

## Printed Circuit Board Workbench for FreeCAD PCB-FreeCAD

marmni ([marmni@onet.eu](mailto:marmni@onet.eu))

Copyright 2013-2021



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



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



<https://www.freecadweb.org/>



<https://www.sqlalchemy.org/>



<https://www.python.org/>



<https://pypi.org/project/PySide/>

## Spis treści

GENERAL INFORMATIONS .....	5
LICENCE .....	6
INTRODUCTION.....	7
Requirements.....	7
Supported softwares .....	8
INSTALLATION.....	9
Manual installation .....	9
Addon manager .....	11
CONFIGURATION .....	12
Setting PCB module as main workbench .....	13
CUSTOMIZING WORKBENCH .....	14
General.....	15
Export board .....	17
Colors .....	18
ACCESSING THE WORKBENCH .....	19
MENU BAR .....	20
TOOLBARS.....	21
PCB Settings toolbar .....	21
PCB View toolbar .....	22
Displaying toolbars .....	23
SPECIFICATION TREE .....	24
OBJECTS PROPERTIES.....	27
3D models .....	31
ASSIGN MODELS .....	33
Assign models – left column .....	34
Assign models – middle column .....	35
Assign models – right column.....	36
Multi model definition for one part.....	39
Working with workbench .....	47
OPENING/IMPORTING board.....	48
Unit system .....	49
CREATING BOARD FROM SCRATCH .....	50

## Printed Circuit Board Workbench for FreeCAD

CREATING GLUE PATHS.....	51
ADDING ANNOTATIONS.....	52
ADDING NEW MODELS .....	53
UPDATING models .....	54
CREATING CONSTRAINST AREAS .....	56
GENERATING BOUNDING BOX.....	57
CREATING SECTION CUTS .....	58
EXPORTING HOLE LOCATIONS .....	59
EXPORTING HOLE LOCATIONS REPORT .....	60
CREATING DRILLING MAP .....	61
BOM .....	62
CENTROID .....	63
EXPORTING BOARD .....	64
VIEW OPTIONS .....	65
Display modes.....	66
Grouping parts .....	68
LAYERS .....	69
Cut to board outline.....	70
HOLES SETTINGS .....	72
SIGNALS MARKING.....	73
EXPLODE .....	74
Bounding box .....	75
RENDERS .....	77
Kerkythea.....	78
POV-RAY.....	79
OTHER .....	80
GENERATE MODELS .....	81
SCRIPTS .....	82
Eagle.....	82
EXAMPLES .....	83

# GENERAL INFORMATION

# 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	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
gEDA	pcb	✓	✗	✓	✗	✓	✗	✓	✓	✓	✗	✓
FreePCB	fpc	✓	✓	✓	✗	✓	✗	✓	✓	✓	✓	✓
KiCad	kicad_pcb	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
FidoCadJ	fcd	✗	✗	✗	✗	✗	✗	✗	✗	✗	✗	✗
Razen	rzp	✗	✗	✗	✗	✗	✗	✗	✗	✗	✗	✗
IDF v2	idf	✓	✓	✓	✗	✗	✓	✗	✗	✗	✓	✗
IDF v3	idf	✓	✓	✓	✗	✗	✓	✗	✗	✗	✓	✓
IDF v4		✗	✗	✗	✗	✗	✗	✗	✗	✗	✗	✗
HyperLynx	HYP	✗	✗	✗	✗	✗	✗	✗	✗	✗	✗	✗

Yes	Yes
No	No
Never	Never
In progress	In progress
Future	Future



# INSTALLATION

There are two methods 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 subfolders).

---

```
chmod 777 -R PCB
```

---

## Printed Circuit Board Workbench for FreeCAD

- Windows

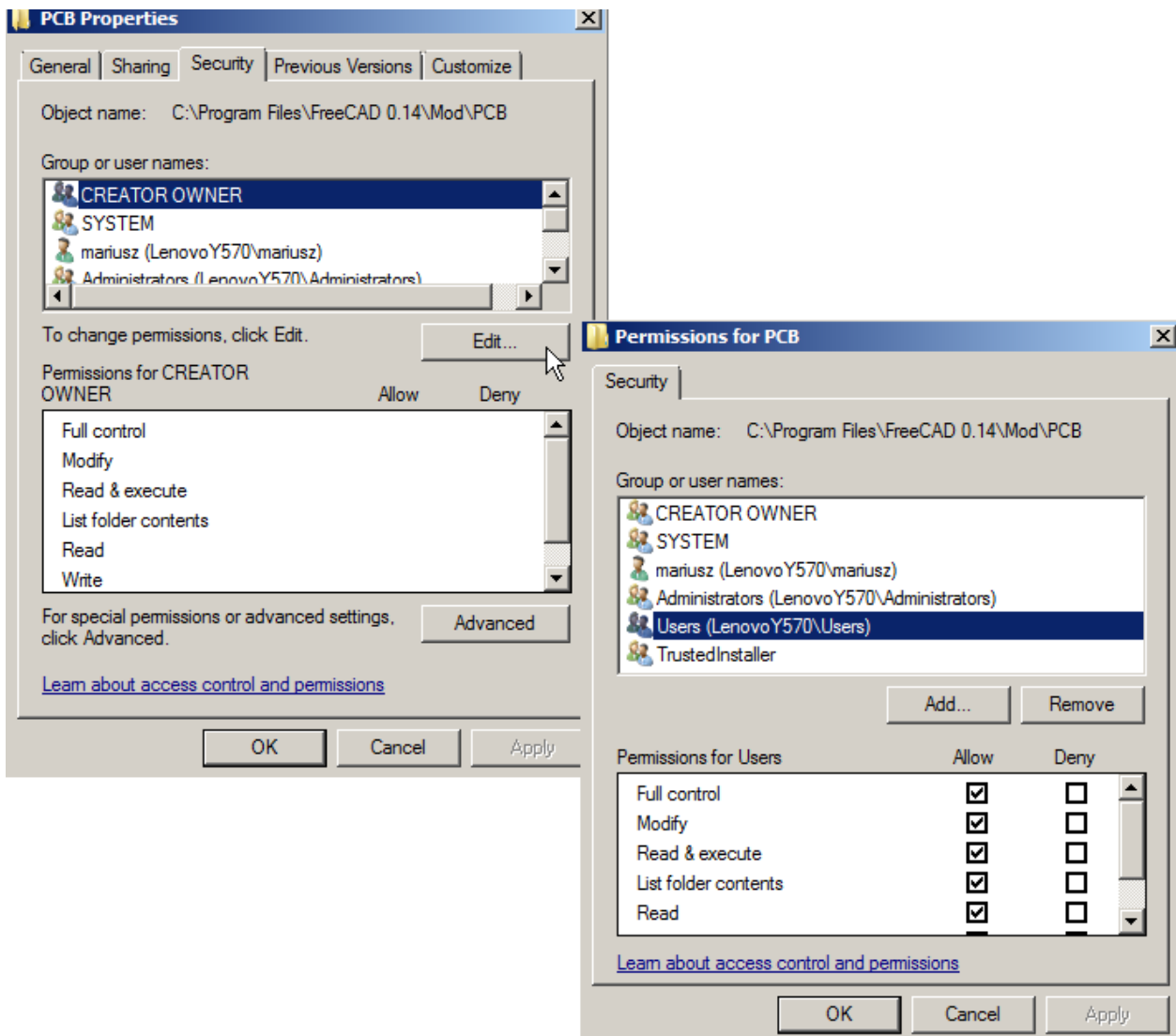
---

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

---

Replace **username** with our user name.

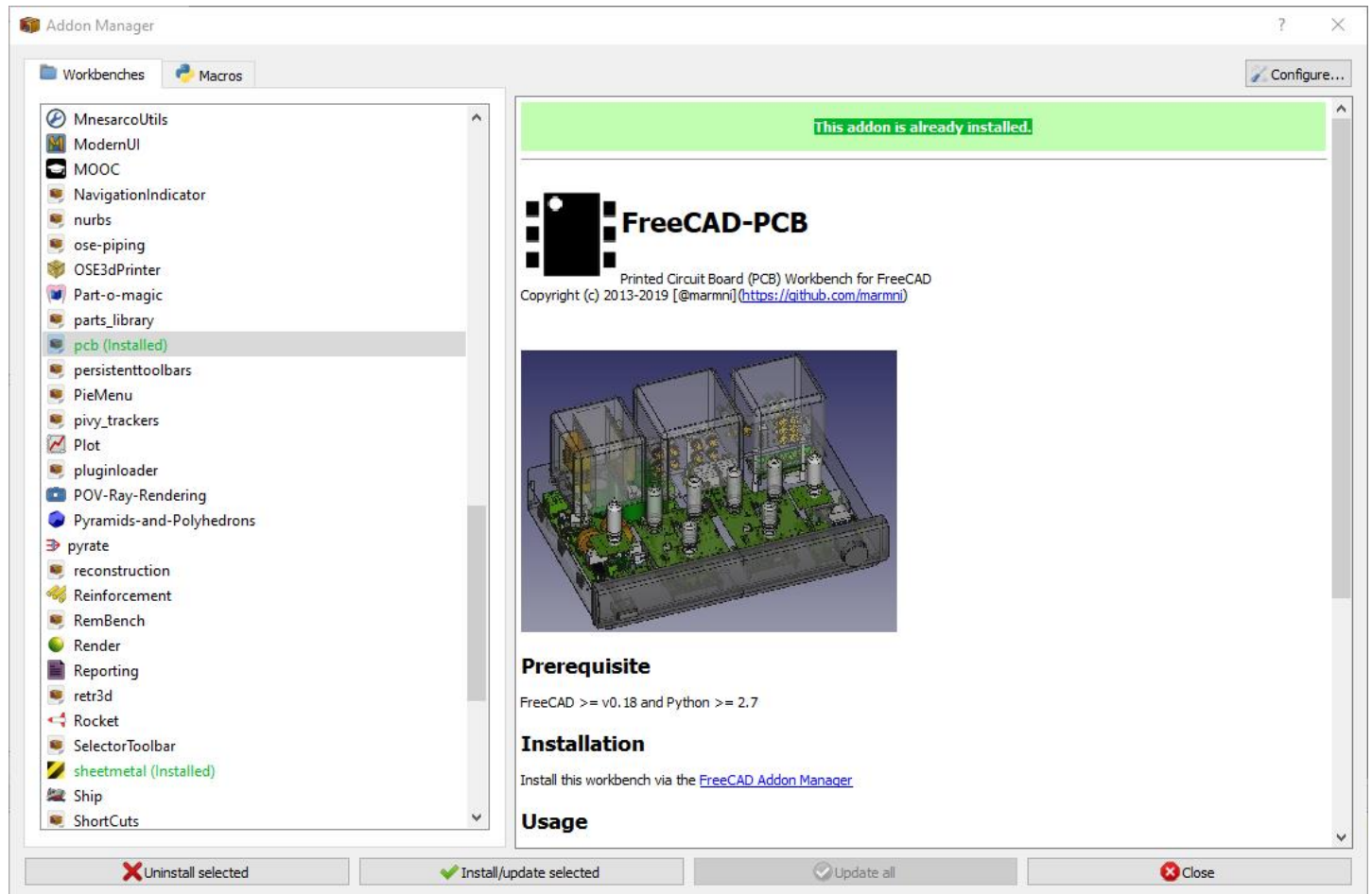
It is recommended also to change read/write permission for all users. Click right button on PCB folder and select Properties → Security → Edit → Users and mark all checkboxes under 'Allow' option.



## Printed Circuit Board Workbench for FreeCAD

### Addon manager

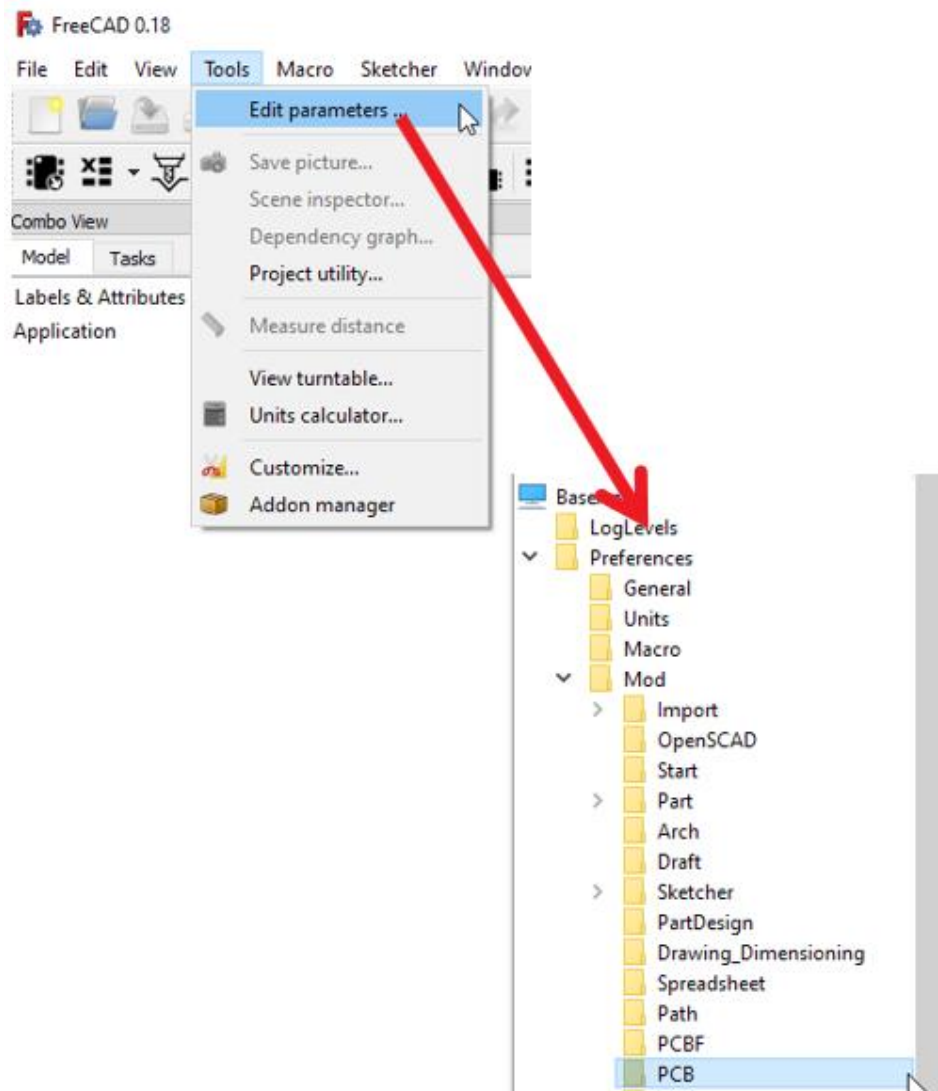
FreeCAD-addons is a standard part of FreeCAD which allows you to automatically install new workbenches/macros for FreeCAD. You will find it in the main menu Tools -> Addon manager



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

# CONFIGURATION

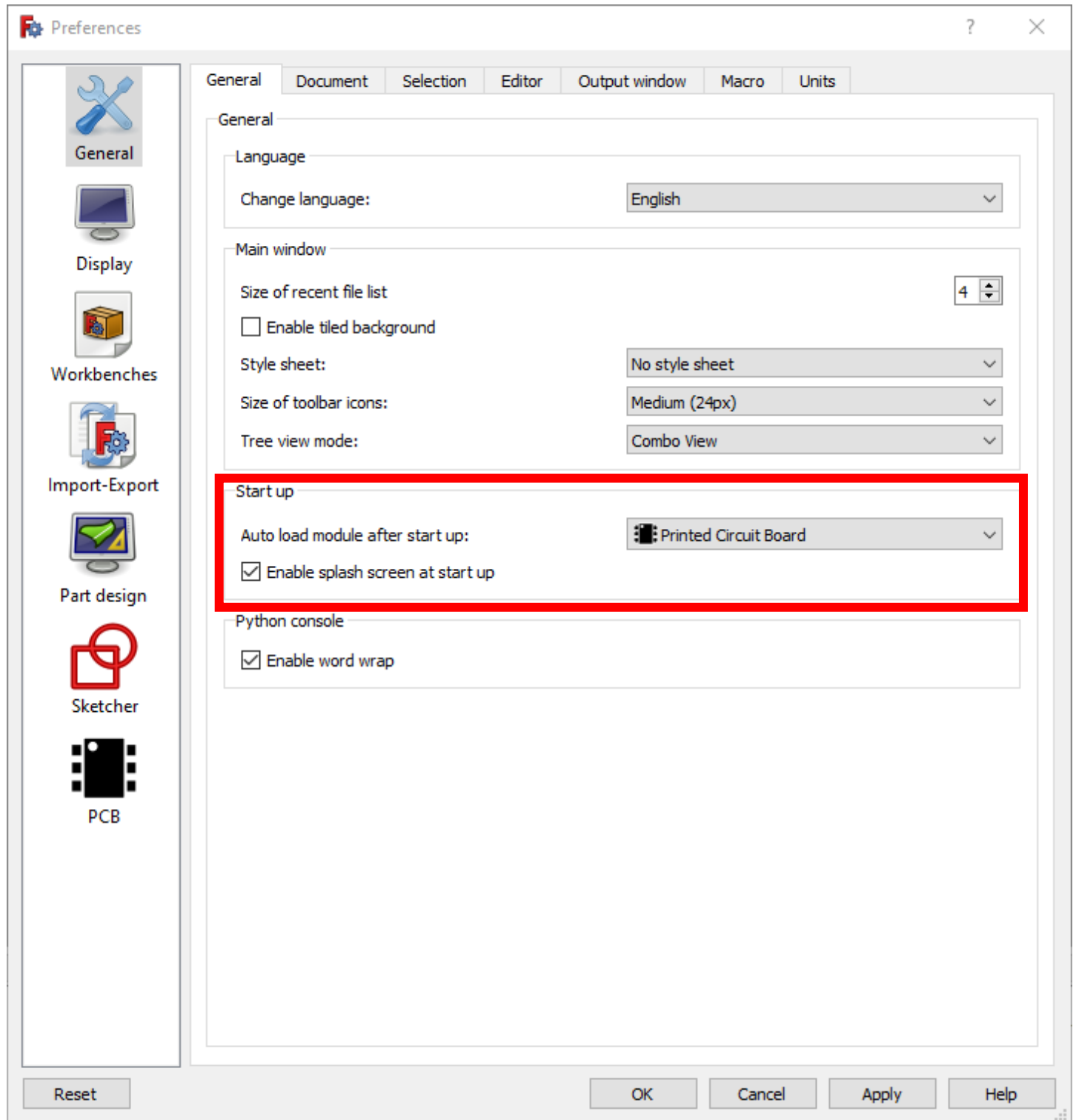
Most of variables/parameters are stored in FreeCAD configure files. You can find a parameter editor in the main menu.



**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.



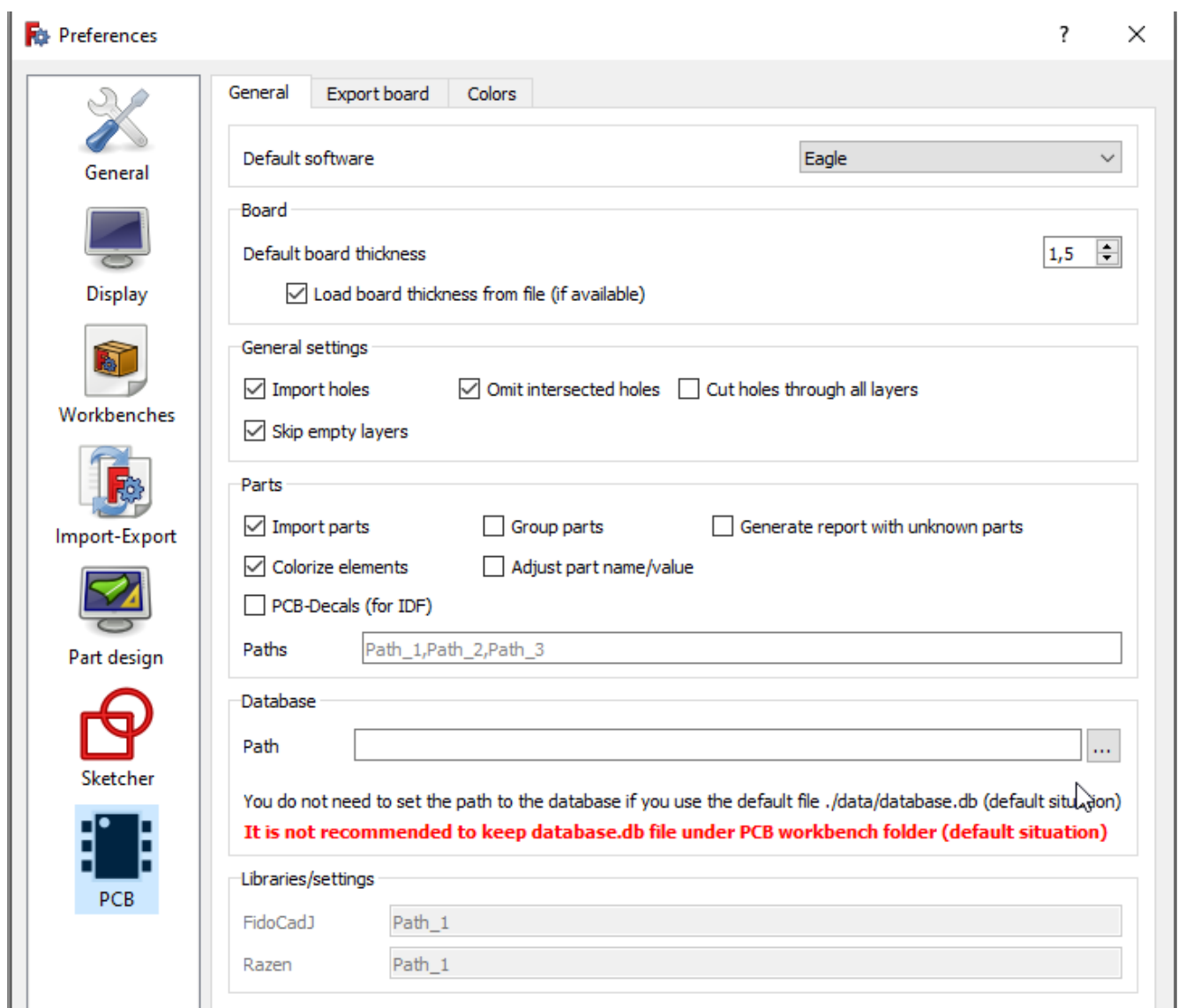
# CUSTOMIZING WORKBENCH

Workbench is fully configurable - you can set various variables which are directly connected with importing/exporting PCB boards. All settings 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.

The image shows the 'General' tab of the PCB Workbench settings dialog. It is divided into six sections, each highlighted with an orange border and a numbered orange square on the left:

- 1** **Default software**: A dropdown menu set to 'Eagle'.
- 2** **Board**:
  - Default board thickness**: A numeric input field set to '1,5'.
  - ☒ **Load board thickness from file (if available)**
- 3** **General settings**:
  - ☒ **Import holes**
  - ☒ **Omit intersected holes**
  - ☐ **Cut holes through all layers**
  - ☒ **Skip empty layers**
- 4** **Parts**:
  - ☒ **Import parts**
  - ☐ **Group parts**
  - ☐ **Generate report with unknown parts**
  - ☒ **Colorize elements**
  - ☐ **Adjust part name/value**
  - ☐ **PCB-Decals (for IDF)**
  - Paths**: A text input field containing 'Path\_1,Path\_2,Path\_3'.
- 5** **Database**:
  - Path**: A text input field with a browse button ('...').
  - Text: 'You do not need to set the path to the database if you use the default file ./data/database.db (default situation)'
  - Text: **It is not recommended to keep database.db file under PCB workbench folder (default situation)**
- 6** **Libraries/settings**:
  - FidoCadJ**: A text input field containing 'Path\_1'.
  - Razen**: A text input field containing 'Path\_1'.

## Printed Circuit Board Workbench for FreeCAD

- 1 Set default software which you are using.
- 2 Set default board 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.

The screenshot shows the 'Export board' dialog box with three tabs: 'General', 'Export board' (selected), and 'Colors'. The dialog is organized into five sections, each corresponding to a different export format. Each section contains a list of layers with checkboxes to select which layers to include in the export.

Format	Annotations	Dimensions	Holes	Glue paths
Eagle	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
KiCad	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
gEDA	<input type="checkbox"/>		<input type="checkbox"/>	
FreePCB			<input type="checkbox"/>	
IDF v3			<input type="checkbox"/>	

## Colors

Default colors for imported layer.

General
Export board
Colors

Board

Color

Constraint areas

Place Outline Top

Place Outline

Route Outline Top

Route Outline

Route Keepout Top

Via Keepout

Place Keepout Top

Place Outline Bottom

Route Outline Bottom

Route Keepout Bottom

Place Keepout Bottom

Layers

Path

Annotations

Glue

Silk

Measures

Pad

Center drill

External files

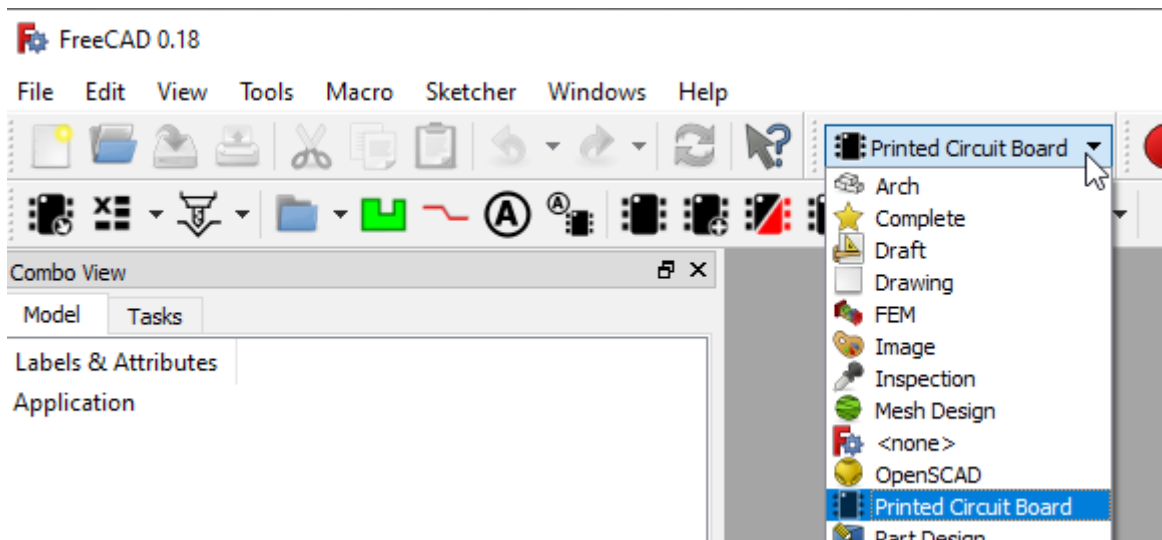
gEDA colors

...

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

## Printed Circuit Board Workbench for FreeCAD

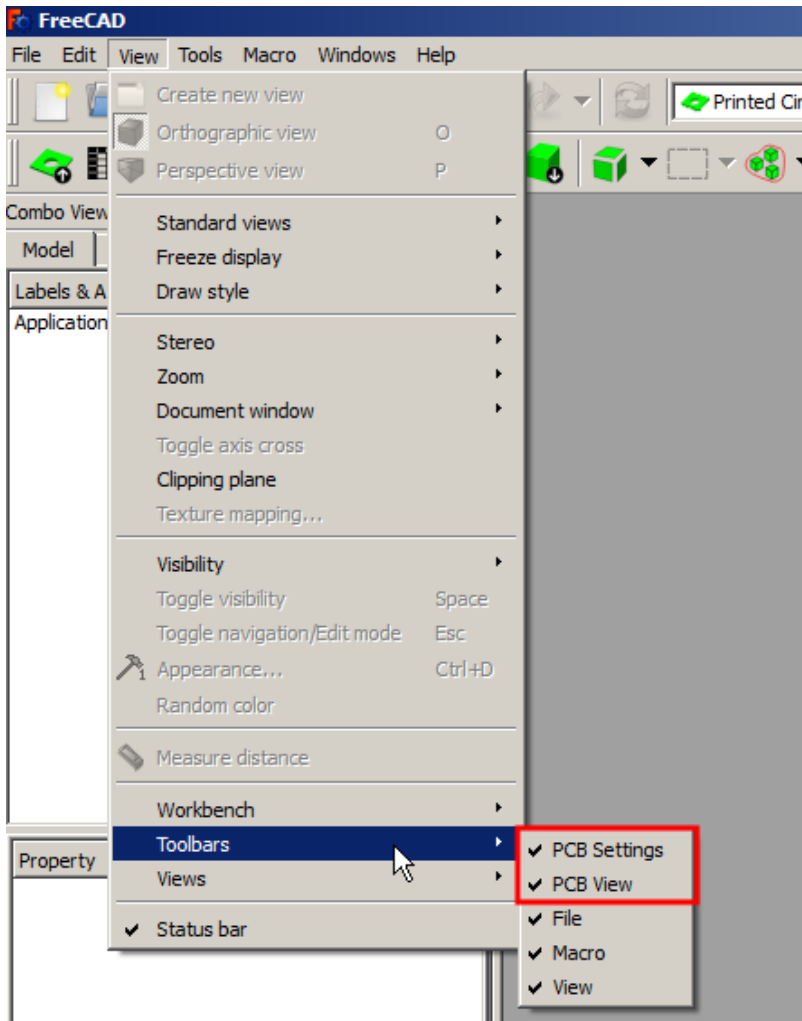
	Option	Description
	Explode	<a href="#">Check Explode section</a>
	Create constraint area	<a href="#">Check Create constraint area section</a>
	Bounding box	<a href="#">Check Bounding box section</a>
	Section cuts	<a href="#">Check Section cuts section</a>

### PCB View toolbar



	Option	Description
	Change display mode to Shaded	<a href="#">Check Display modes section</a>
	Change display mode to Flat Lines	<a href="#">Check Display modes section</a>
	Change display mode to Wireframe	<a href="#">Check Display modes section</a>
	Change display mode to Internal View	<a href="#">Check Display modes section</a>
	Layers settings	<a href="#">Check Layers section</a>
	Cut holes through all layers ON/OFF	<a href="#">Check Cutting holes through all layers section</a>
	Cut to board outline	<a href="#">Check Cut to board outline section</a>
	Show signals	<a href="#">Check Signals marking section</a>
	Group/Ungroup models in 'Parts' folder	<a href="#">Check Grouping parts section</a>
	3D rendering: export to Kerkythea	<a href="#">Check Kerkythea section</a>
	3D rendering: export to POV-RAY	<a href="#">Check POV-RAY section</a>
	Instructions	




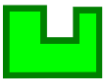



### Displaying toolbars




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.

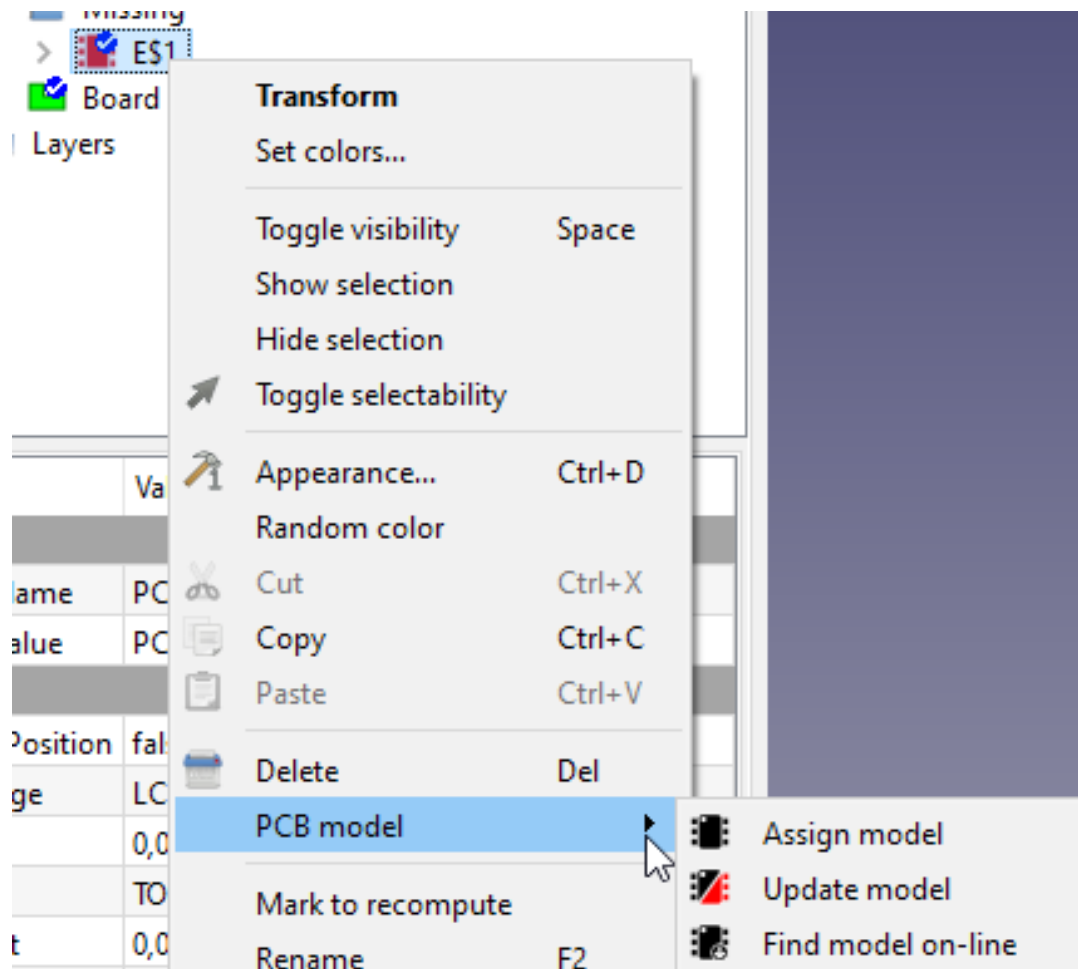
# SPECIFICATION TREE

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.

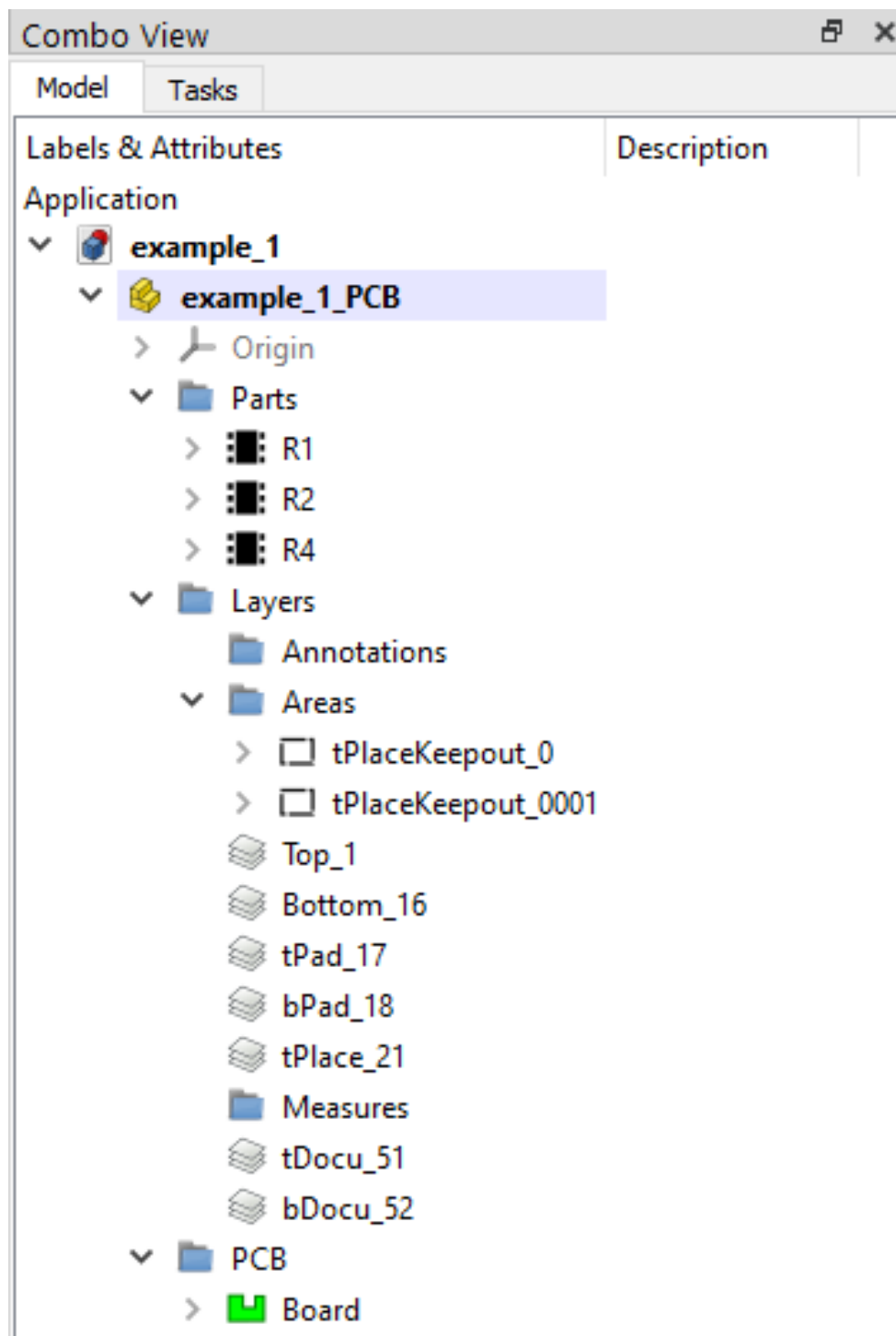
	Object	Context menu
	Explode objects	<ul style="list-style-type: none"> <li>· Edit</li> </ul>
	3D representation found in database	<ul style="list-style-type: none"> <li>· Update model</li> <li>· Placement model</li> </ul>
	The 3D representation was not found in the database	<ul style="list-style-type: none"> <li>· Assign model</li> <li>· Update model</li> <li>· Find model on-line</li> </ul>
	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.**



# 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

Property	Value
<b>Base</b>	
Auto Update	true
Group	[PCBAnnotation_0000, PCBAnnotation_0000...
Parent	example_1_PCB
<b>Holes</b>	
Display	true
Holes	PCB_Holes
<b>PCB</b>	
Border	PCB_Border
Thickness	1,50



## 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 length (for specific W/H)

**Volume:** for specific W/H

Property	Value
<b>Base</b>	
> Placement	[(0,00 0,00 1,00); 0,00 °; (0,00 mm 0,...
Label	Glue_0
Base	Sketch
Flat	false
Height	7,00 mm
Width	6,20 mm
<b>Info</b>	
Length	15,22 mm
Volume	872,06



## 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

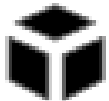
**Side:** part position on board (top/bottom side)

**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

**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 font size)

**Size:** font size

**Spin:** if parameter set to True text will keep rotation, parameter works for angle value  $\geq 90^\circ$

**Text:** text displayed by annotation object

**Tracking:** distance between letters

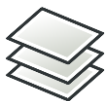
**Rot:** rotation value around Z axis

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

**X:** text position in X direction

**Y:** text position in Y direction

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



## Layer

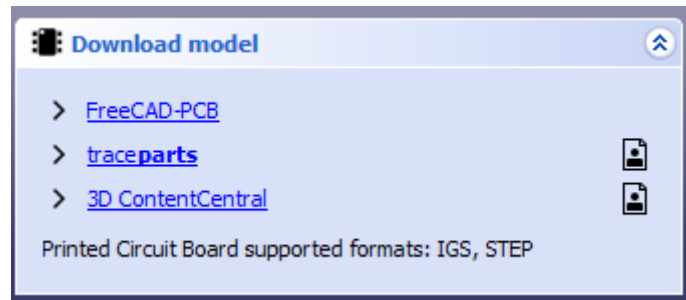
**Cut:** show/blank holes

**Cut To Board:** cut shape to board outline

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

## 3D MODELS

Workbench comes without 3D model so it is necessary to download them separately. 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.**



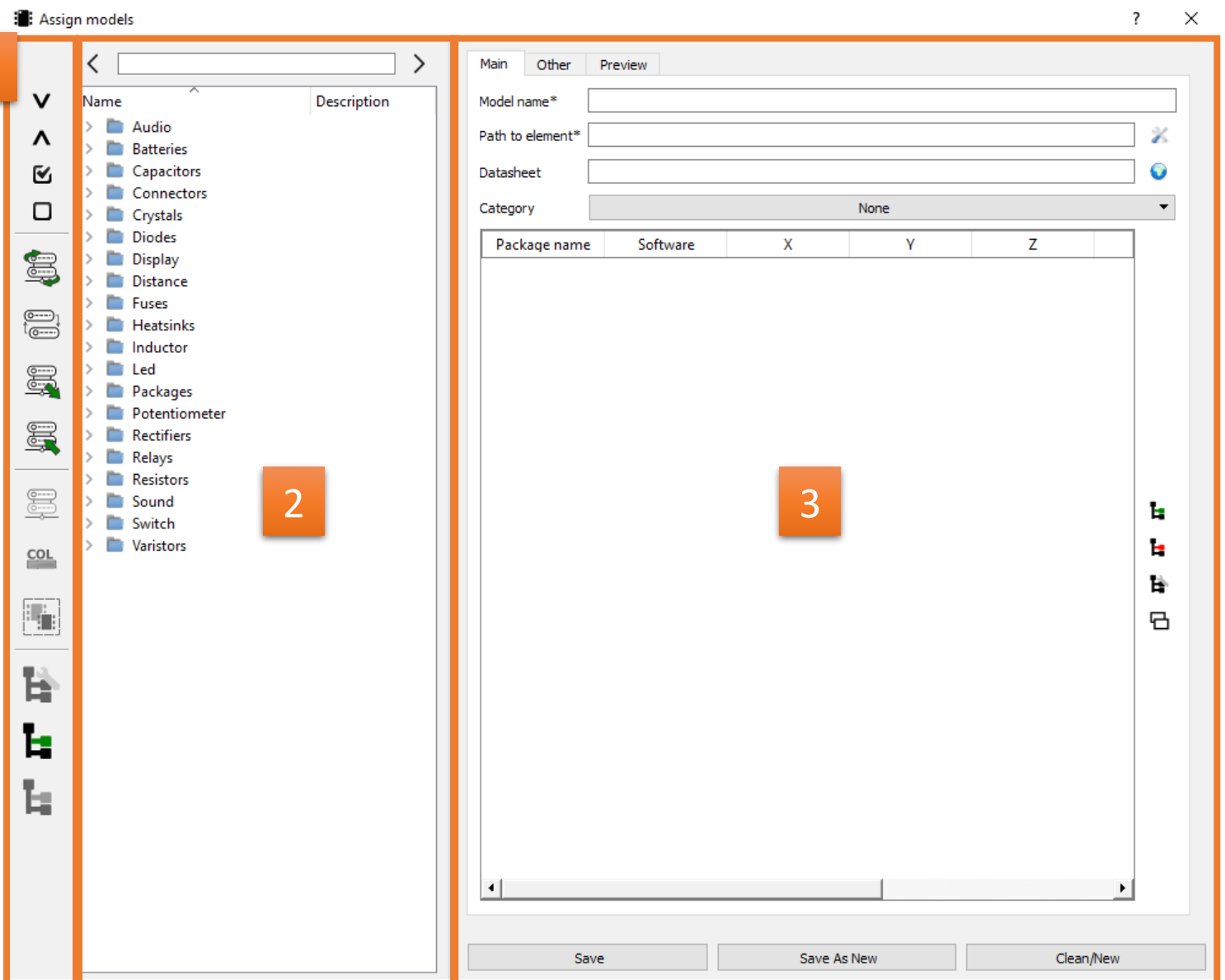
**It is recommended to keep parts outside PCB folder - to avoid data lost during workbench update.**



# ASSIGN MODELS

The 'Assign models' window allow for assigning 3D models to corresponding part from one of the supported software. It is necessary to assign 3D models and specific packages after downloading new components (not applicable for models downloaded from FreeCAD-PCB site).

You can bring up the 'Assign models' window from the PCB Settings toolbar.



## Printed Circuit Board Workbench for FreeCAD

Window consists of three main areas:

1. Left column: contains functions necessary to manage parts in database
2. Middle column: contains list of all categories/packages saved in the database
3. Right column: allows to view/edit parameters for selected package

### Assign models – left column

Collapse/Expand all items from Package list



Check/uncheck all items from Package list



Reload database



Convert packages between softwares



Export database copy



Import database copy



Delete all selected packages from database



Delete \*.col files for selected packages



Set one category for selected models



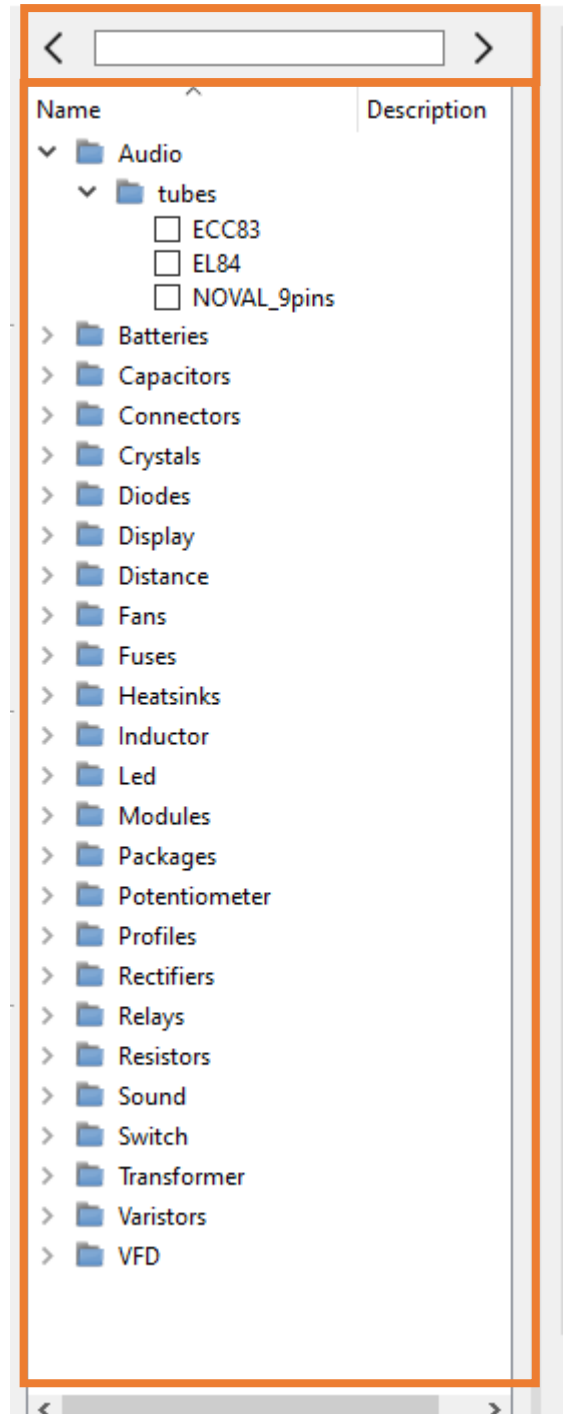
Add/edit/delete category



## Assign models – middle column

Search block contains prev/next button and entry field for searching specified package

All models are grouped under categories (blue folder).



## Assign models – right column

Area is splitted to three main blocks:

- Main
- Other
- Preview

Model name\*

Path to element\*

Datasheet

Category

Package name	Software	X	Y	Z
--------------	----------	---	---	---

Save Save As New Clean/New

## Main tab

The screenshot shows the 'Main' tab of the 'Printed Circuit Board Workbench' in FreeCAD. The interface is divided into three main sections, highlighted by orange boxes and numbered 1, 2, and 3.

**Section 1:** This section contains four input fields and a dropdown menu. The fields are labeled 'Model name\*', 'Path to element\*', 'Datasheet', and 'Category'. The 'Category' dropdown is currently set to 'None'. There are also small icons for file operations (a blue 'X' and a globe icon) next to the 'Path to element\*' and 'Datasheet' fields.

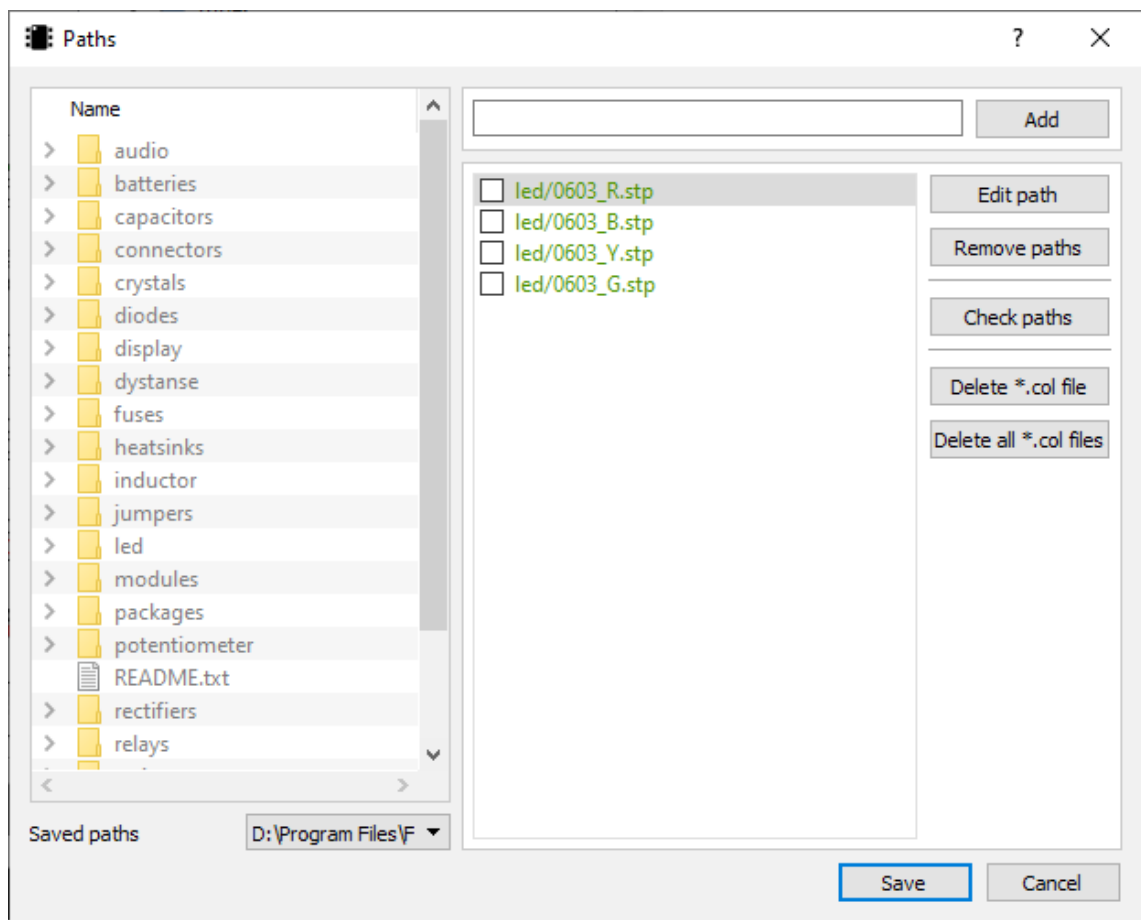
**Section 2:** This section contains a large table for component data. The table has five columns: 'Package name', 'Software', 'X', 'Y', and 'Z'. The table is currently empty, with only the header row visible. To the right of the table, there are four small icons for file operations (a green 'X', a red 'X', a blue 'X', and a globe icon).

**Section 3:** This section contains three buttons: 'Save', 'Save As New', and 'Clean/New'.

## Printed Circuit Board Workbench for FreeCAD

### 1. Basic settings

- **Model name:** any name for model
- **Datasheet:** you can specific path to datasheet for package (url or path to pdf file)
- **Category:** define under what category model will be placed. To keep it without category select 'None'
- **Path to element:** path to assigned 3D models – it is available to assign more then one 3D representation for one model. To add/edit/delete path click button on the right – new window will appear



To add new 3D model just select it in the list on the left and click 'Add'.

Option 'Check paths' will check if previously picked paths still exist (green color → YES, red color → NO).



**If 3D model is under one of pre defined paths (in PCB Workbench preferences) you will see only relative path (no absolute).**



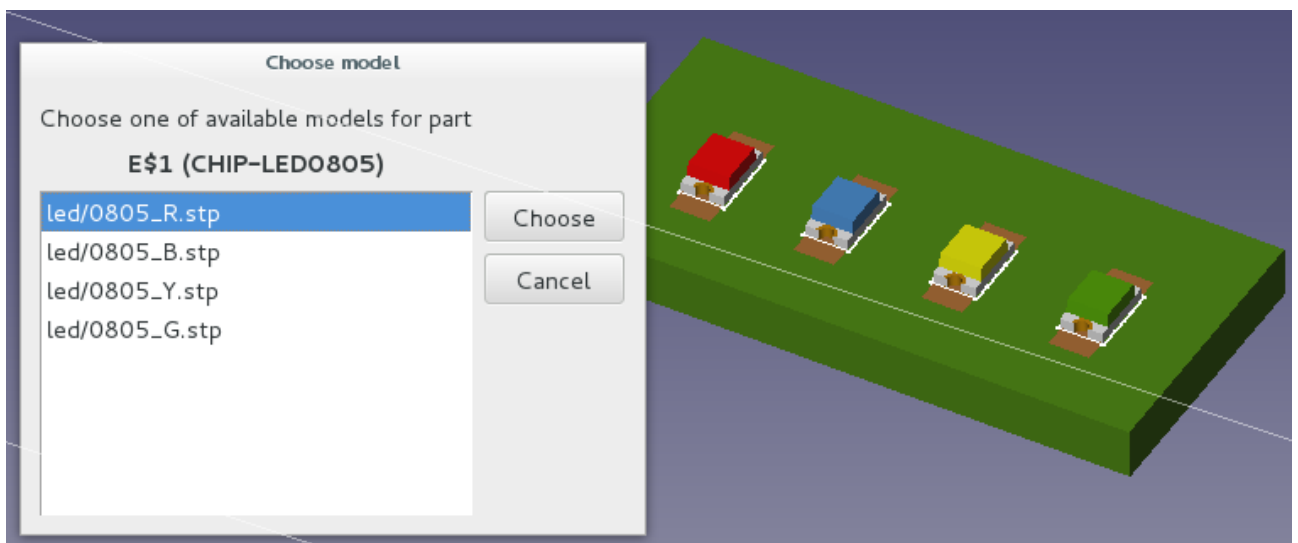
**If something will be not ok with model after loading board (for example no colors) or model will not automatically update despite the new 3D file use function 'Delete \*.col file'.**



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

## Multi model definition for one part

This function is useful for parts which only different is color – the same correction values are set for all models. For packages where we set multi models, special window will appear during board loading or parts updating.




## Printed Circuit Board Workbench for FreeCAD

### 2. Models definitions

For each model you can define as many packages as you need (there are no limits).

Package name	Software	X	Y	Z
R0603-ROUND	Eagle	0.0	0.0	0.2
R0603	Eagle	0.0	0.0	0.2
r_s1608	Razen	0.0	0.0	0.0
r_s1608	FidoCadJ	0.0	0.0	0.0
r_0603	KiCad	0.0	0.0	0.2
SMD0603	IDF	0.0	0.0	0.0
SMD0603_R	IDF	0.0	0.0	0.0
R_0603	KiCad	0.0	0.0	0.2



Buttons from right side will helps you in managing packages:

- 'Add' special window will appear, that allow you to set parameters for new package.
- 'Edit' button will appear window, that will contain all settings for current selected model.
- 'Delete' button will delete from database selected entry.
- Last button allows you to copy existing entry and save it in database under new name



## Adding new package

This field contains package type name taken from software used by you to create PCB boards. →

From drop-down list You need to choose software name for with this entry will be connected. →

X , Y, Z, RX, RY, RZ parameters are correction values used to correctly placement 3D model →

Add new model

Package name

Software Eagle

X

Y

Z

RX

RY

RZ

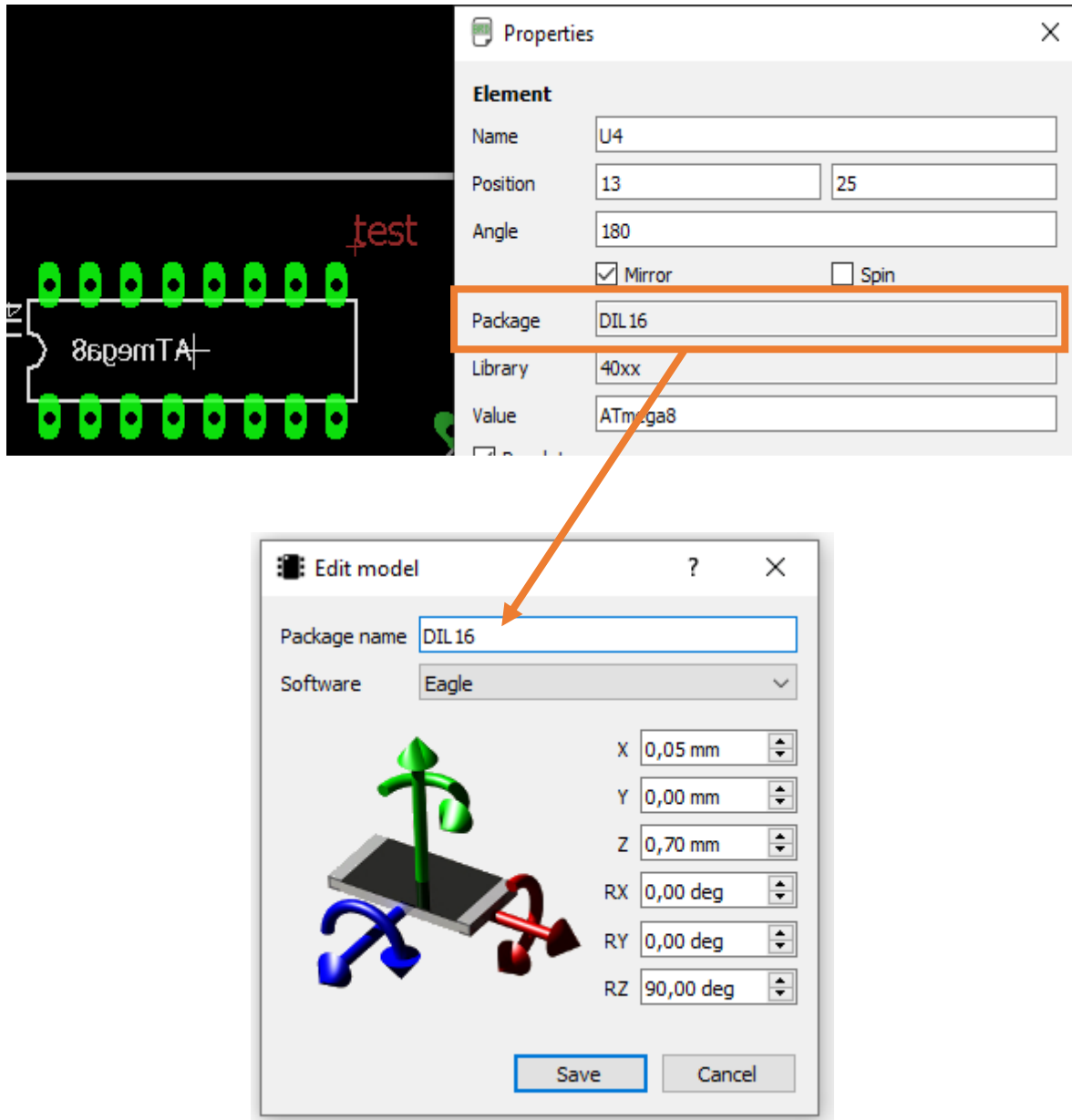
Add Cancel



**Package name is closely related to the software which from PCB files comes**

## Example: Defining a new package DIL16

Eagle: package name = DIL16



### 3. Buttons

To save specified model in database you need to use one of the available on the bottom buttons.

Save Save As New Clean/New

Close button will appear only for GNU/Linux users.

1

MainOtherPreview

	Parameter	Visible	X	Y	Z
<input checked="" type="checkbox"/>	Name	True	-7,54mm	0,08mm	2,57mm
<input checked="" type="checkbox"/>	Value	True	0,51mm	0,63mm	2,57mm

☐ Add socket

SocketPW80\_14FF\_2Z

☐ Set as socket

Height10,00 mm

Description

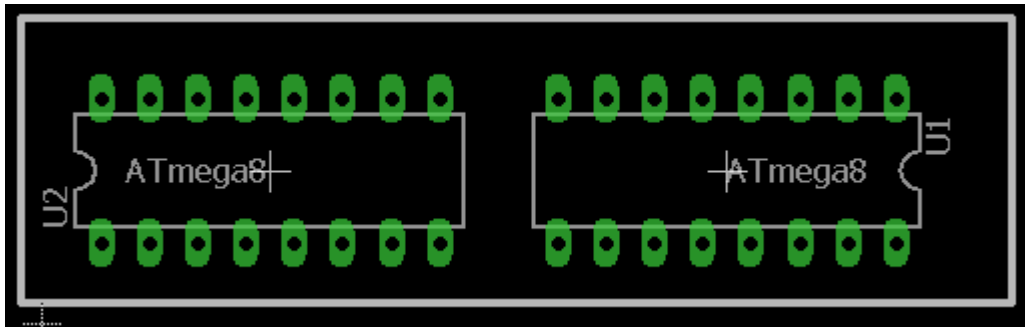
2

3

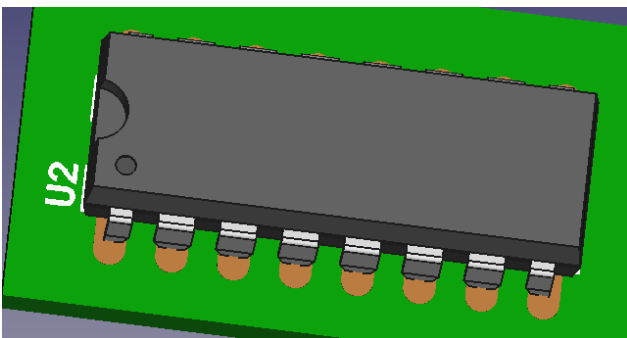
4

1. **'Adjust part name/value':** option allows to automatic placing objects name/value in specific position.

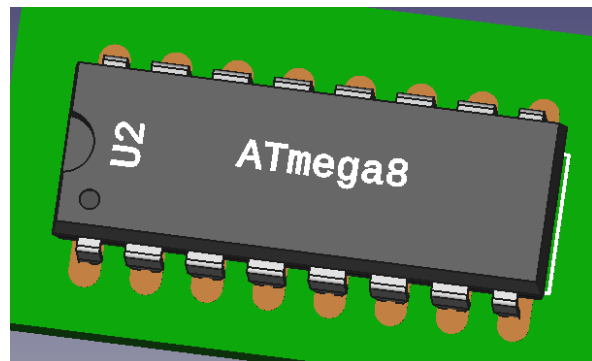
**Example for DIL16:**



Board created in external software.



'Adjust part name/value' = OFF



'Adjust part name/value' = ON

Board view after importing to FreeCAD.

	Parameter	Visible	X	Y	Z	RZ	
<input checked="" type="checkbox"/>	Name	True	-7,54mm	0,08mm	2,57mm	-270,00deg	1
<input checked="" type="checkbox"/>	Value	True	0,51mm	0,63mm	2,57mm	0,00deg	1

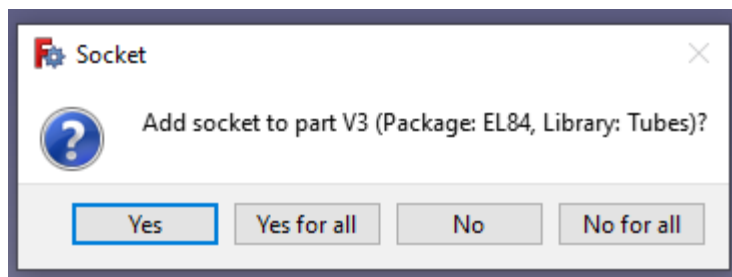
Parameter activation

## 2. Set socket for model

To add socket for model just mark checkbox for 'Add socket' and from drop down list choose socket 3D model name. In drop down list you will find only models marked before as sockets

☐ **Add socket**  
Socket

For model where socket was specified special window will appear.



## 3. Set model as socket

To set model as socket just mark checkbox for 'Set as socket' sign. Enter the height of the socket in the spinbox.

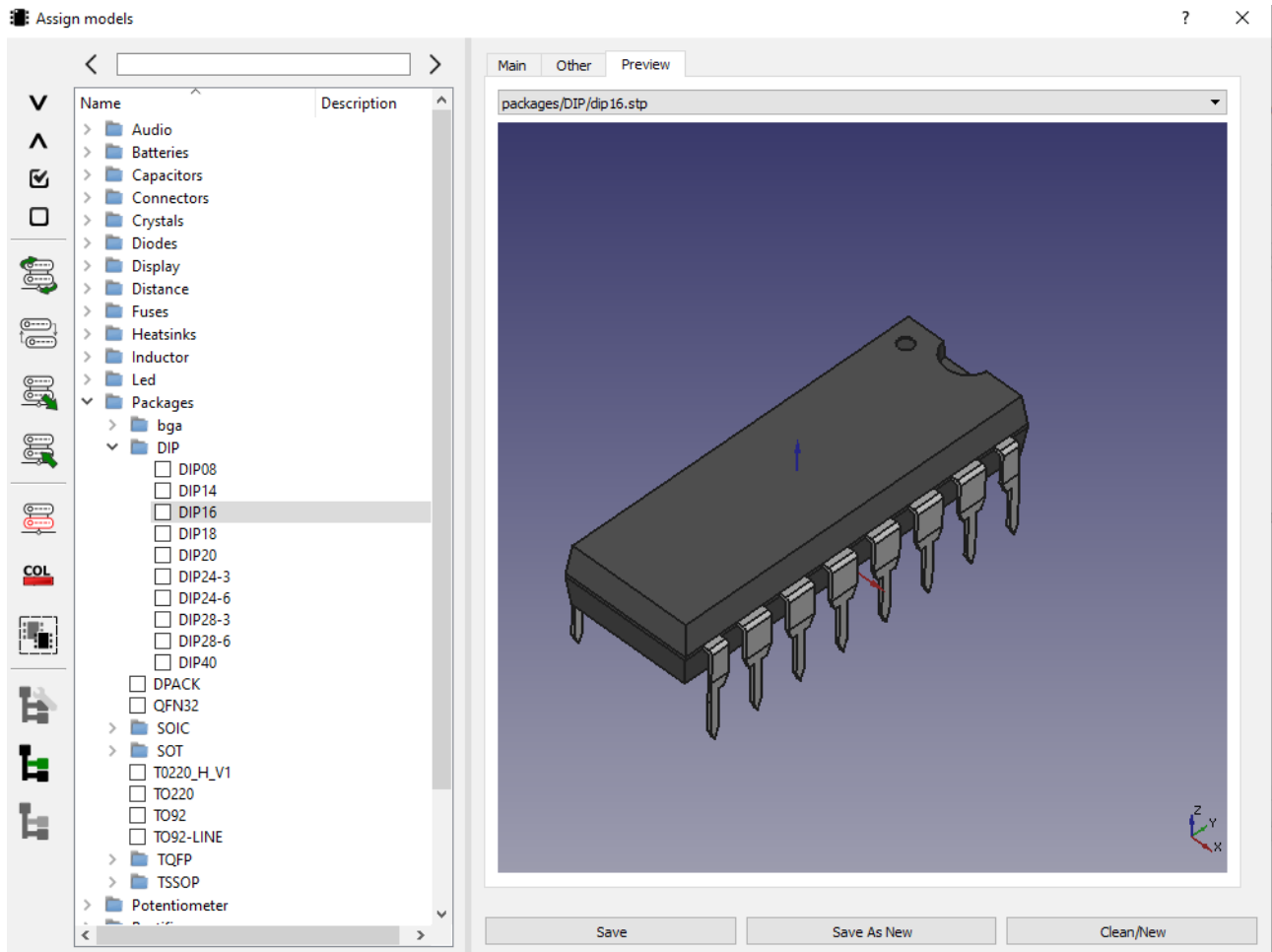
☐ **Set as socket**  
Height

## 4. Description

## Printed Circuit Board Workbench for FreeCAD

### Preview tab

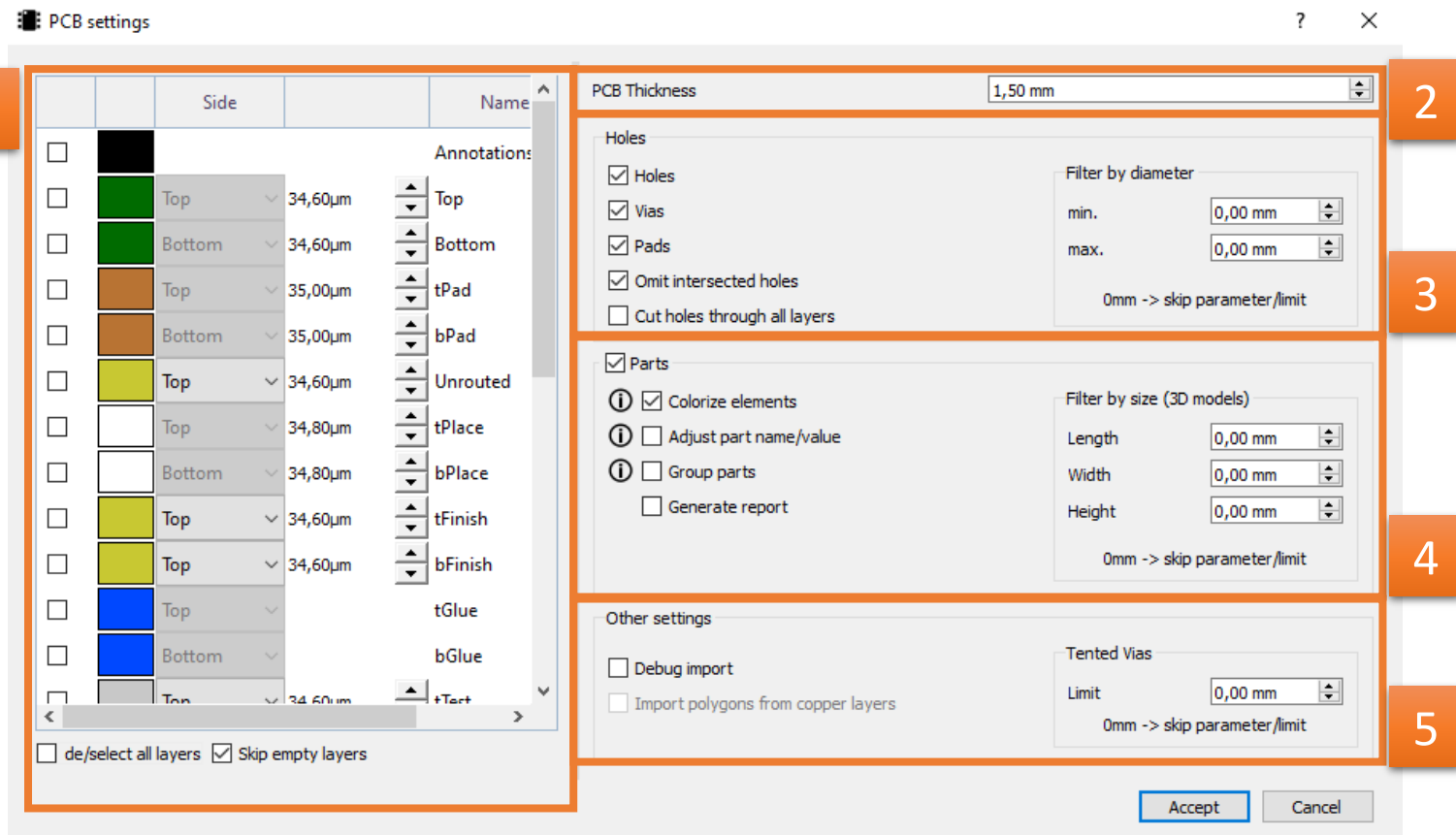
It is possible to see 3D model in last tab - 'Preview'.



## WORKING WITH WORKBENCH

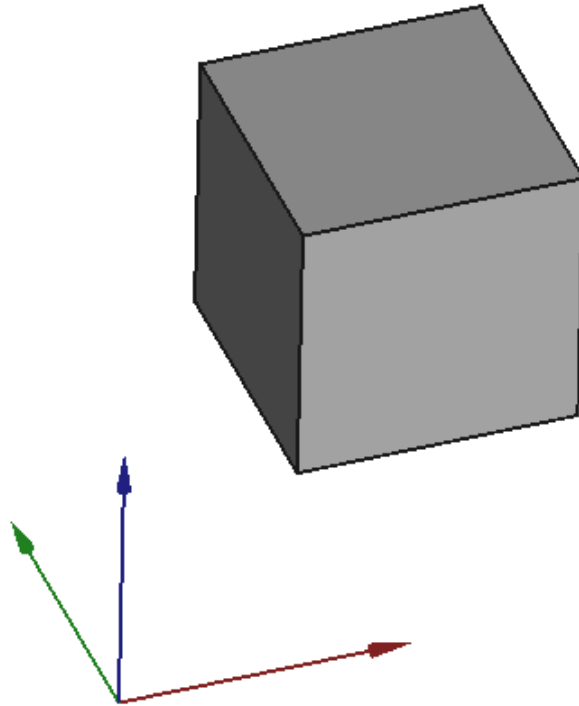
# OPENING/IMPORTING BOARD

During the opening / importing process, a special window will appear in which you can set the basic parameters of the board.



1. In first section You can choose, which layers will be loaded. Available layers depends from loading file type. Layer name and color are editable.
2. This section allow You to set PCB thickness. If file contain board thickness this value will be displayed in this field. Default value is 1.5[mm].
3. Third section contain basic settings about importing holes. Here You can decide what type of holes You want to import (hole/vias/pads) and set imported holes diameter range (min/max). Both parameter can be set separately.
4. Fourth area contains basic settings about importing parts. Here You can decide if You want to import parts, decide if they should contain colors, etc. Fields L/W/H allow You to decide about minimum length/width/height of 3D models which will be imported. All three parameter can be set separately.





## 5. Other settings

### Unit system

During board loading process units are changed to millimeters [mm].



**To skip a specific filter just set it to 0.**

# CREATING BOARD FROM SCRATCH

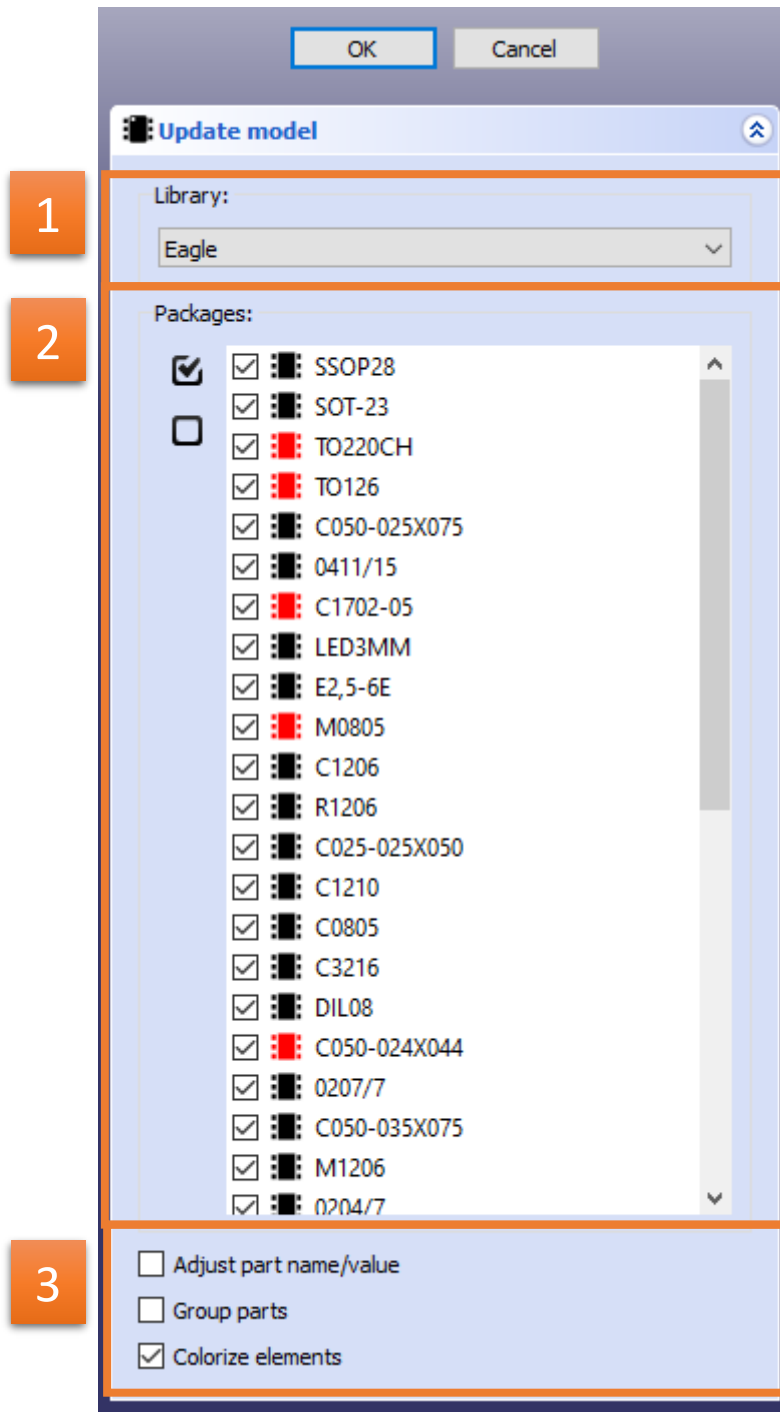
## CREATING GLUE PATHS

## ADDING ANNOTATIONS

## ADDING NEW MODELS

## UPDATING MODELS

Update models window will reload/load 3D model/settings for used in project components.



## Printed Circuit Board Workbench for FreeCAD

Update models tab contain three sections:

1. Library: during update process, script will search settings (eg. X, Y, Z values) in specific library,
2. Packages: contain listbox with used in project components. Checked checkbox next to model type mean that this part will be updated.
3. Configuration options:
  - Adjust part name/value – set Name/Value annotation values according to settings set in database.
  - Group parts: grouping parts in tree according to Categories.



**When selected component does not appear in specified library,  
model will be not updated.**

## CREATING CONSTRAINST AREAS



# GENERATING BOUNDING BOX

## CREATING SECTION CUTS

## EXPORTING HOLE LOCATIONS

## EXPORTING HOLE LOCATIONS REPORT

## CREATING DRILLING MAP

# BOM

# CENTROID

# EXPORTING BOARD



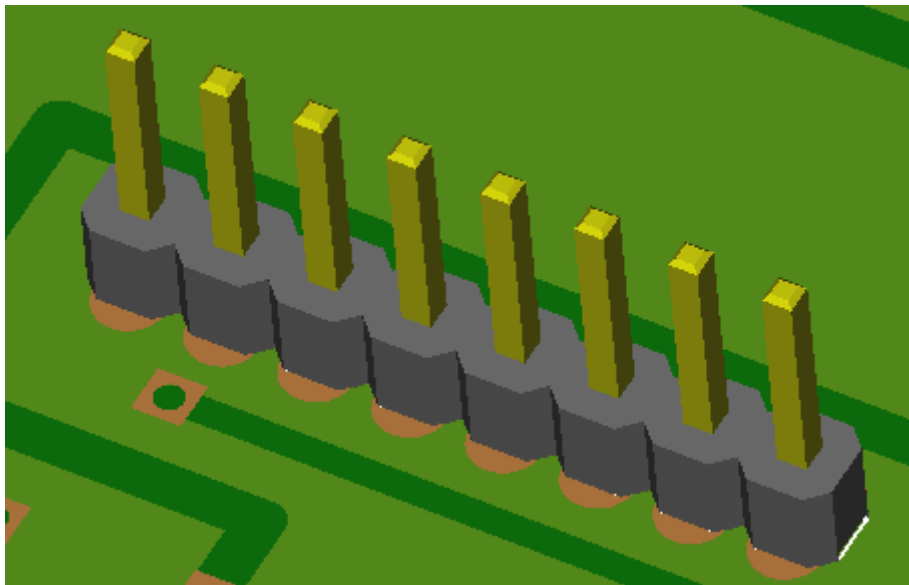
## VIEW OPTIONS

# DISPLAY MODES

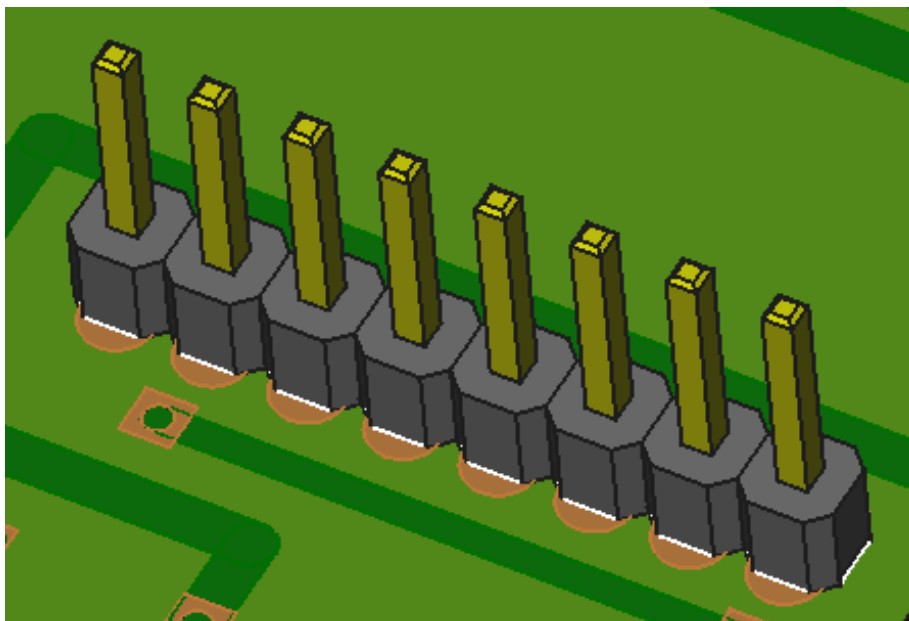
The Display Modes function allows you to quickly and easily change the display representation of shapes in your project.

Available types:

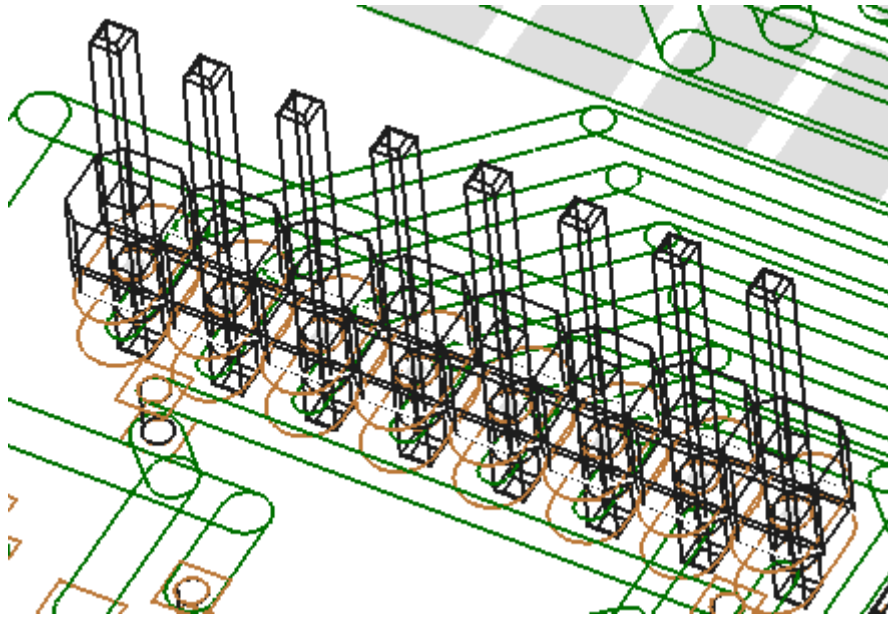
1. Shaded: border lines are hidden.



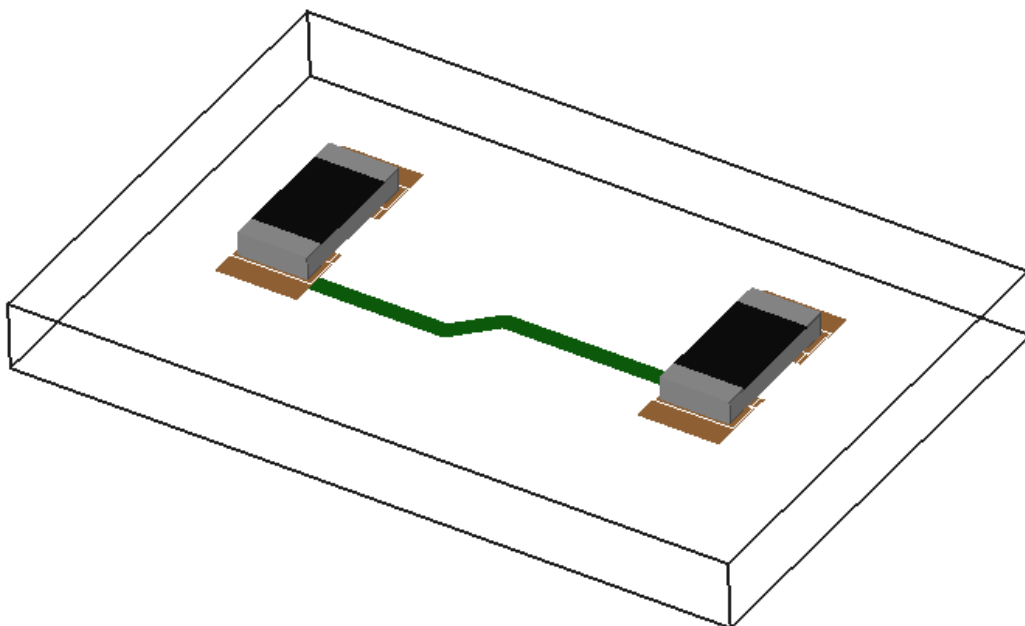
2. Flat lines: surfaces and border lines are displayed in one time.



3. Wireframe: only border lines are displayed.

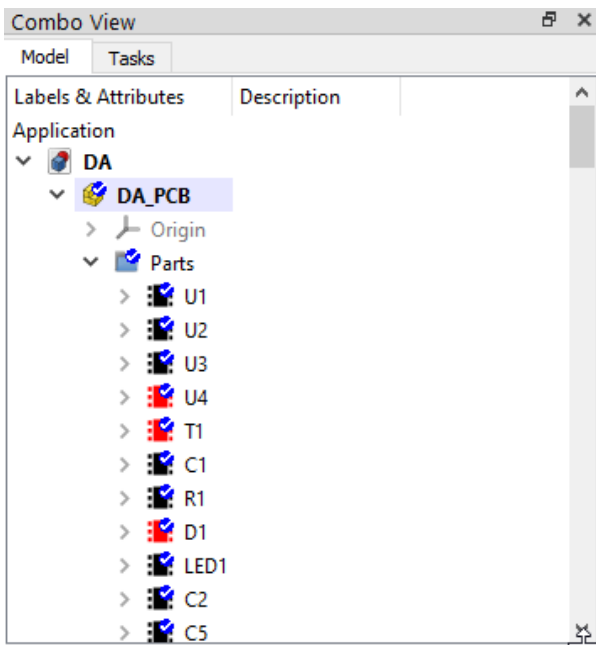


4. Internal View: for board only border lines are displayed, rest is displayed in Flat lines mode

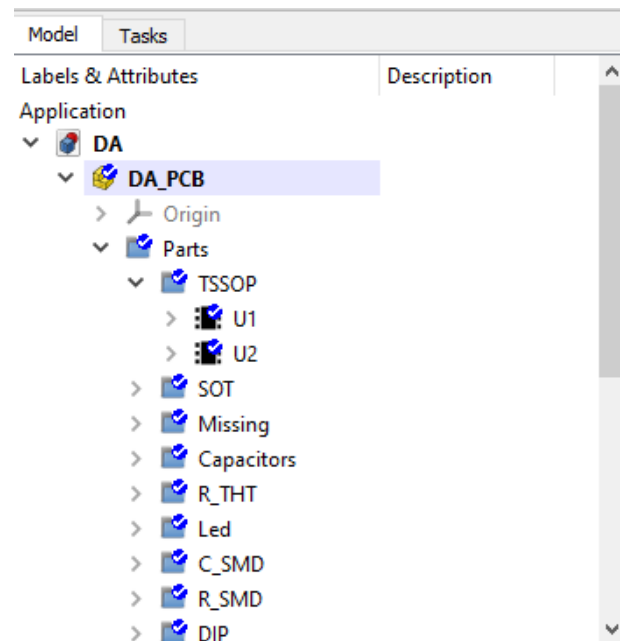


# GROUPING PARTS

These options allow you to group/ungroup parts according to the categories they belong to (parameters stored in the database).



Ungrouped parts



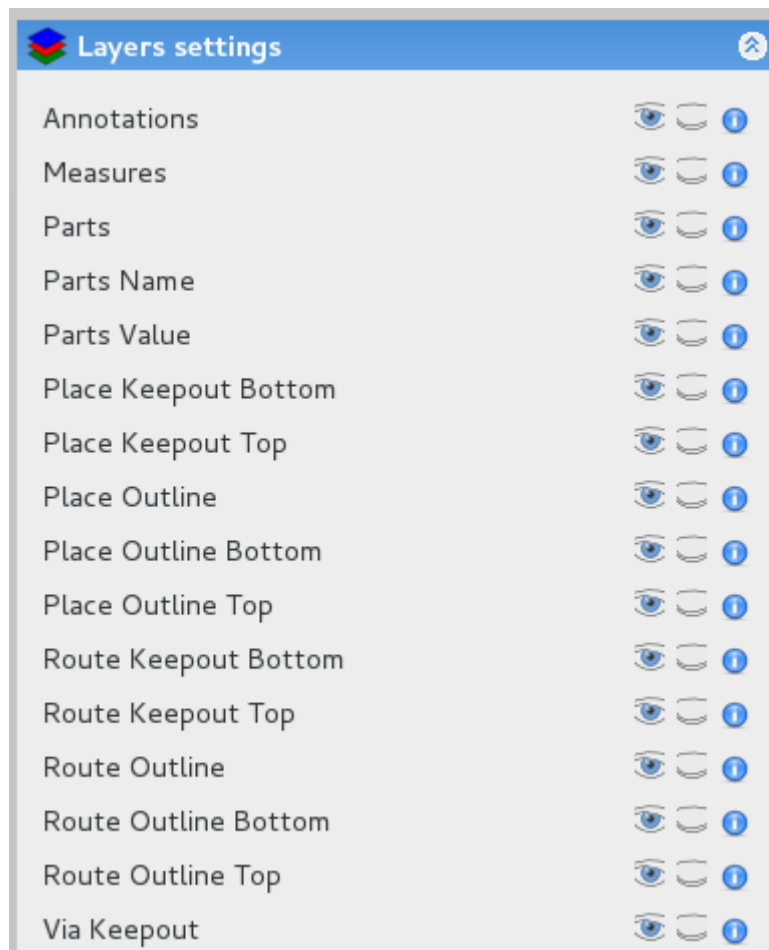
Grouped parts

These options are also available in:

- open/import window,
- update parts window,
- add new model window.

# LAYERS

The layer settings window helps you manage the currently displayed layers of the board. The layer settings window appears on the Task tab.



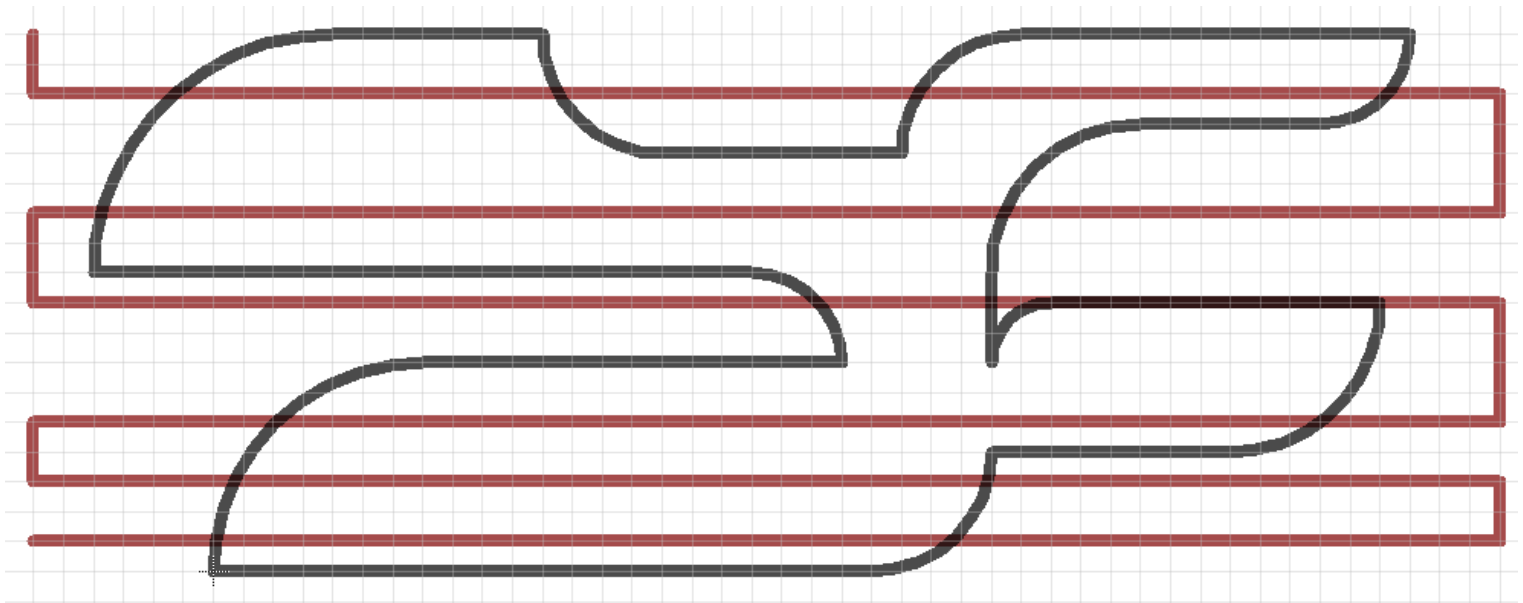
Each line consists of four parts:

- Layer name,
- Button Show All – show all objects of this type,
- Button Hide All – hide all objects of this type,
- Information button – display information about layer.

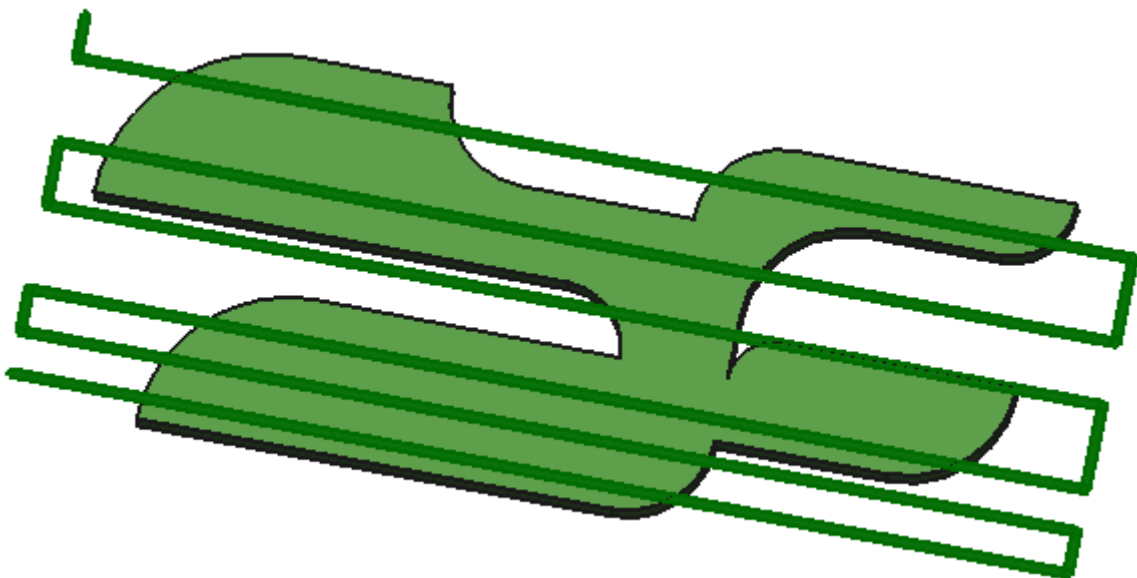
## CUT TO BOARD OUTLINE

Sometimes it is necessary to display board like it will look after manufacturing. To do this just use option 'Cut to Board Outline'. Function will automatically blank/display all layers/paths that are outside of the board.

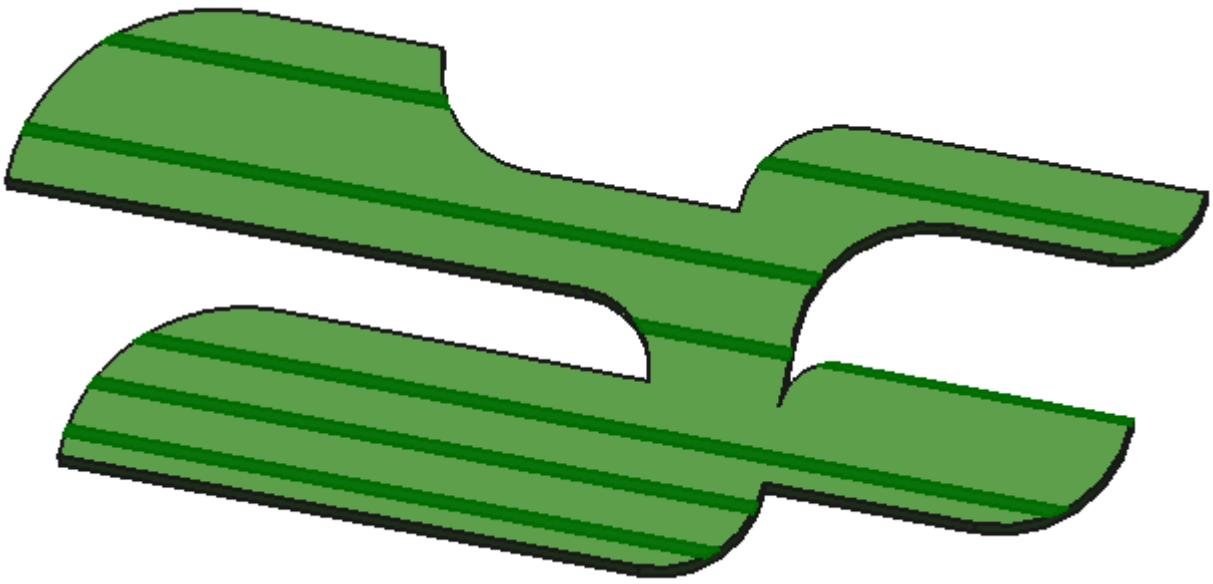
### Example



Board created in Eagle



Cut to Board Outline = OFF



Cut to Board Outline = ON

# HOLES SETTINGS



# SIGNALS MARKING

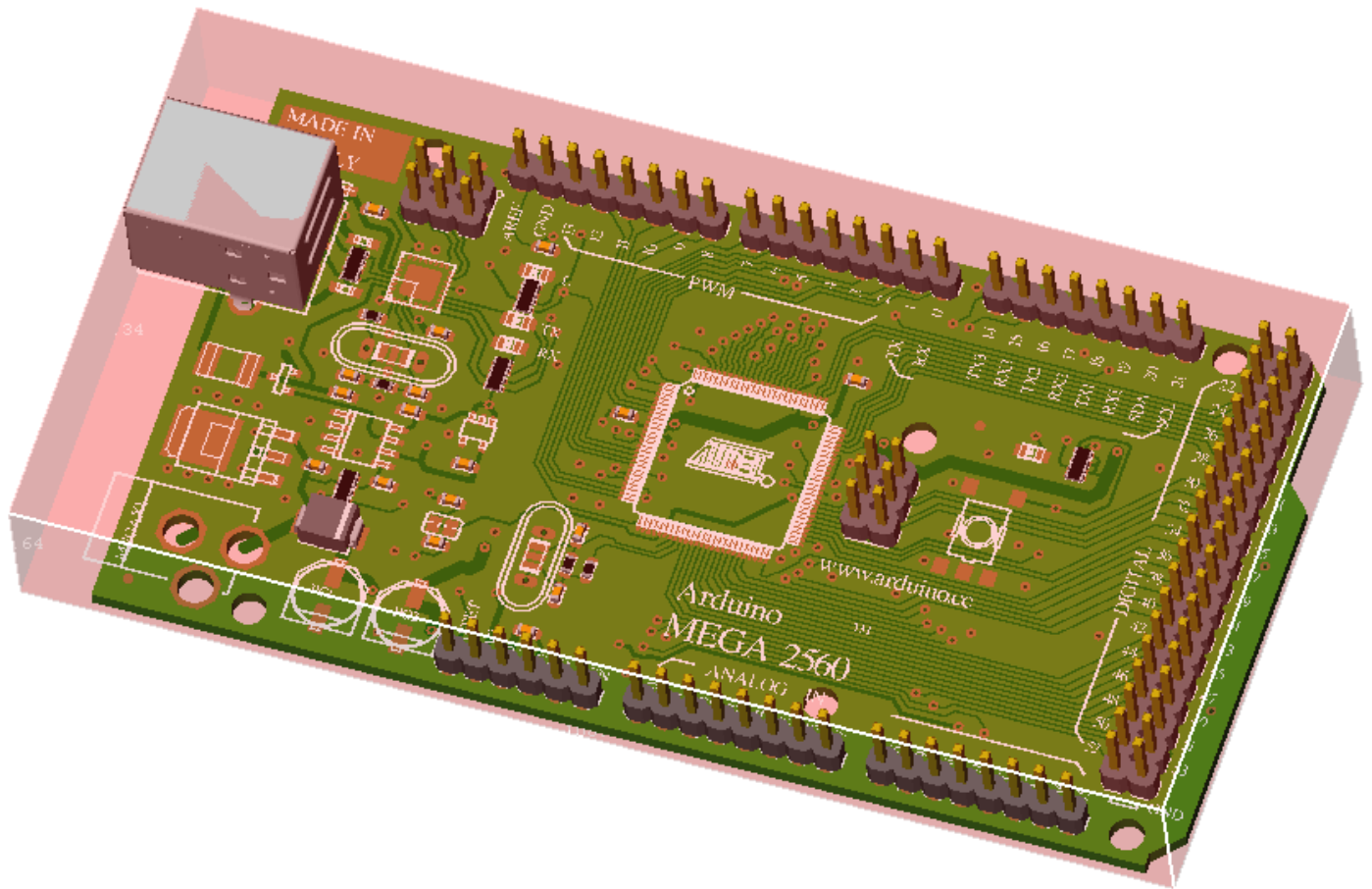
# EXPLODE

## BOUNDING BOX

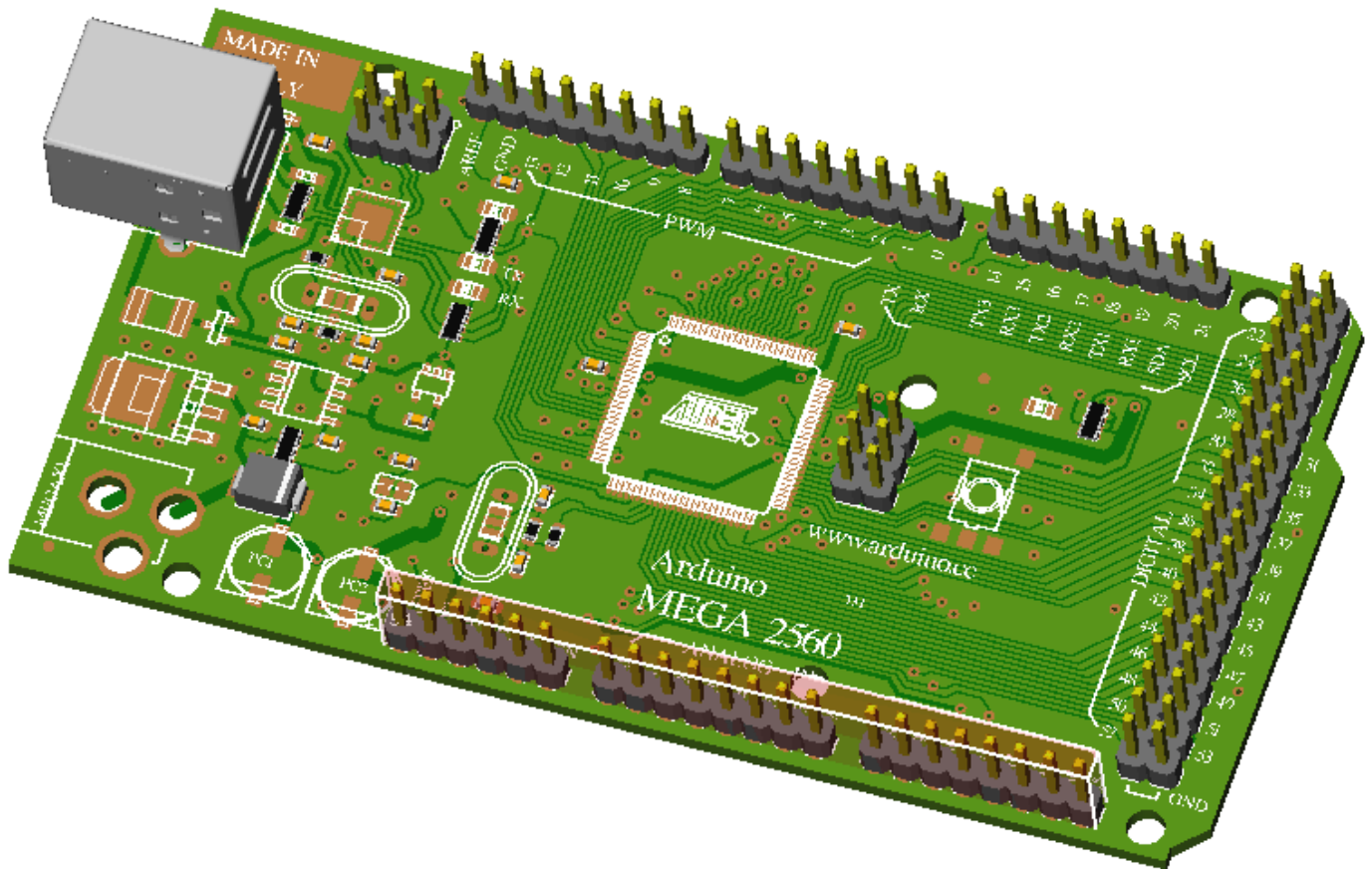
Bounding box is a smallest cuboid completely surrounds the object.

Printed Circuit Board workbench contain two function to generate bounding box:

- Bounding box – generate box for all board (board, parts, paths),
- Bounding box from selection – generate box for selected components.



Bounding box generated for whole board



Bounding box generated for selected components

Generated boxes are normal cubes so it is possible to work with them in FreeCAD.



**You can generate as many bounding boxes, as you need..**

## RENDERS

# KERKYTHEA

# POV-RAY

## OTHER



## GENERATE MODELS

# SCRIPTS

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

## Eagle

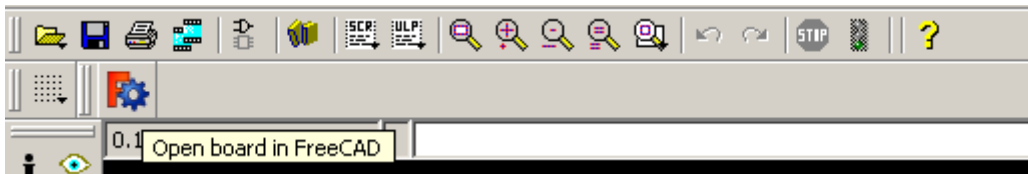
Directly exporting boards from Eagle to FreeCAD [path: scripts/eagle]

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.

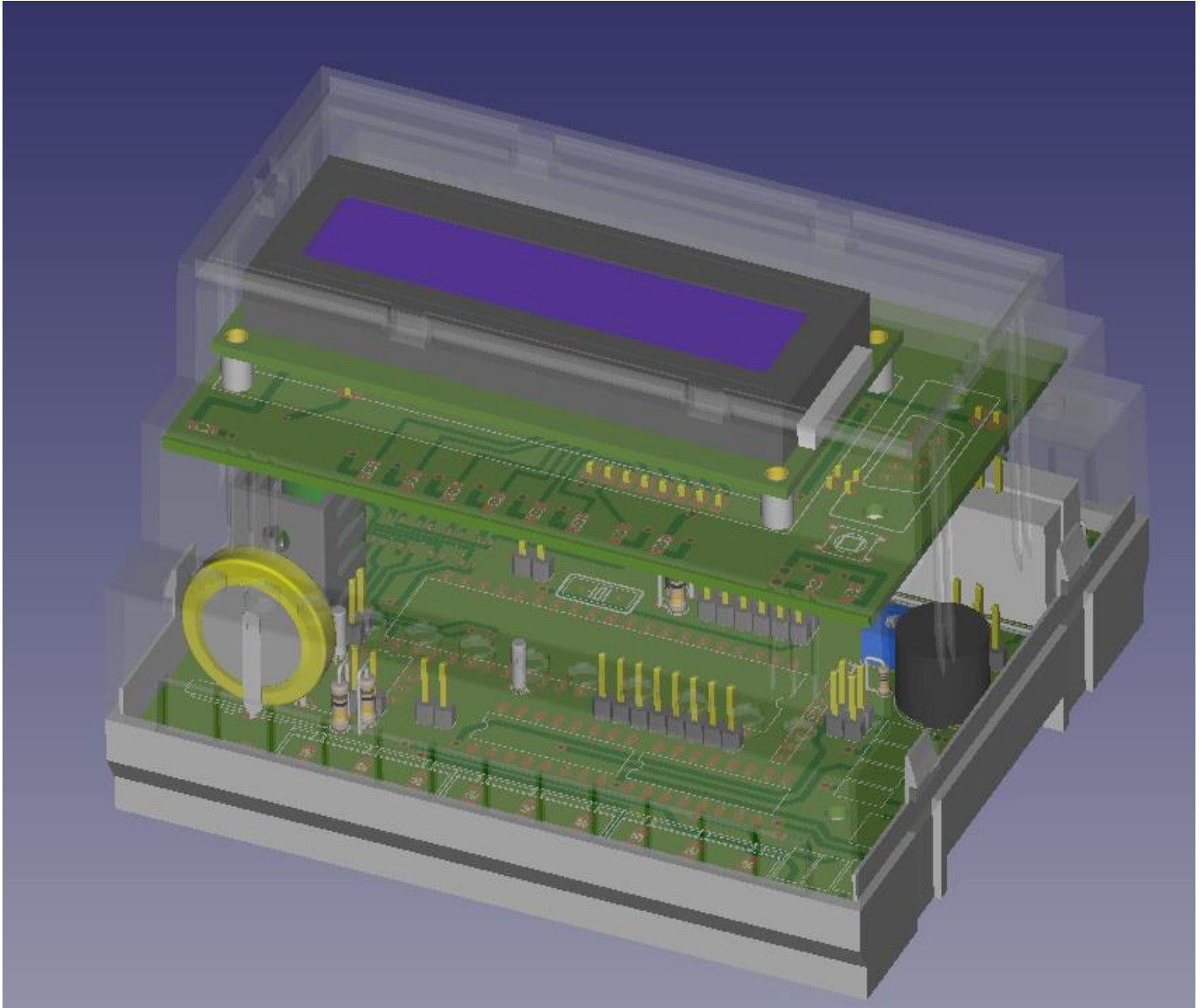


On Linux to set path to FreeCAD change value of var 'programPath\_LIN' in file freecad.ulp.

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



## Printed Circuit Board Workbench for FreeCAD

