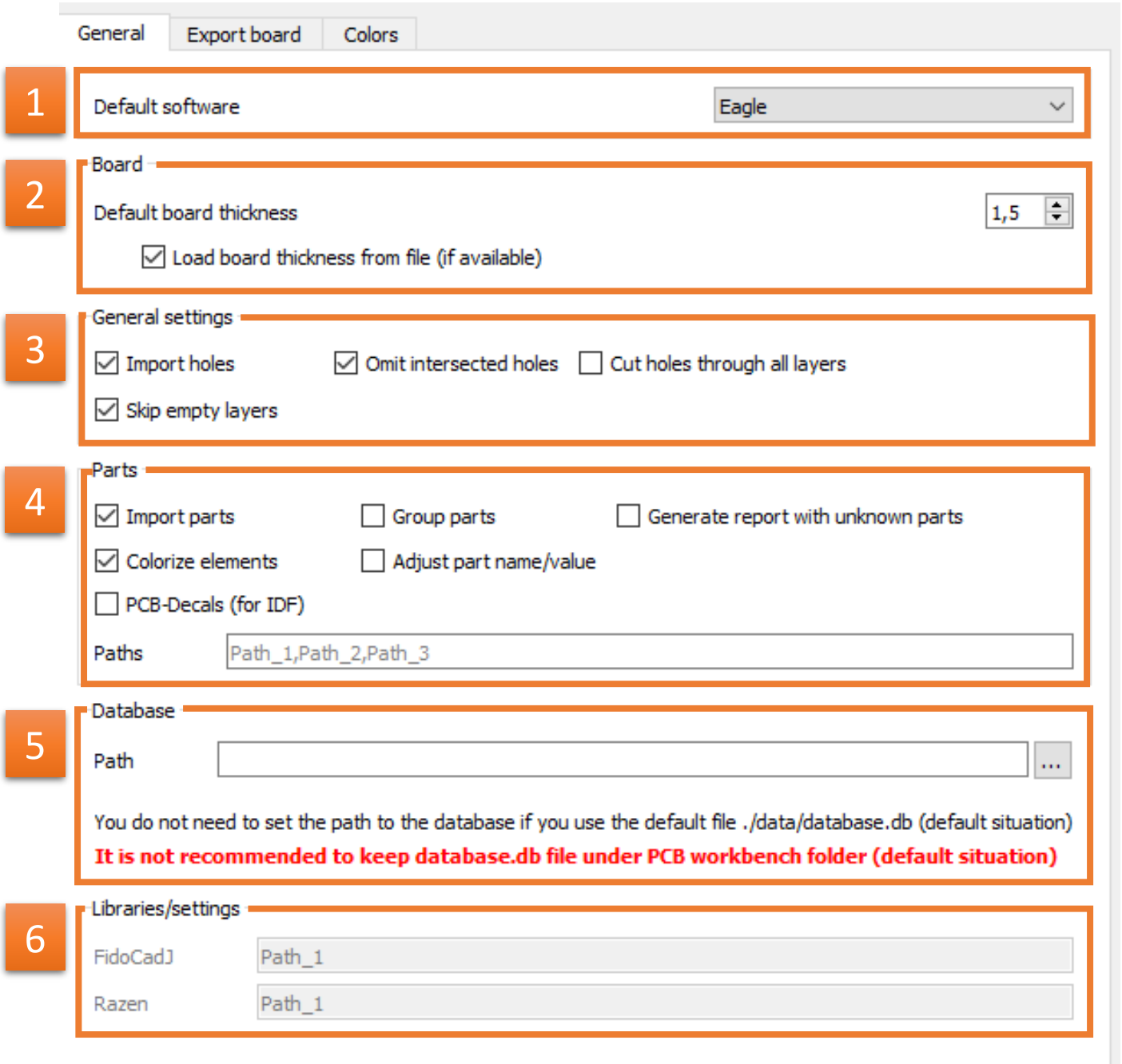
In the PCB section you can find all the configuration settings that are included in three tabs:

- General
- Export board
- Colors



General

In this tab all settings are arranged under six sections.



1 Set default software which you are using.

2 Set default boart thickness (parameter can be changed even after importing the board)

3 **Import holes** – import holes from file (if checked)

Omit intersected holes - to avoid problems during loading PCB board this option should be always checked

Cut holes through all layers – it is possible to show/hide holes in layers (for example paths/pads, parameter can be changed even after importing the board)

Skip empty layers – this option decrease time necessary to generate 3D representation of the PCB board – empty layers will not be generated

4 **Import parts** – import or skip 3D models of the parts

Group parts – group imported 3D models in categories

Generate report with unknown parts – if 3D representation for imported package will not be recognize you can generate report (txt file) which will contain missing 3D models

Colorize elements – import colorized 3D models (if selected) or gray scale models (if option is not selected)

Adjust part name/value -

PCB-Decals – check this option if you will import IDF files

Paths – add here a path under which you are storing 3D models. Do not change anything (leave empty) if you are using standard localization (PCB/parts) . Separate different paths by comma

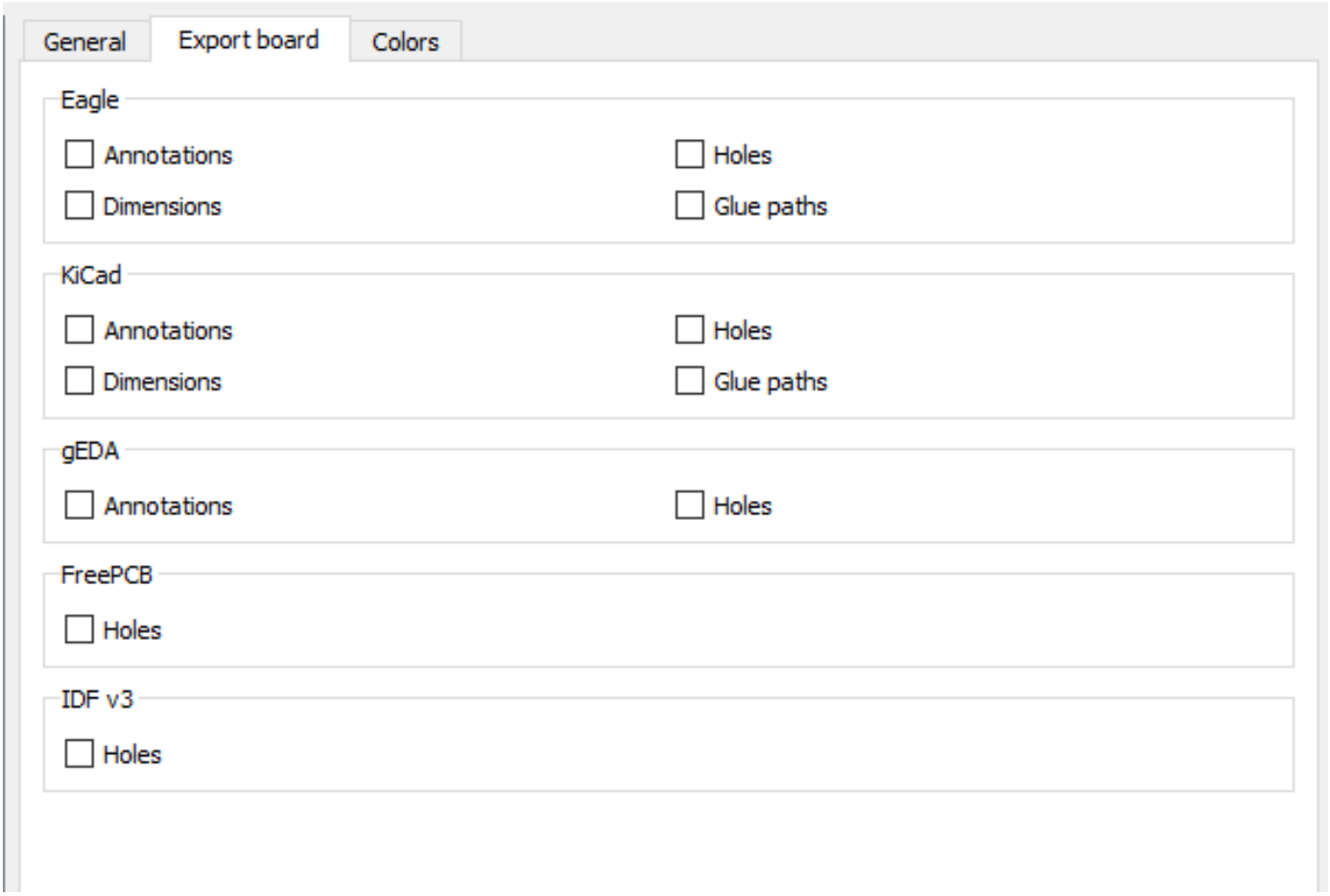
5 **Path** – path to database.db file (only one). Do not change anything (leave empty) if you are using standard localization (PCB/data/atabase.db)

It is not recommended to keep database.db file under PCB workbench folder (default situation)

6 Libraries – for future

Export board

In this section you can set the default layers that will be included when exporting the board to one of the supported formats.



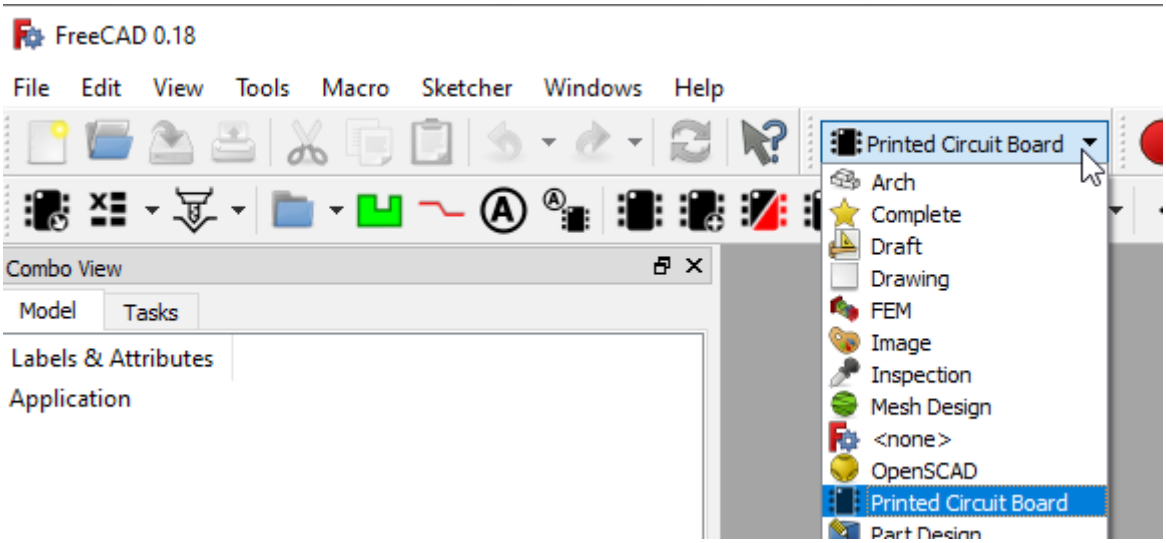
Colors

Default colors for imported layer.

ACCESSING THE WORKBENCH

There are two methods to access to the PCB workbench:

- In one of the available toolbars locate drop down list and choose ’Printed Circuit Board’.



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

MENU BAR

There are no menu bars dedicated for PCB workbench.

TOOLBARS

Two special toolbars are available:

- PCB View.
- PCB Settings.

This section describes the various icons available in mentioned toolbars.

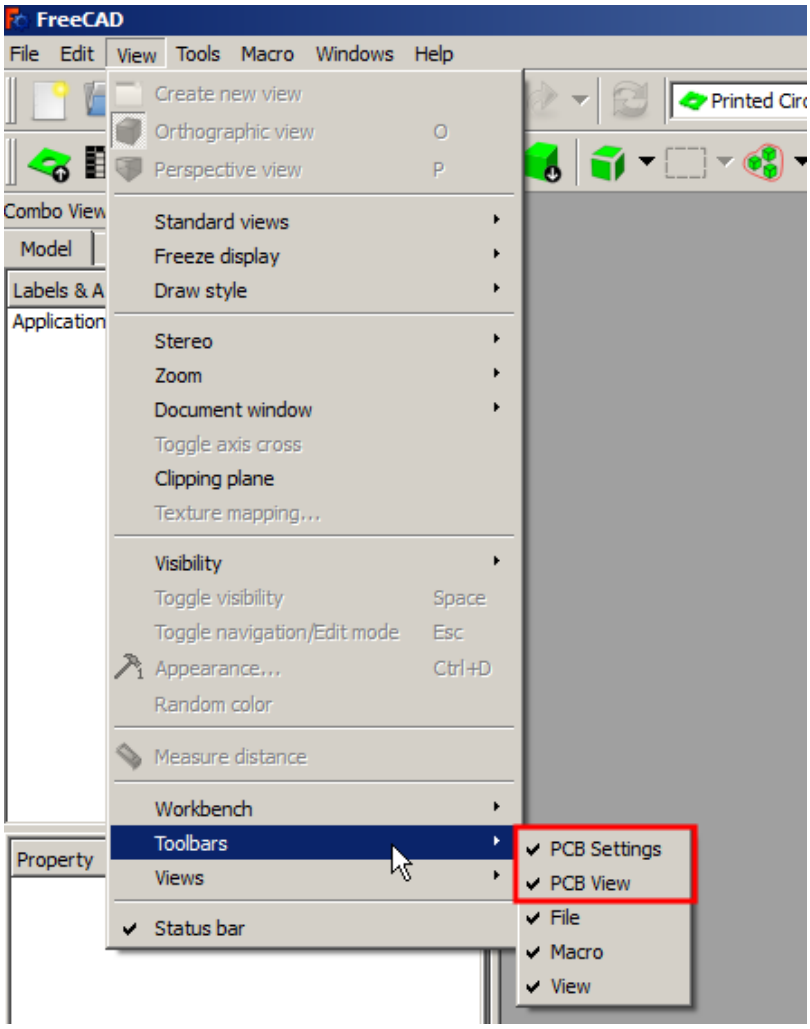
PCB Settings toolbar

	Option	Description
	Export PCB	Check Export board section
	Export BOM	Check Export Bill Of Materials (BOM) section
	Centroid	Check centroid section
	Export hole locations	Check Export hole locations section
	Export hole locations report	Check Export hole locations report section
	Create drilling map	Check Create drilling map section
	Create PCB	Check Create PCB section
	Create glue path	Check Create glue path section
	Add annotation	Check Add annotation section
	Store Name/Value as parm	
	Assign models	Check Assign models section
	Add model	Check Add model section
	Update models	Check Update models section
	Download models	Check Download models section
	Generate models	Check generate models section

	Option	Description
	Explode	Check Explode section
	Create constraint area	Check Create constraint area section
	Bounding box	Check Bounding box section
	Section cuts	Check Section cuts section

PCB View toolbar

Displaying toolbars

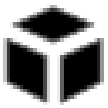
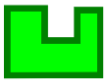


When mentioned toolbars are not displaying automatically after choosing PCB workbench in the main window, you need to do it manually.

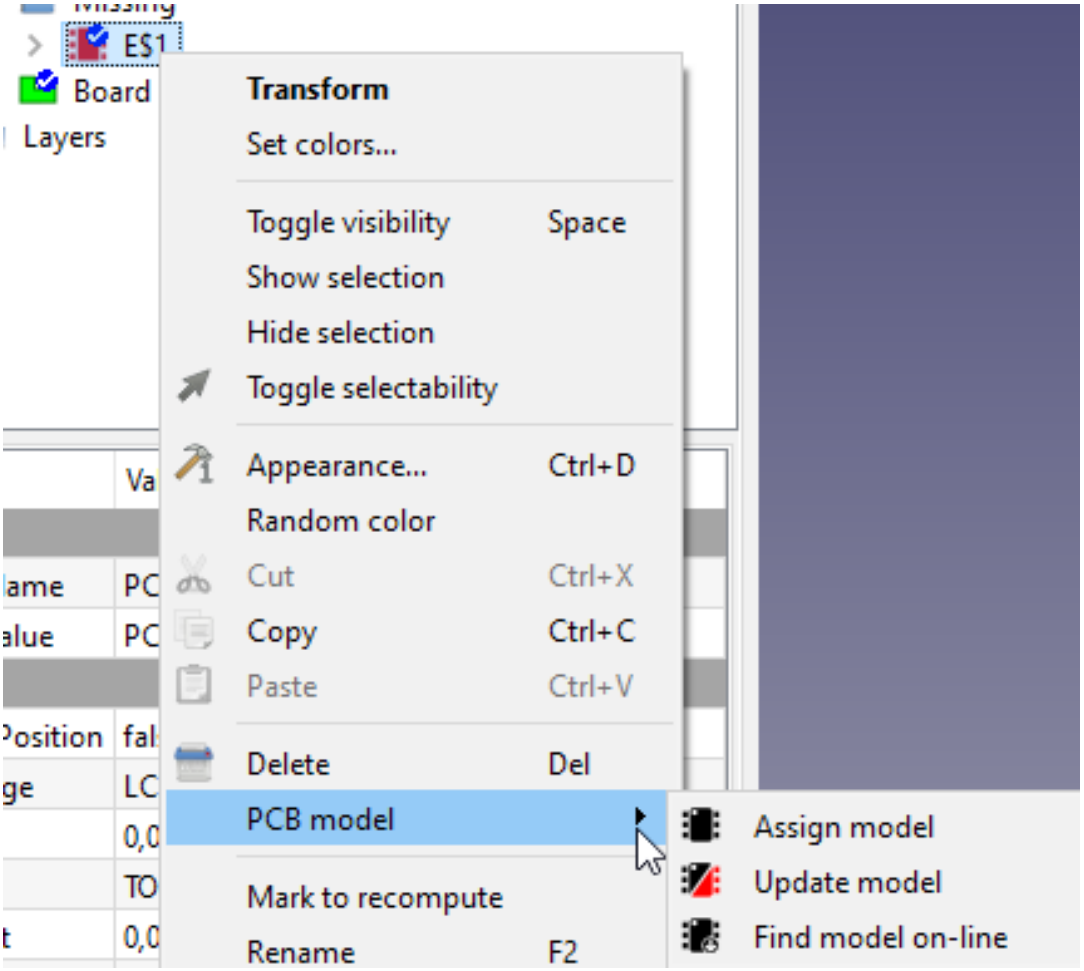
From top menu bar choose View → Toolbars and mark toolbars from Printed Circuit Board workbench.

SPECIFICATION TREE

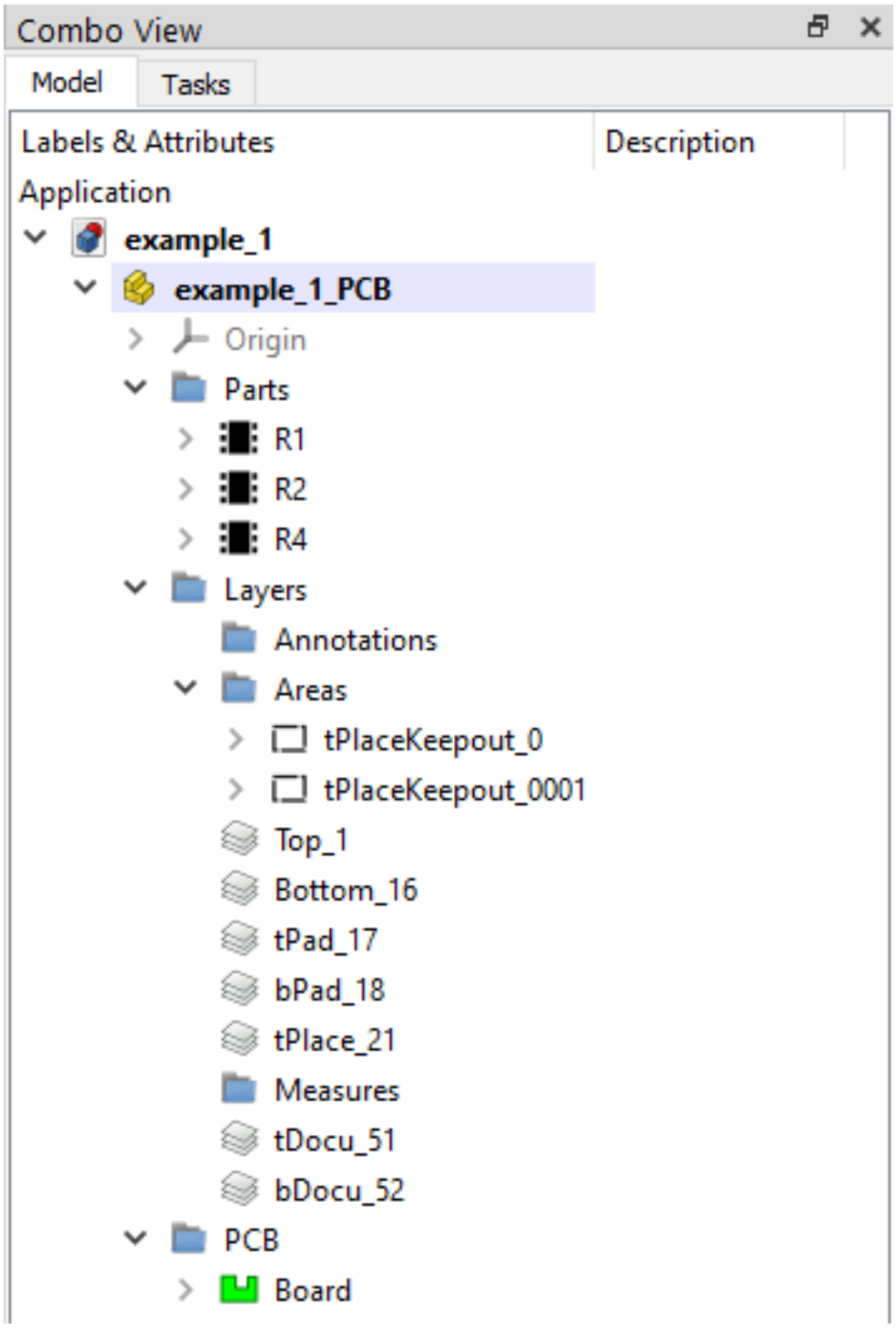
There are several types of objects directly related to the PCB workbench. You can see them in the "Combo View" and recognize them by the corresponding icons. For some of them specific context menu is available.

	Object	Context menu
	Explode objects	· Edit
	3D representation found in database	· Update model · Placement model
	The 3D representation was not found in the database	· Assign model · Update model · Find model on-line
	Board	----------
	Constraint area	----------
	Layer	----------
	Annotation/Object Name/Object Value	----------

	Object	Context menu
	Glue path	----------

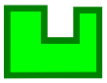


Context menu for specific objects is only available in the PCB environment.

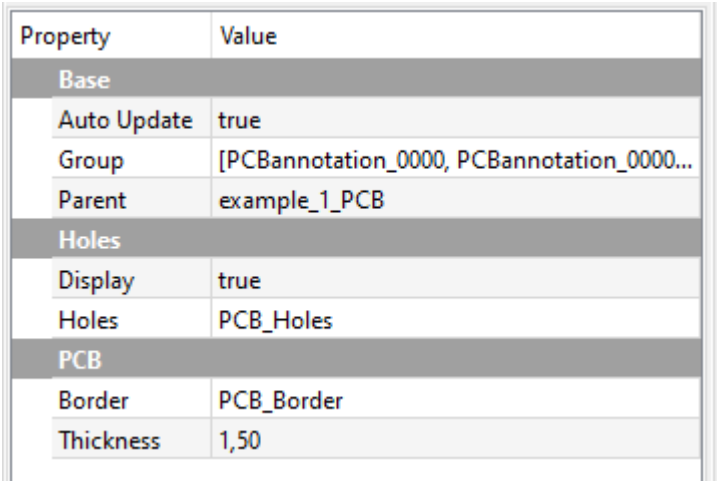

Combo View
Model
Tasks
Labels & Attributes
Description
Application
example_1
example_1_PCB
Origin
Parts
R1
R2
R4
Layers
Annotations
Areas
tPlaceKeepout_0
tPlaceKeepout_0001
Top_1
Bottom_16
tPad_17
bPad_18
tPlace_21
Measures
tDocu_51
bDocu_52
PCB
Board

OBJECTS PROPERTIES

Each object created in PCB workbench has unique parameters that can be set in the Property View (View or Data tab).



Board



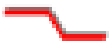
Group: all objects which are directly connected with board

Display Holes: show/blank holes

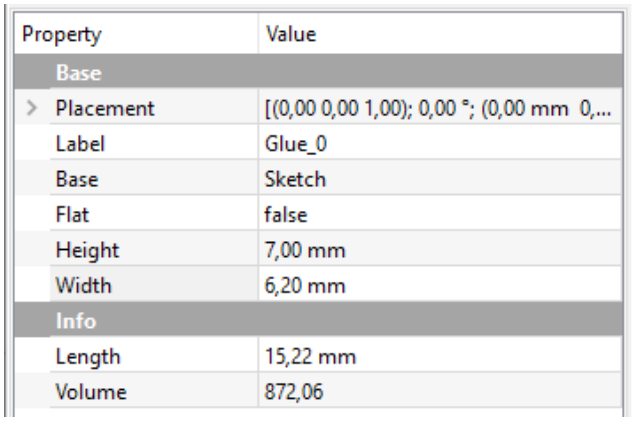
Holes: reference to sketch that containing holes

Border: reference to sketch that containing board outline

Thickness: board thickness



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

Length: glue seam lenght (for specific W/H)

Volume: for specific W/H

Part model founded 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

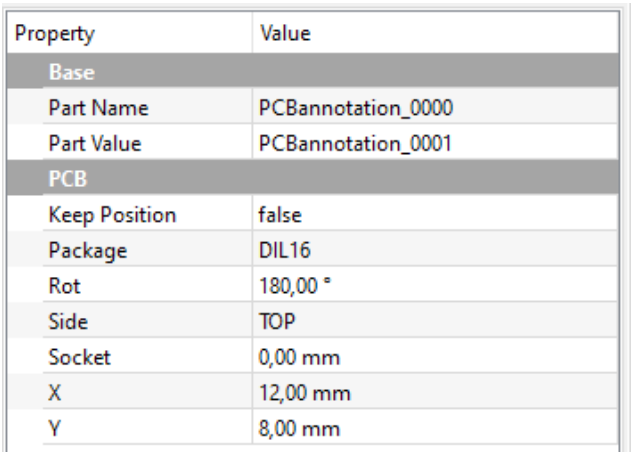
Rot: rotation value around Z axis

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

Socket: socket height / model position in Z direction

X: model position in X direction

Y: model position in Y direction

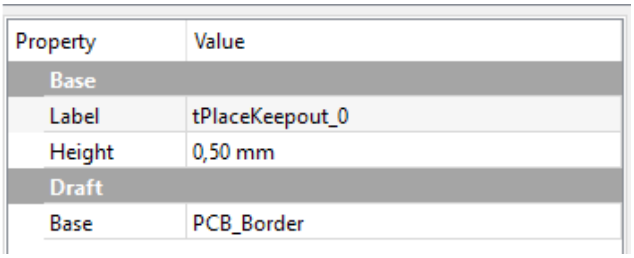


Property	Value
Base	
Part Name	PCBannotation_0000
Part Value	PCBannotation_0001
PCB	
Keep Position	false
Package	DIL16
Rot	180,00 °
Side	TOP
Socket	0,00 mm
X	12,00 mm
Y	8,00 mm

Constraint area

Height: area height, parameter available only for some constraints areas type (on TOP/BOTTOM side)

Base: reference to sketch that containing area outline



Property	Value
Base	
Label	tPlaceKeepout_0
Height	0,50 mm
Draft	
Base	PCB_Border

Part model not founded 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)

Socket: socket height / model position in Z direction

X: model position in X direction

Y: model position in Y direction

Property	Value
Base	
Part Name	PCBannotation_0010
Part Value	PCBannotation_0011
PCB	
Keep Position	false
Package	LCC20
Rot	0,00 °
Side	TOP
Socket	0,00 mm
X	57,00 mm
Y	9,00 mm

Explode objects

Active: turn of/off explode effect

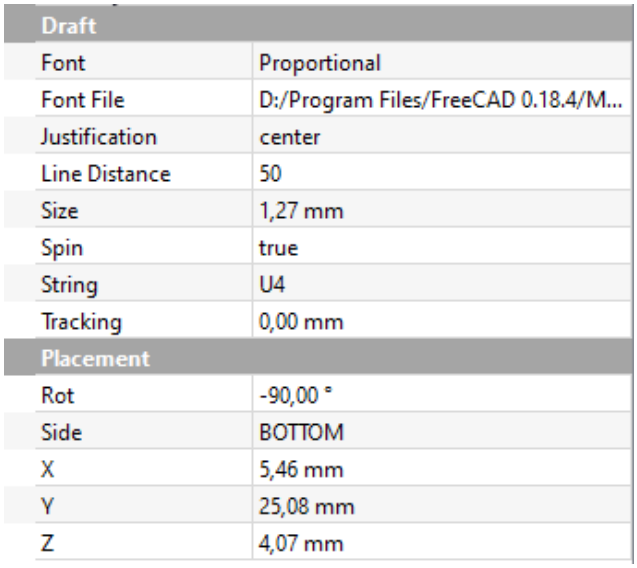
Bottom Step Size: distance between parts placed on bottom side of board

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	
Label	Explode
Active	true
Bottom Step Size	10,00
Inverse	false
Top Step Size	10,00

Annotation/Object Name/Object Value



Draft	
Font	Proportional
Font File	D:/Program Files/FreeCAD 0.18.4/M...
Justification	center
Line Distance	50
Size	1,27 mm
Spin	true
String	U4
Tracking	0,00 mm
Placement	
Rot	-90,00 °
Side	BOTTOM
X	5,46 mm
Y	25,08 mm
Z	4,07 mm

Font: font name

Font file: it is possible to use own font

Justification: text position according to X, Y values

Line distance: distance between lines (in % according to fonf size)

Size: font size

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

Text: text displayed by annotation object

Tracking: distance between letters

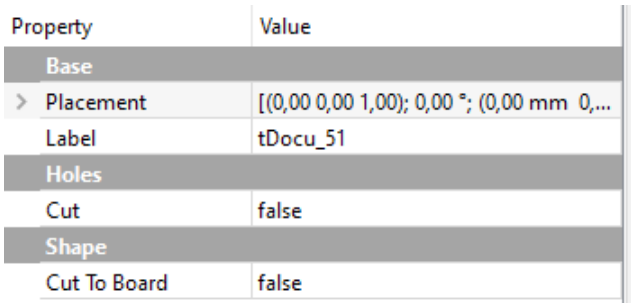
Rot: rotation value around Z axis

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

X: text position in X direction

Y: text position in Y direction

Layer



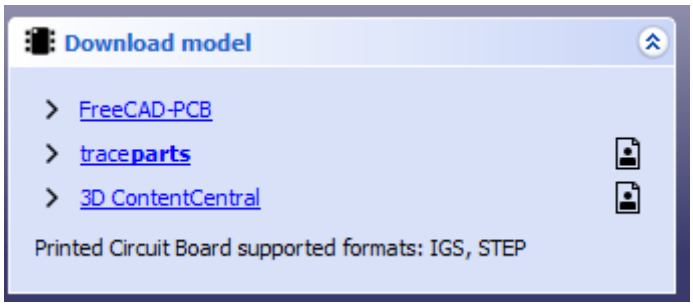
Property	Value
Base	
Placement	[(0,00 0,00 1,00); 0,00 °; (0,00 mm 0,...
Label	tDocu_51
Holes	
Cut	false
Shape	
Cut To Board	false

Cut: show/blank holes

Cut To Board: cut shape to board outline

3D MODELS

Workbench comes without 3D model so it is necessery to dowload them separatelly. You can bring up the 'Download models' window from the PCB Settings toolbar. Mentioned window contains links to sites when you can find free 3D models.



Registration is necessary to download models

There is also possibility to search for concrete model. To do this just right click on missing model in specification tree and choose PCB model → Find model on-line

Models from FreeCAD-PCB (github site) are directly connected with default database.db file.

To add/remove paths you need to open Preferences window. More info you can find in section Customizing workbench.

Workbench supports 3D models saved in one of the following formats: STP/IGS

The default path is set to the “/Parts" folder which is located in the main PCB Workbench folder.

It is recommended to keep parts outside PCB folder - to avoid data lost during workbench update.

ASSIGN MODELS

WORKING WITH WORKBENCH

OPENING/IMPORTING BOARD

CREATING BOARD FROM SCRATCH

CREATING GLUE PATHS

ADDING ANNOTATIONS

ADDING NEW MODELS

UPDATING MODELS

CREATING CONSTRAINST AREAS

GENERATING BOUNDING BOX

CREATING SECTION CUTS

EXPORTING HOLE LOCATIONS

EXPORTING HOLE LOCATIONS REPORT

CREATING DRILLING MAP

BOM

CENTROID

EXPORTING BOARD

VIEW OPTIONS

DISPLAY MODES

GROUPING PARTS

LAYERS

CUT TO BOARD OUTLINE

HOLES SETTINGS

SIGNALS MARKING

EXPLODE

BOUNDING BOX

RENDERS

KERKYTHEA

POV-RAY

OTHER

GENERATE MODELS

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.

The script is useful for Eagle versions lower than 7.

EXAMPLES