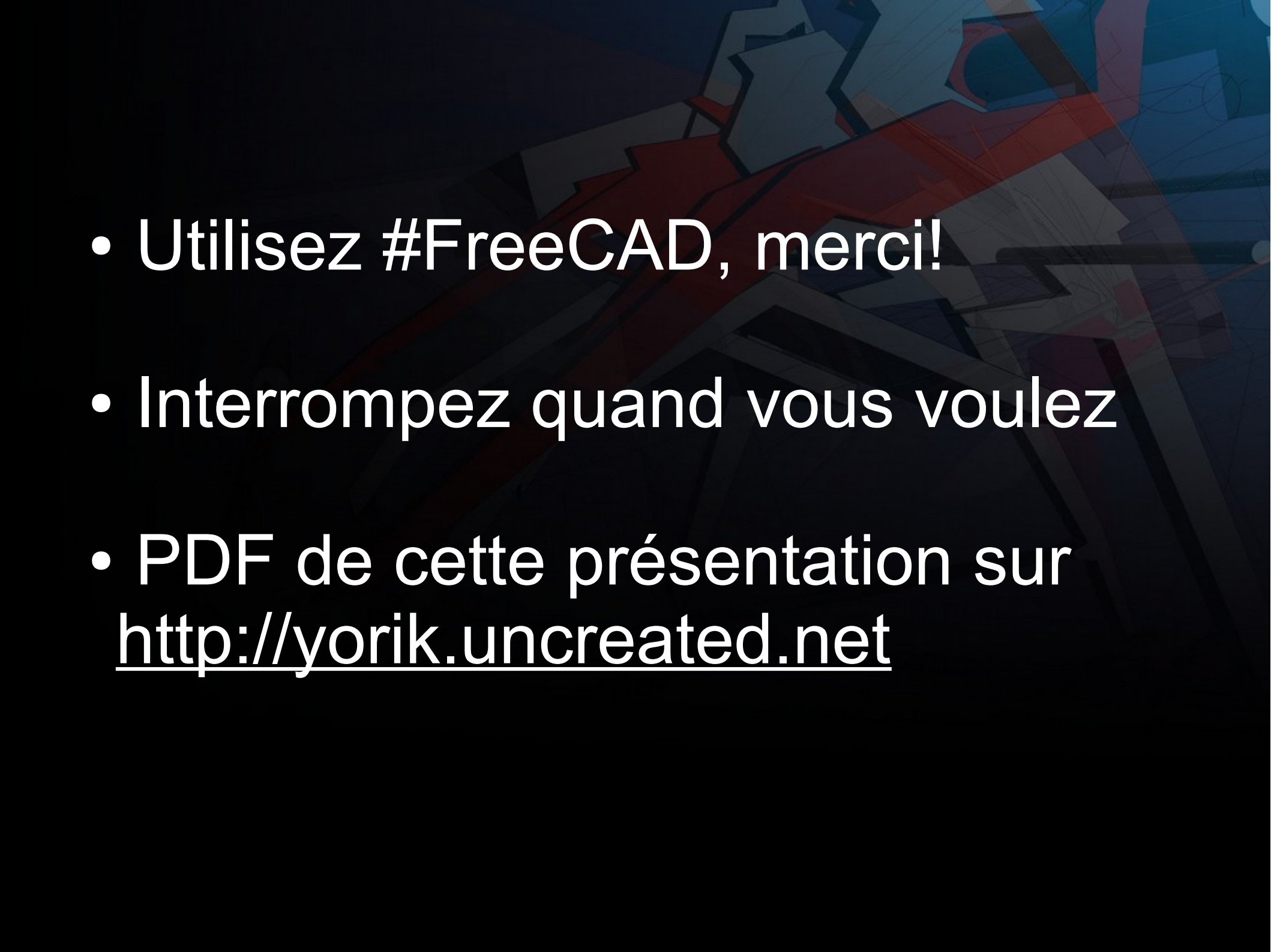




FreeCAD

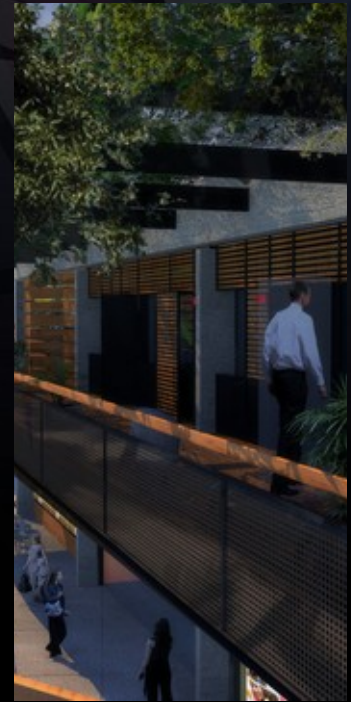
Une plateforme de design hackable

Yorik van Havre
Capitole du Libre, Toulouse, 2015

- 
- Utilisez #FreeCAD, merci!
 - Interrompez quand vous voulez
 - PDF de cette présentation sur <http://yorik.uncreated.net>

Yorik van Havre

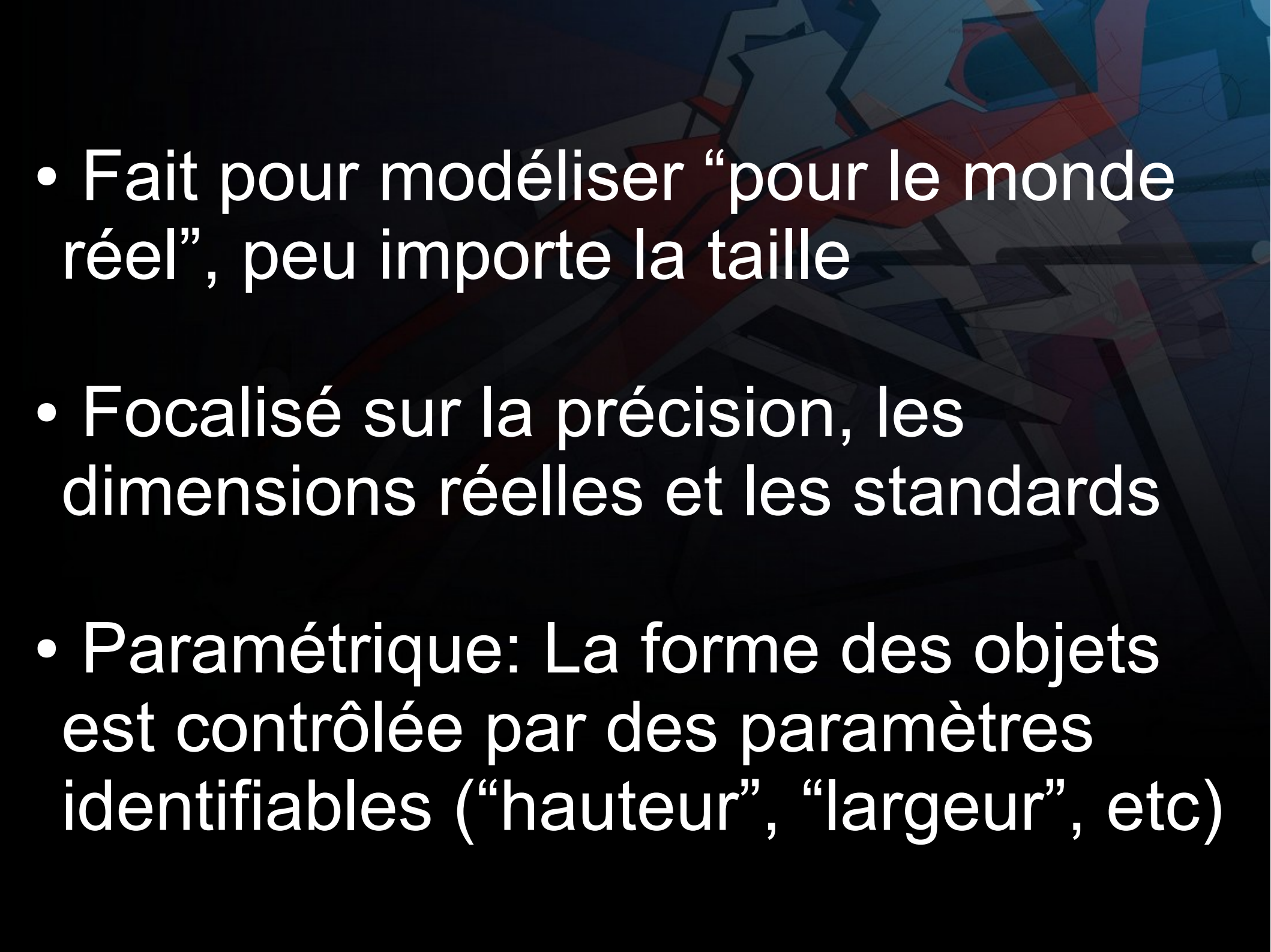
- Architecte (de bâtiments, pas d'information!), et un des développeurs de FreeCAD
- <http://yorik.uncreated.net>
- yorik@uncreated.net
- [@yorikvanhavre](#)

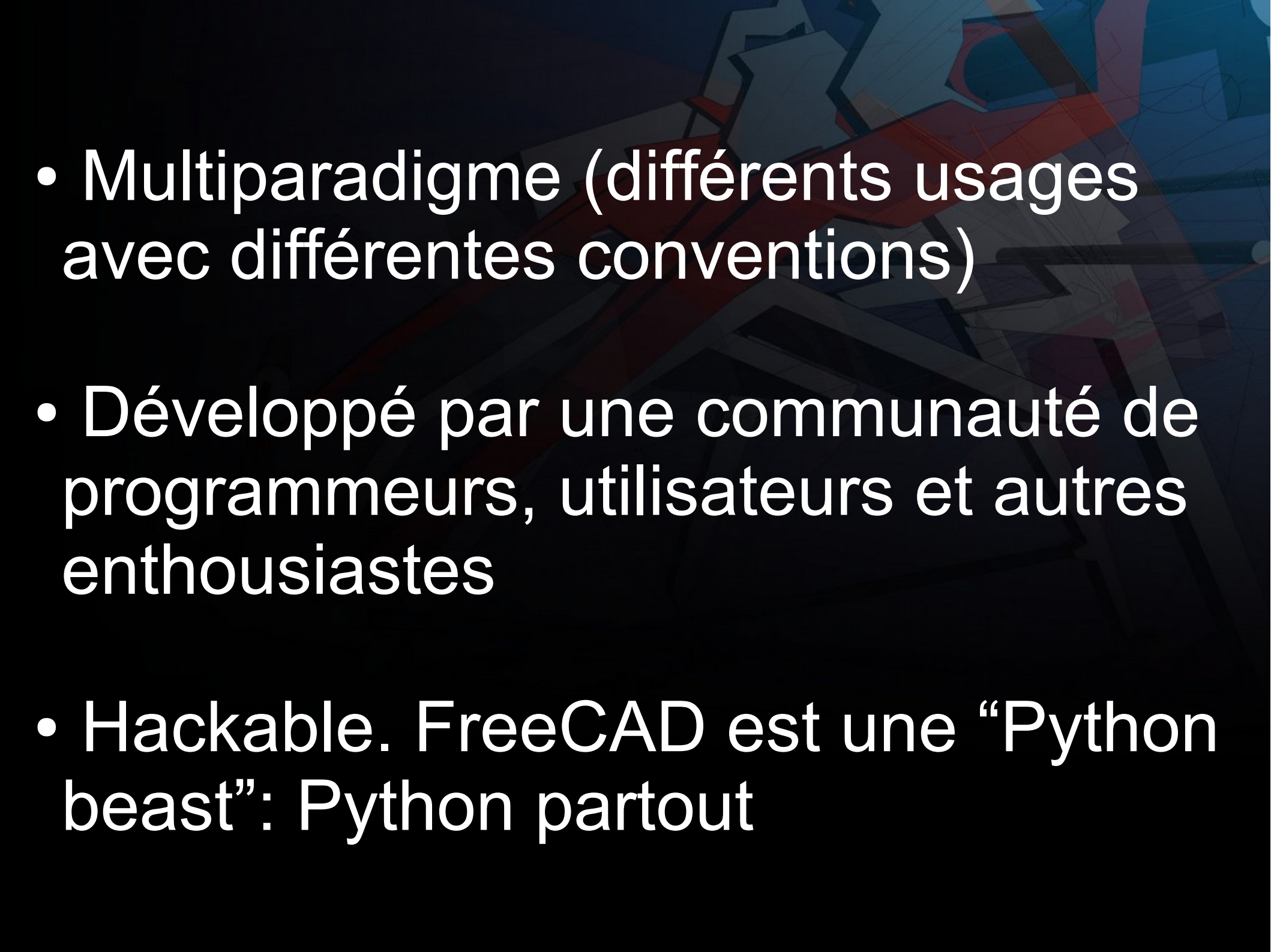


<http://www.uncreated.net>

FreeCAD

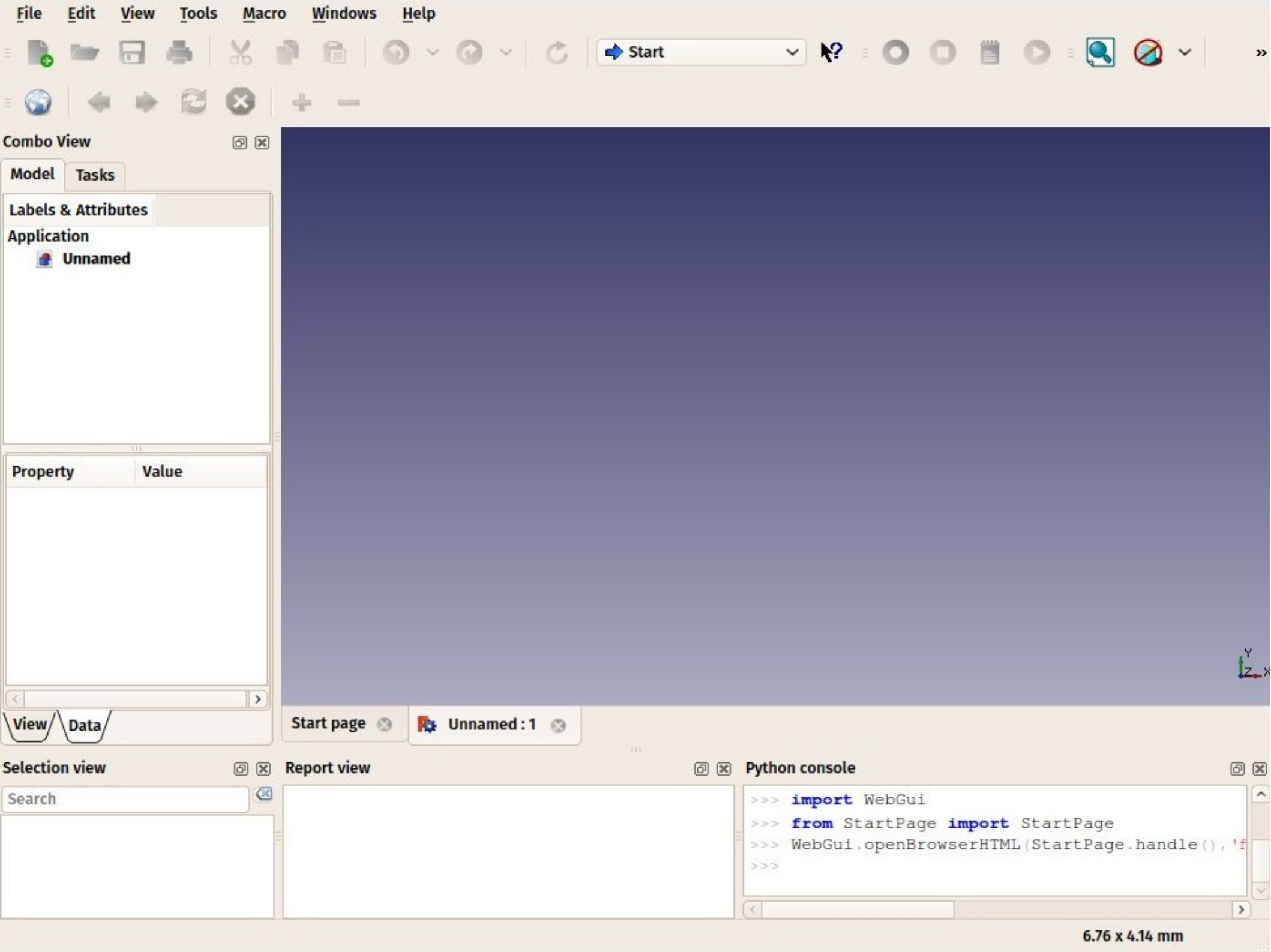
- Application de modélisation 3D
- Libre (license LGPL)
- Multiplateforme (Mac, Windows, Linux)

- 
- Fait pour modéliser “pour le monde réel”, peu importe la taille
 - Focalisé sur la précision, les dimensions réelles et les standards
 - Paramétrique: La forme des objets est contrôlée par des paramètres identifiables (“hauteur”, “largeur”, etc)

- 
- Multiparadigme (différents usages avec différentes conventions)
 - Développé par une communauté de programmeurs, utilisateurs et autres enthousiastes
 - Hackable. FreeCAD est une “Python beast”: Python partout

<http://www.freecadweb.org>

- Homepage avec info, téléchargement, etc.
- Documentation (wiki)
- Forum TRÈS actif (← allez-y!)
- Bug tracker





Combo View



Model Tasks

Labels & Attributes

Application

Unnamed

Property Value

View Data

Selection view



Search

Report view

Python console

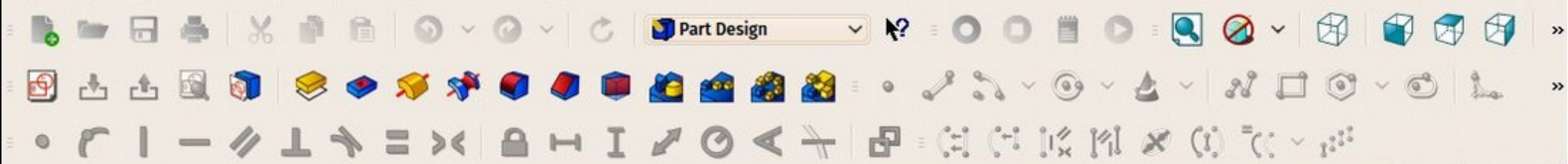


```
>>> import WebGui
>>> from StartPage import StartPage
>>> WebGui.openBrowserHTML(StartPage.handle(), 'f
>>>
```

- Arch
- Assembly
- Complete
- Draft
- Drawing
- FEM
- Image
- Inspection
- Mesh Design
- <none>
- OpenSCAD
- Part Design
- Part
- Path
- Plot
- Points
- Raytracing
- Reverse Engineering
- Robot
- Ship
- Sketcher
- Spreadsheet
- Start**
- Test framework
- Web

Start page

Unnamed : 1

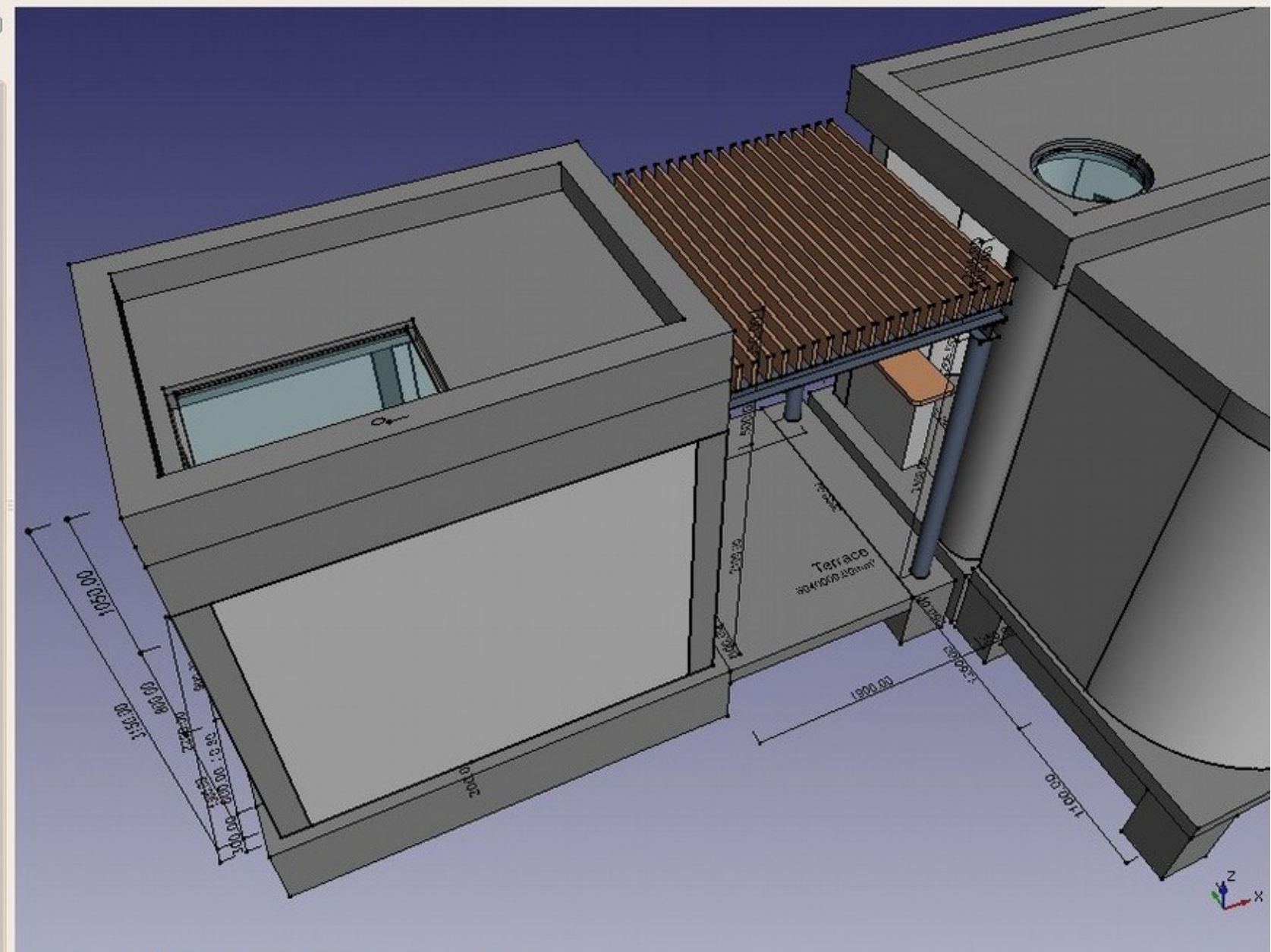


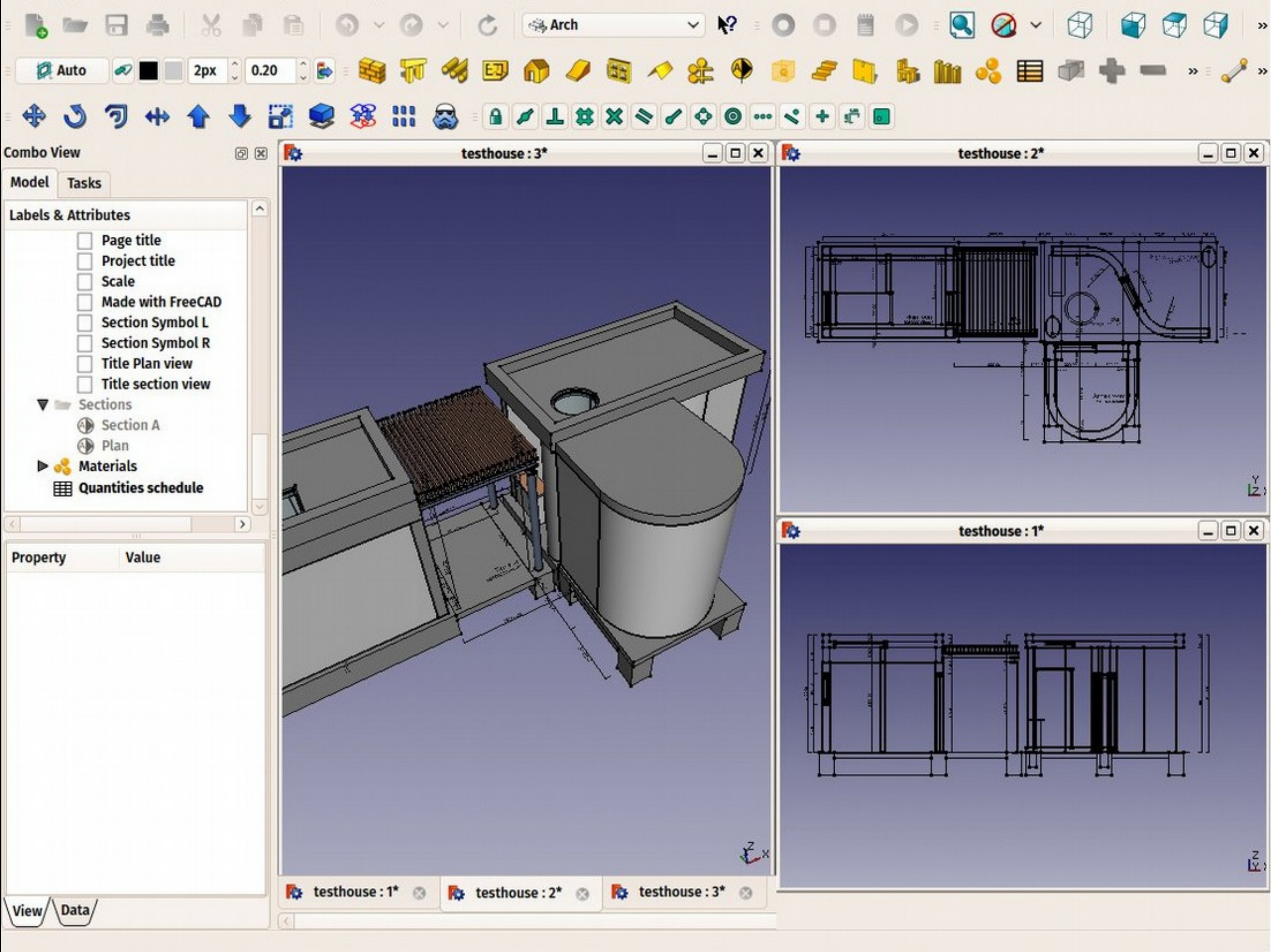
Combo View

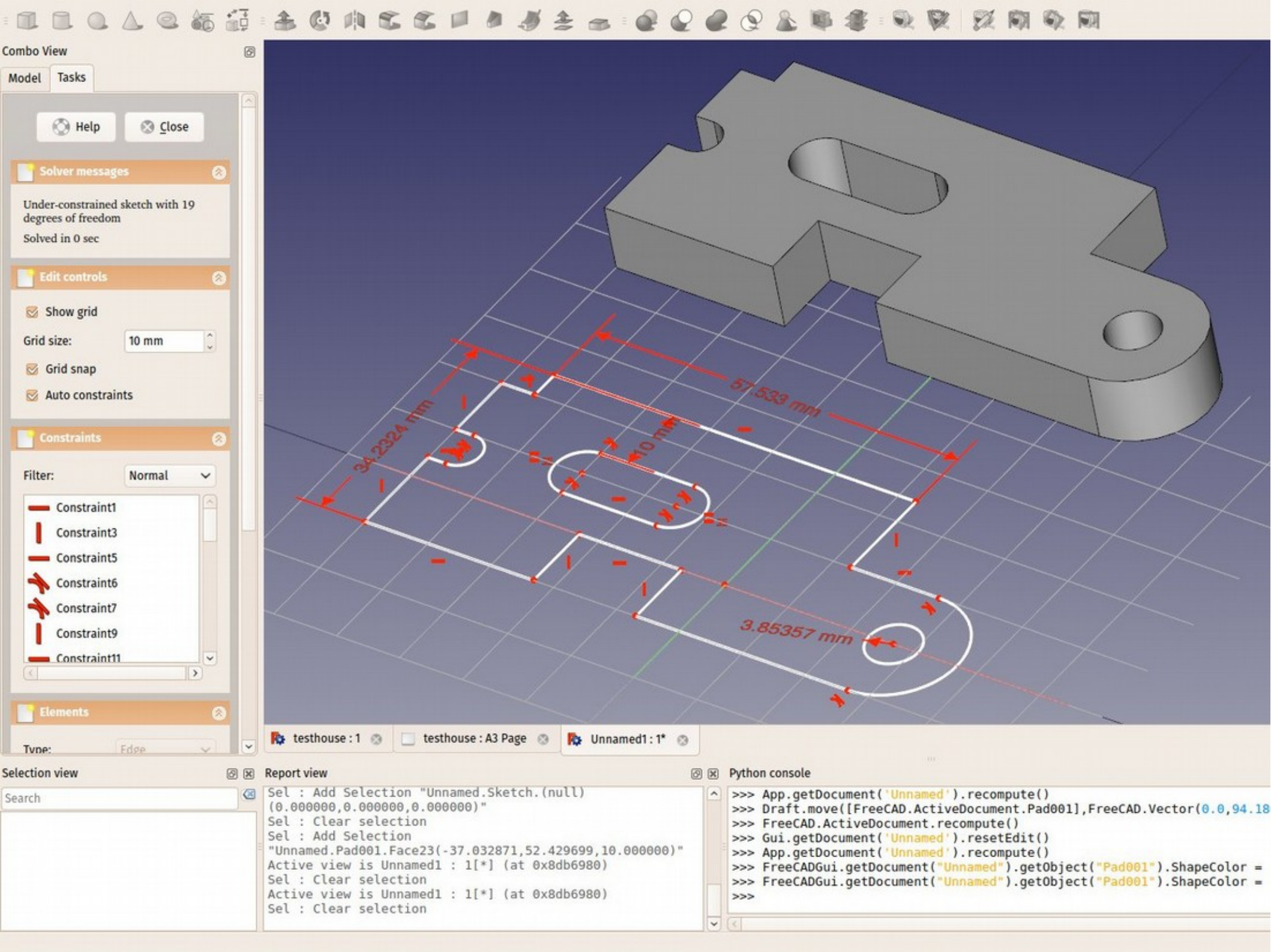
Model Tasks

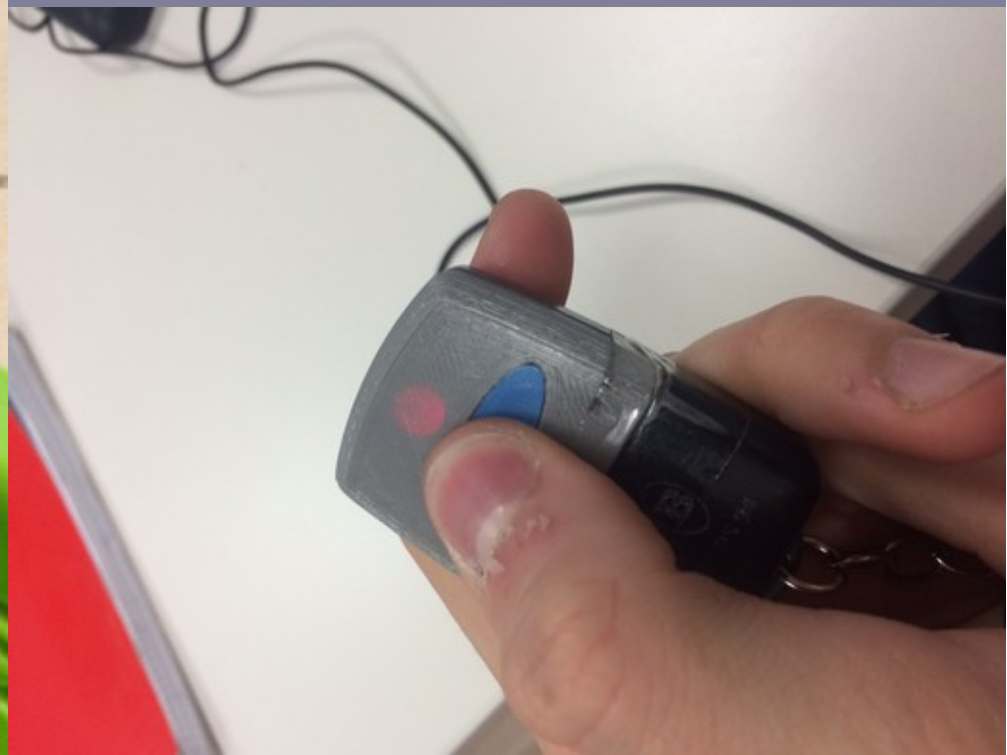
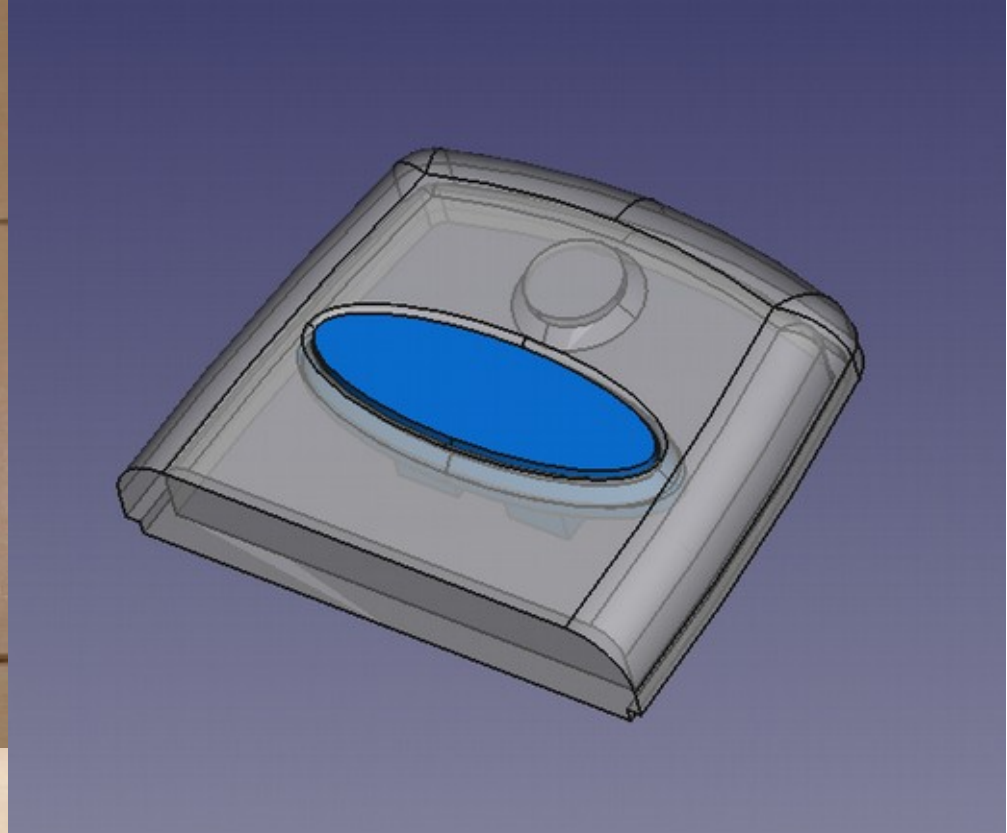
Create Geometry

- Create sketch
- Cube
- Cylinder



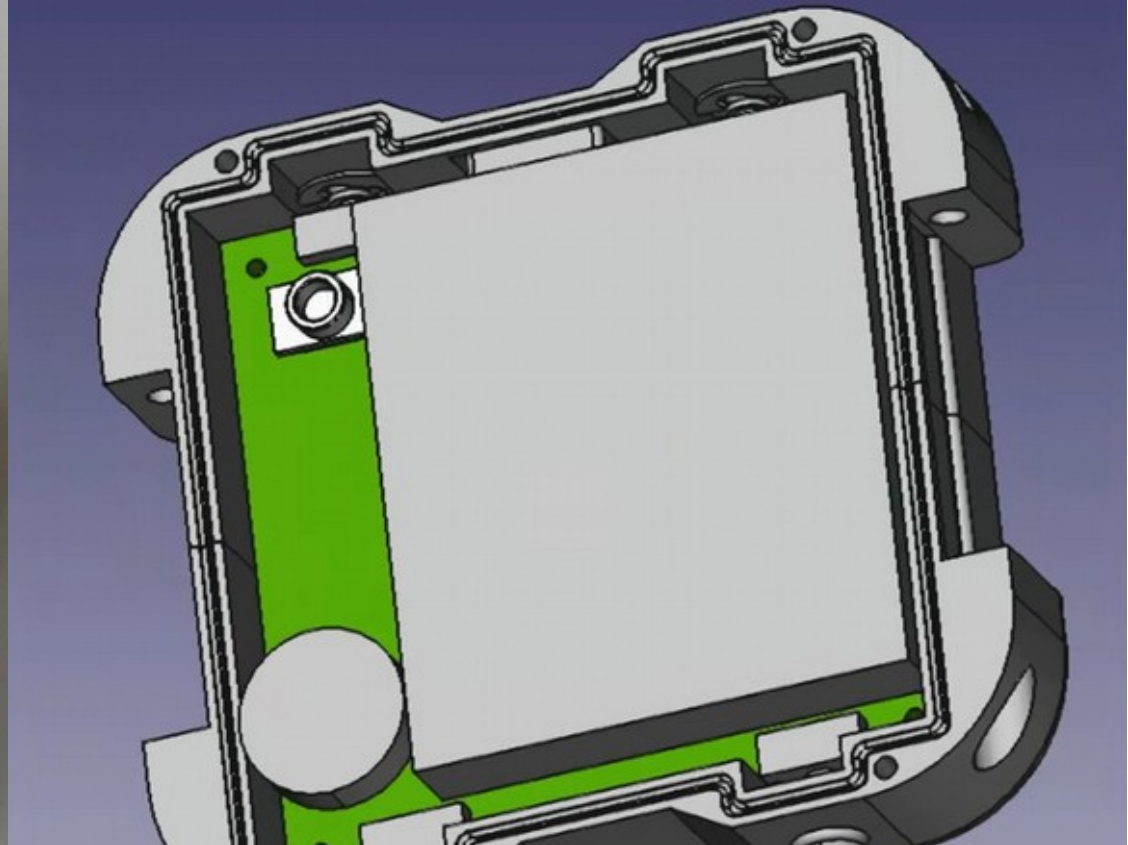


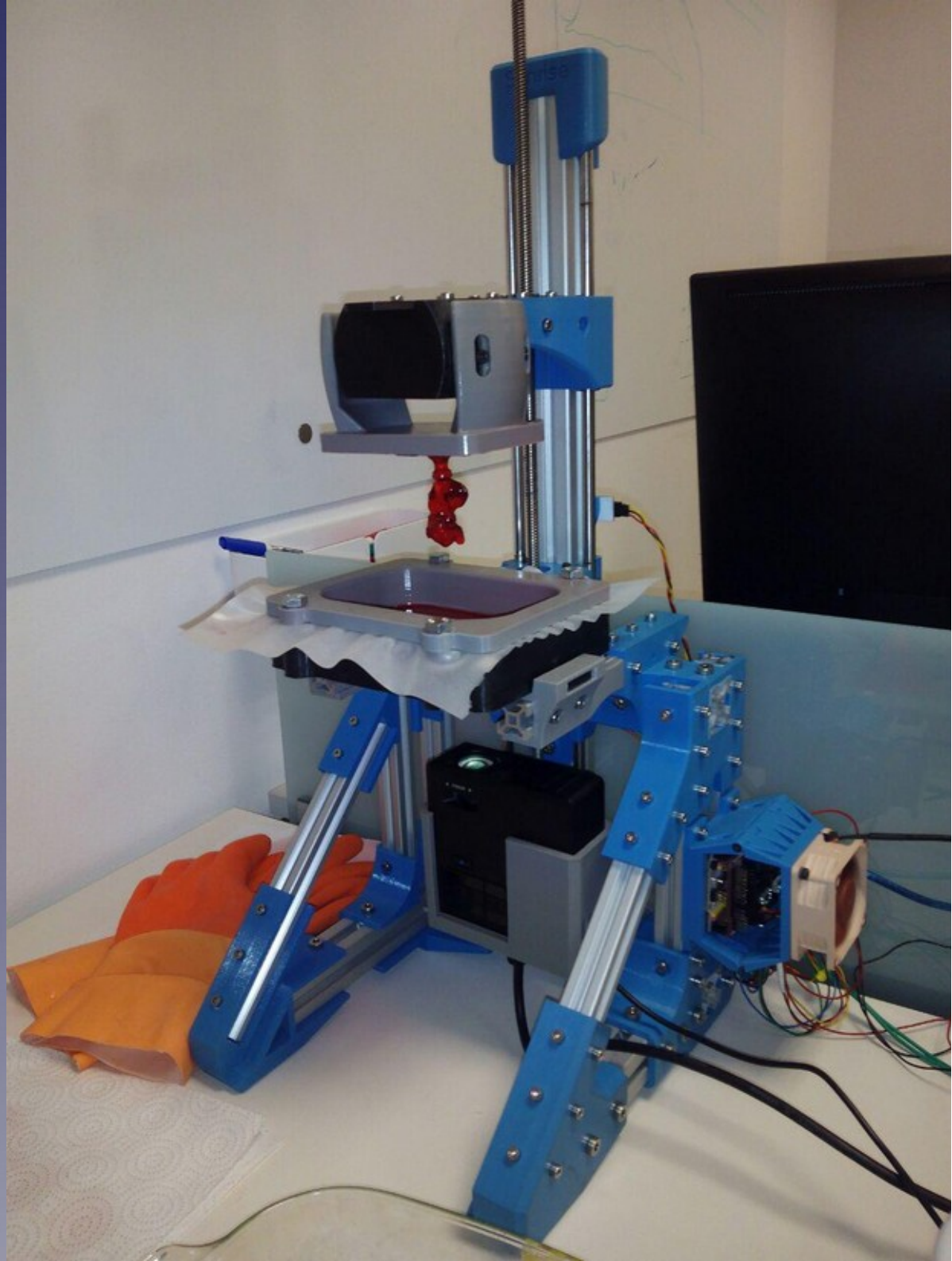
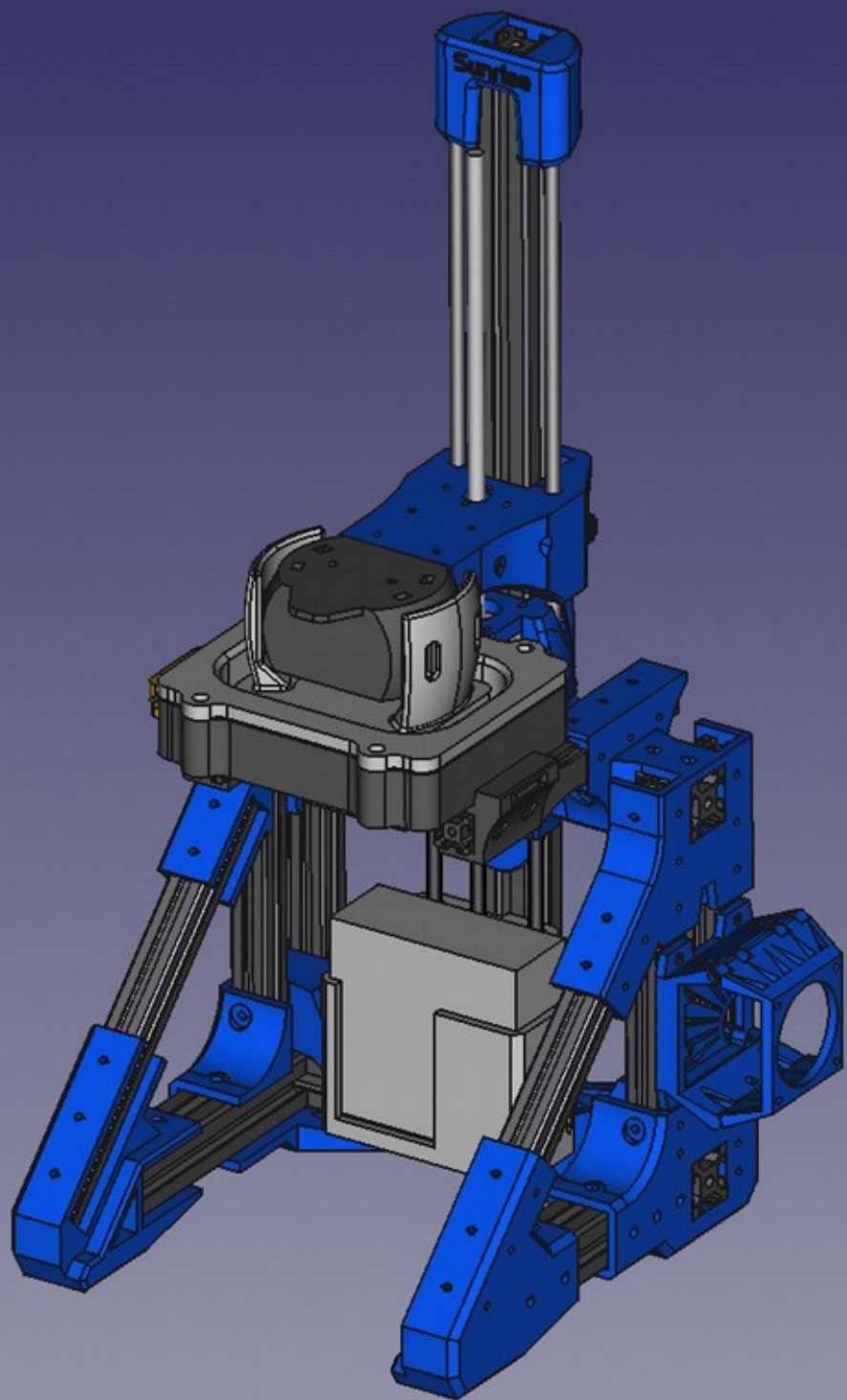


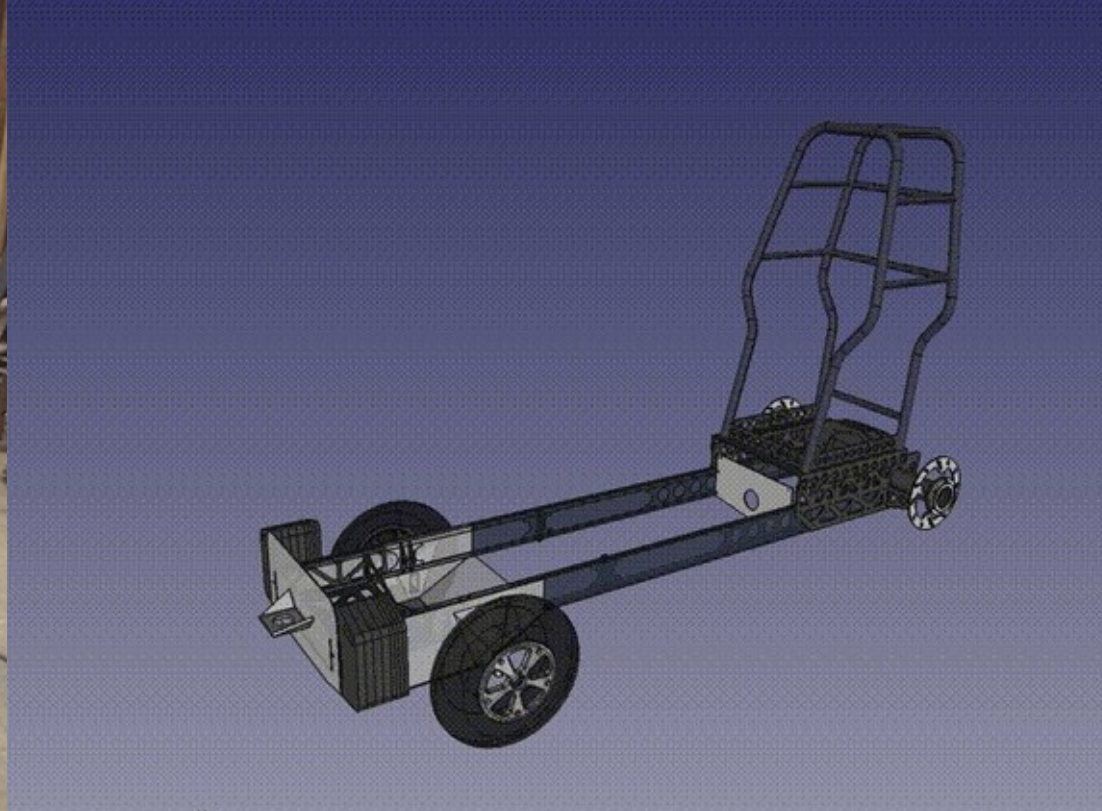




f*watch

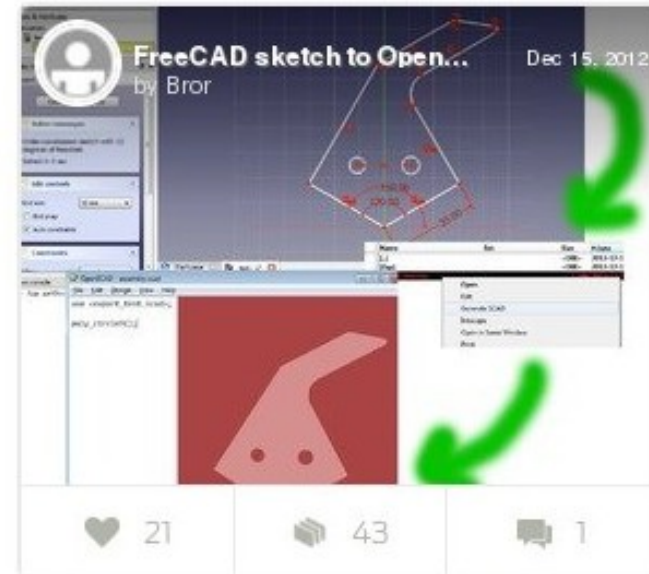
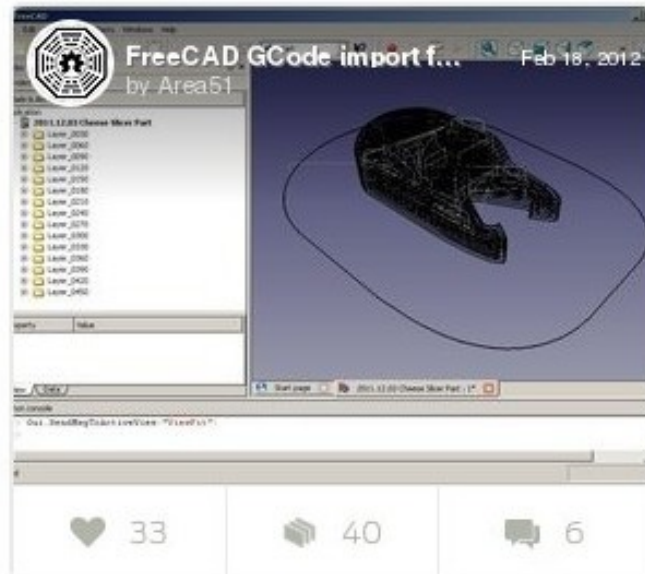


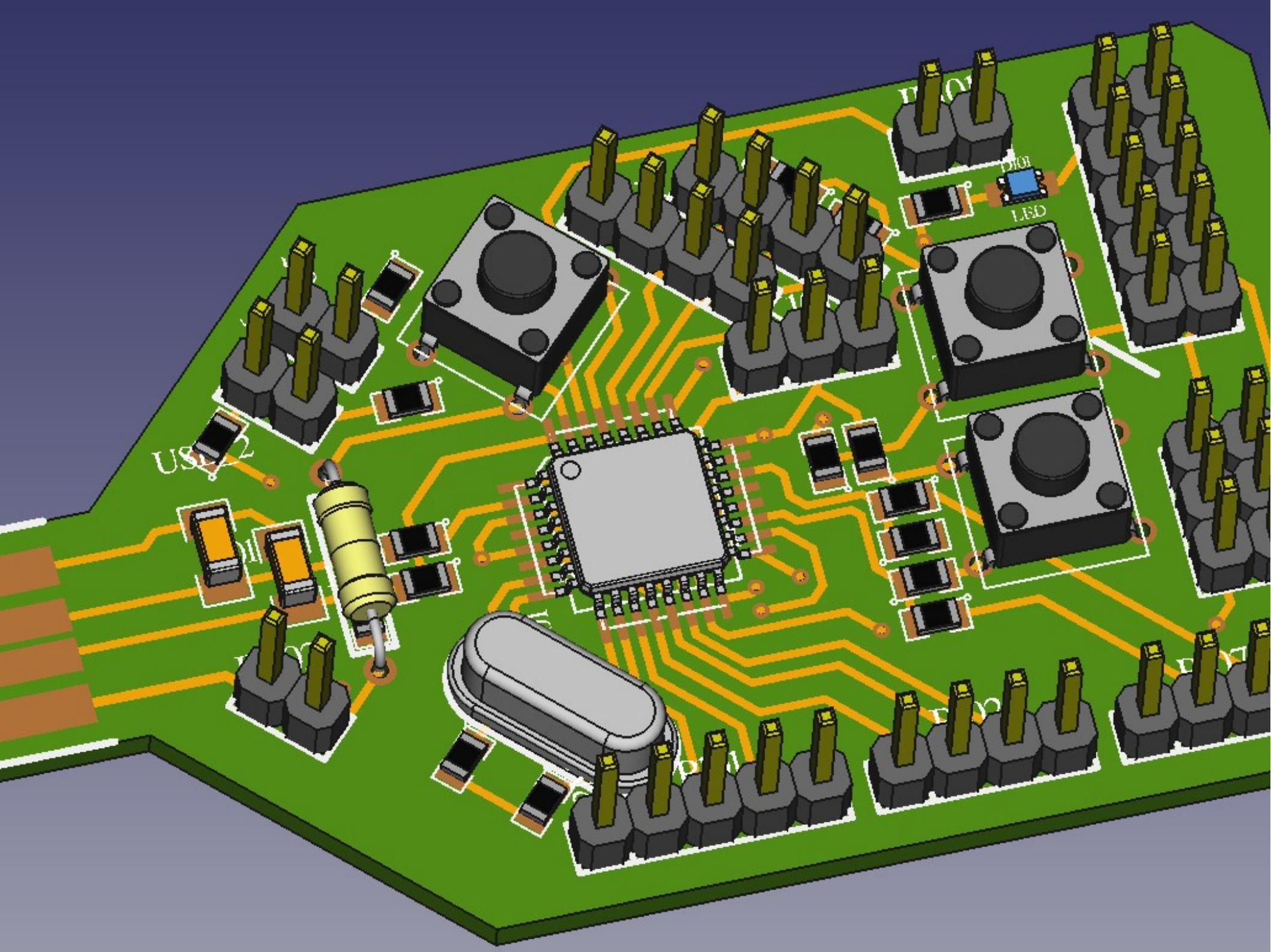


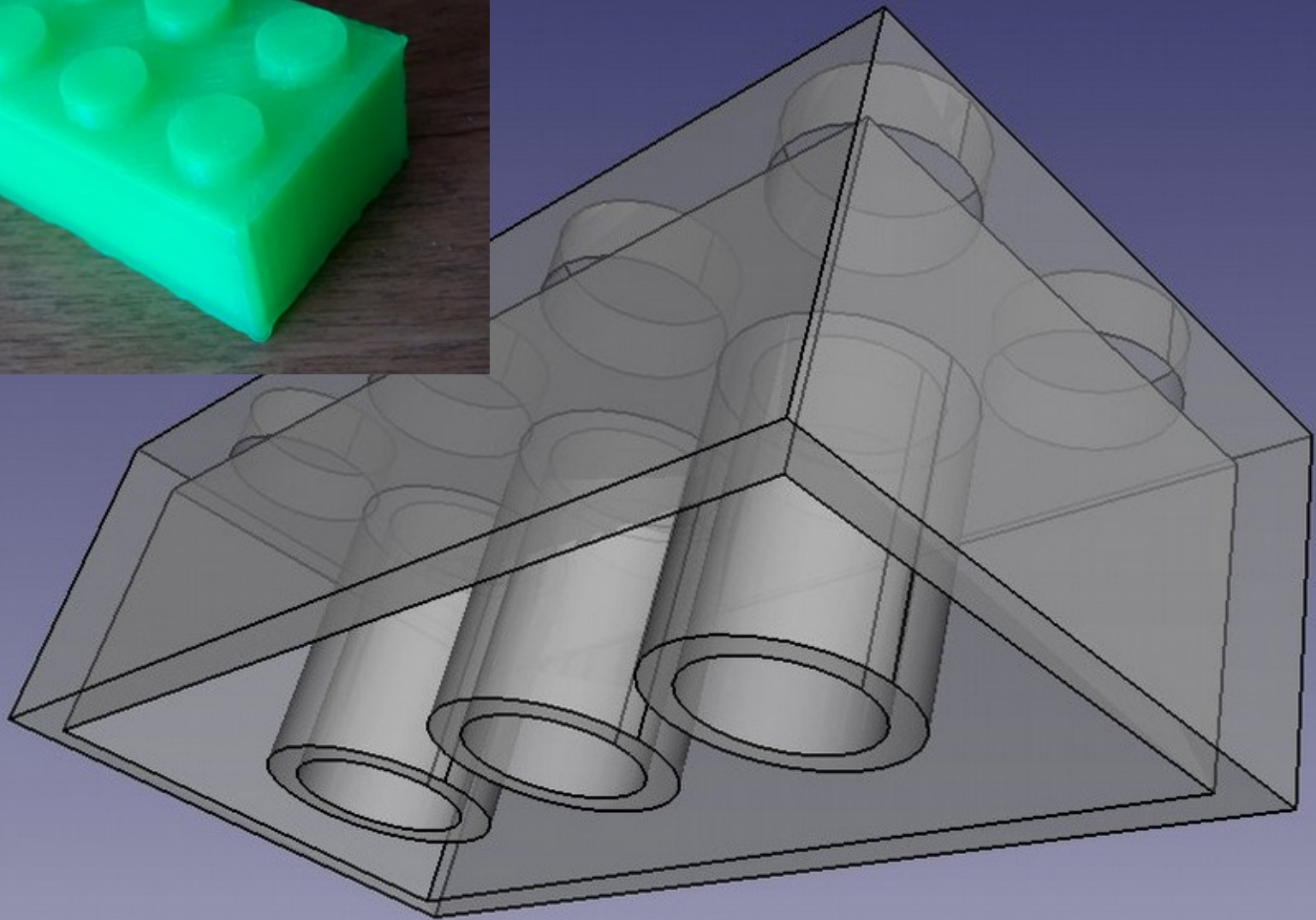
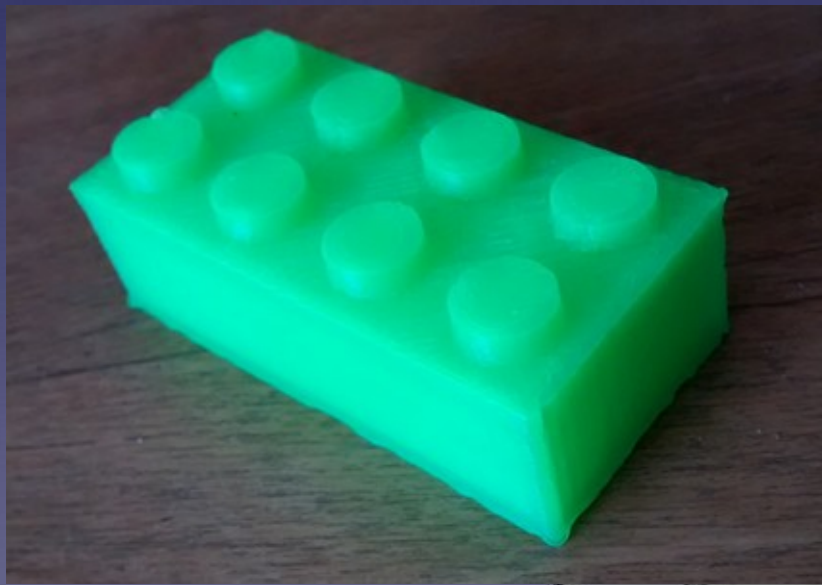


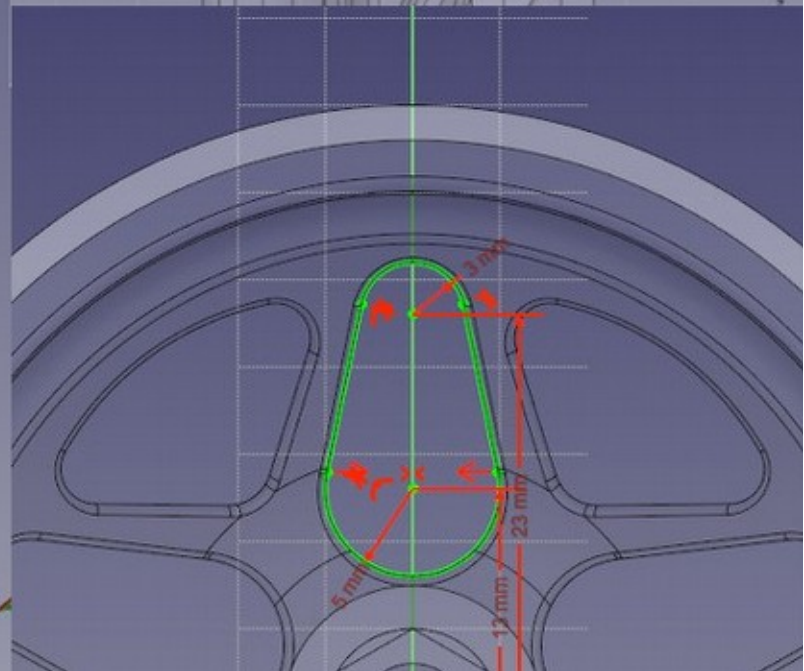
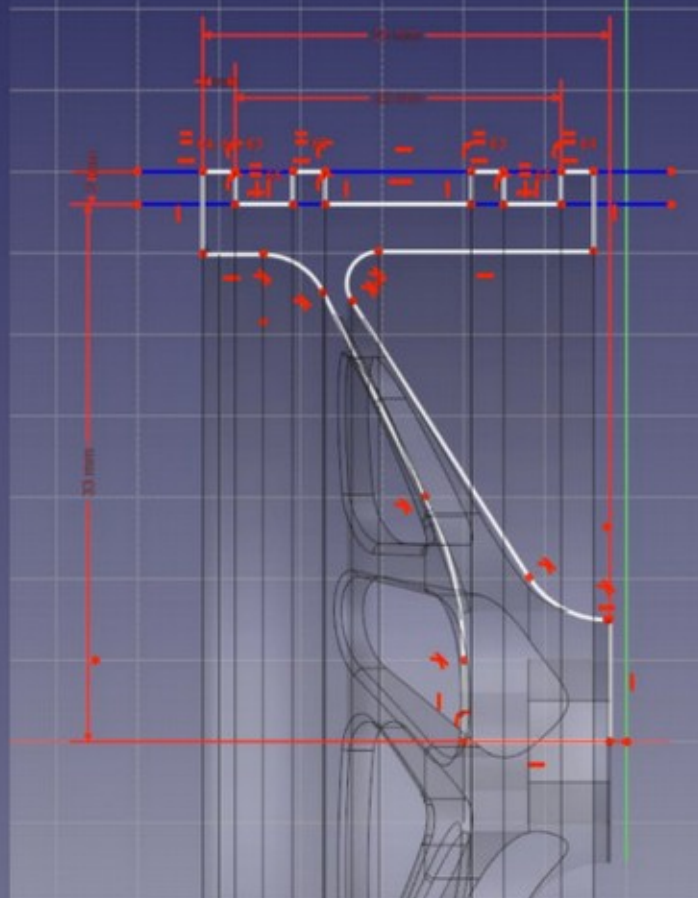
629 results for "freecad": Things

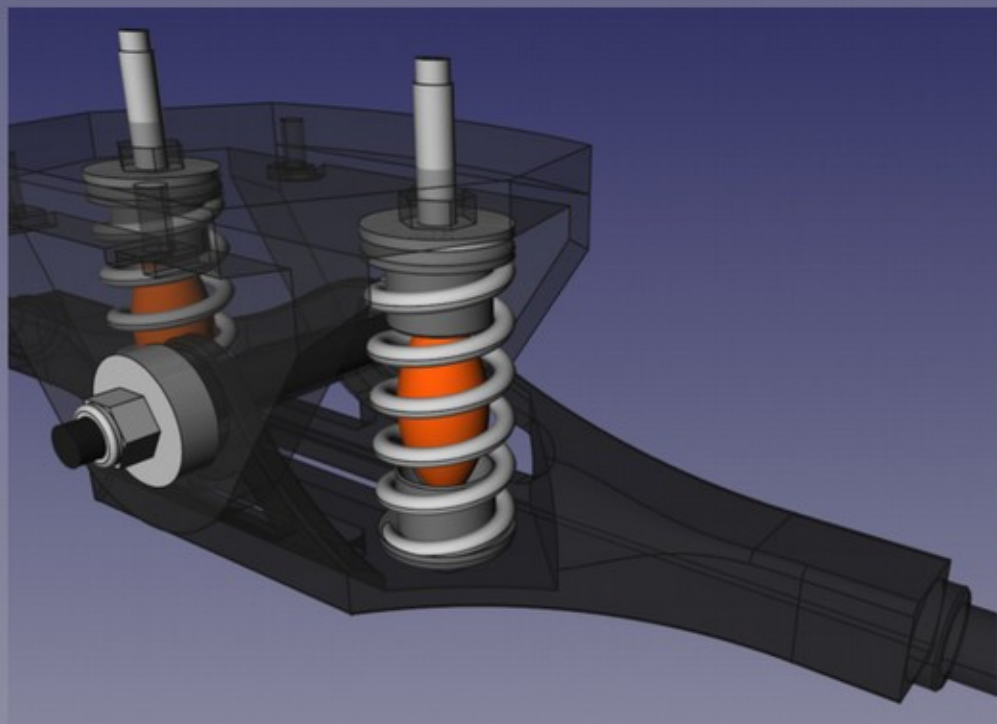
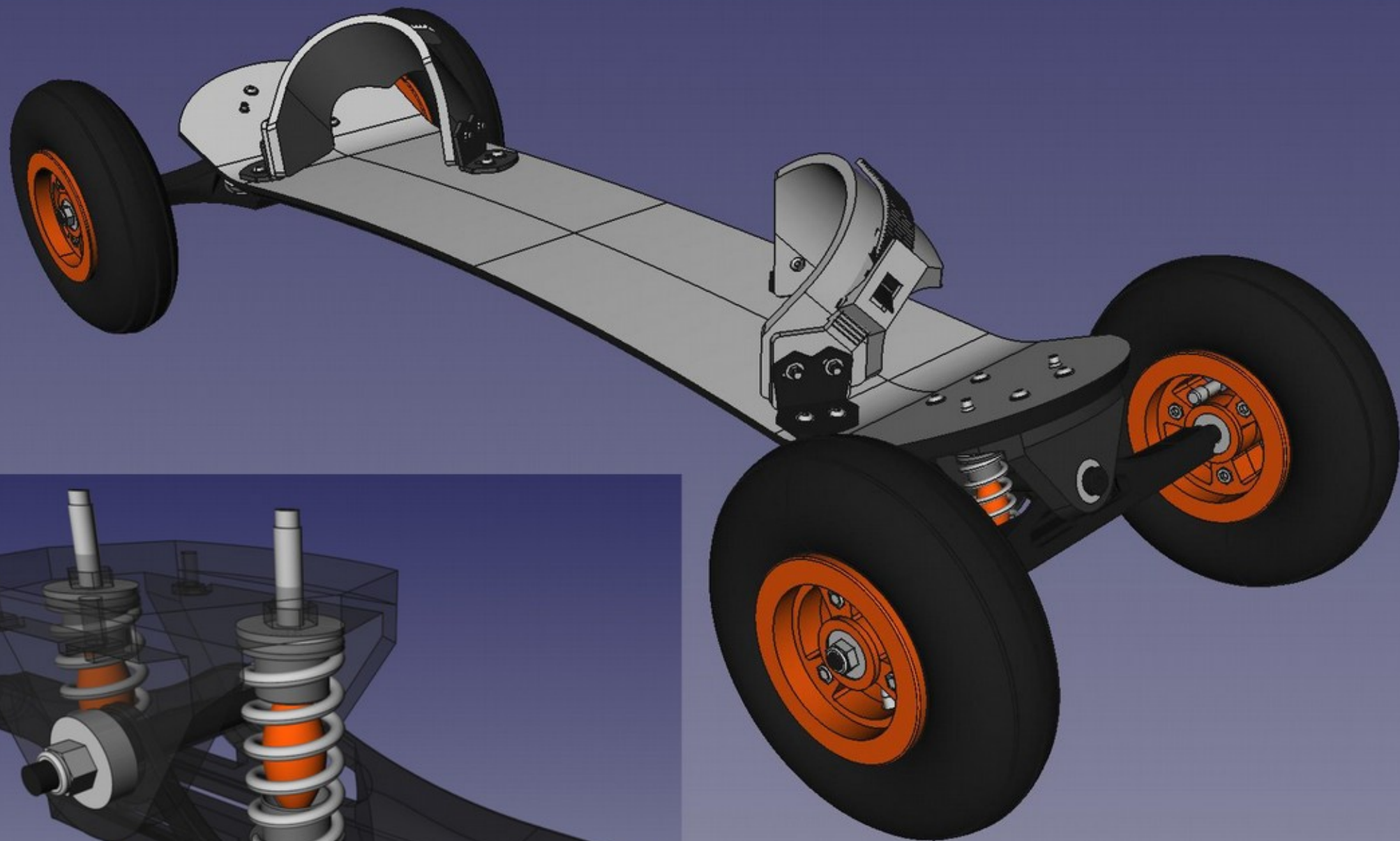
Sort: Relevant

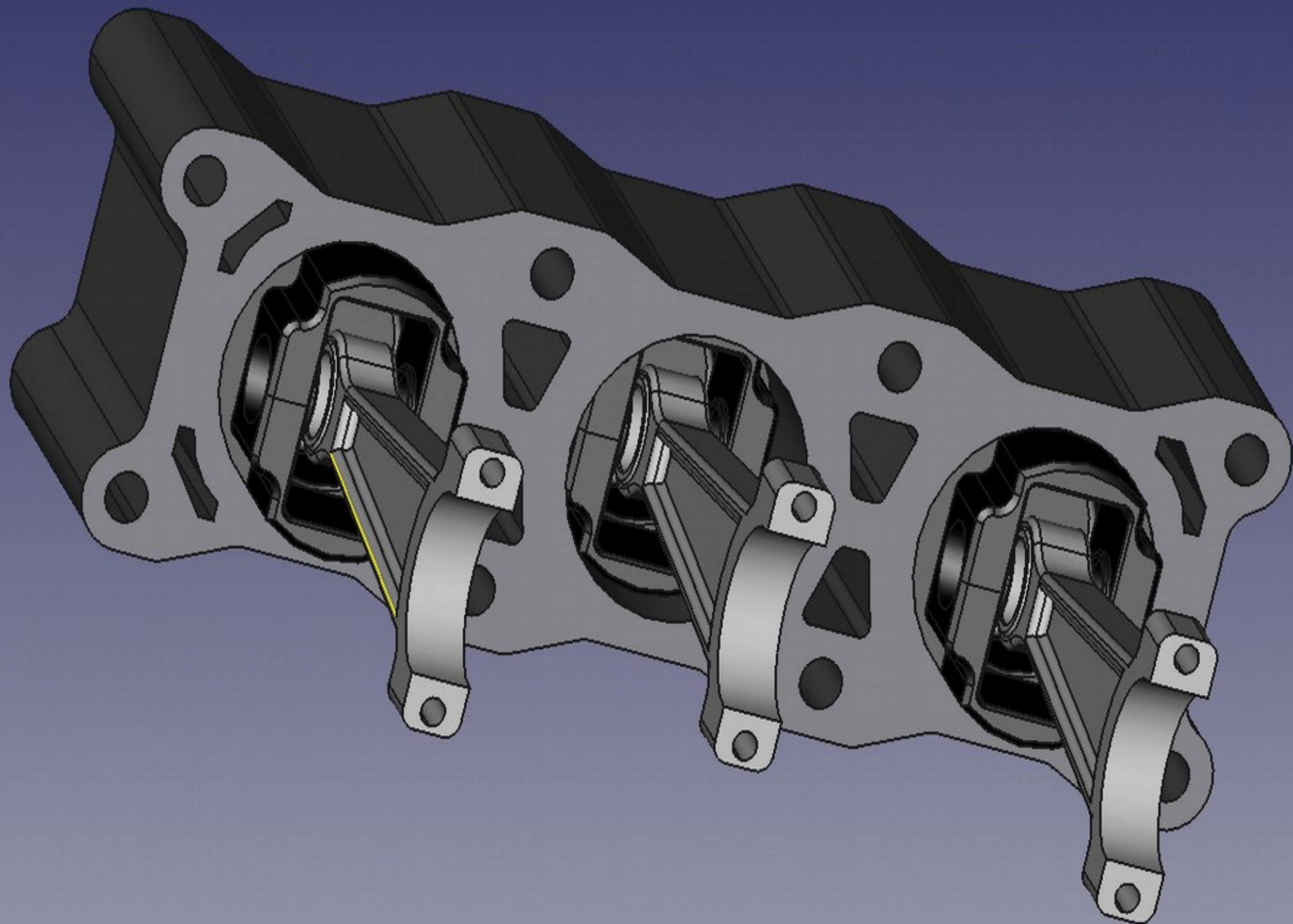


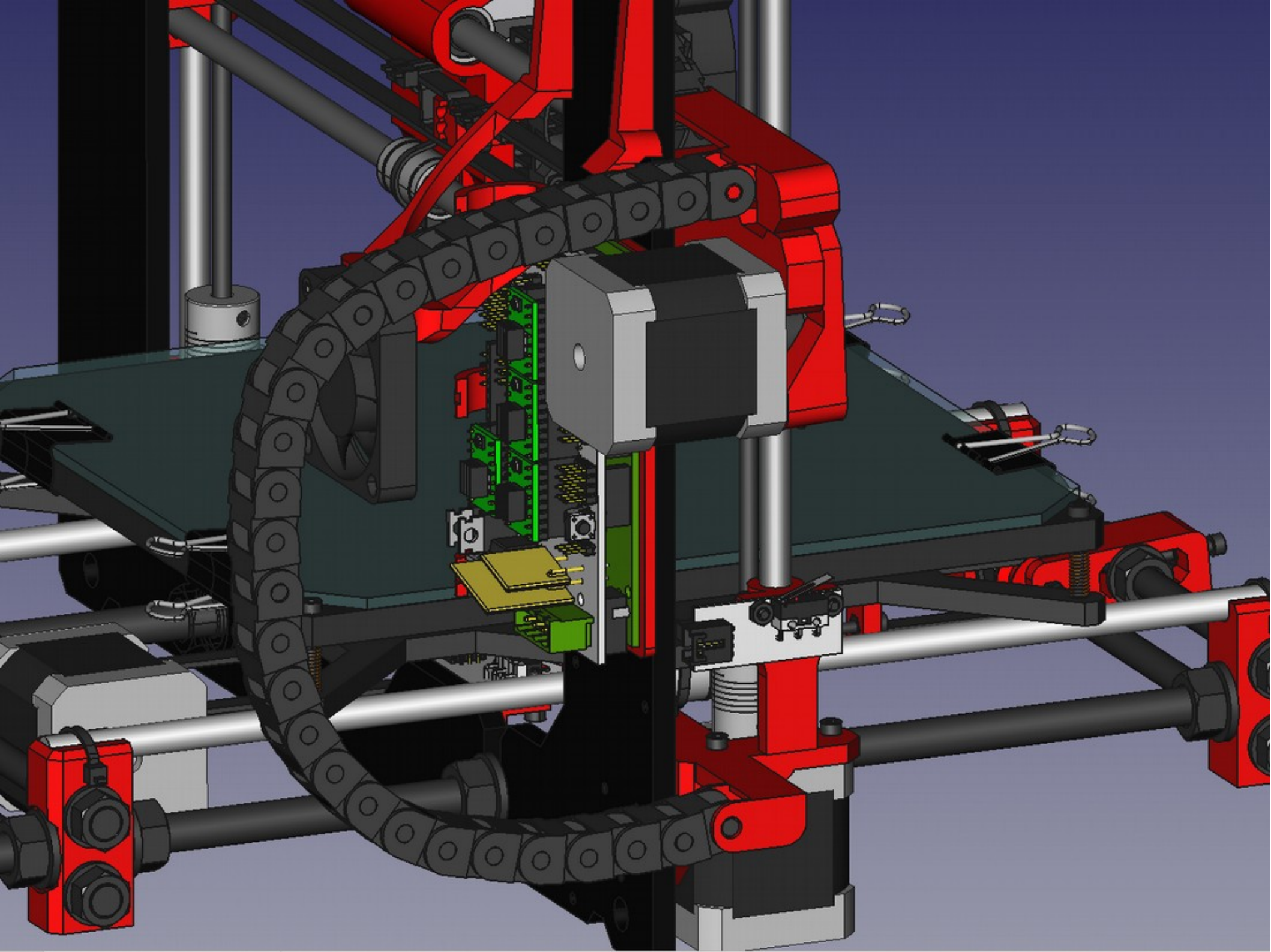


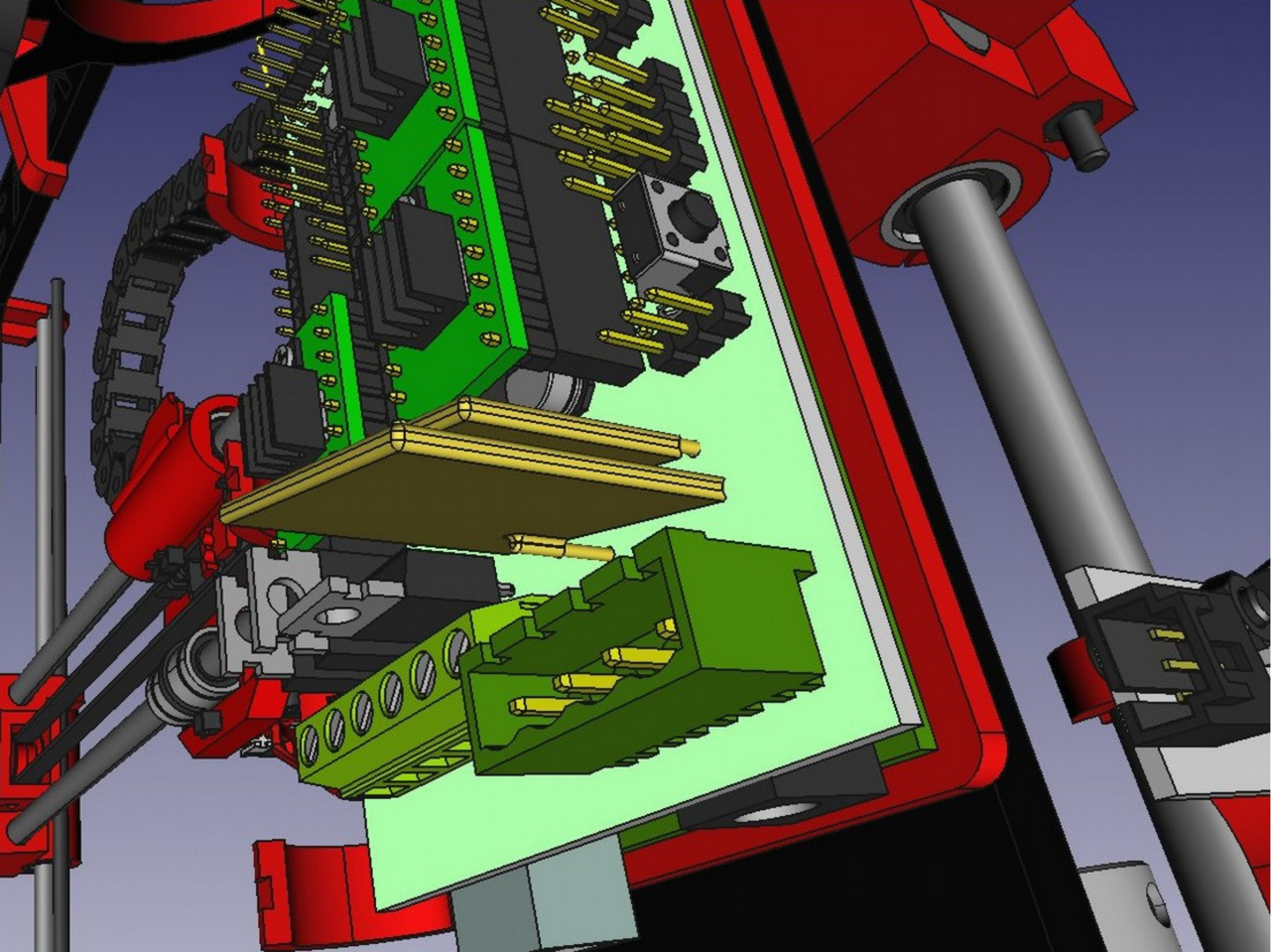


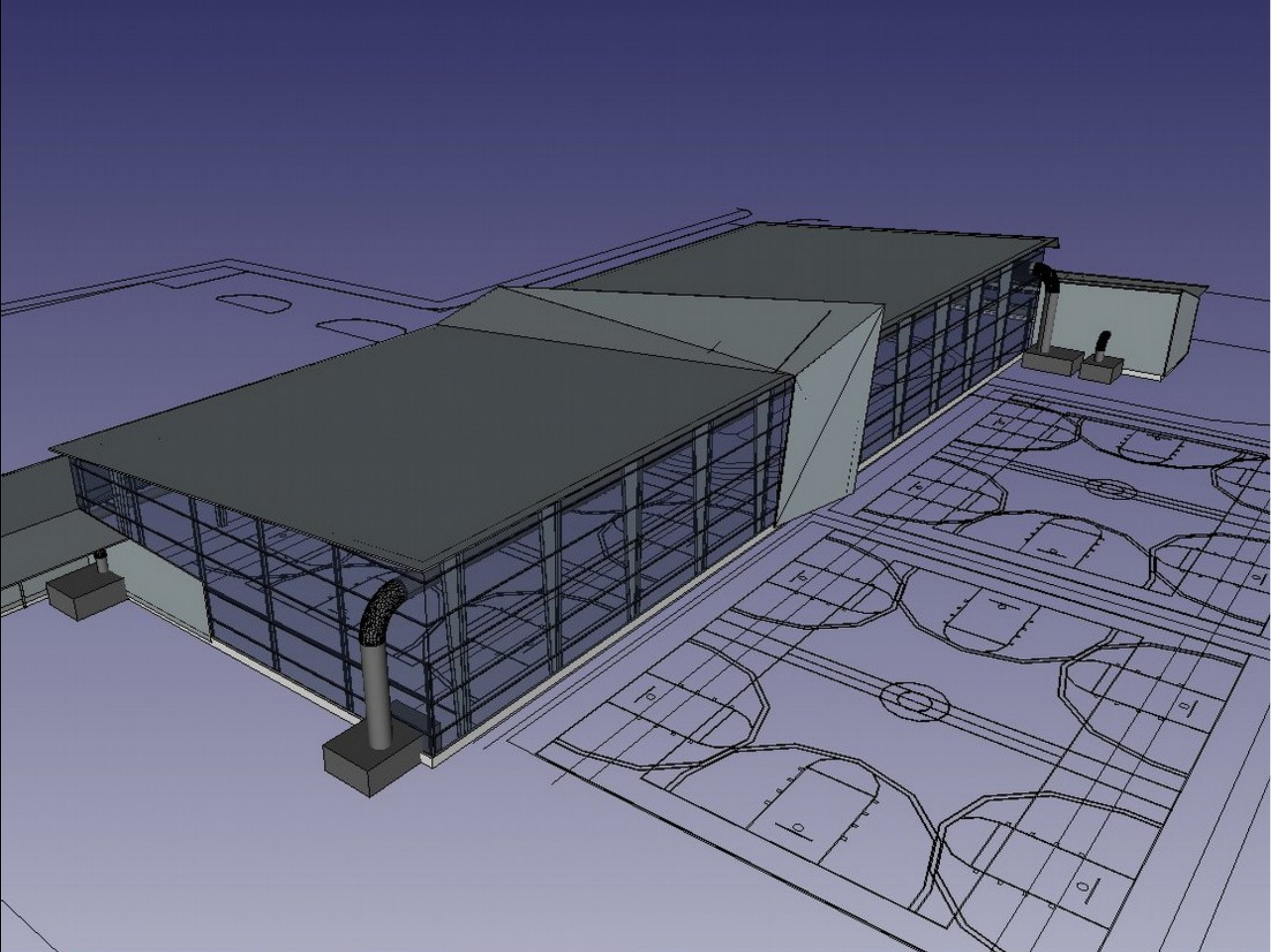


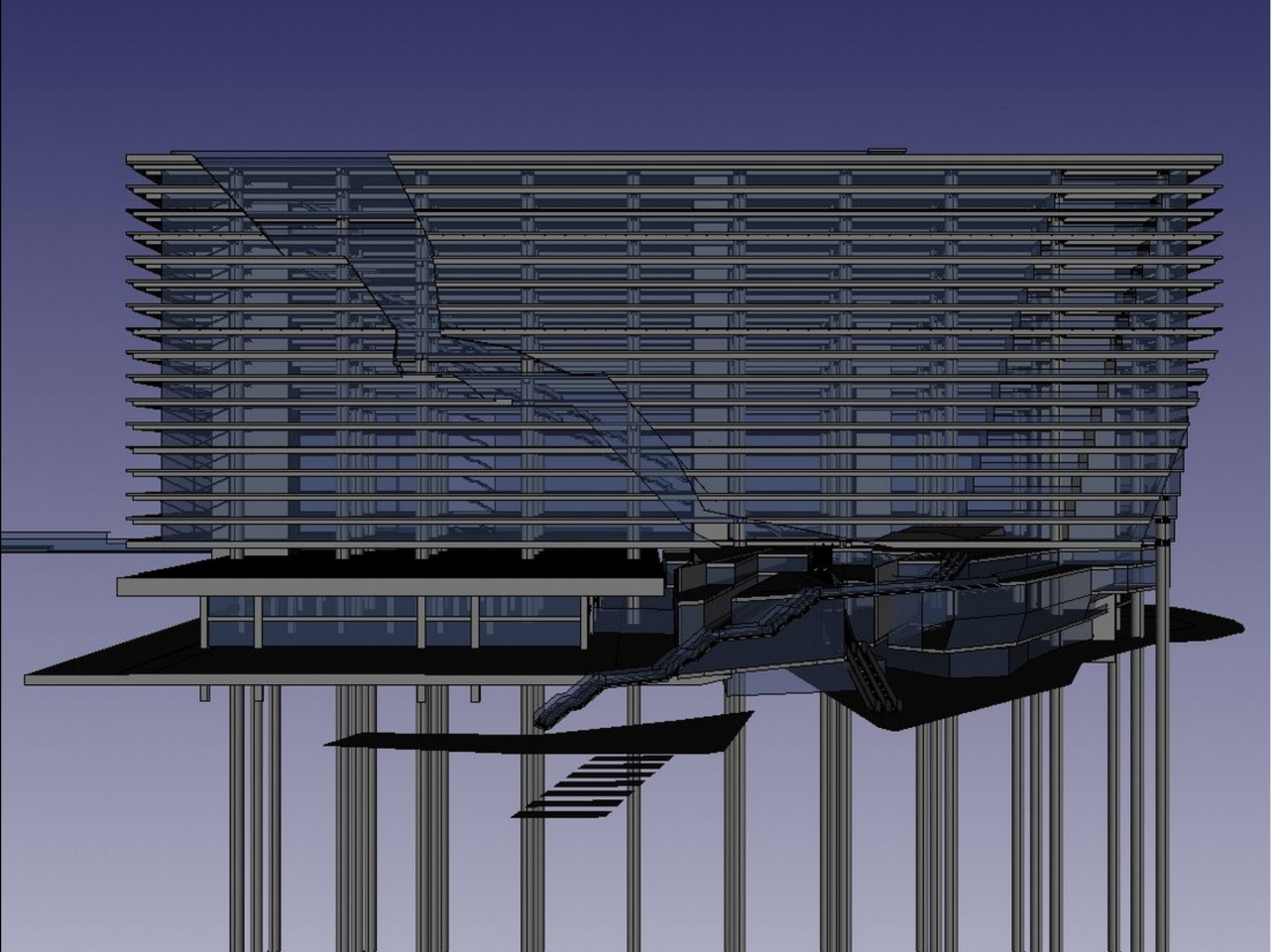


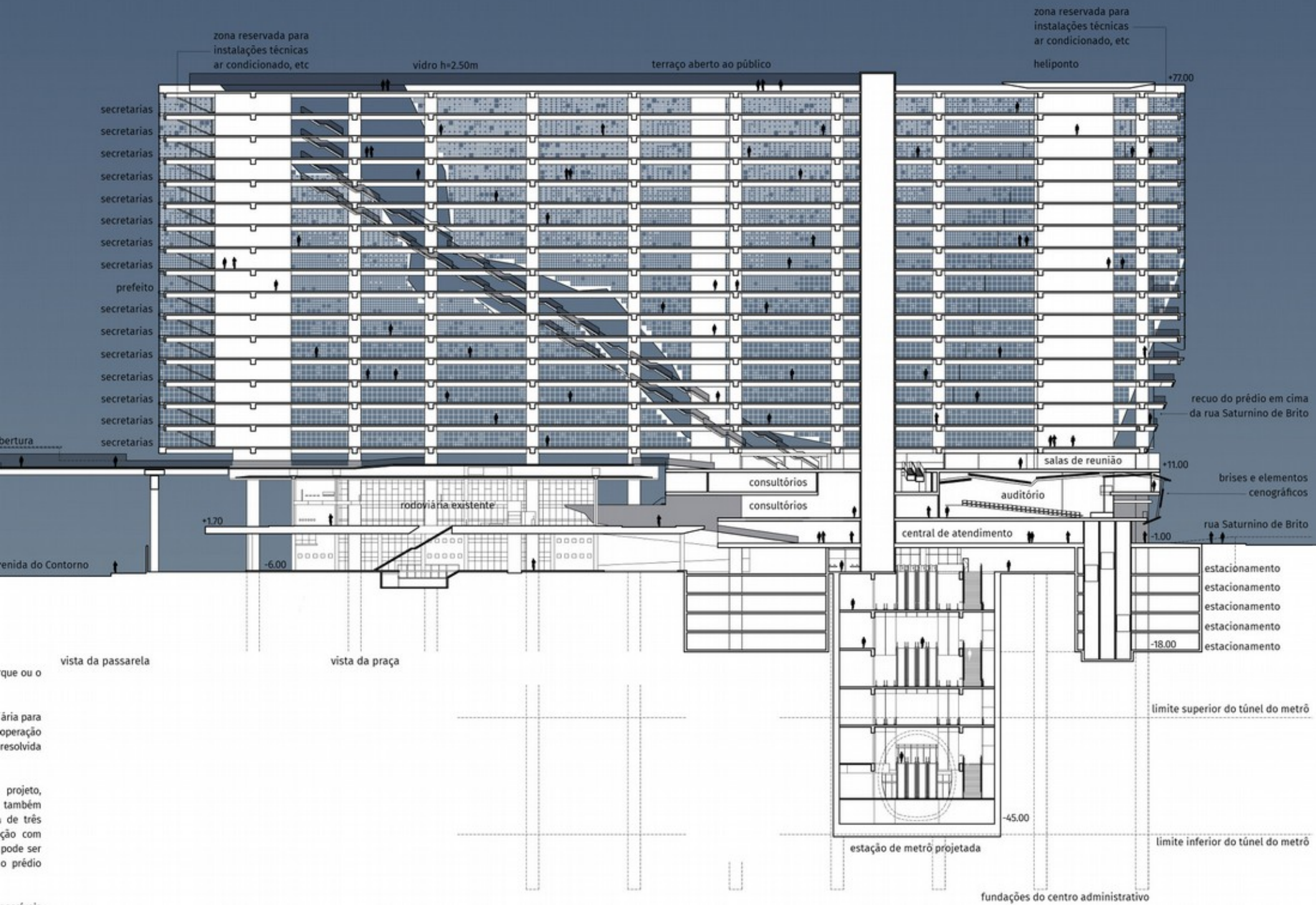


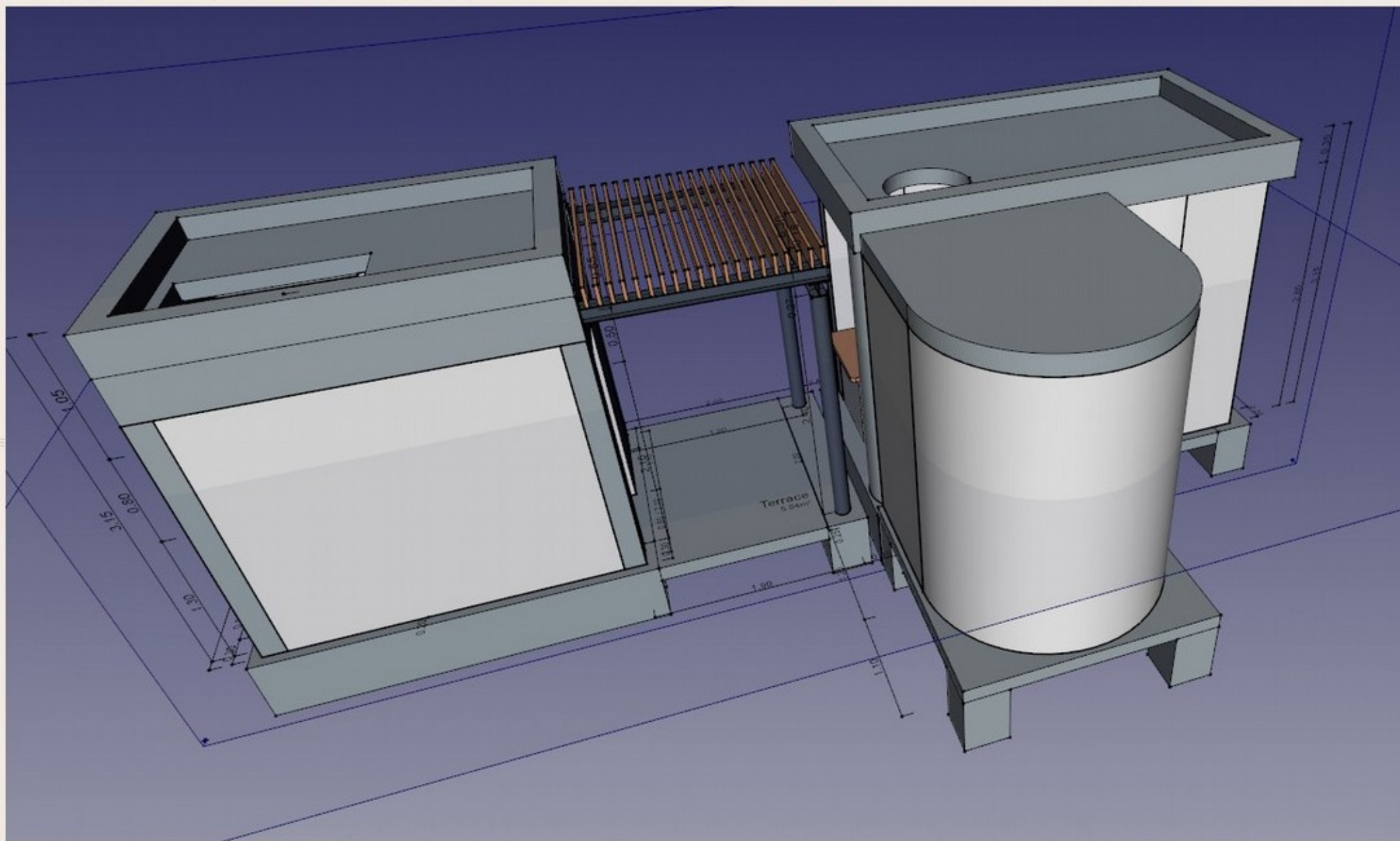












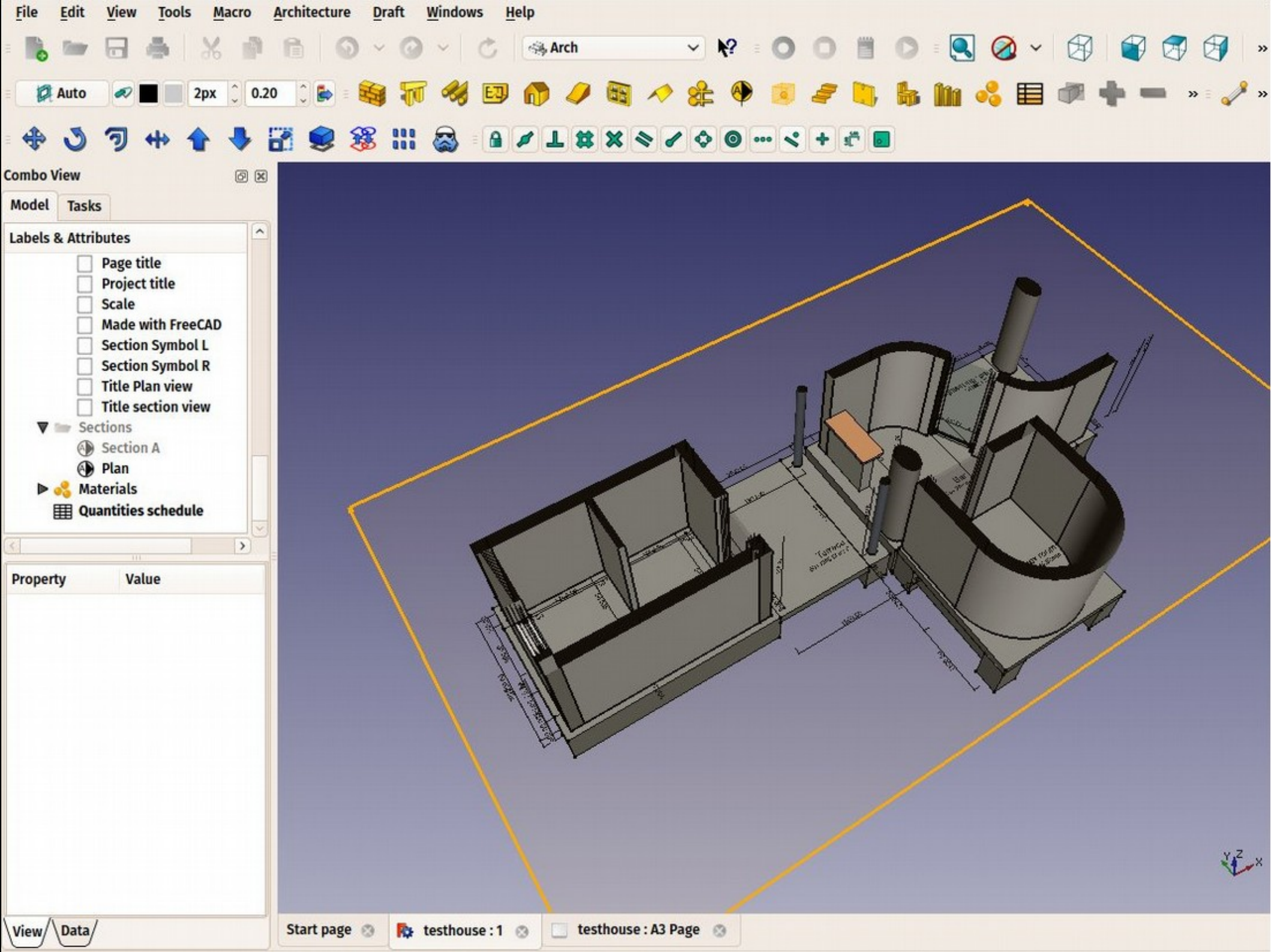
testhouse : 1 testhouse : A3 Page

Report view

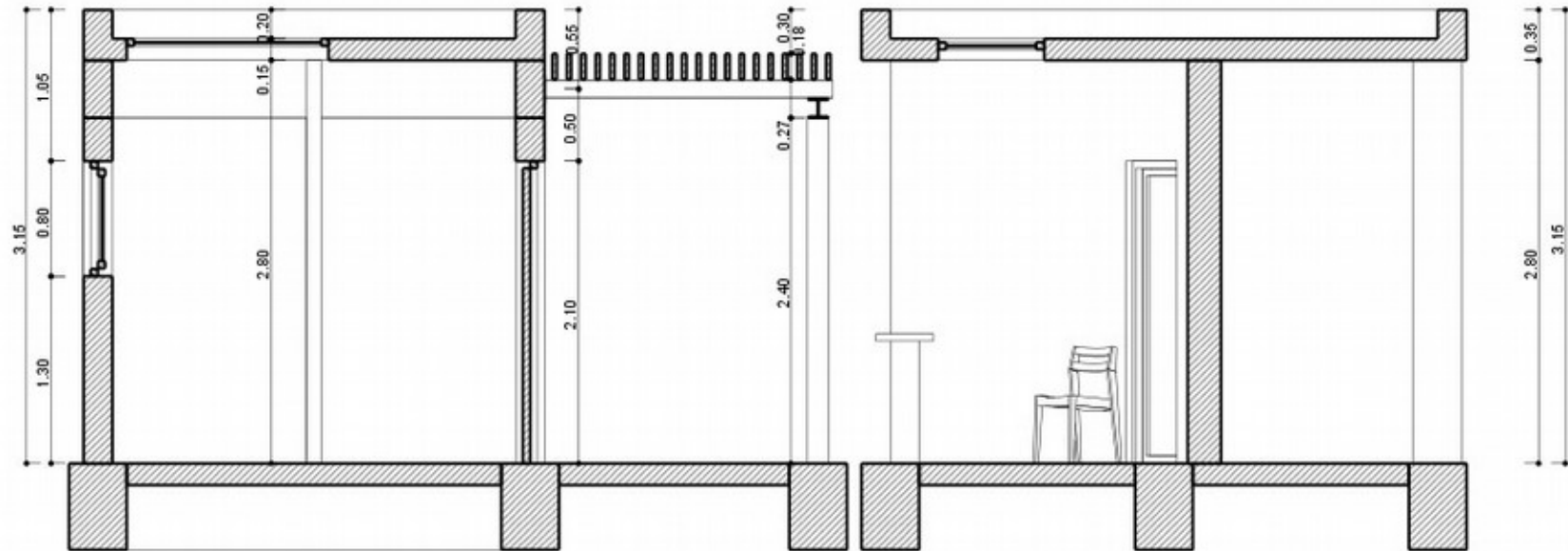
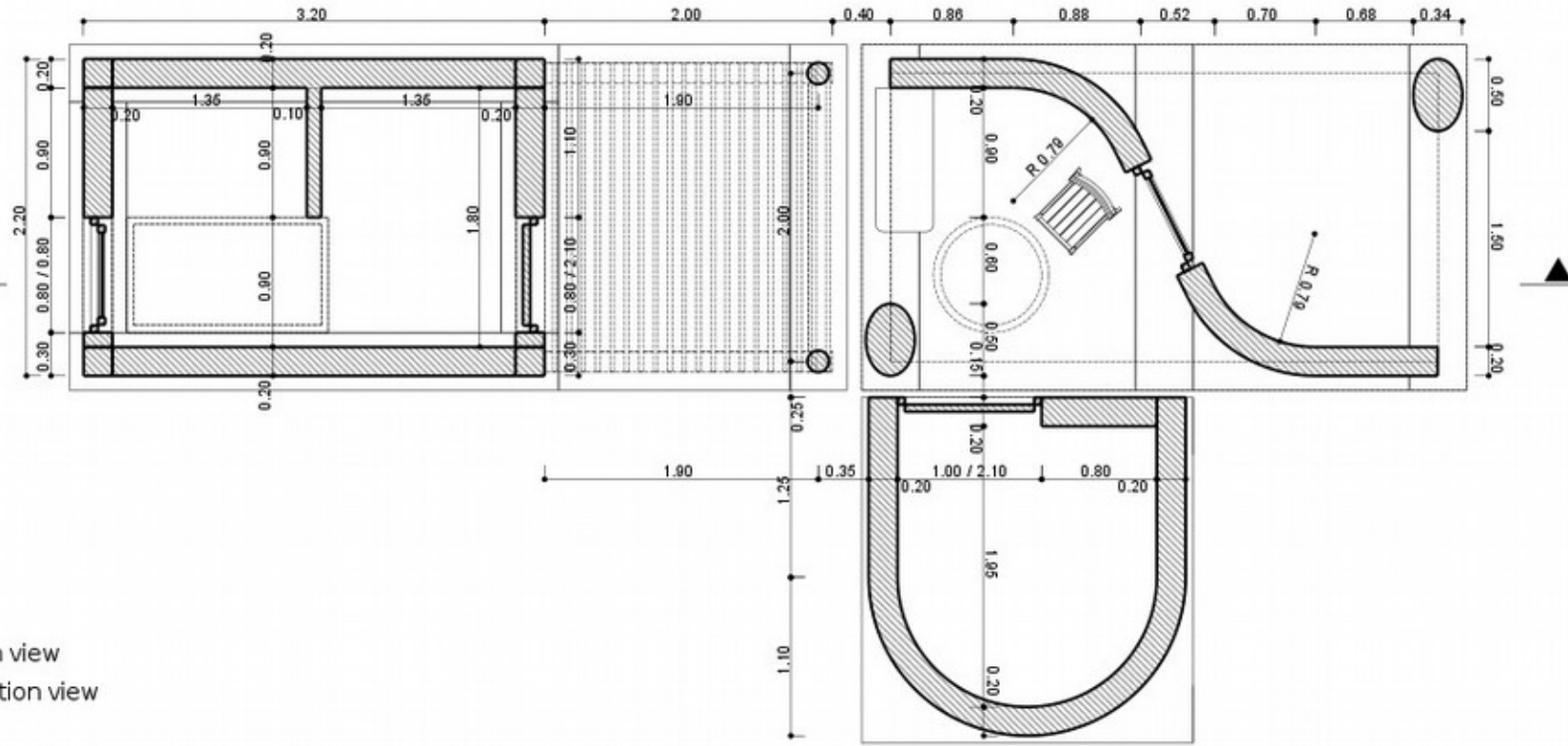
Active view is Unnamed : 1[*] (at 0x47ec120)
Active document is Unnamed (at 0x47ea180)
Active view is Unnamed : 1[*] (at 0x47ec120)
Sel : Clear selection
No active document
Active view is Unnamed : 1[*] (at 0x638a580)

Python console

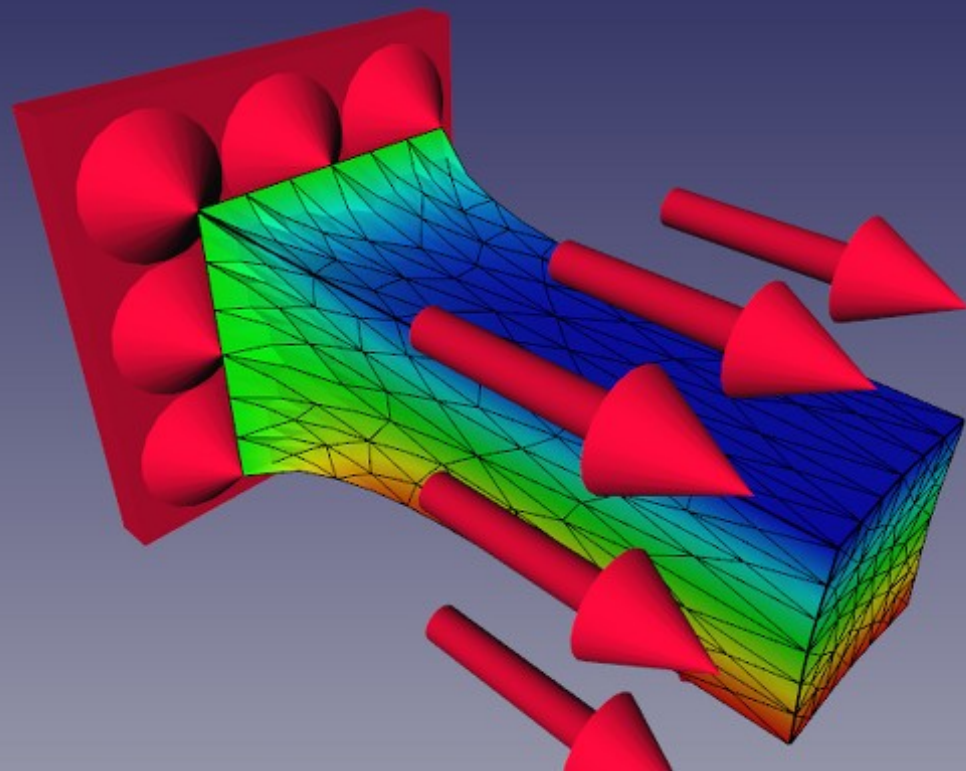
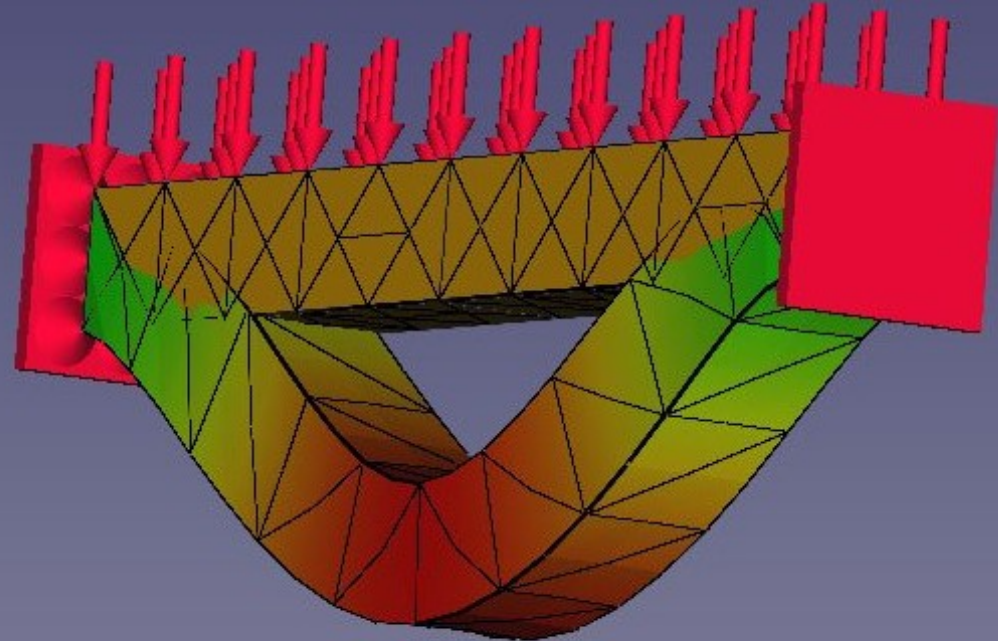
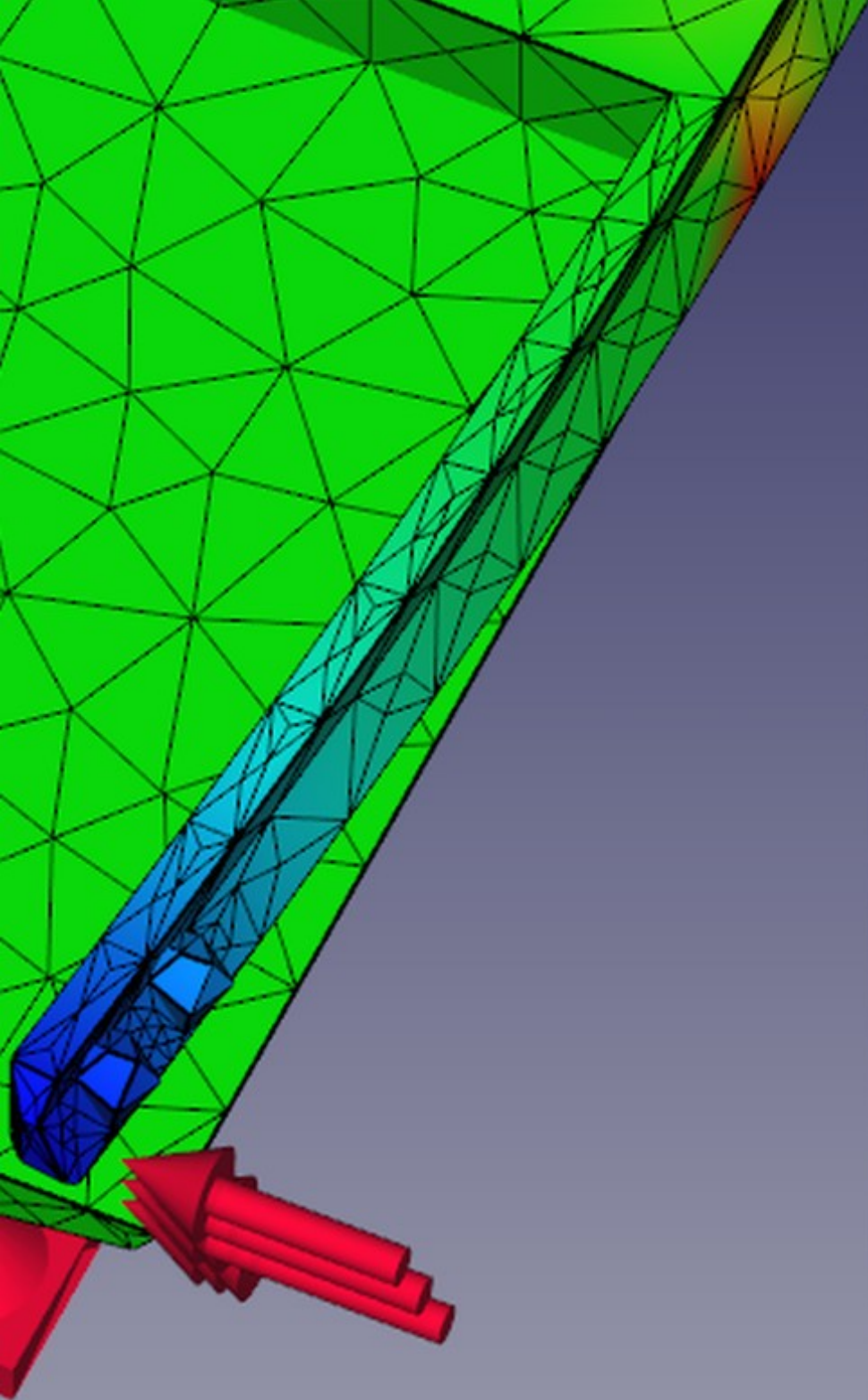
Type 'help', 'copyright', 'credits' or 'license' for more information.
>>> App.setActiveDocument("Unnamed")
>>> App.ActiveDocument=App.getDocument("Unnamed")
>>> Gui.ActiveDocument=Gui.getDocument("Unnamed")
>>> App.closeDocument('Unnamed')



Plan view
Section view



Test House
Plan and Section
Scale: 1:33





Tasks

- ☐ Page title
- ☐ Project title
- ☐ Scale
- ☐ Made with FreeCAD
- ☐ Section Symbol L
- ☐ Section Symbol R
- ☐ Title Plan view
- ☐ Title section view

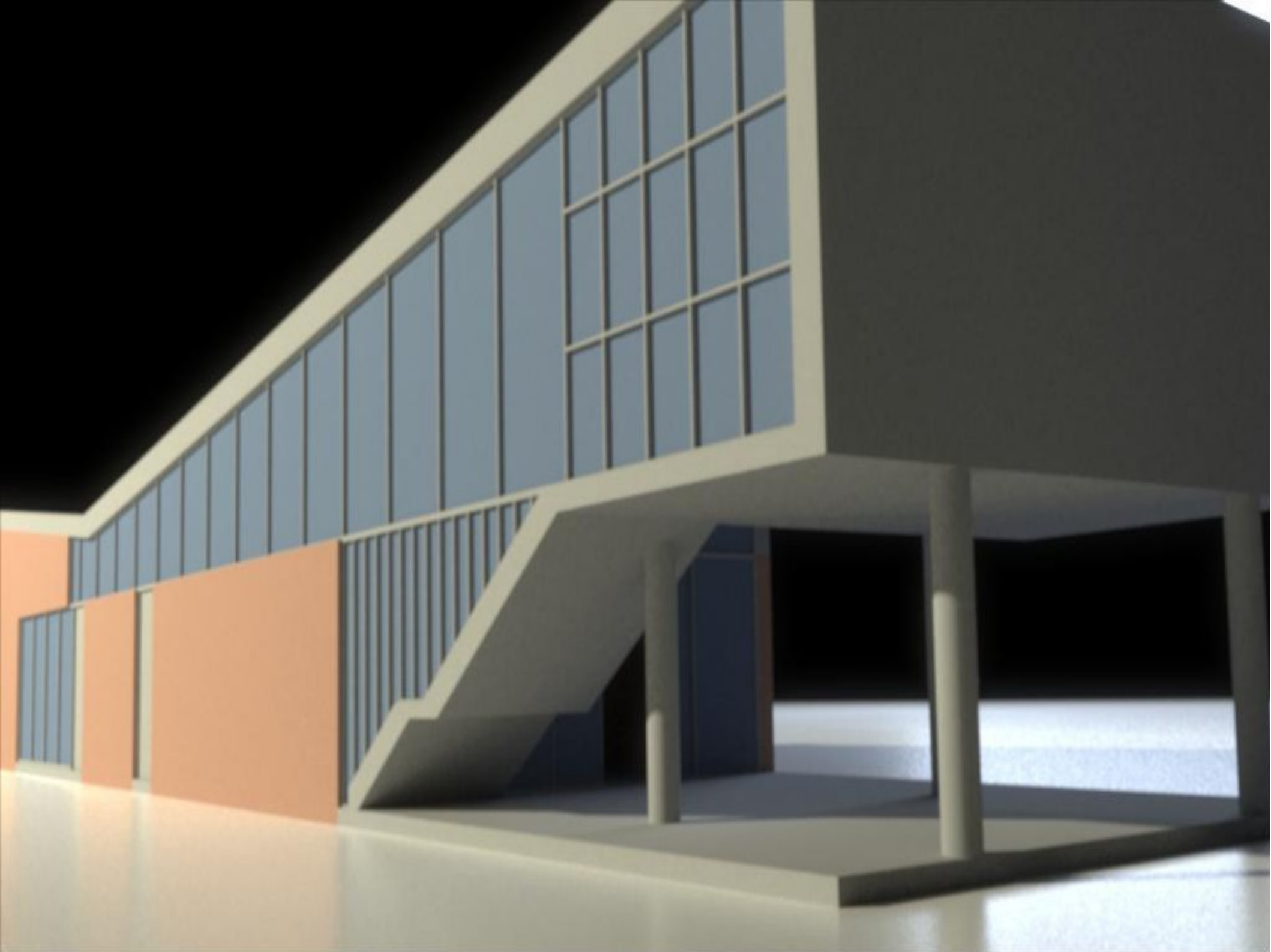
Section A

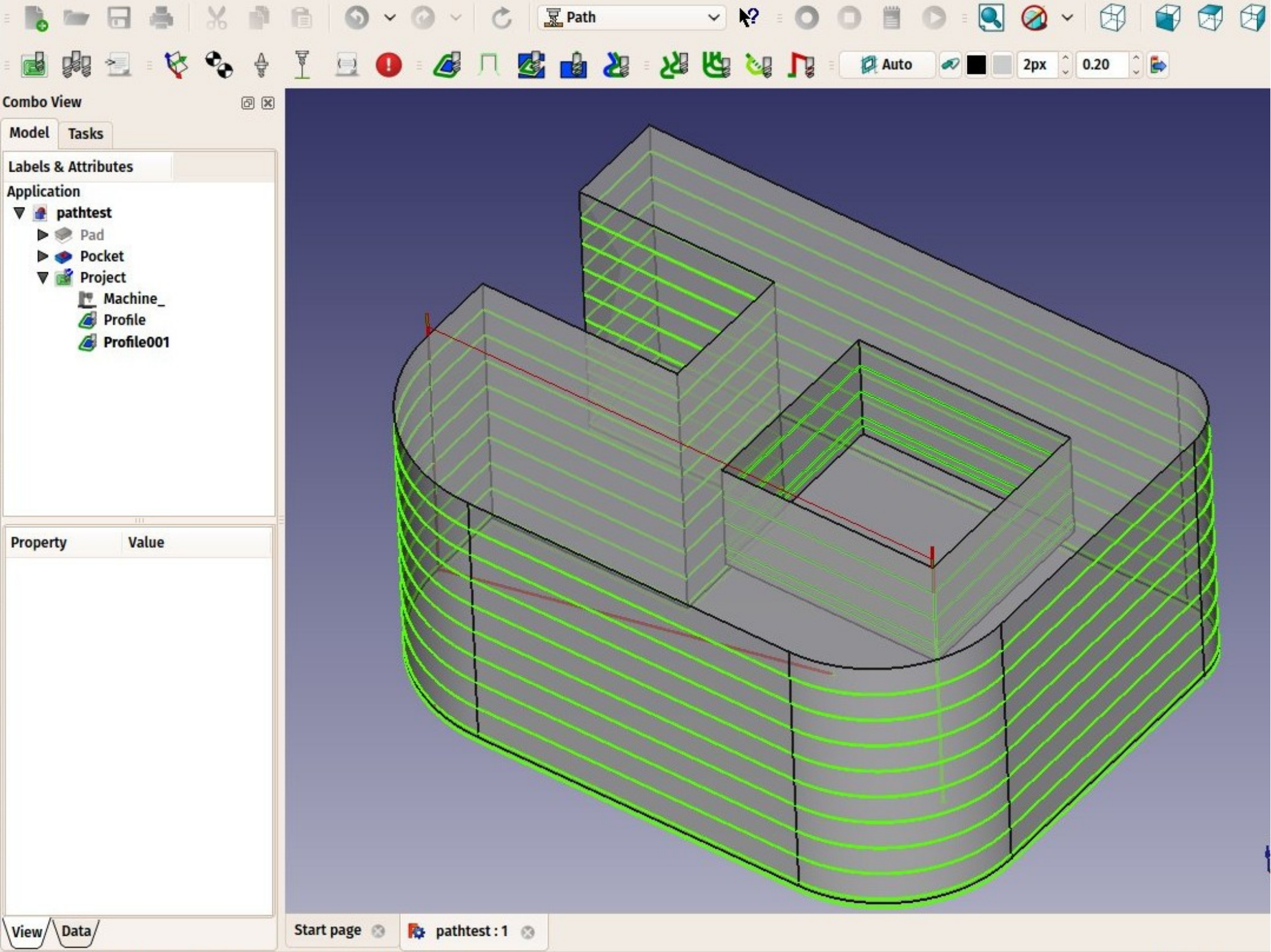
 **Plan**

Quantities schedule

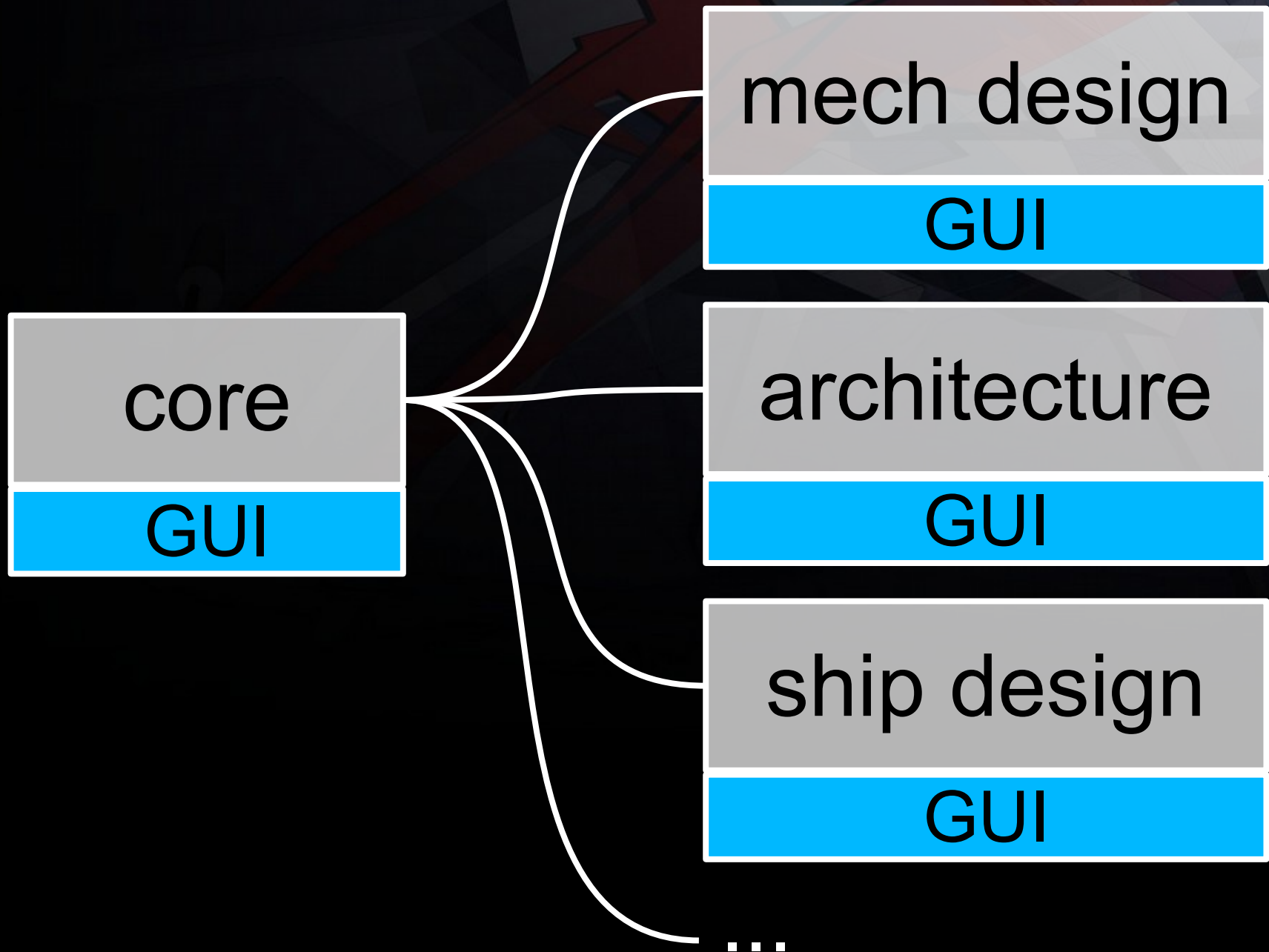
Property	Value
Base	
Display Mode	Spreadsheet
Visibility	true

	A	B	C	D	E	F	G	
1	Quantities schedule							
2	Project: testhouse							
3	Date: Sun A...							
4								
5		Material	Description	Color	Item	Volume (m³)		
6								
7	1. Columns							
8								
9		Steel	Structural steel, prot...		Metal column 1	0.0424115		
10					Metal column 2	0.0424115		
11					Total	0.084823		
12								
13		Concrete	25MPa cast-in-place structural concrete		Column 1	0.096		
14					Column 2	0.096		
15					Column 3	0.096		
16					Column 4	0.096		
17					Column Extension 1	0.37385		
18					Column Extension 2	0.37385		
19					Total	1.1317		





Modularité



Multiparadigme

- Core: simple conteneur avec vue 3D
- Modules: Toutes sortes de types d'objets peuvent cohabiter dans un même document
- Modules: Différents outils et manières de travailler sur un même document

Dépendences lourdes!

- Qt (interface)
- Python (script & autres)
- OpenCasCade (noyau géométrique)
- OpenInventor/Coin3D (affichage 3D)
- Beaucoup d'autres, chaque module a les siennes: KDL, IfcOpenShell, matplotlib, openSCAD, etc...

Python partout

- “colle” entre Core et GUI
- Certains modules écrits 100% en python
- L'utilisateur de python a les pouvoirs d'un ~~dieu~~ programmeur de C++
- Apprentissage transparent

Python est SUPER FACILE!

```
>>> print "Geeks rulez"  
Geeks rulez
```

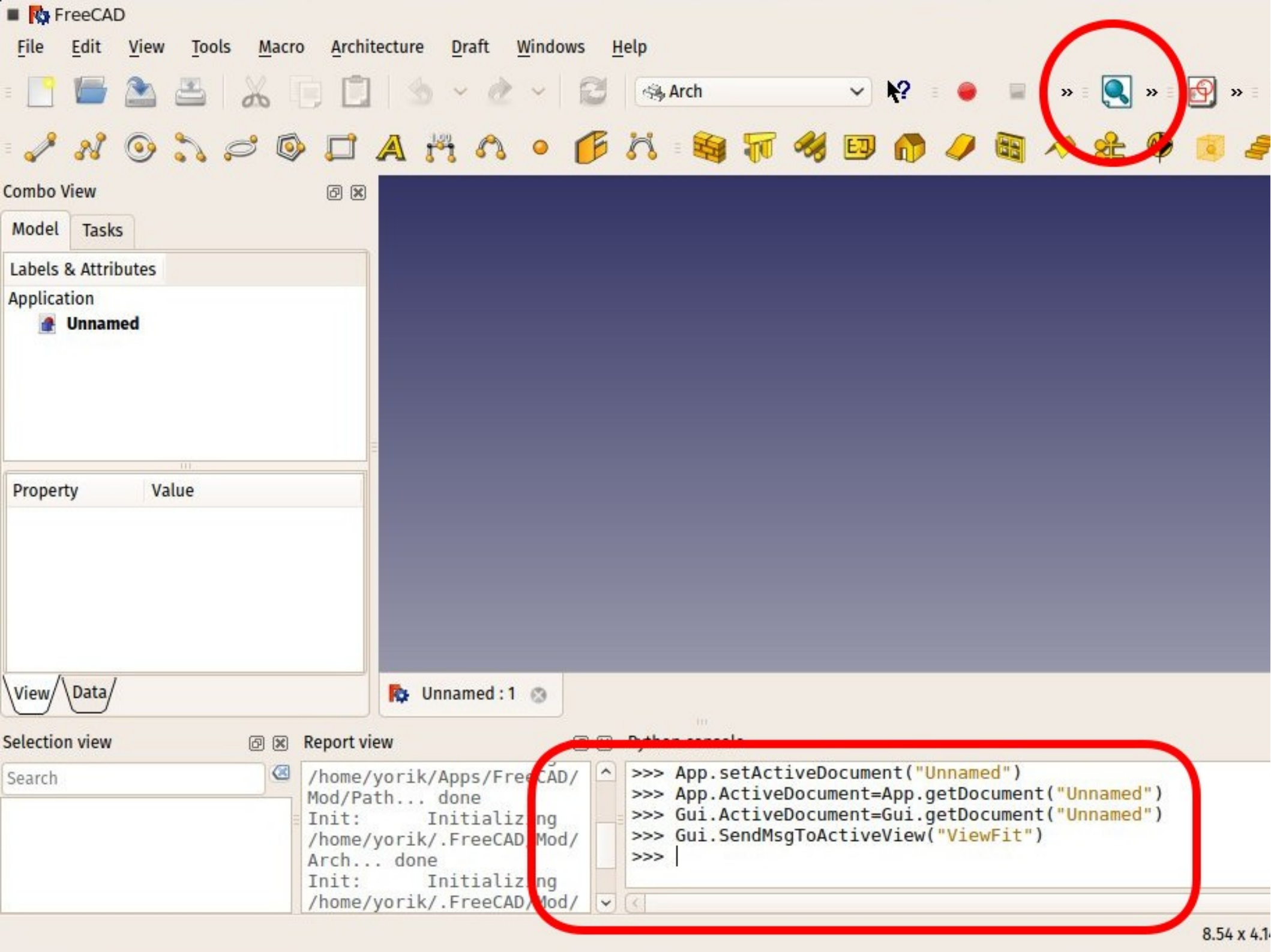
```
>>> resultat = 1 + 1  
>>> print resultat  
2
```

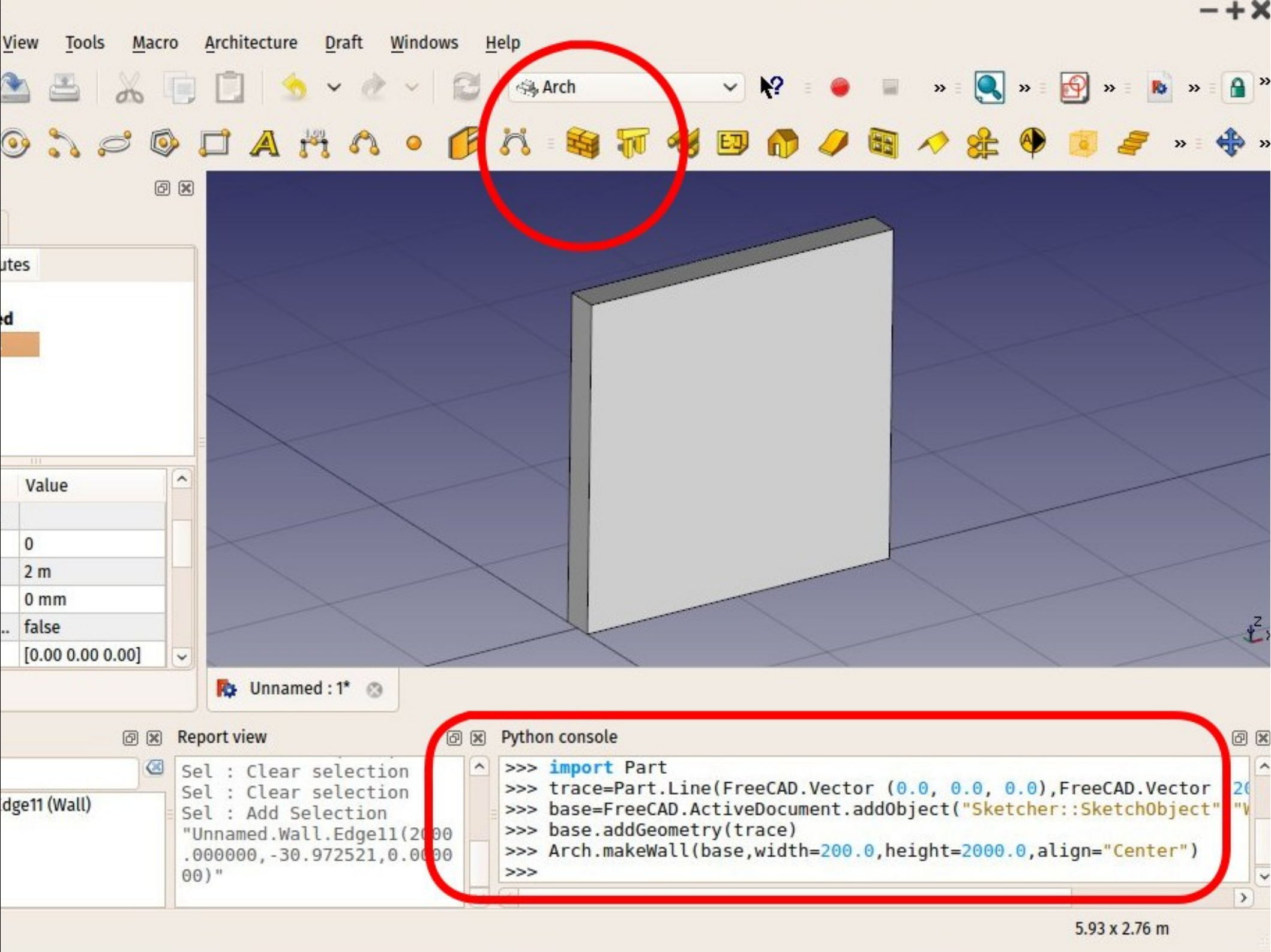
```
>>> print resultat + 3  
5
```

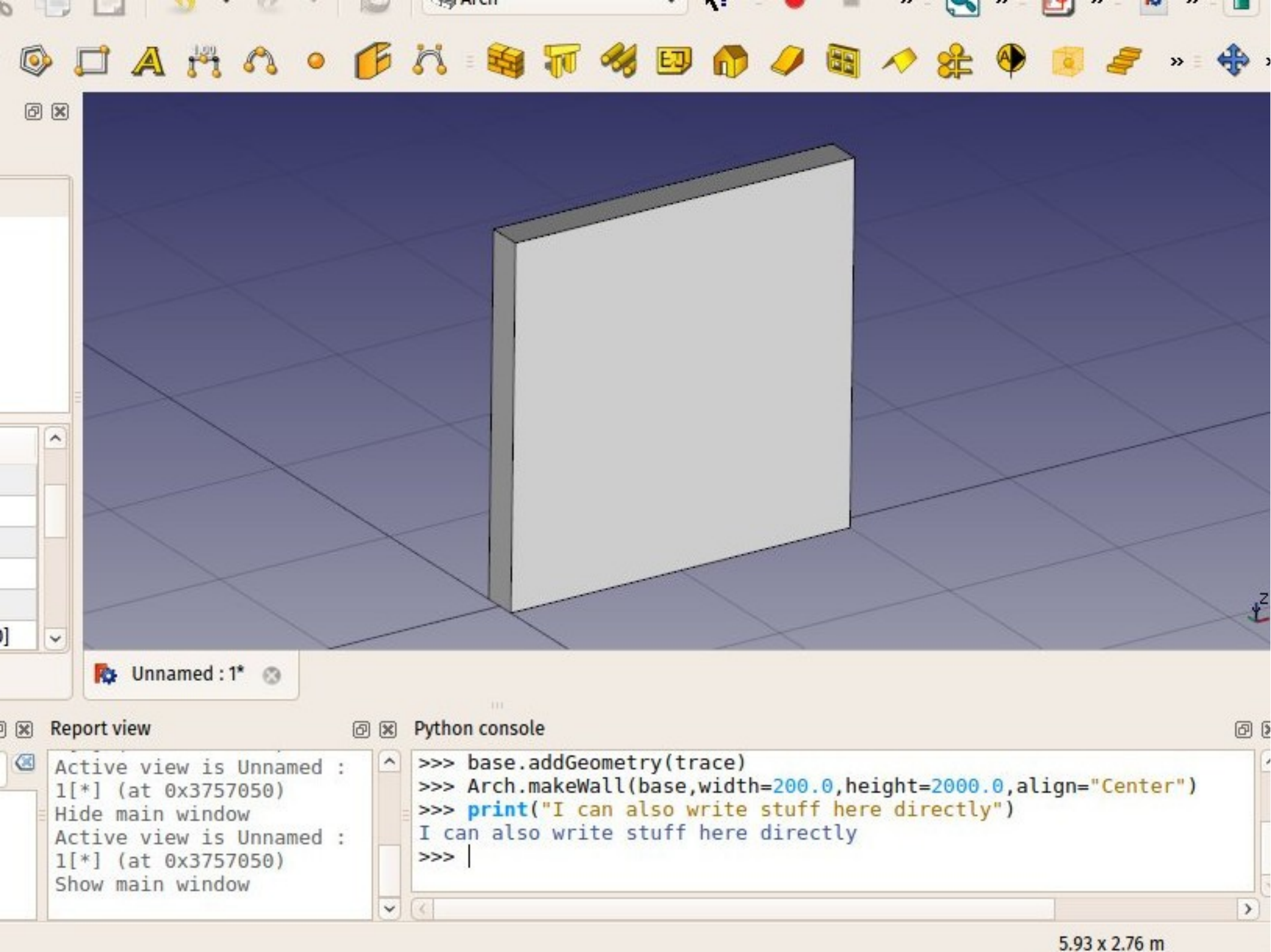
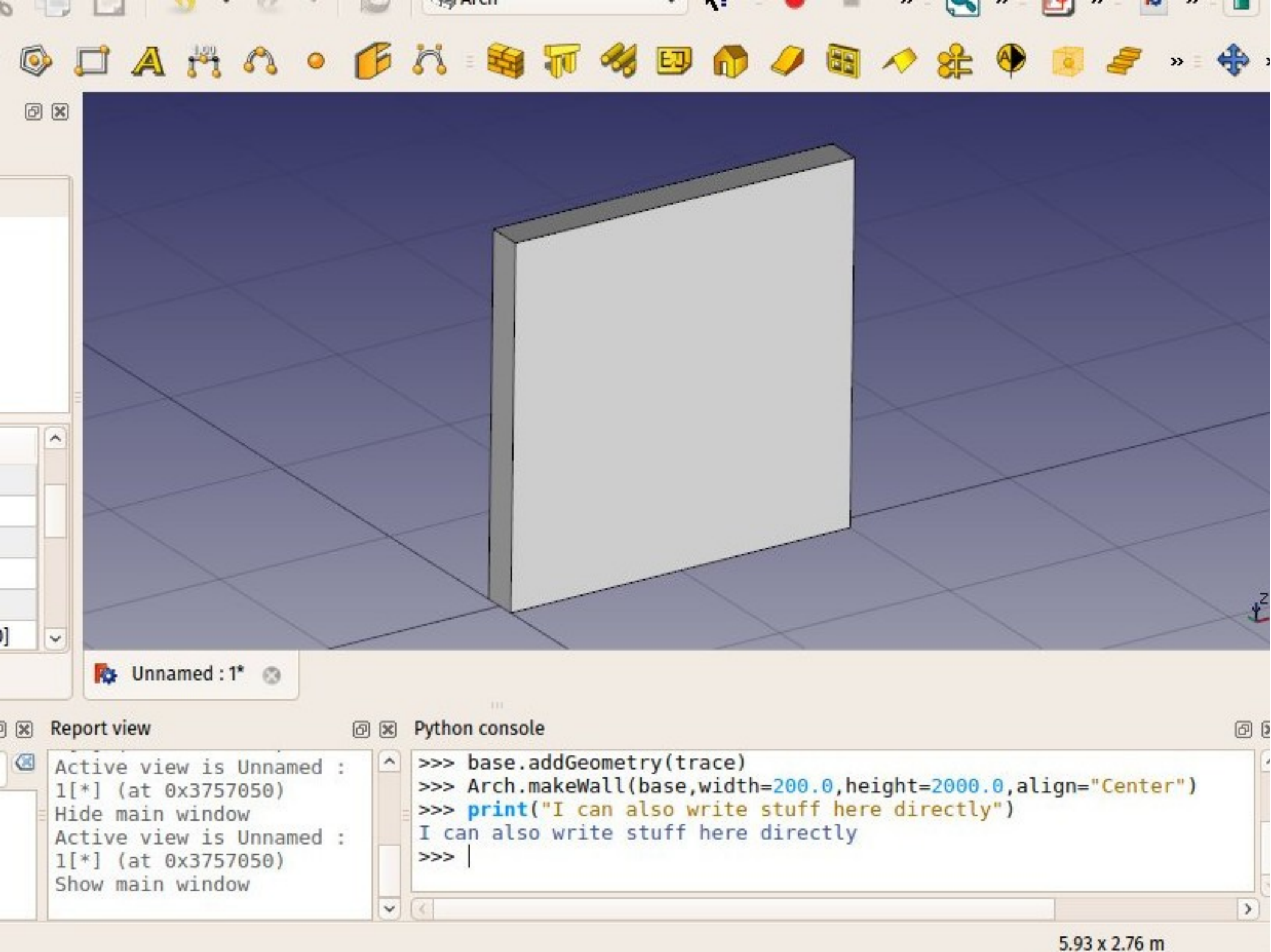
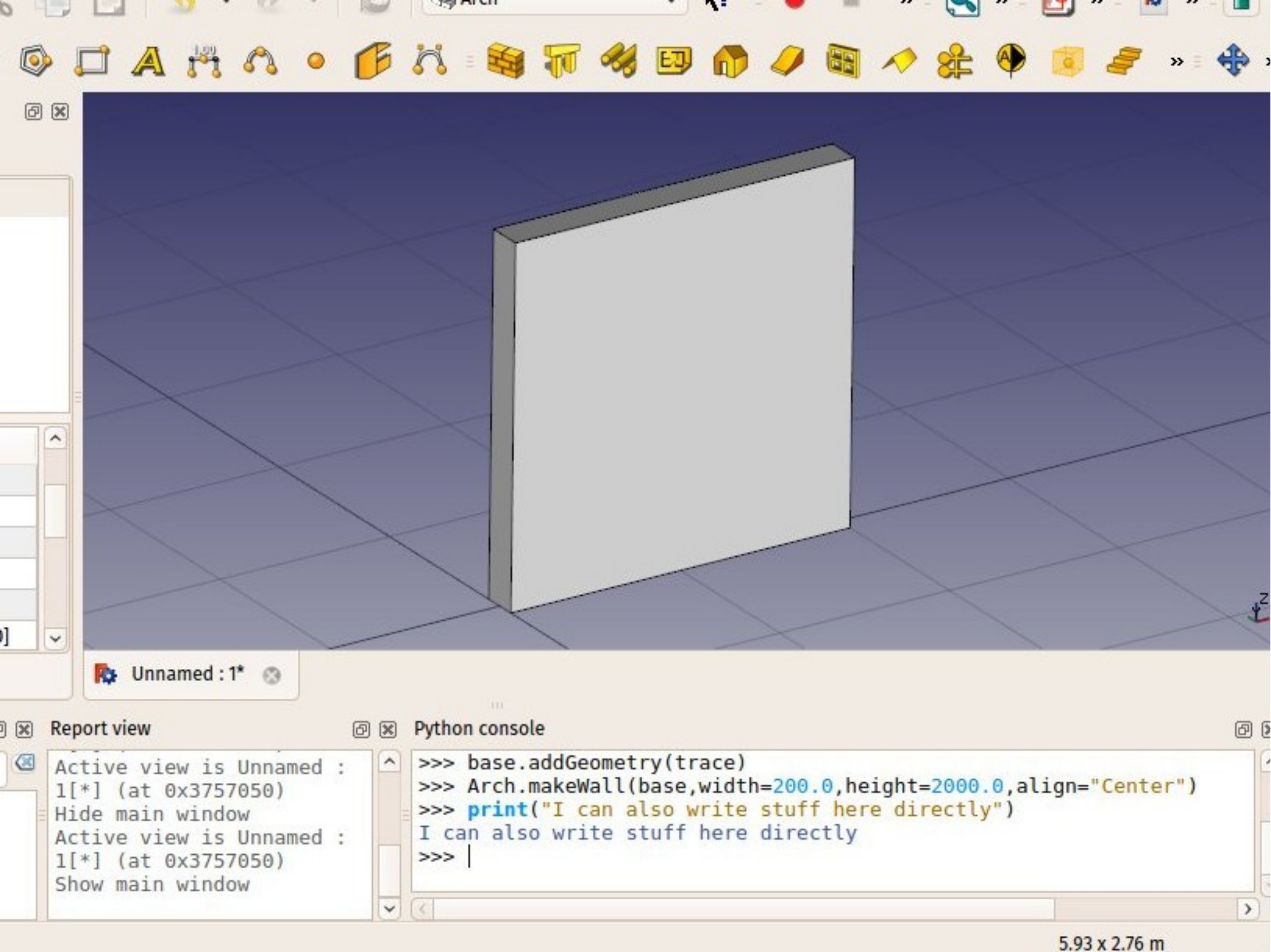


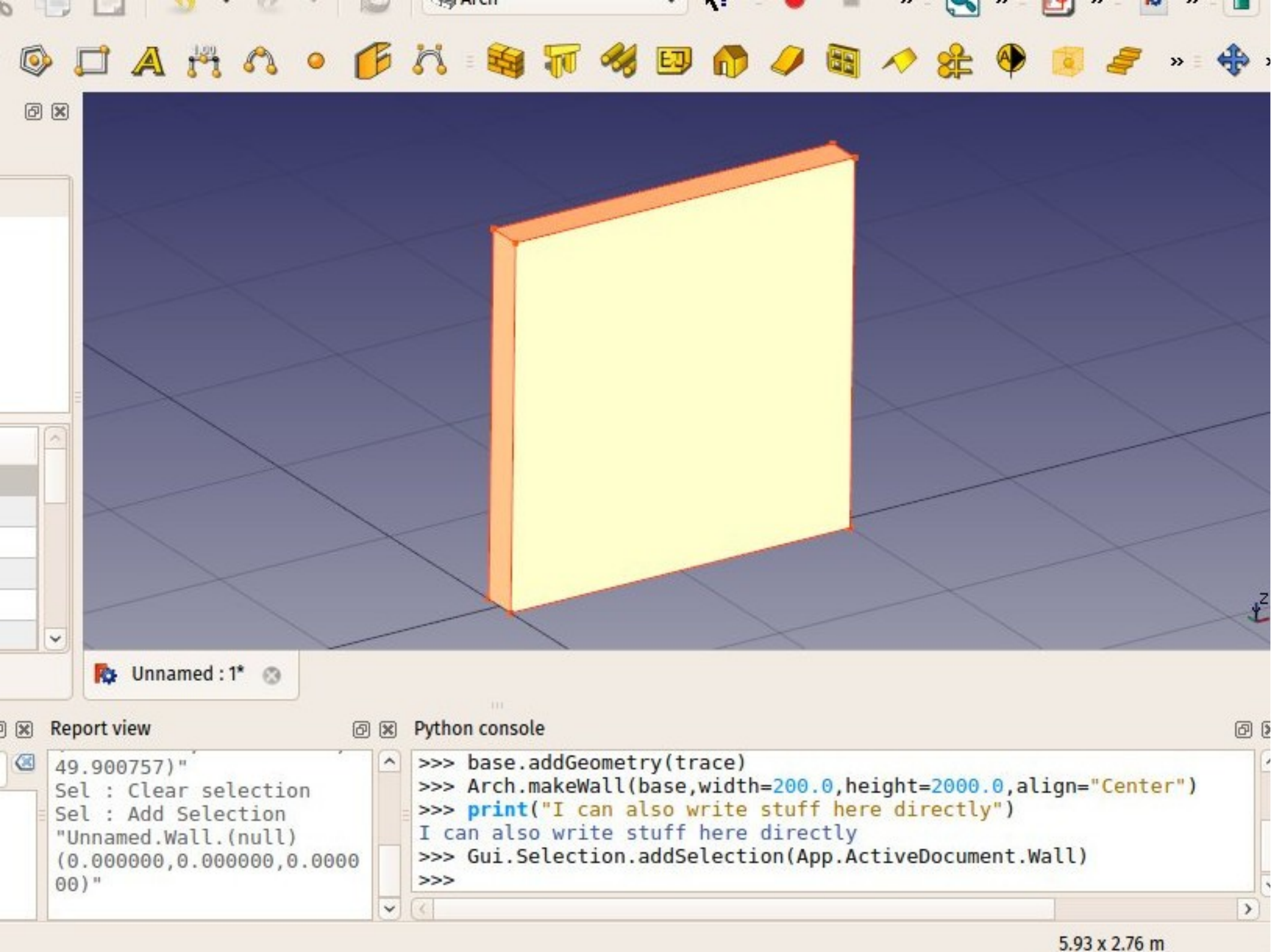
Python console

```
[GCC 4.9.2] on linux2
Type 'help', 'copyright', 'credits' or 'license' for more information.
>>> App.setActiveDocument("Unnamed")
>>> App.ActiveDocument=App.getDocument("Unnamed")
>>> Gui.ActiveDocument=Gui.getDocument("Unnamed")
>>>
>>> print "Geeks will rule the world MESMO"
Geeks will rule the world MESMO
>>> resultado = 1 + 1
>>> print resultado
2
>>> print resultado + 3
5
>>> |
```









Unnamed : 1*

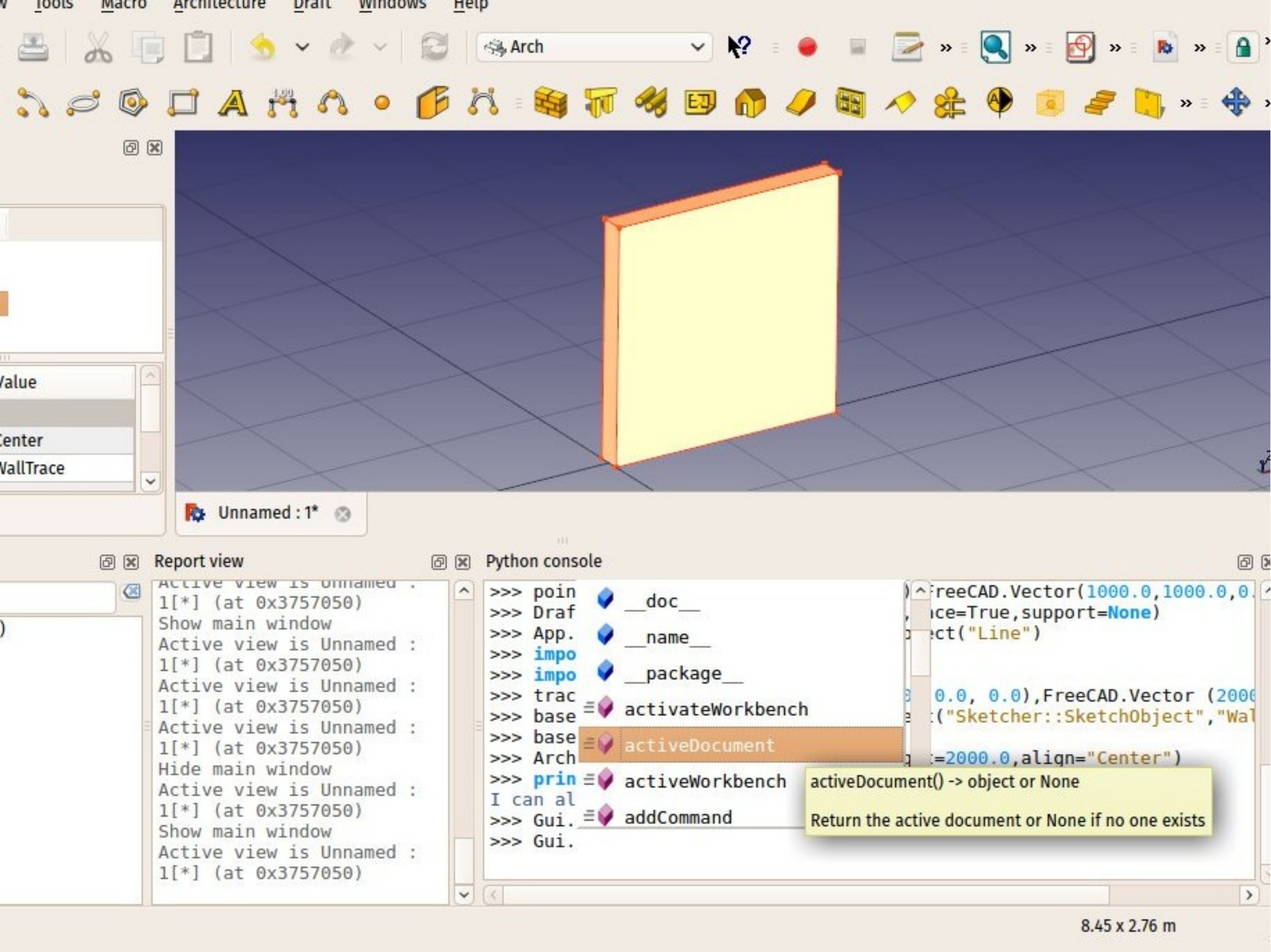
Report view

Python console

```
49.900757)"
Sel : Clear selection
Sel : Add Selection
"Unnamed.Wall.(null)
(0.000000,0.000000,0.0000
00)"
```

```
>>> base.addGeometry(trace)
>>> Arch.makeWall(base,width=200.0,height=2000.0,align="Center")
>>> print("I can also write stuff here directly")
I can also write stuff here directly
>>> Gui.Selection.addSelection(App.ActiveDocument.Wall)
>>>
```

5.93 x 2.76 m



Unnamed : 1*

Report view

```
Active view is Unnamed : 1[*] (at 0x3757050)
Show main window
Active view is Unnamed : 1[*] (at 0x3757050)
Active view is Unnamed : 1[*] (at 0x3757050)
Active view is Unnamed : 1[*] (at 0x3757050)
Active view is Unnamed : 1[*] (at 0x3757050)
Hide main window
Active view is Unnamed : 1[*] (at 0x3757050)
Show main window
Active view is Unnamed : 1[*] (at 0x3757050)
```

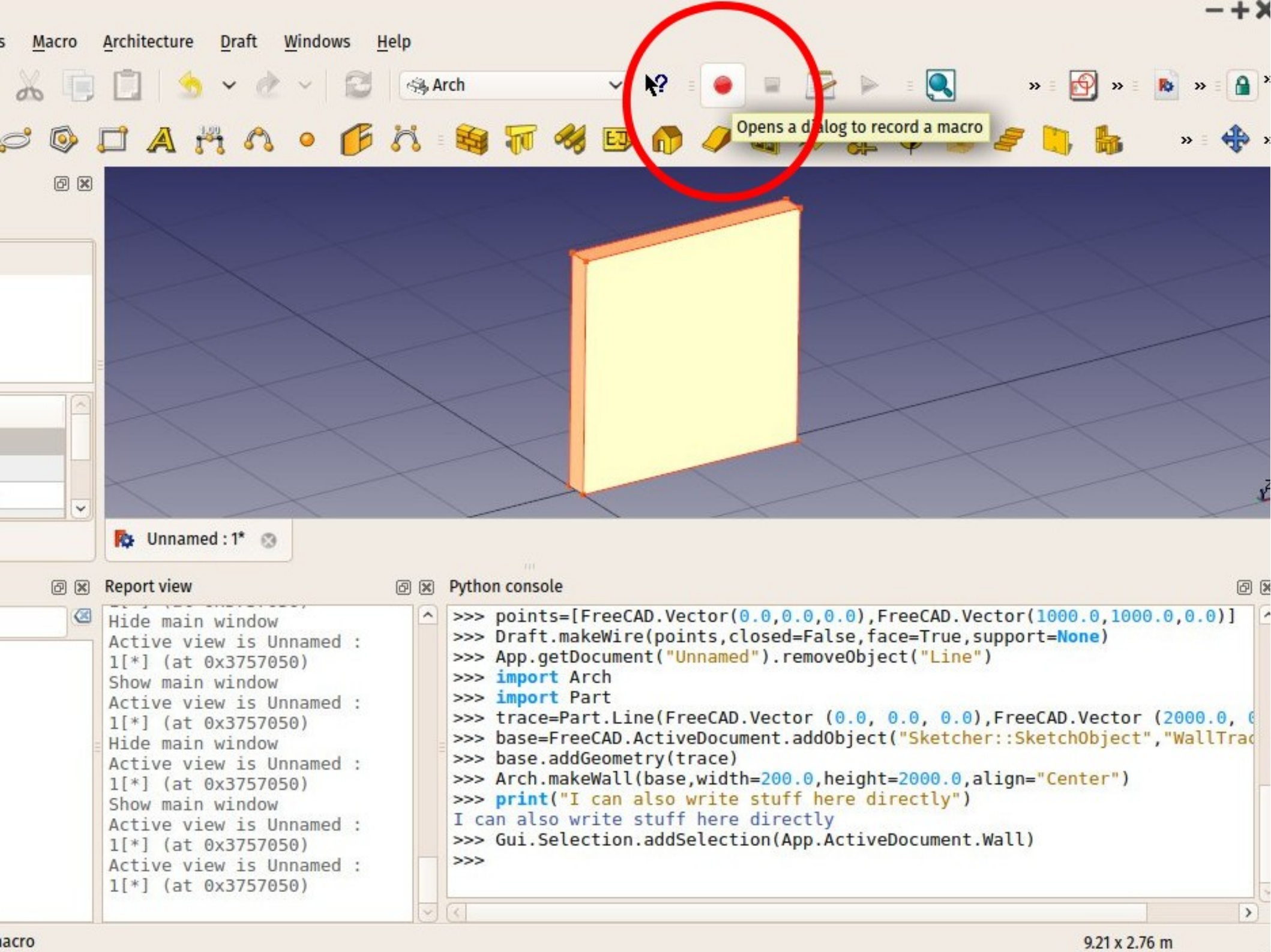
Python console

```
>>> poin  _doc_
>>> Draf  _name_
>>> App.   _package_
>>> impo   activateWorkbench
>>> trac   activeDocument
>>> base   activeWorkbench
>>> Arch   addCommand
>>> prin
>>> Gui.
>>> Gui.
```

activeDocument() -> object or None

Return the active document or None if no one exists exists

8.45 x 2.76 m





Macro recording

Macro name:
house-o-matic

Macro path:
/home/yorik/.FreeCAD/ ...

Record Stop Cancel

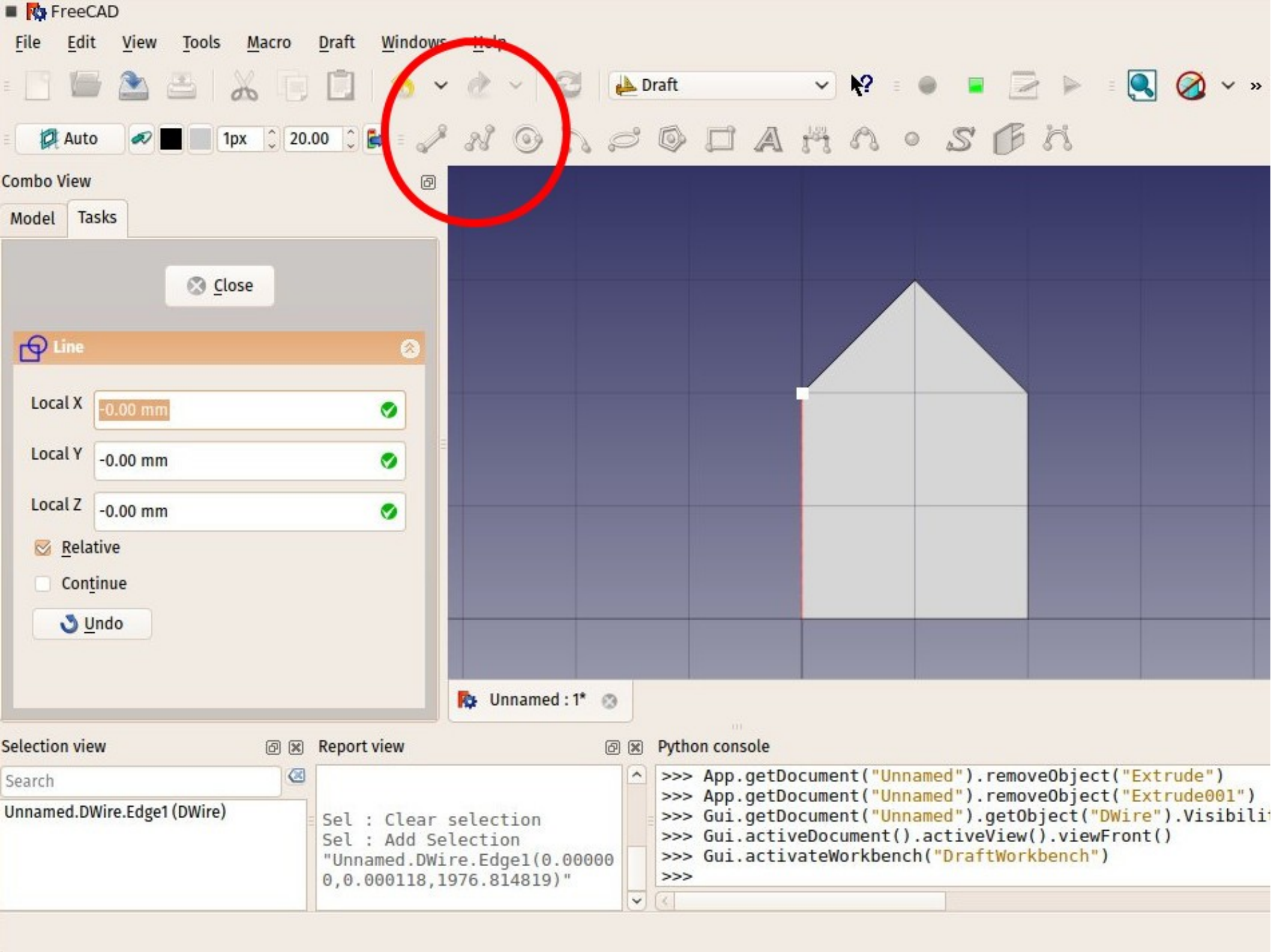
Unnamed : 1*

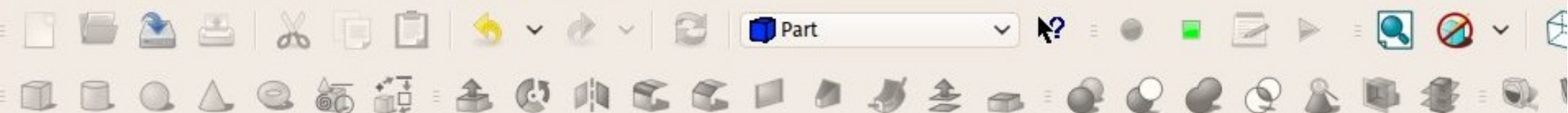
Report view

```
Show main window
Active view is Unnamed :
1[*] (at 0x3757050)
Sel : Clear selection
Sel : Rmv Selection
"Unnamed.Wall.(null)"
```

Python console

```
I can also write stuff here directly
>>> Gui.Selection.addSelection(App.ActiveDocument.Wall)
>>> App.getDocument("Unnamed").removeObject("Wall")
>>> App.getDocument("Unnamed").removeObject("WallTrace")
>>> Gui.activeDocument().activeView().viewTop()
>>>
```





Combo View

Model

Tasks

Apply

Close

OK



Extrude

Direction

X: 0.00

Y: 0.00

Z: 1.00

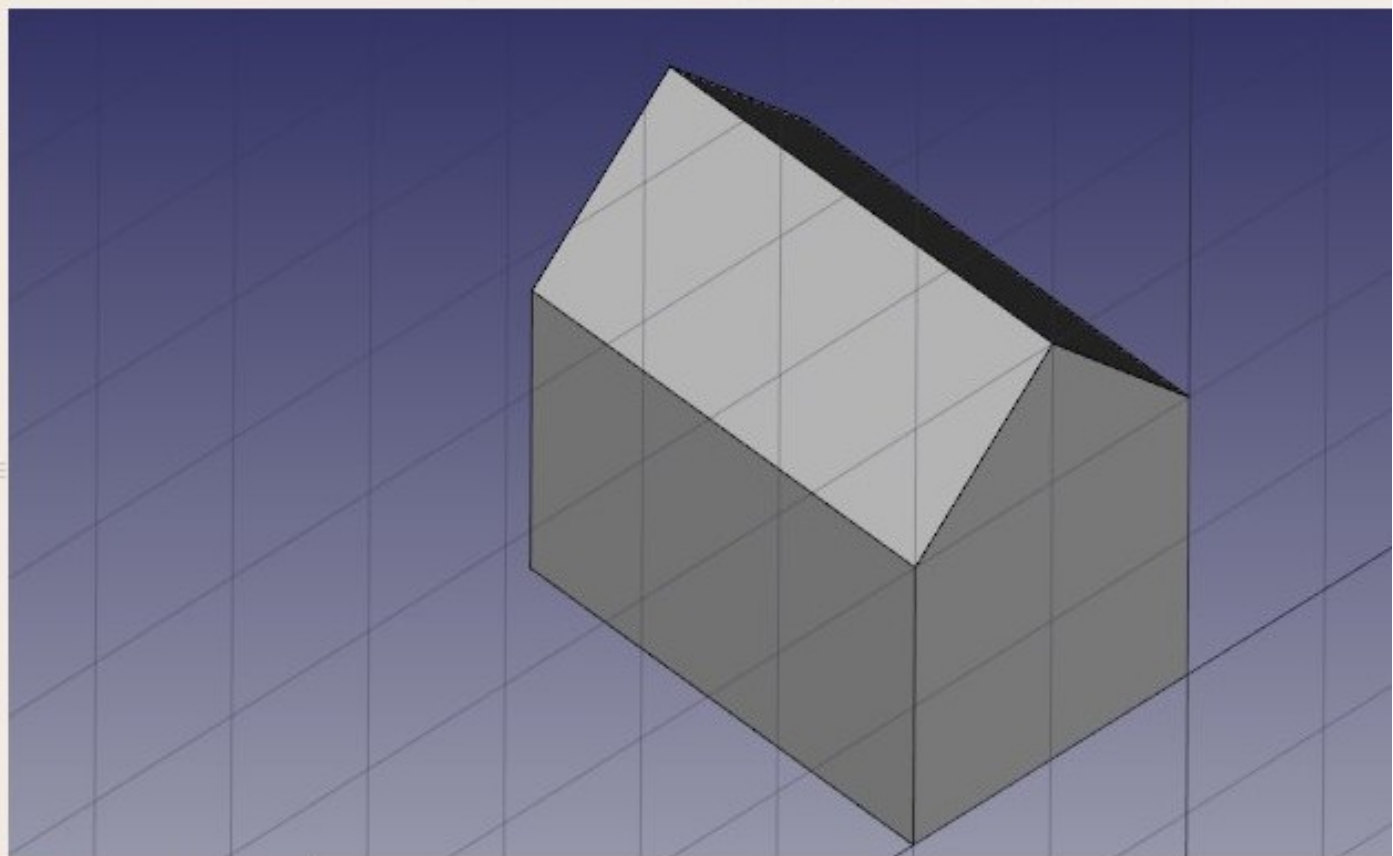
Length: 3000.00

☒ Along normal

Note: This option works for planes only

☒ Create solid

Taper outward angle: 0 deg



Unnamed: 1*

Selection view

Search

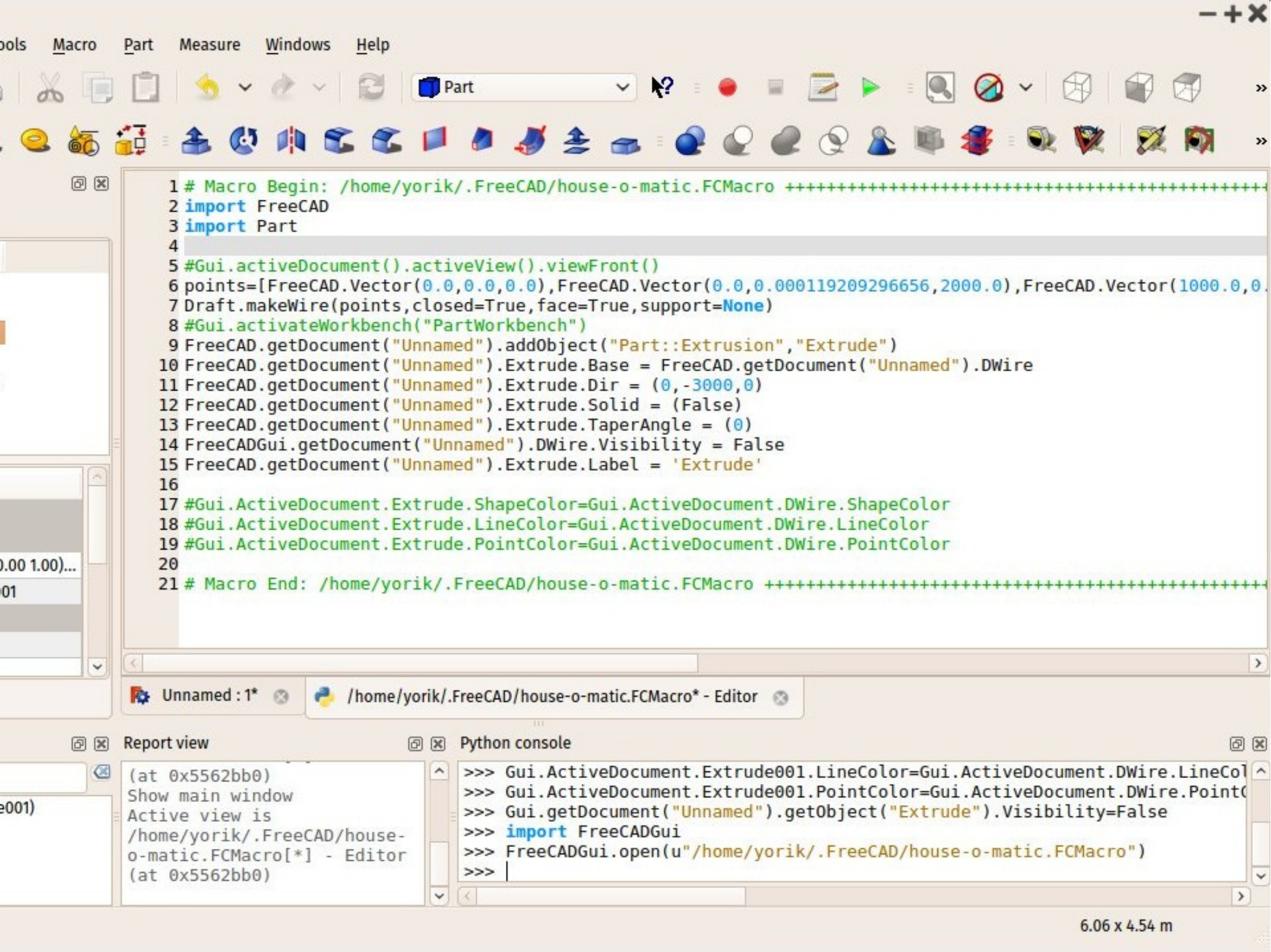
Unnamed.Extrude (Extrude)

Report view

```
1[*] (at 0x3757050)
Sel : Clear selection
Sel : Add Selection
"Unnamed.DWire.Face1(1319.80
3467,0.000072,1208.478760)"
```

Python console

```
>>>
>>> Gui.ActiveDocument.Extrude001.ShapeColor=Gui.ActiveDocum
>>> Gui.ActiveDocument.Extrude001.LineColor=Gui.ActiveDocume
>>> Gui.ActiveDocument.Extrude001.PointColor=Gui.ActiveDocum
>>> Gui.getDocument("Unnamed").getObject("Extrude").Visibili
>>>
```



```
1 # Macro Begin: /home/yorik/.FreeCAD/house-o-matic.FCMacro ++++++
2 import FreeCAD
3 import Part
4
5 #Gui.activeDocument().activeView().viewFront()
6 points=[FreeCAD.Vector(0.0,0.0,0.0),FreeCAD.Vector(0.0,0.000119209296656,2000.0),FreeCAD.Vector(1000.0,0.0,0.0)]
7 Draft.makeWire(points,closed=True,face=True,support=None)
8 #Gui.activateWorkbench("PartWorkbench")
9 FreeCAD.getDocument("Unnamed").addObject("Part::Extrusion","Extrude")
10 FreeCAD.getDocument("Unnamed").Extrude.Base = FreeCAD.getDocument("Unnamed").DWire
11 FreeCAD.getDocument("Unnamed").Extrude.Dir = (0,-3000,0)
12 FreeCAD.getDocument("Unnamed").Extrude.Solid = (False)
13 FreeCAD.getDocument("Unnamed").Extrude.TaperAngle = (0)
14 FreeCADGui.getDocument("Unnamed").DWire.Visibility = False
15 FreeCAD.getDocument("Unnamed").Extrude.Label = 'Extrude'
16
17 #Gui.ActiveDocument.Extrude.ShapeColor=Gui.ActiveDocument.DWire.ShapeColor
18 #Gui.ActiveDocument.Extrude.LineColor=Gui.ActiveDocument.DWire.LineColor
19 #Gui.ActiveDocument.Extrude.PointColor=Gui.ActiveDocument.DWire.PointColor
20
21 # Macro End: /home/yorik/.FreeCAD/house-o-matic.FCMacro ++++++
```

Unnamed : 1* /home/yorik/.FreeCAD/house-o-matic.FCMacro* - Editor

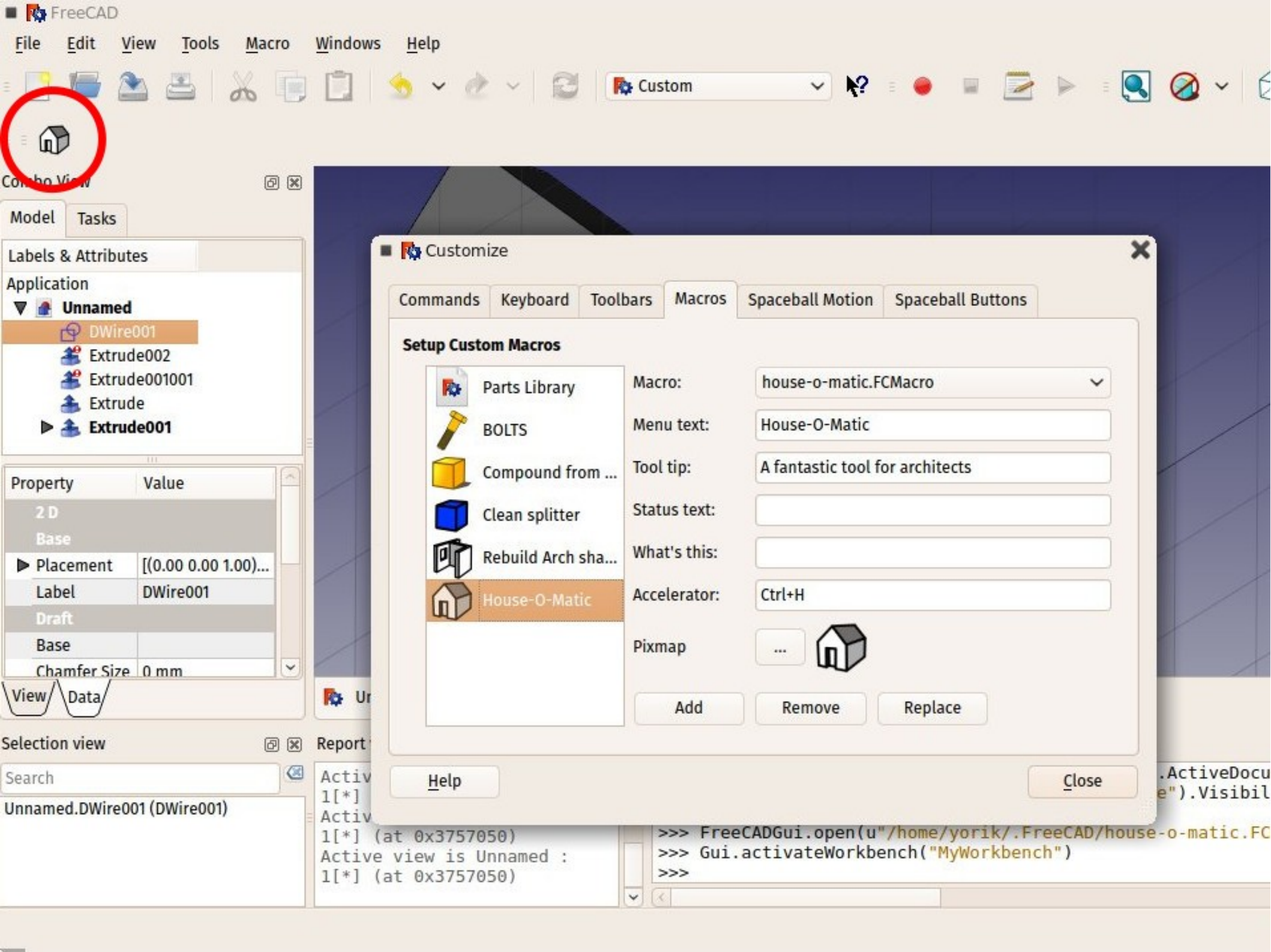
Report view

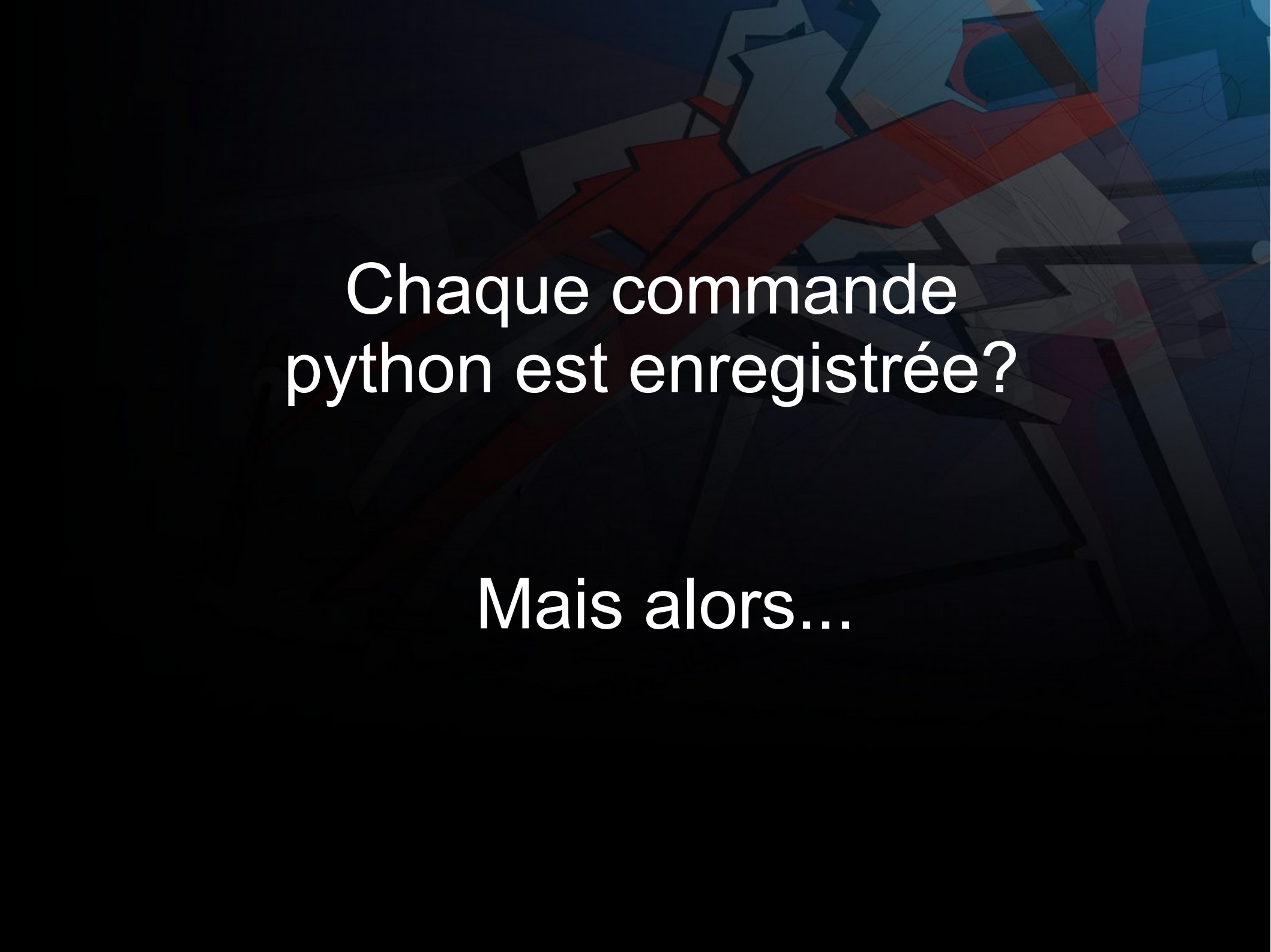
```
(at 0x5562bb0)
Show main window
Active view is
/home/yorik/.FreeCAD/house-
o-matic.FCMacro[*] - Editor
(at 0x5562bb0)
```

Python console

```
>>> Gui.ActiveDocument.Extrude001.LineColor=Gui.ActiveDocument.DWire.LineColor
>>> Gui.ActiveDocument.Extrude001.PointColor=Gui.ActiveDocument.DWire.PointColor
>>> Gui.getDocument("Unnamed").getObject("Extrude").Visibility=False
>>> import FreeCADGui
>>> FreeCADGui.open(u"/home/yorik/.FreeCAD/house-o-matic.FCMacro")
>>>
```

6.06 x 4.54 m





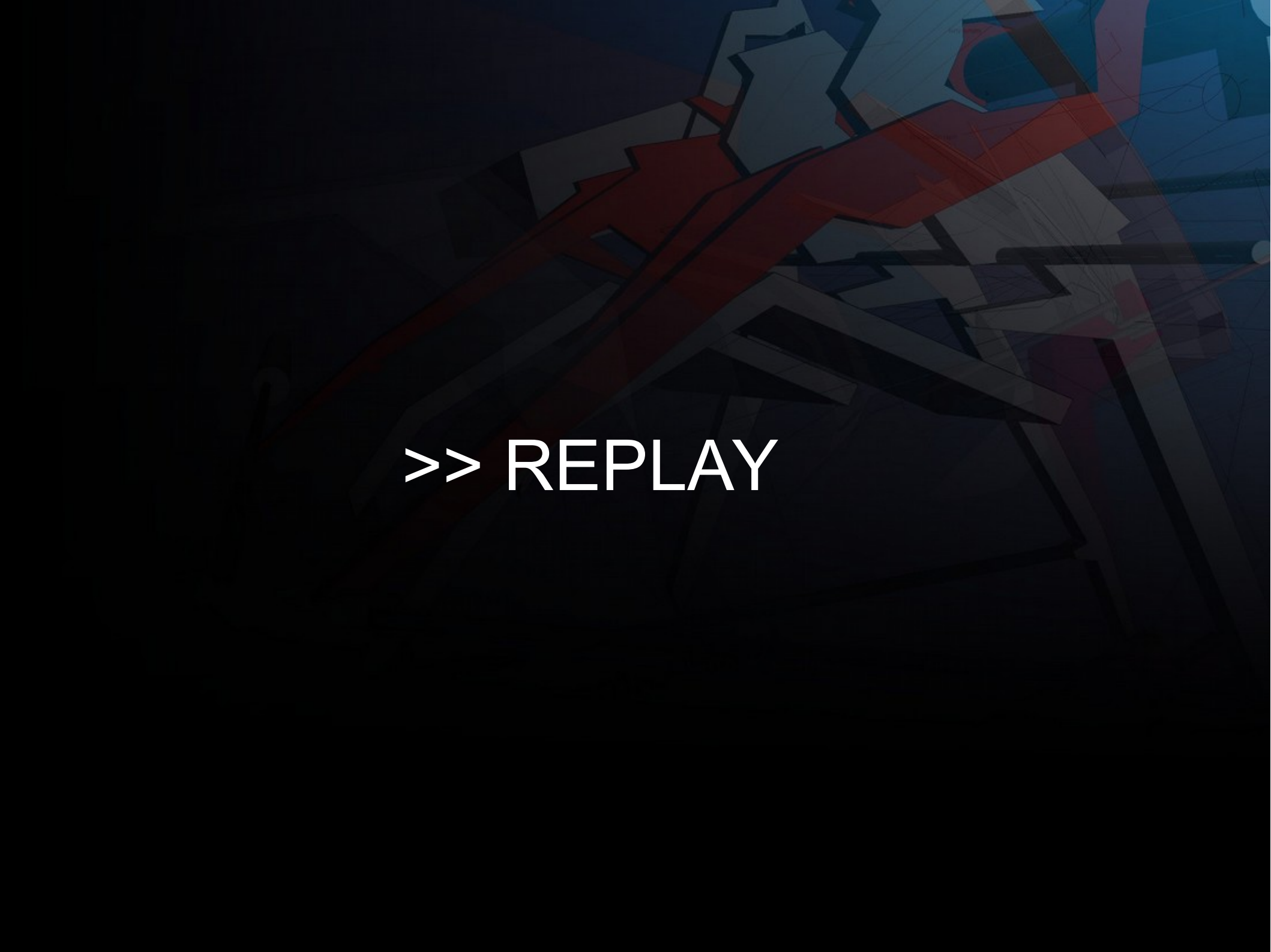
Chaque commande
python est enregistrée?

Mais alors...

A fatal exception 0E has occurred at 0028:C00069F8 in UxD UMM(01) + 000059F8. The current application will be terminated.

- * Press any key to terminate the current application.
- * Press CTRL+ALT+DEL to restart your computer. You will lose any unsaved information in all applications.

Press any key to continue _

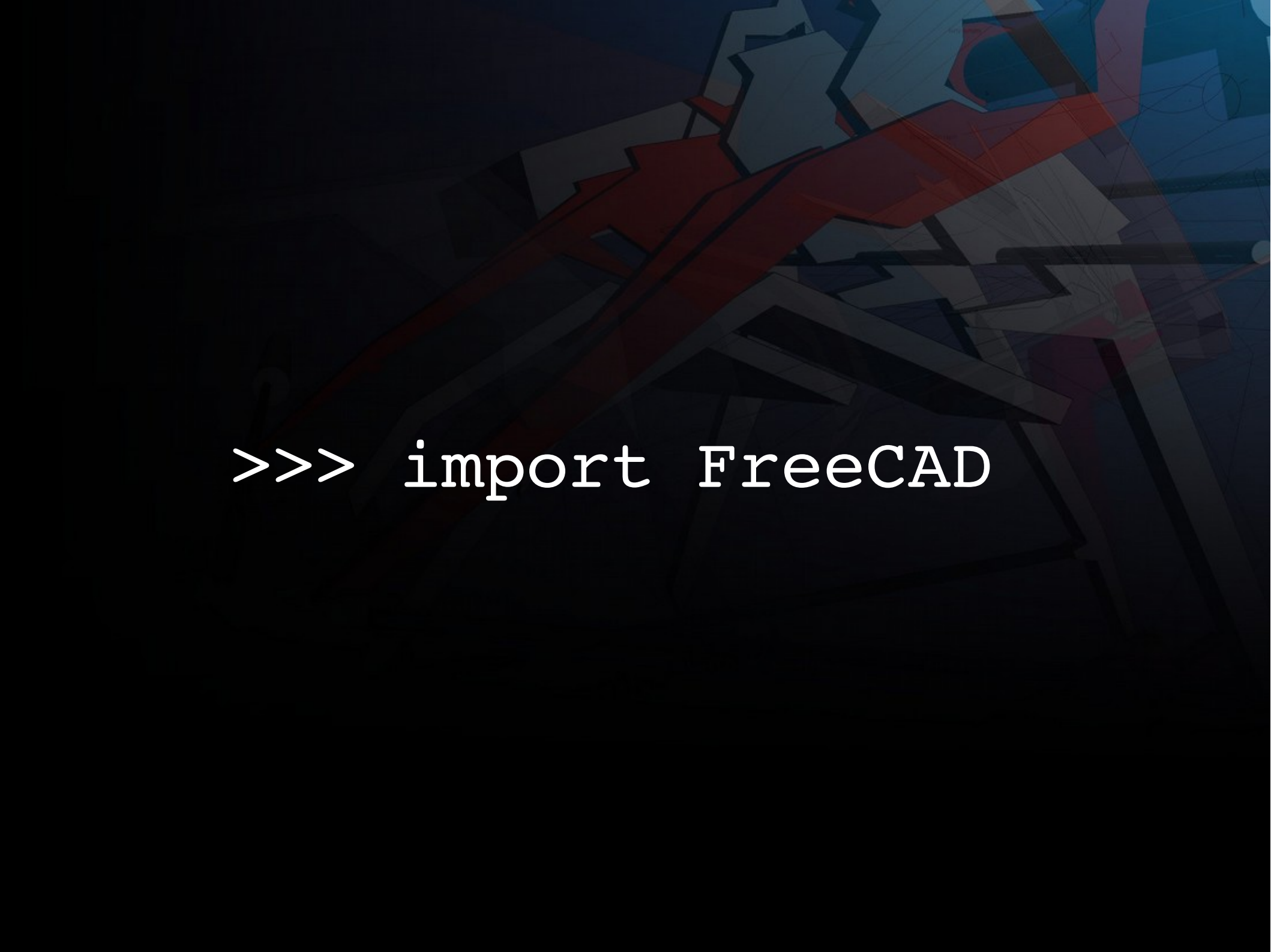


>> REPLAY



Séparation entre Core and GUI?

Mais alors...



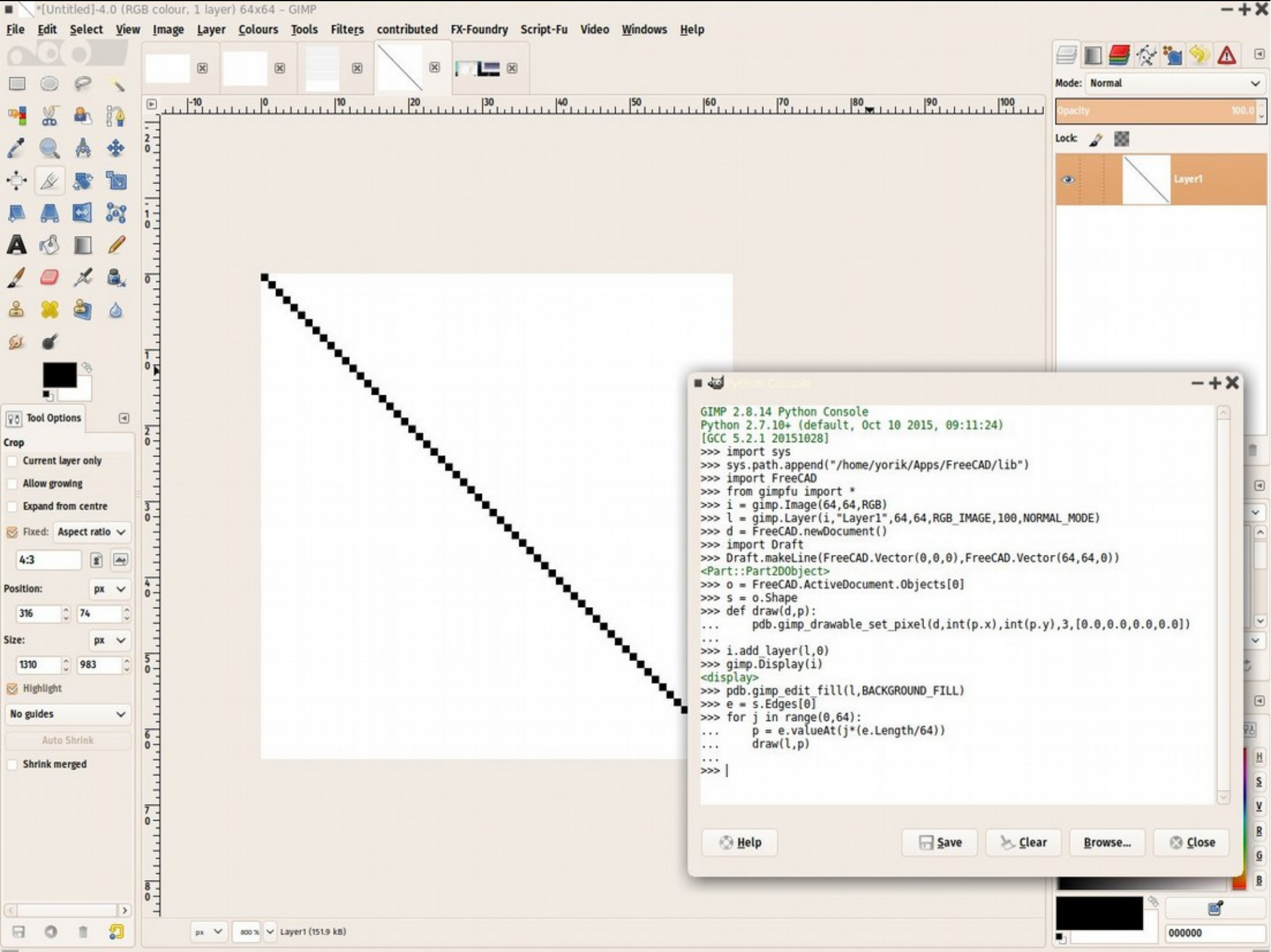
```
>>> import FreeCAD
```

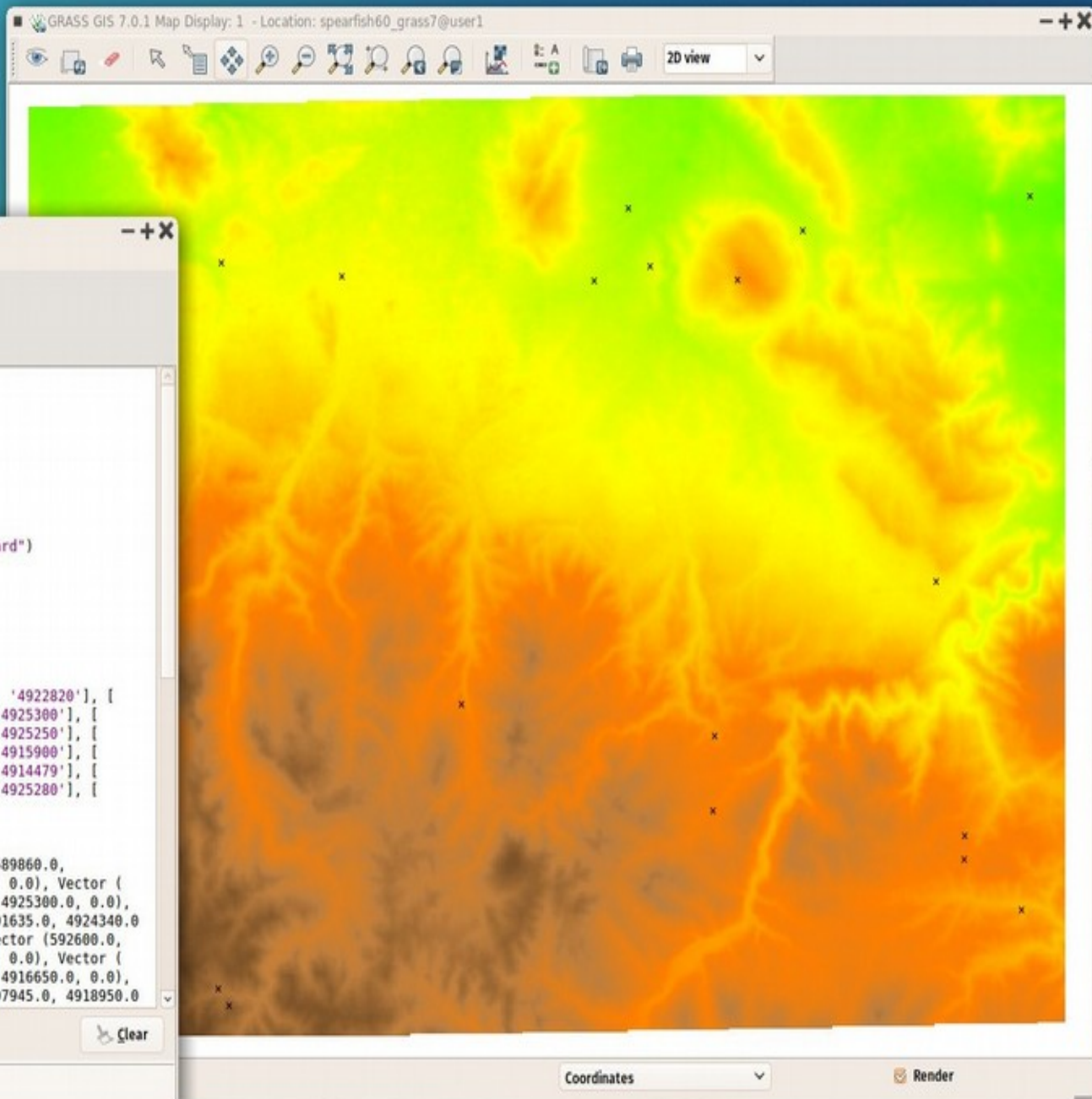
FreeCAD tourne sans son interface (mode text / console) ou comme module python

```
~ freecad -c 0 12:32 h
FreeCAD 0.16, Libs: 0.16R5875 (Git)
© Juergen Riegel, Werner Mayer, Yorik van Havre 2001-2015
#####          #####  ###  #####
#                #      # #  #    #
#      ##  #####  #####  #      #  #  #  #
#####  #  #  #  #  #  #  #  #####  #  #
#      #  #####  #####  #      #  #  #  #
#      #  #      #      #      #  #  #  #  ##  ##  ##
#      #  #####  #####  ###  #      #  #####  ##  ##  ##

[FreeCAD Console mode <Use Ctrl-D (i.e. EOF) to exit.>]
>>> FreeCAD.newDocument()
<Document object at 0x17f1400>
>>> import Draft
>>> p1 = FreeCAD.Vector(0,0,0)
>>> p2 = FreeCAD.Vector(10,0,0)
>>> Draft.makeLine(p1,p2)
<Part::Part2DObject>
>>> □
```

- Intégrable et utilisable dans n'importe quelle application qui utilise Python, en une seule ligne de code
- Tous les modules fonctionnent (également sans leur interface)
- Peut fonctionner comme serveur, Web entre autres
- Peut avoir d'autres interfaces





Welcome to wxGUI Interactive Python Shell 0.9.8

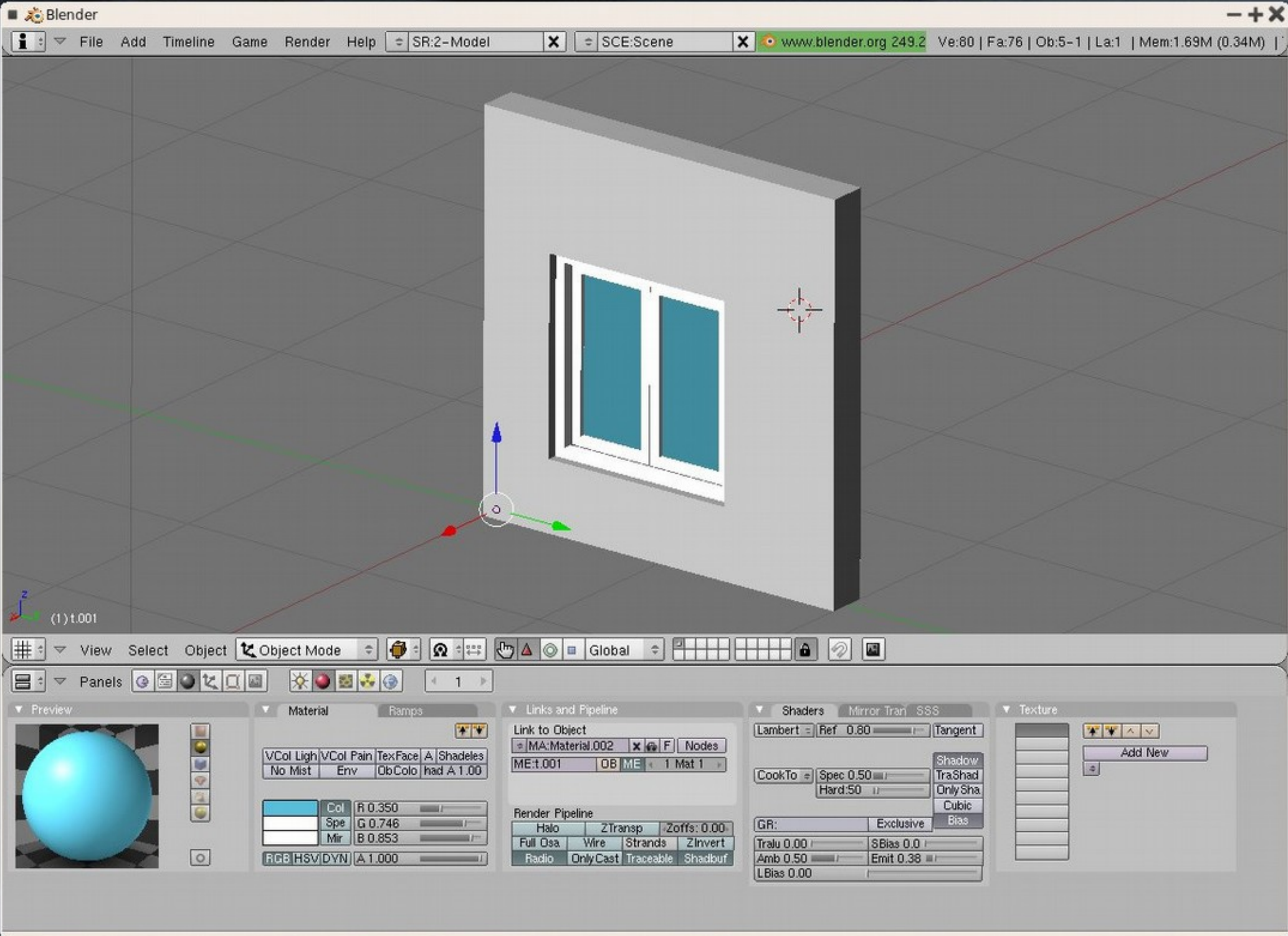
Type "help(grass)" for more GRASS scripting related information.
Type "AddLayer()" to add raster or vector to the layer tree.

Python 2.7.10+ (default, Oct 10 2015, 09:11:24)
[GCC 5.2.1 20151028] on linux2
Type "help", "copyright", "credits" or "license" for more information.

```
>>> import FreeCAD
>>> buffer = grass.read_command("v.out.ascii", input="archsites", format="standard")
>>> buffer = buffer.split("\n")
>>> buffer = buffer[:-1]
>>> buffer = [l for l in buffer if l[0] == " "]
>>> buffer = [l for l in buffer if l[1] != "1"]
>>> buffer = [l.strip() for l in buffer]
>>> buffer = [l.split(" ") for l in buffer]
>>> buffer
[['593493', '4914730'], ['591950', '4923000'], ['589860', '4922000'], ['590400', '4922820'], [
'593549', '4925500'], ['600375', '4925235'], ['606635', '4920773'], ['595755', '4925300'], [
'601000', '4926310'], ['602550', '4917370'], ['591635', '4924340'], ['603000', '4925250'], [
'592100', '4921200'], ['592600', '4922000'], ['604194', '4925980'], ['608200', '4915900'], [
'601400', '4925450'], ['591000', '4916000'], ['607150', '4916650'], ['593690', '4914479'], [
'608355', '4926490'], ['597945', '4918950'], ['607159', '4917000'], ['591583', '4925280'], [
'602580', '4918480']]
>>> buffer = [FreeCAD.Vector(int(l[0]),int(l[1]),0) for l in buffer]
>>> buffer
[Vector (593493.0, 4914730.0, 0.0), Vector (591950.0, 4923000.0, 0.0), Vector (589860.0,
4922000.0, 0.0), Vector (590400.0, 4922820.0, 0.0), Vector (593549.0, 4925500.0, 0.0), Vector (
600375.0, 4925235.0, 0.0), Vector (606635.0, 4920773.0, 0.0), Vector (595755.0, 4925300.0, 0.0),
Vector (601000.0, 4926310.0, 0.0), Vector (602550.0, 4917370.0, 0.0), Vector (591635.0, 4924340.0
, 0.0), Vector (603000.0, 4925250.0, 0.0), Vector (592100.0, 4921200.0, 0.0), Vector (592600.0,
4922000.0, 0.0), Vector (604194.0, 4925980.0, 0.0), Vector (608200.0, 4915900.0, 0.0), Vector (
601400.0, 4925450.0, 0.0), Vector (591000.0, 4916000.0, 0.0), Vector (607150.0, 4916650.0, 0.0),
Vector (593690.0, 4914479.0, 0.0), Vector (608355.0, 4926490.0, 0.0), Vector (597945.0, 4918950.0
```

Clear

Map layers | Command console | Search modules | Python shell



Le web!

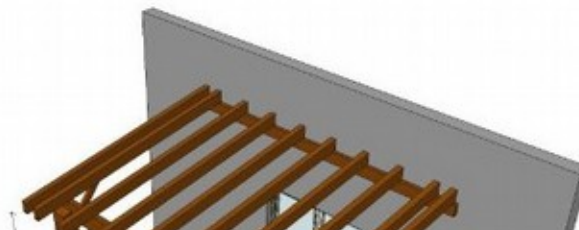
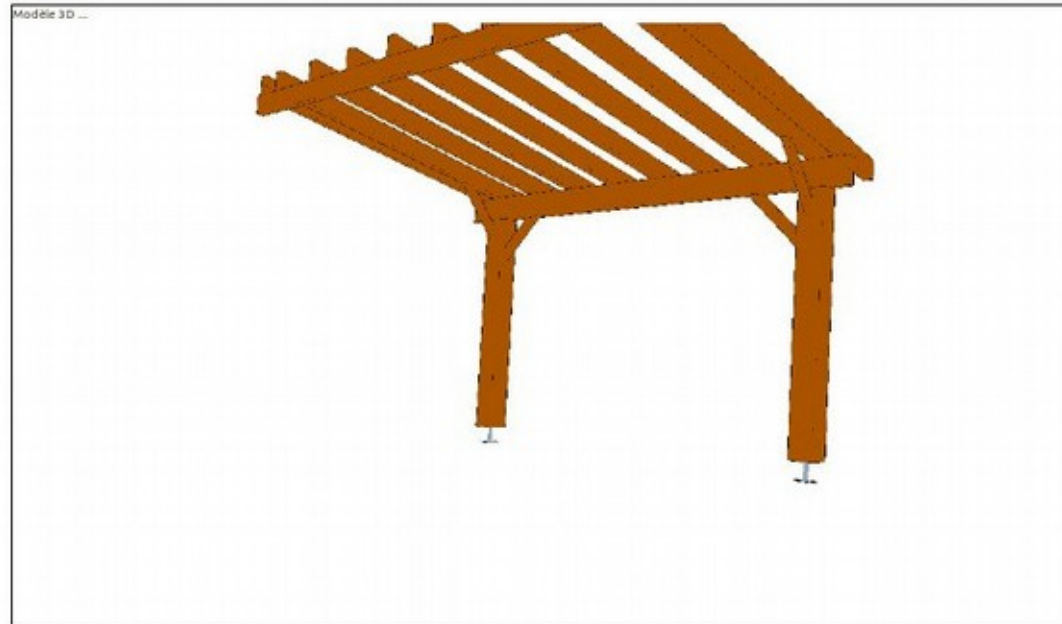
- Une application Web peut utiliser FreeCAD pour générer du contenu
- FreeCAD pourrait avoir une interface WebGL
- N'importe quoi d'autre auquel on a pas encore pensé

Kit Bois Appentis Murale 3M00 x 4M00 x 10°

Appentis Bois sur mesure en Kit	1470 € livraison inclus
Longeur x Longueur x Hauteur (m)	3 x 4 x 2,2 m
Angle de la pente	10 °
Débard des poutres et des solives	20 cm et 30 cm
Longeur x Longueur à l'extérieur des poteaux	2,4 x 3,4 m
Quincaillerie fournis	Pieds de poteaux réglables, vis et boulons
Matériaux, Essence, Traitement, Finition	Bois massif, Pin, Traitement Classe 3 de couleur Brun, Brut

Pour naviguer dans le modèle 3D :

- **Rotation** : clic gauche maintenu et déplacement de la souris de gauche à droite et de bas en haut.
- **Zoom** : clic molette maintenu et déplacement de la souris de bas en haut.
- **Translation** : clic droit maintenu et déplacement de la souris de gauche à droite et de bas en haut.



Pourquoi Python, au fait?

- Excellente API C, facile à intégrer a un programme écrit en C/C++
- Le jour d'aujourd'hui, n'importe quelle bibliothèque possède une extension Python
- La communauté du Libre l'adore, l'adoption est facile

Applications avec Python:

libres

- Blender: oui
- Inkscape: oui
- OpenSCAM: oui
- GRASS: oui
- Salome: oui
- OpenSCAD: oui

...

propriétaires

- Autocad: oui
- SolidWorks: non
- Catia: oui
- Rhino: oui
- SolidEdge: oui
- Revit: oui

...

Et l'utilisateur “normal”?

- La fabrication d'objets demande déjà beaucoup de bricolage
- La modélisation 3D précise devient vite très complexe (l'automatisation peut beaucoup aider)
- Les paradigmes évoluent, le code reste!



Au sérieux maintenant!

Accéder au noyau géométrique

via la propre API de FreeCAD

```
>>> import Part
```

```
>>> l = Part.makeLine((0,0,0),(2,2,0))
```

```
>>> Part.show(l)
```

```
>>> c = Part.makeBox(2,2,2)
```

```
>>> Part.show(c)
```

en manipulant OpenCasCade directement (PythonOCC)

```
>>> import OCC
>>> from OCC.BRepPrimAPI
        import BrepPrimAPI_MakeBox

>>> box = BrepPrimAPI_MakeBox(10,20,30)
>>> shape = box.Shape()
>>> part = Part._fromPythonOCC_(shape)

>>> Part.show(part)
```

A

1024

Accéder à l'interface

directement via Qt

```
>>> from PySide import QtGui
>>> win = FreeCADGui.getMainWindow()
>>> w = QtGui.QDockWidget()
>>> w.setWidget(QtGui.QCalendarWidget())
>>> win.addDockWidget( w )
```

FreeCAD

File Edit View Tools Macro Architecture Draft Windows Help

Arch

Auto

Combo View

Model Tasks

Labels & Attributes

Application

Unnamed

Property Value

View Data

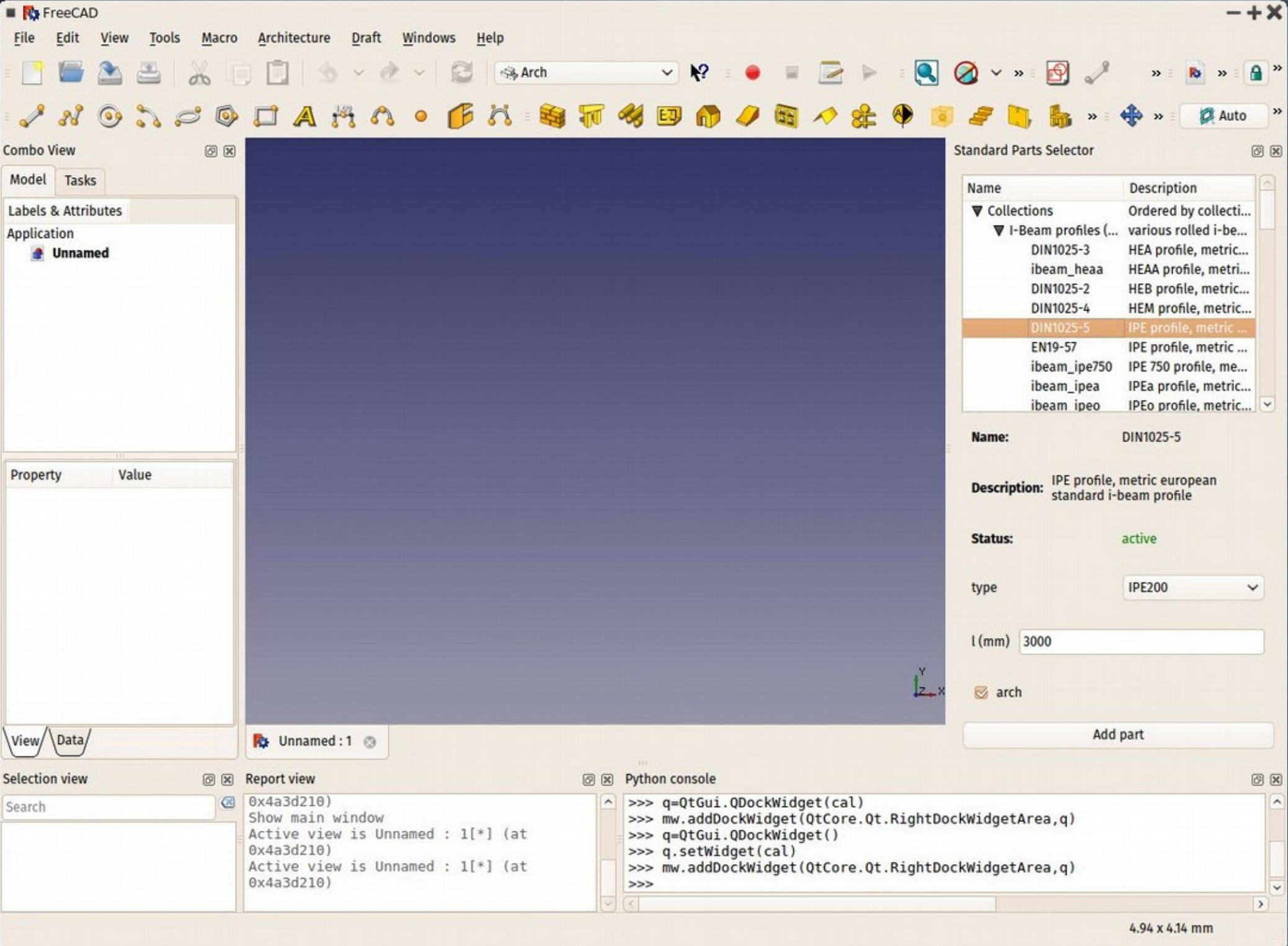
Selection view

Report view

Python console

5.28 x 4.14 mm

january 2015						
Sun	Mon	Tue	Wed	Thu	Fri	Sat
1	28	29	30	31	1	2
3	4	5	6	7	8	9
10	11	12	13	14	15	16
17	18	19	20	21	22	23
24	25	26	27	28	29	30
31	1	2	3	4	5	6
7						



0x4a3d210)
Show main window
Active view is Unnamed : 1[*] (at
0x4a3d210)
Active view is Unnamed : 1[*] (at
0x4a3d210)

```
>>> q=QtGui.QDockWidget(cal)
>>> mw.addDockWidget(QtCore.Qt.RightDockWidgetArea,q)
>>> q=QtGui.QDockWidget()
>>> q.setWidget(cal)
>>> mw.addDockWidget(QtCore.Qt.RightDockWidgetArea,q)
>>>
```

Name	Description
▼ Collections	Ordered by collecti...
▼ I-Beam profiles (...)	various rolled i-be...
DIN1025-3	HEA profile, metric...
ibeam_heaa	HEAA profile, metri...
DIN1025-2	HEB profile, metric...
DIN1025-4	HEM profile, metric...
DIN1025-5	IPE profile, metric ...
EN19-57	IPE profile, metric ...
ibeam_ipe750	IPE 750 profile, me...
ibeam_ipea	IPEa profile, metric...
ibeam_ipeo	IPEo profile, metric...

Name: DIN1025-5

Description: IPE profile, metric european
standard i-beam profile

Status: active

type IPE200

l (mm) 3000

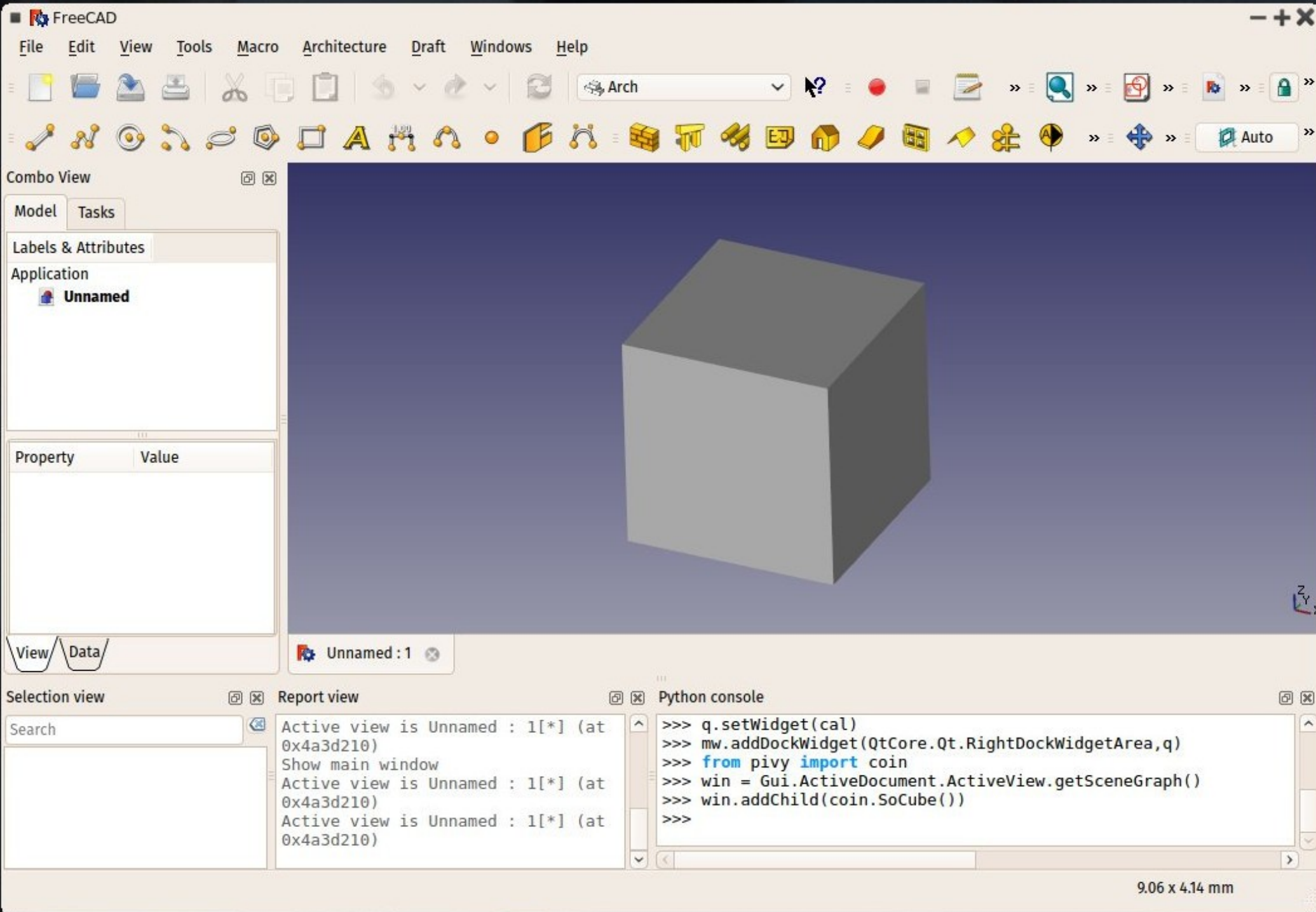
arch

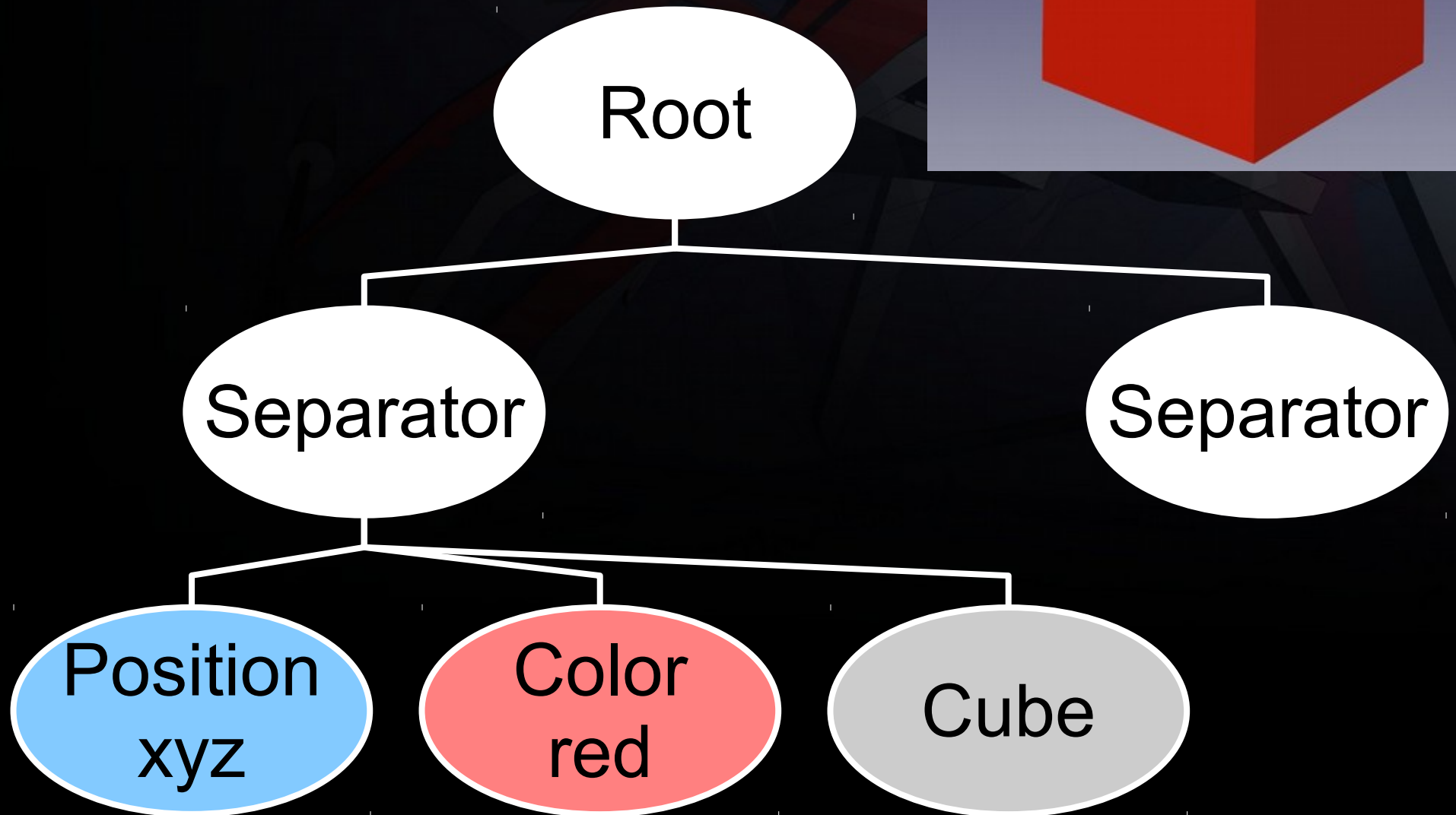
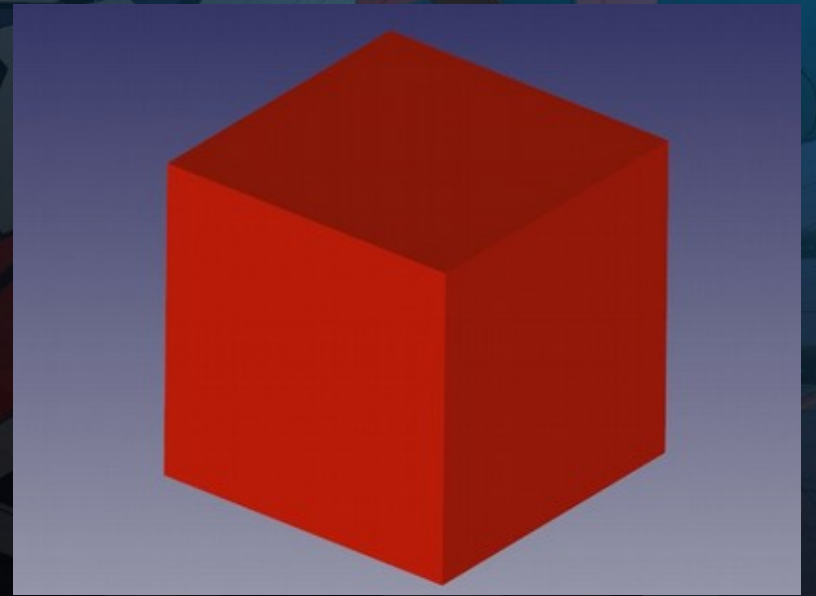
Add part

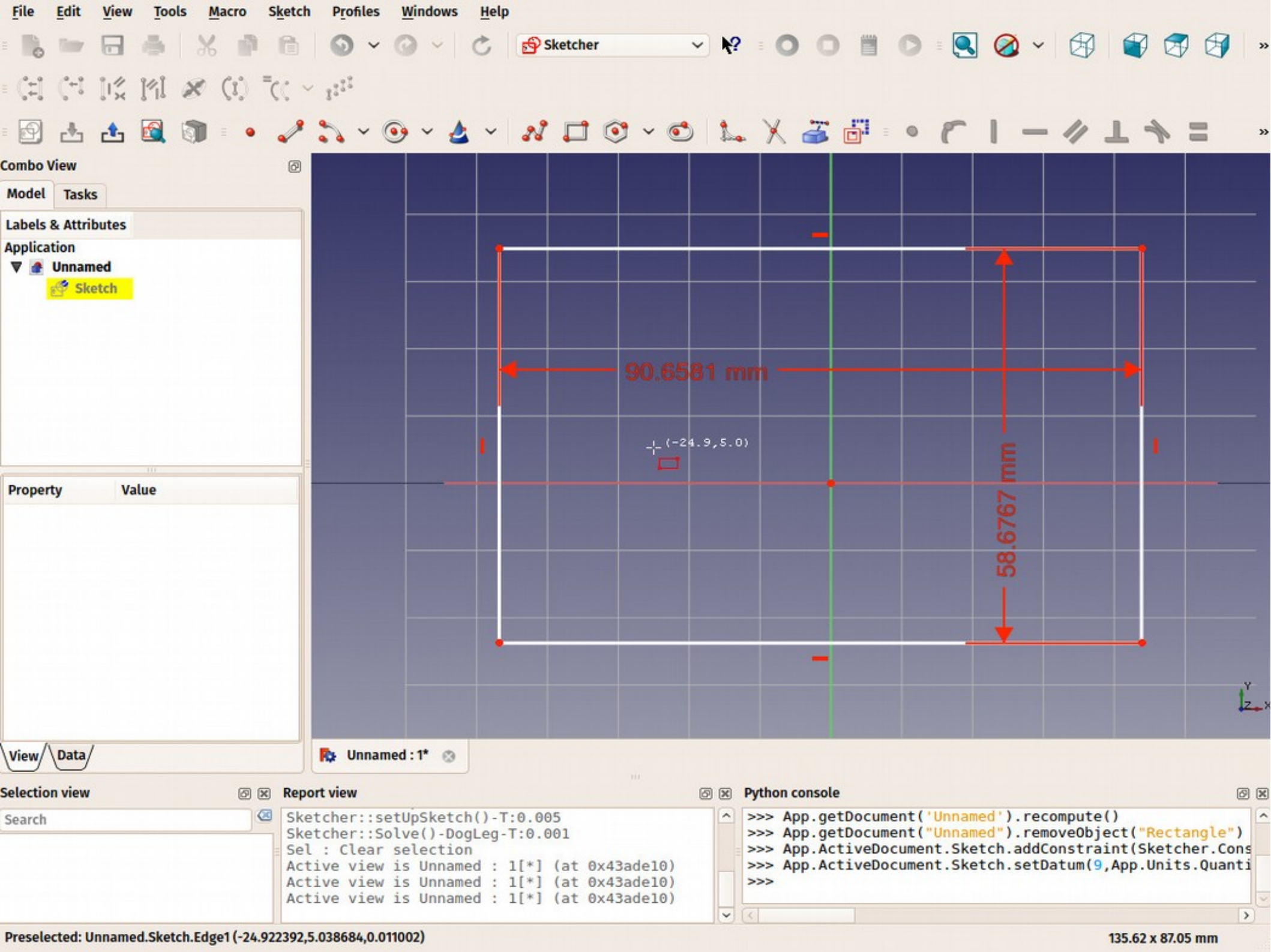
Accéder à la vue 3D

via Coin3D (pivy)

```
>>> from pivy import coin
>>> sg = FreeCADGui.ActiveDocument.  
        ActiveView.getSceneGraph()
>>> cube = coin.SoCube()
>>> sg.addChild( cube )
```

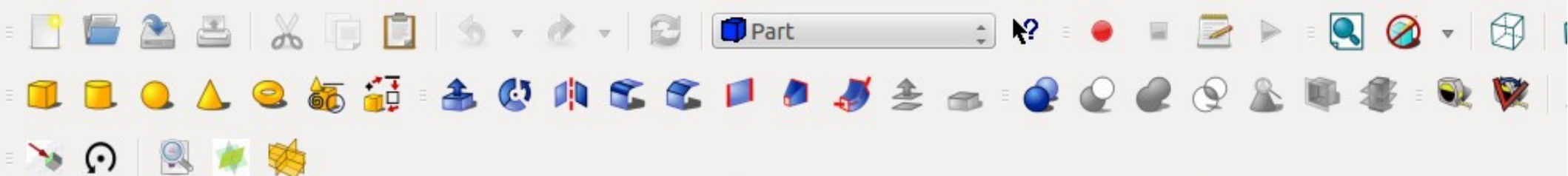






Création d'un outil:

- Enregistrement de quelques actions (macro)
- Modification du code, si nécessaire
- Création d'éléments d'interface (dialogues, boutons, etc)
- Création d'interaction dans la vue 3D



Vue combinée

Modèle Tâches Work Features

Origin

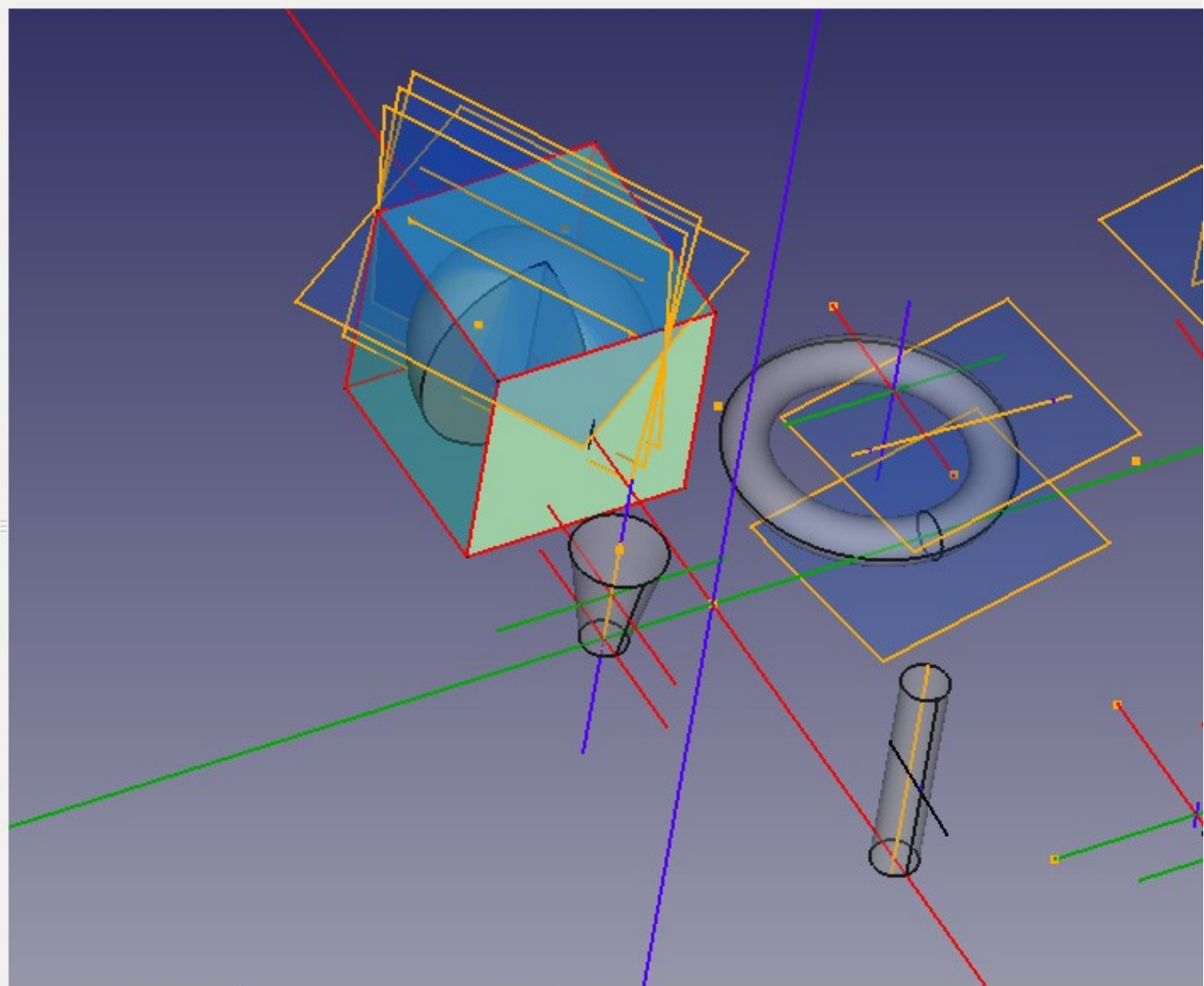
Point

Axis

Plane

Boxes

- Object(s) Center
- Line(s) Center
- Line(s) Extrema
- Circle(s) Center
- Point along Line
- Face(s) Center
- Point=(Line,Face)
- Point=(Line,Point)
- Point=(Point,Face)
- Object(s) Base Point



Apprentissage communautaire

- Ma première macro!
- Publication sur le forum
- Les autres utilisateurs testent, critiquent, veulent plus de fonctions
- La macro devient un outil complexe

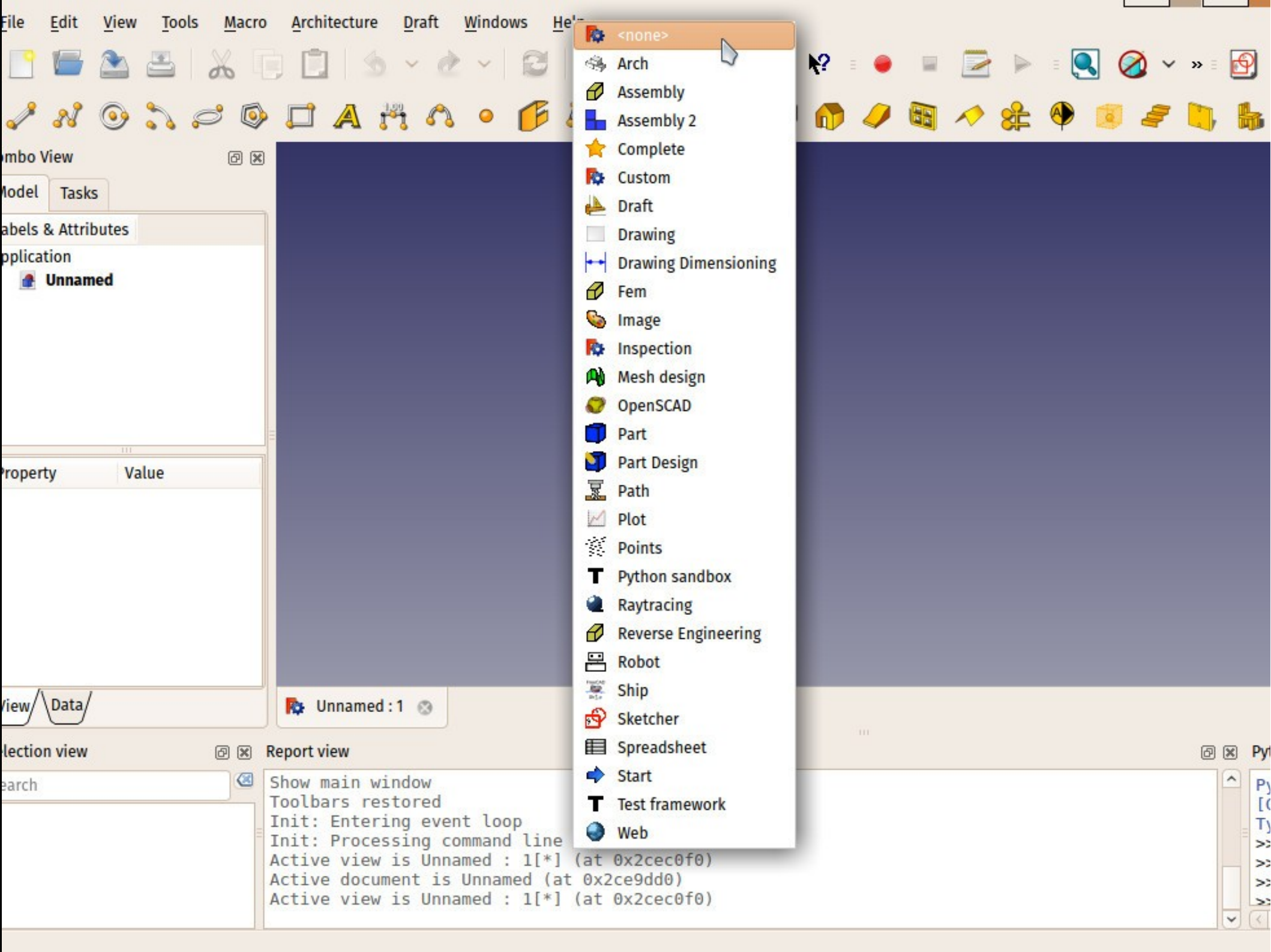
Board index

Users

 Help on using FreeCAD Topics: 4495, Posts: 35761	Re: making polar patterned te... by bejant → Mon Nov 09, 2015 12:34 pm
 Python scripting and macros Topics: 822, Posts: 4852	Re: Get Arcs from BSpline by deavik → Mon Nov 09, 2015 3:14 am
 Install / Compile Topics: 887, Posts: 7582	Re: CMake not finding boost_p... by sgrogan → Mon Nov 09, 2015 1:00 pm
 Open discussion Topics: 1311, Posts: 13362	Re: Plugin Loader by microelly2 → Mon Nov 09, 2015 12:22 pm
 Feature Announcements Topics: 137, Posts: 1856	Re: FEM: Frequency analysis by PrzemoF → Wed Nov 04, 2015 12:59 pm
 Users Showcase Topics: 258, Posts: 1491	Re: buggy by godblessfq → Fri Nov 06, 2015 2:44 pm
 Job offers Topics: 4, Posts: 28	Re: Paid FreeCAD work. by sgrogan → Thu Oct 22, 2015 1:46 pm

Development

 Developers corner Topics: 1444, Posts: 17677	Re: eigen2 support disabled by abdullah → Mon Nov 09, 2015 11:37 am
 Pull Requests Topics: 332, Posts: 1994	Re: Fixed for issue #2302 by wmayer → Mon Nov 09, 2015 7:50 am
 Part Design module development Topics: 67, Posts: 756	Re: Starting parts by pinguindark → Thu Oct 15, 2015 2:16 am
 Assembly Topics: 93, Posts: 1335	Re: Part Design Roadmap by DeepS01C → Sun Nov 08, 2015 4:52 am



File Edit View Tools Macro Architecture Draft Windows Help



Combo View

Model Tasks

Labels & Attributes

Application

Unnamed

Property Value

View Data

Selection view

Search

Report view

Show main window
Toolbars restored
Init: Entering event loop
Init: Processing command line
Active view is Unnamed : 1[*] (at 0x2cec0f0)
Active document is Unnamed (at 0x2ce9dd0)
Active view is Unnamed : 1[*] (at 0x2cec0f0)

- <none>
- Arch
- Assembly
- Assembly 2
- Complete
- Custom
- Draft
- Drawing
- Drawing Dimensioning
- Fem
- Image
- Inspection
- Mesh design
- OpenSCAD
- Part
- Part Design
- Path
- Plot
- Points
- Python sandbox
- Raytracing
- Reverse Engineering
- Robot
- Ship
- Sketcher
- Spreadsheet
- Start
- Test framework
- Web



Python console

Python console output area with a scrollbar.

Le développement à la FreeCAD

- Pas de plan. Pas d'agenda. Pas d'organisation. Pas de loi.
- Nouvelles versions quand on le sent (“It's done when it's done”)
- Pas de “relation développeur – utilisateur” traditionnelle

Qui développe?

- Tout le monde (vous faites une macro, bravo, vous êtes développeur de FreeCAD)
- Les utilisateurs expérimentés programment, testent ou révisent
- 3 personnes (développeurs originaux) contrôlent le code officiel

Pas de pyramide

- Toutes les décisions sont prises sur le forum
- La direction est maintenue par consensus
- La nature ouverte de FreeCAD permet à peu près n'importe quelle innovation

Exemple: Le module Path

- Module hybride (C++ / Python)
- Base simple et robuste en C++
- Extensible en Python
- Permet l'intervention de programmeurs de tous niveaux et expériences

FreeCAD

FileEditViewToolsMacroPathWindowsHelp

Path

Combo View

ModelTasks

Labels & Attributes

Application

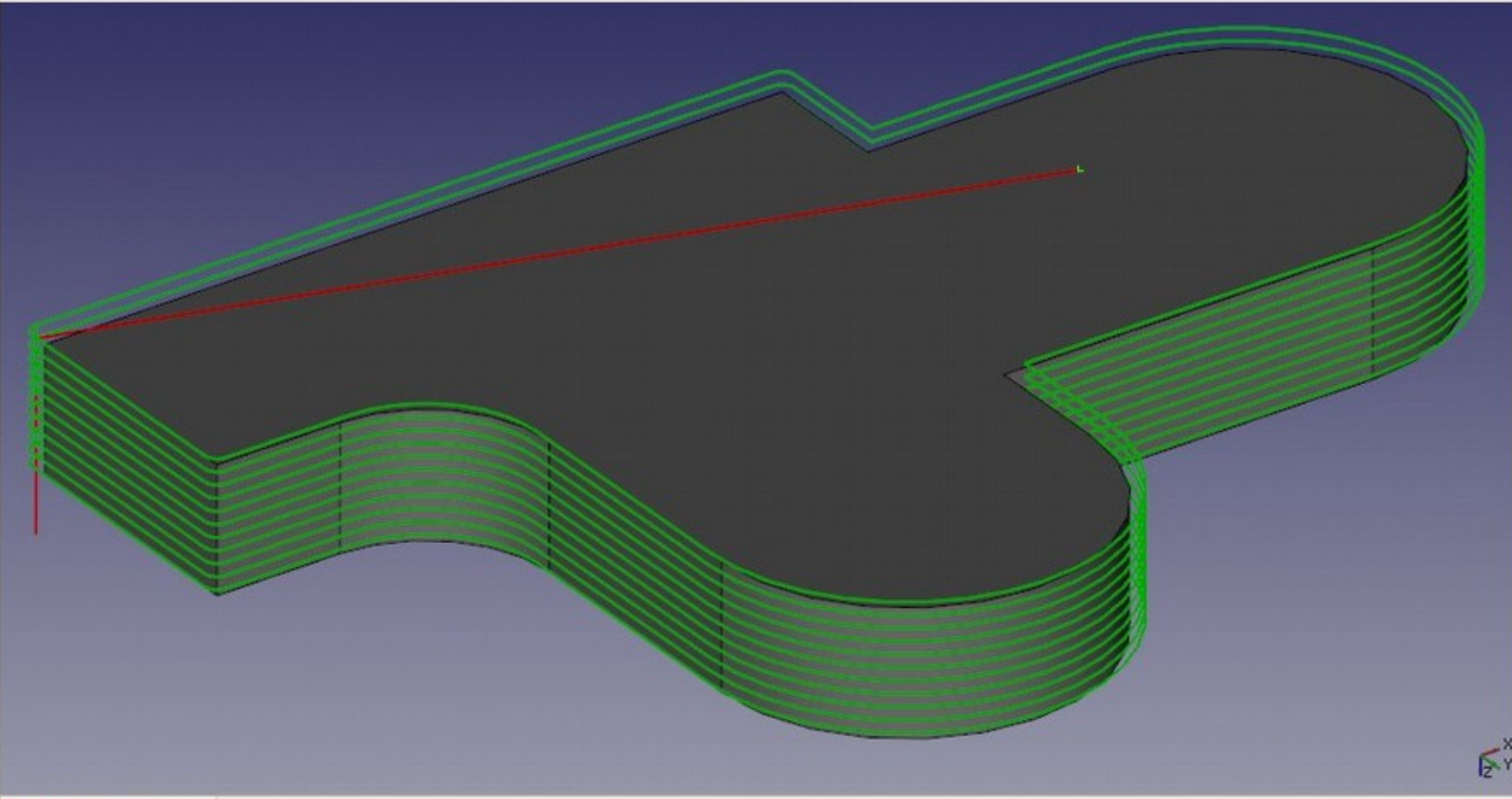
▼ Unnamed

▶ Pad

Profile

Property	Value
Depth Parameters	
Clearance Height	15.00
Final Depth	-1.00
Retract Height	0.00
Start Depth	9.00
Step Down	1.00
Use Start Depth	false
Path	
Path Closed	true
Profile Parameters	
Direction	CW
▶ End Point	[0.00 0.00]
Extend At End	0.00
Extend At Start	0.00
Horiz Feed	0.00

ViewData



Selection view

Search

Unnamed.Profile (Profile)

Report view

Loading PathCopy... done

Loading Path workbench... done

Path workbench activated

There are edges selected

Sel : Clear selection

Sel : Clear selection

Active view is Unnamed : 1[*] (at 0x3304db0)

Python console

>>> App.ActiveDocument.recompute()

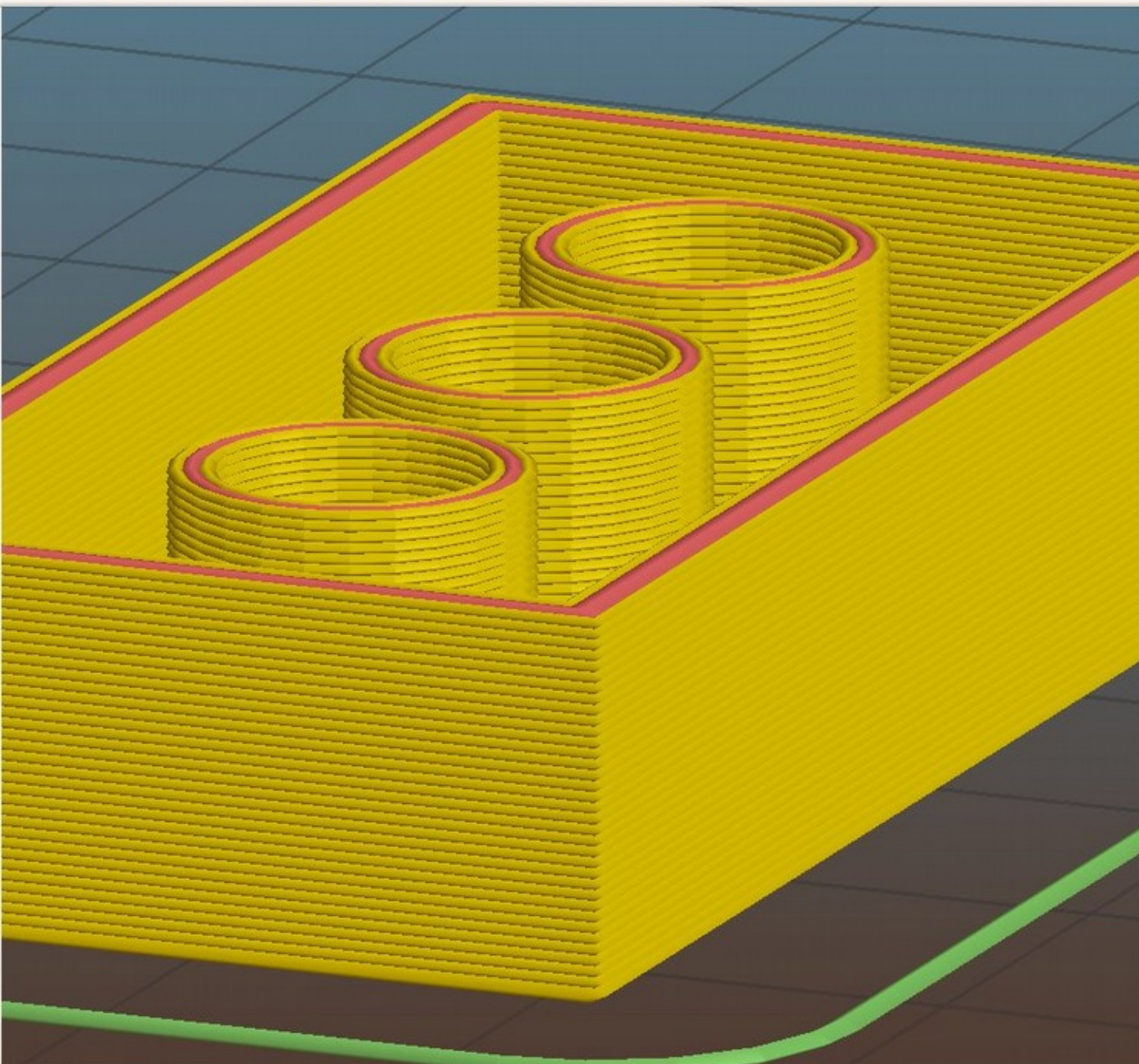
>>> Gui.activeDocument().resetEdit()

>>> Gui.activateWorkbench("PathWorkbench")

>>> import PathScripts.PathProfile

>>> FreeCADGui.getDocument("Unnamed").getObject("Profile")

>>>

[Export STL...](#) [Export G-code...](#)

Name	Copies	Scale
lego.stl	1	100%

Info

Size: 47.70 x 23.70 x 17.10 Volume: 7258.83

Facets: 1428 (1 shells) Materials: 1

Manifold: Yes

GCODE



Machines CNC

G0 X0 Y0

G1 X0 Y10

G1 X5 Y5

Imprimantes 3D

G1 X0 Y0

G1 X0 Y10 E2.0

G1 X5 Y5 E2.0

Dans FreeCAD

```
>>> import Path
>>> myPath = Path.Path()
>>> m1 = Path.Command("G0 X0 Y0")
>>> m2 = Path.Command("G1 X0 Y10")
>>> myPath.addCommands([m1,m2])
>>> print myPath.toGCode()
```

G0X0Y0

G1X0Y10

SHAPEOKO 2

A factory for your desktop.

SEE PRICING

This is Ryan. He needed a powerful machine he could take on the road.

Explore Ryan's story

Please note: Kits ordered today are scheduled to ship within 12 business days.

MECHANICAL KIT

Not for the faint of heart, this kit includes only the skeleton of the machine.

This is **recommended if you are an expert and your goal is to build a custom machine**. You'll need to supply your own electronics, pulleys, belts, and M3 hardware.

[Kit contents](#)

\$300

ADD TO CART

FULL KIT

The Full Kit comes with everything you need to get started with 3D carving, including tools and a milling bit.

After you assemble the Full Kit you will have a working bare-bones machine that can carve parts. This is **recommended for hobbyists who see building and upgrading the machine as part of the fun**.

[Which voltage kit should I buy?](#)

[Kit contents](#)

- 110V  \$650
- 220V  \$695

ADD TO CART

THE WORKS

This package includes the Full Kit + a number of popular upgrades including a Quiet Cut spindle, a waste board with clamps, and an electronics enclosure.

This setup is recommended if you want to order these popular upgrades and accessories for the machine all at one time.

[Which voltage kit should I buy?](#)

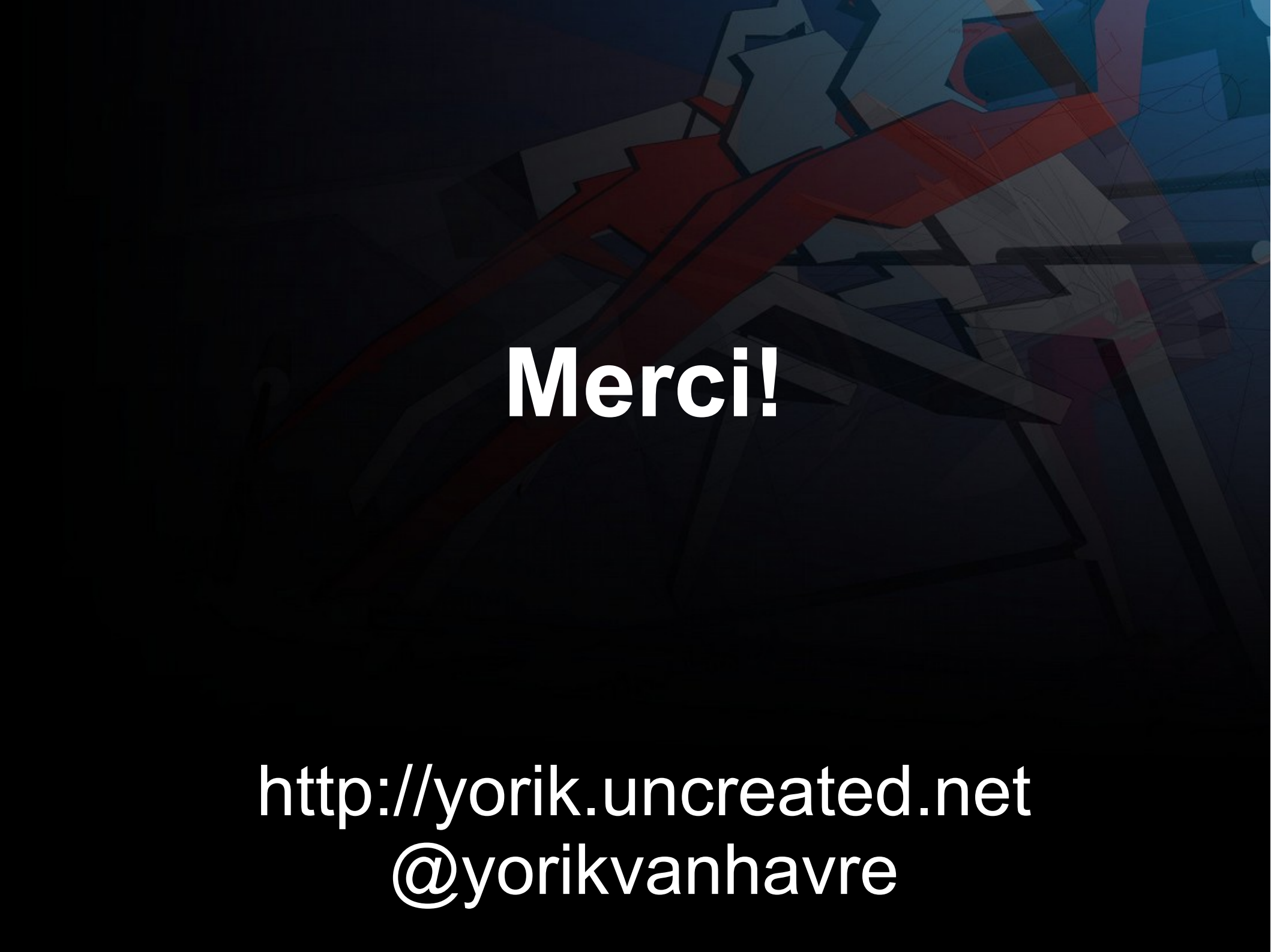
- 110V  \$1,022
- 220V  \$1,061

ADD TO CART

“We're moving into this future where the factory is everywhere, and the design team is everyone. That is an industrial revolution”

“Nous entrons dans ce futur où la fabrique est partout, et l'équipe de design est tout le monde. C'est une nouvelle révolution industrielle”

Alastair Parvin (TED talk)
<http://www.wikihouse.cc>



Merci!

<http://yorik.uncreated.net>
@yorikvanhavre