

FreeCAD ~ a free CAD system that is excellent ~ notesFreecadPython.doc

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www.illustratingshadows.com

FreeCAD is a free CAD system that supports 3d and 2d, it is more focused on using scripting, however the Illustrating Shadows Python scripts show FreeCAD is an equal to DeltaCAD in many cases. It is not well suited to manual drafting where DeltaCAD and TurboCAD excell.

First, go to this web page:-

http://www.freecadweb.org/wiki/index.php?title=Main_Page




click on DOWNLOADS which is:-

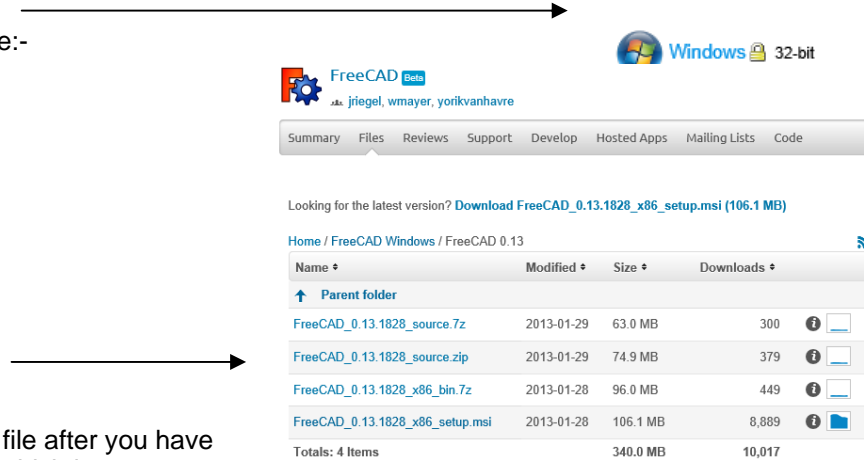
<http://www.freecadweb.org/wiki/index.php?title=Download>

Official Installers

Stable FreeCAD installers

The FreeCAD team provides ready-to-install packages for  Windows (32 and 64bits). Below are links to the Stable installation files for the various features being worked on, but might crash more often or contain features

select the .msi package:-

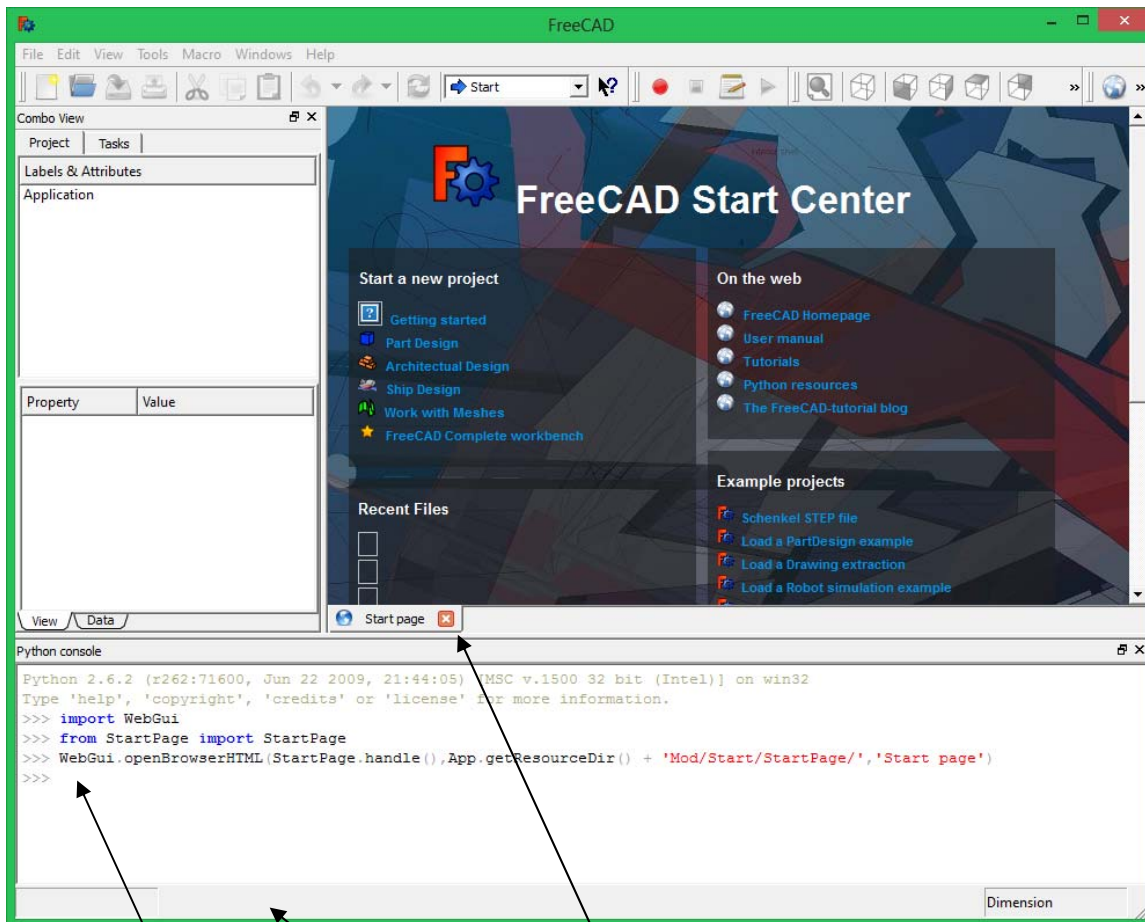


Then execute the .msi file after you have verified it is virus free, which it was.

At this point, you now have FreeCAD installed.

Useful scripting web pages are something like:- <http://www.freecadweb.org/wiki>

Bring up FreeCAD



click to get rid of unwanted screens

paste the Python scripts here

Syntax error messages appear at the screen bottom but only for a very short time. If the error is unbalanced parentheses, then the line indicated is the line after the line with the error.

Once a macro works, it can be installed:-

click MACRO and in the pull down window,

click MACRO then click

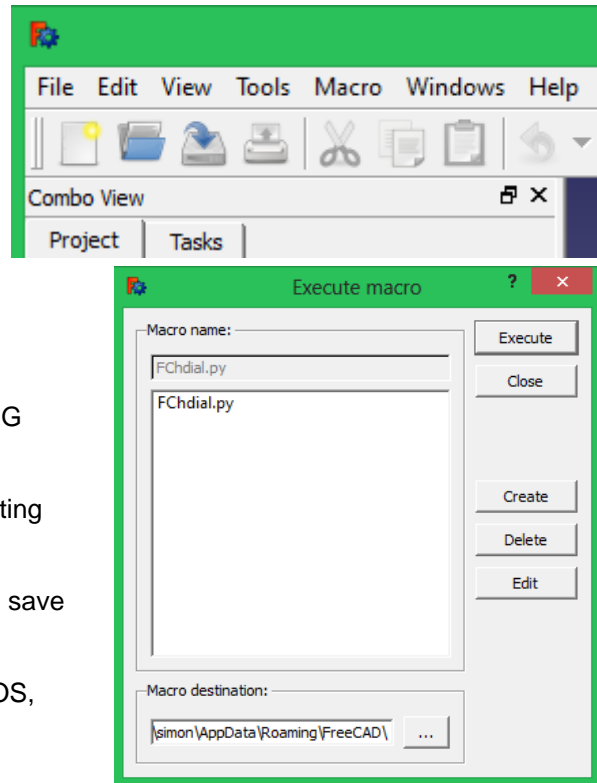
click CREATE and then

enter a macro name which causes an EDITING panel to be created, then

paste your macro or Python script into the editing window

then close the window which will prompt you to save the macro.

From then on you may click MACRO, MACROS, click on the name and either click on EDIT or EXECUTE.



This makes it easy to edit a macro, test it, and subsequently edit it. Useful links are:-

Use of the built in macro editor

<http://www.freecadweb.org/wiki>

Creating user interaction:

<http://www.freecadweb.org/wiki>

Python tutorial for FreeCAD

<http://docs.python.org/2/tutorial/index.html>

TO SET THE BACKGROUND COLOR TO WHITE:-

EDIT

PREFERENCES

DISPLAY

COLORS

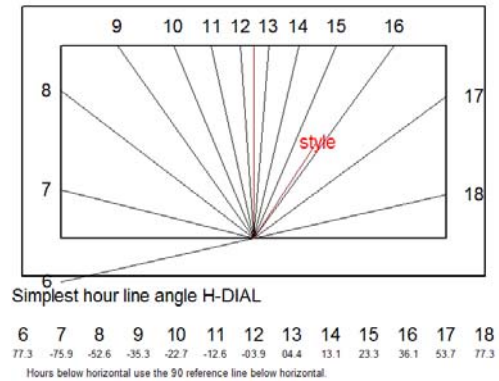
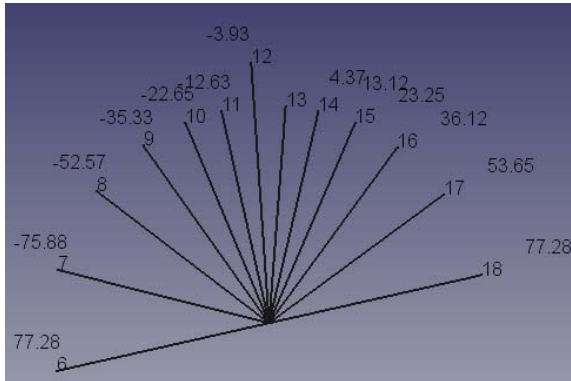
BACKGROUND

choose for example white

For a dial at the specified:

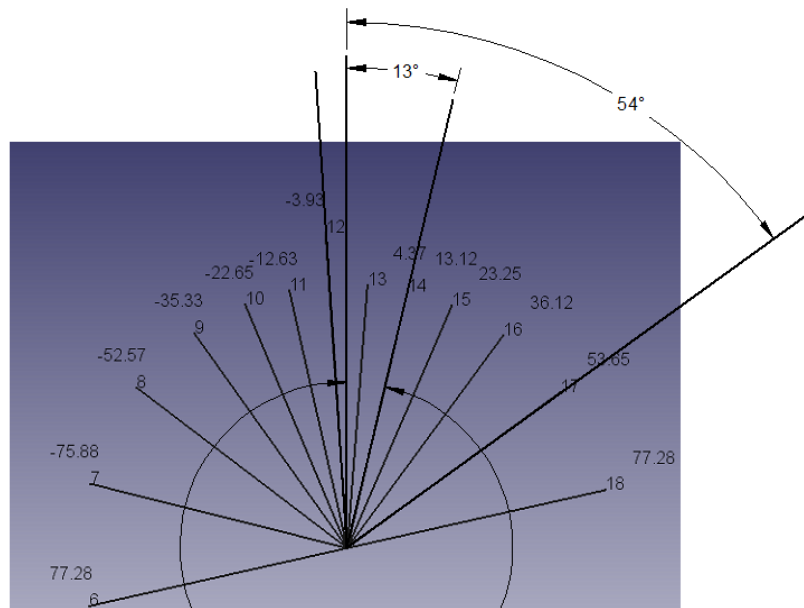
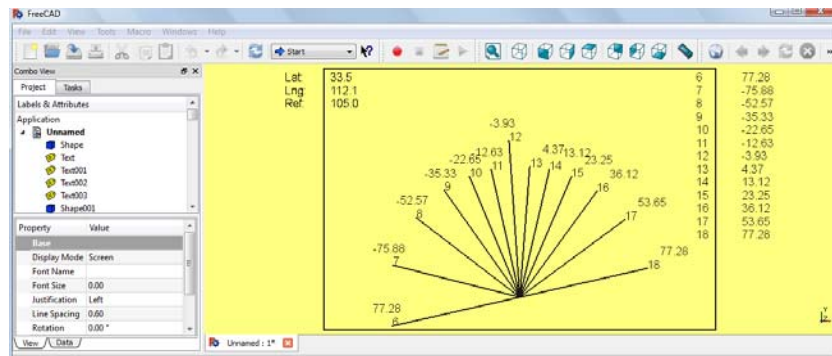
lat=33.5 long=112.1 legal=105

freeCAD then provides using the Python script: FChDial.py (below left)



which compares favorably with the DeltaCAD macro (above right).

The aspect ratio should always be verified, however, when captured with a screen capture and subsequently tested, the angles appeared to be correct.



Hints and kinks of Python and FreeCAD ~ how to fix things

Initial Python macro loading and parsing errors that are displayed, and clues to avoid them. Syntax error messages appear on the bottom of the screen for a second or two.

- Indentation is critical
- Case is critical
- IF a == 2: construct is critical both for the “==” and the “:”
- WHILE construct is equally critical
- If the syntax error is unbalanced parentheses, then the line indicated is the line after the line with the error.
- do not use tabs in the editor, use spaces

Execution errors are hard to detect, here are some clues and ideas

- Never do a Draft..makeText after a mesh= UNLESS you have drafted a graphic object first. If you do, the Python program will halt and you will have no idea what happened.

```
mesh = Mesh.Mesh()
c = Part.makeLine((0,0,0), ( 1,1,0))
Part.show(c)
Draft.makeText("+", FreeCAD.Vector(xc,yc,0),"TRUE")
```

Debugging is frustrating, and one technique is to add “Draft.makeText” at strategic places, and hope for the best. I suggest this as TOGGLE BREAKPOINT and DEUBG MACRO do not seem to function.

Also, to see execution errors, EDIT, PREFERENCES, GENERAL, OUTPUT WINDOW, and check the bottom redirect statements. And make sure the VIEW, VIEWS, REPORT VIEW is checked, when errors appear on the bottom left if execution related. Nowhere does HELP suggest this, at least, not that I could find.

However, this is still an excellent program.

