

OpenSCAD

Modélisation d'objet physique par le code

CAD(en)

Computer-aided design

CAO(fr)

Conception assistée par ordinateur

Différents outils

pour modéliser des objets

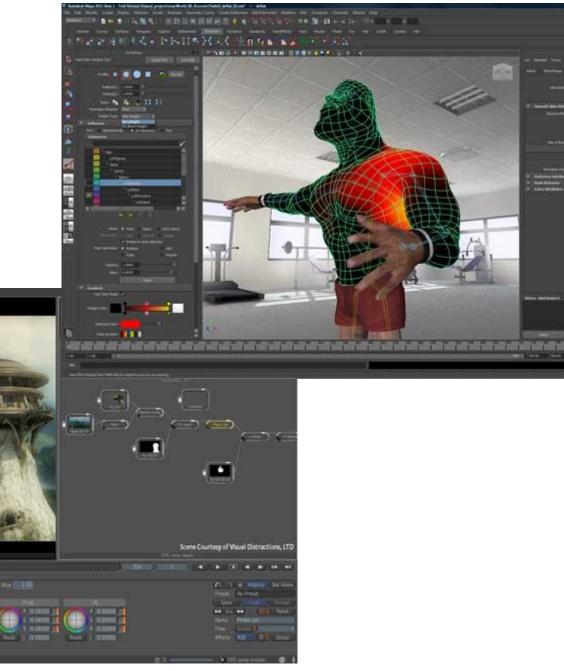
Plutôt artistiques

Blender

Maya

3DS Max

. . .



Plutôt techniques, classiques

FREECAD

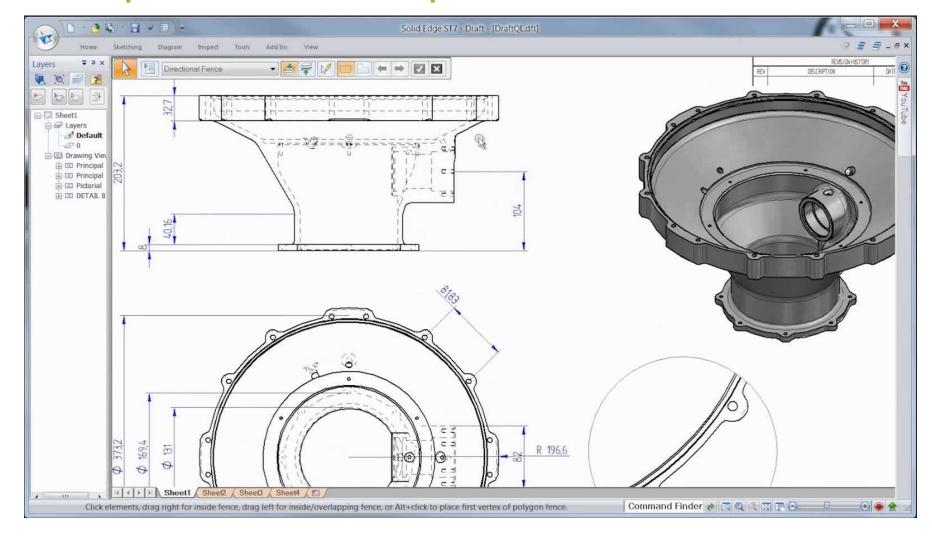
AutoCAD

Catia

SolidEdge

SolidWorks

...



Plutôt techniques, par code

OpenSCAD OpenJSCAD

...

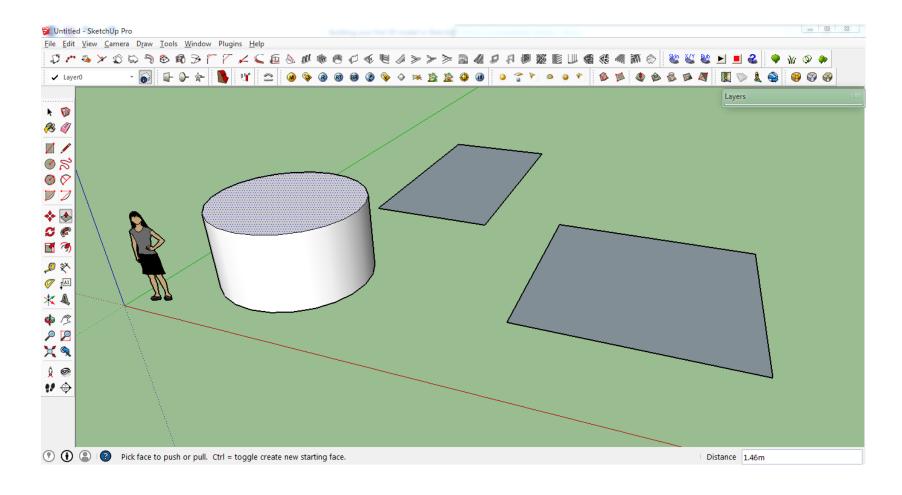
```
9 9
                                                                               example009.scad
                     n a 財 種
podywidth = dxf_dim(file = "example009.dxf", name = "bodywidth");
 2 fanwidth = dxf_dim(file = "example009.dxf", name = "fanwidth");
 3 platewidth = dxf_dim(file = "example009.dxf", name = "platewidth");
 4 fan_side_center = dxf_cross(file = "example009.dxf", layer = "fan_side_center");
 5 fanrot = dxf_dim(file = "example009.dxf", name = "fanrot");
 7 %-linear_extrude(height = bodywidth, center = true, convexity = 10)
     import(file = "example009.dxf", layer = "body");
10 % for (z = [+(bodywidth/2 + platewidth/2),
11 [- · · · - (bodywidth/2 · + · platewidth/2)]) · {
      translate([0, 0, z])
     linear_extrude(height = platewidth, center = true, convexity = 10)
    ····import(file - "example009.dxf", layer - "plate");
15
16 L
17 = intersection() -{
      ·linear_extrude(height = fanwidth, center = true, convexity = 10, twist = -fanrot)
19
     ···import(file = · "example009.dxf", ·layer · = · "fan_top");
20
21
     ·//·NB! ·We ·have · to ·use · the ·deprecated ·module ·here · since · the · "fan_side"
22
     ·//·layer·contains·an·open·polyline, which is not yet supported
23
     ·//·by·the·import()·module.
24
      rotate_extrude(file -- "example009.dxf", layer -- "fan_side",
     ·····origin = fan_side_center, convexity = 10);
25
26
Viewport: translate = [ 0.61 -1.31 -2.07 ], rotate = [ 55.00 0.00 25.00 ], distance = 142.23
                                                                                                                                                            OpenSCAD 2015.03
```

Divers

SketchUP

Rhino3D

...



Standard de modélisation en impression 3D

Modélisation (avec OpenSCAD, 3DS Max, Maya, Catia, ...)

```
⇒ 🖹 ____. ? ? ?
```

Export

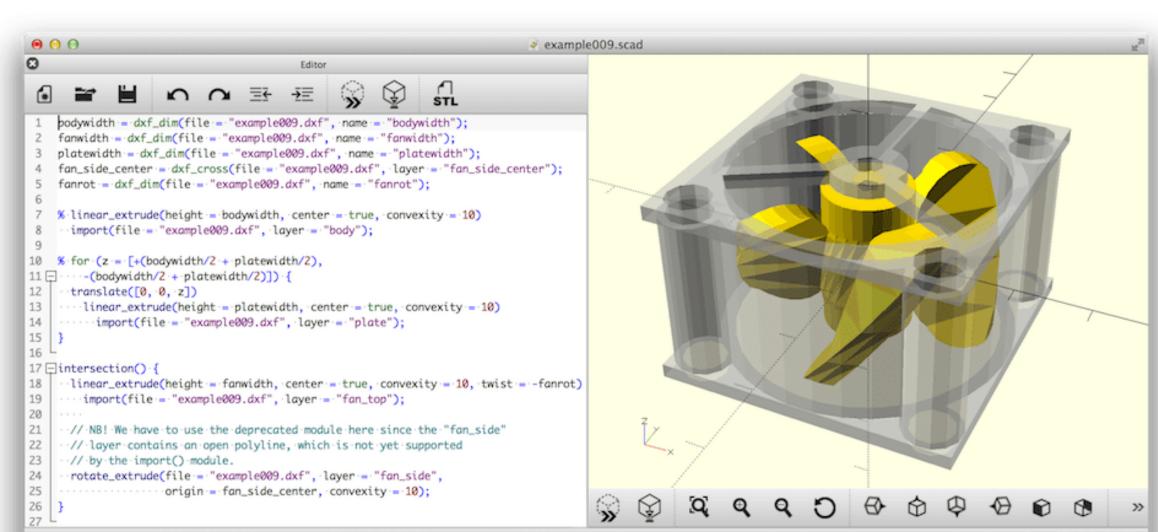
Slicing (avec Cura, Slic3r, simplify3D, ...)



Impression

OpenSCAD

Logiciel libre pour pour créer des objets en 3D, sous Linux/UNIX, Windows et Mac OS.

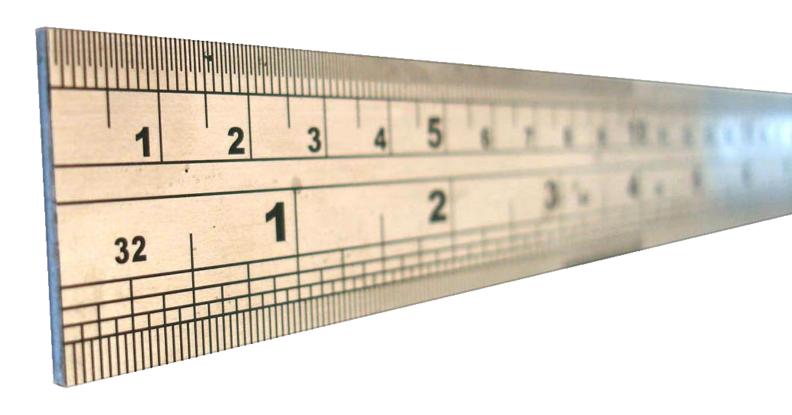


Viewport: translate = [0.61 -1.31 -2.07], rotate = [55.00 0.00 25.00], distance = 142.23

OpenSCAD 2015.03

Pourquoi du code?

Précis



Pourquoi du code?

Facile à modifier, individuellement et collectivement

Gestion de source

Fusion : prendre les améliorations d'untel + celles d'untel



Pourquoi du code?

Facile à structurer et à réutiliser

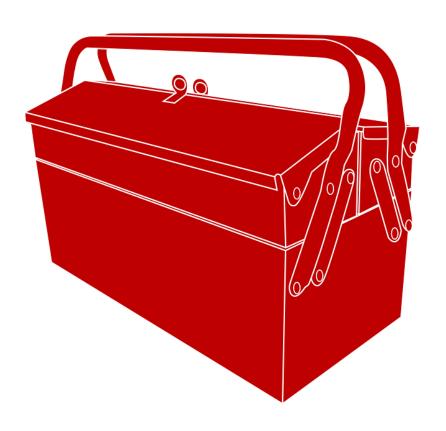
Librairie de fonctions réutilisables

Encoches, trous et découpes pour boulon et vis M2, M3, M4, ... Découpe de puzzle

Paramétrable

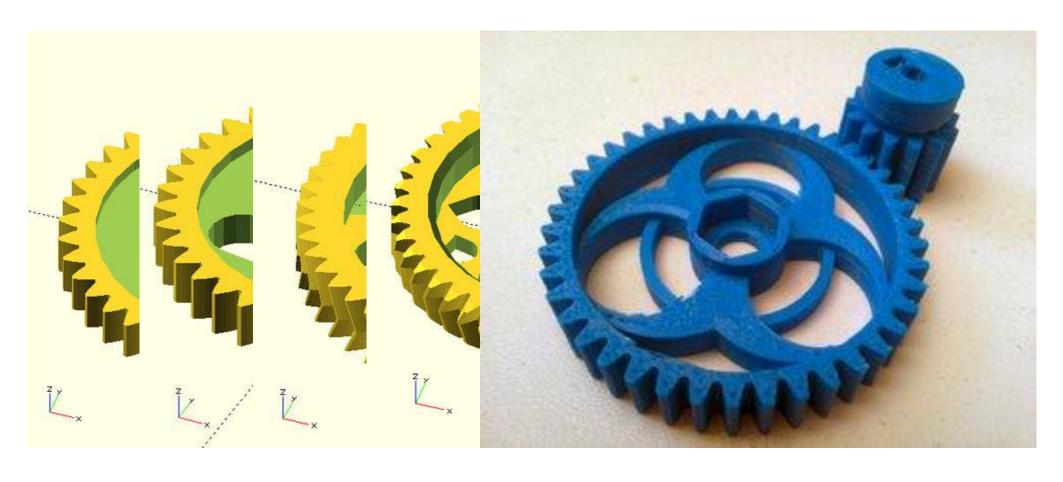
Roue crantée : combien de crans ?

Modèle d'immeuble : combien d'étages ?



Un outil de création collaboratif

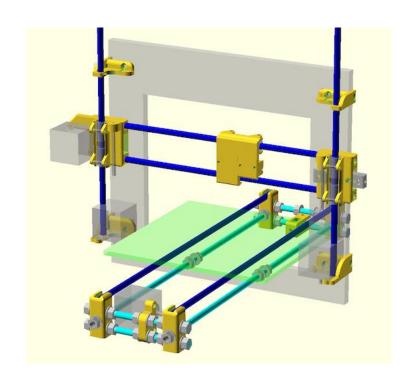
Améliorations fonctionnelles ou cosmétiques



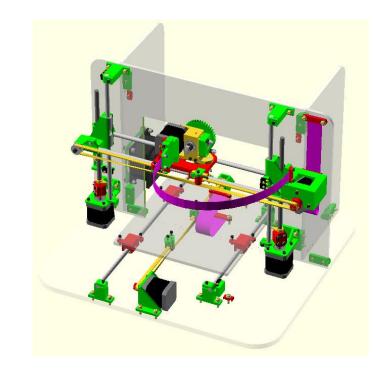
Un standard dans le monde de l'impression 3D



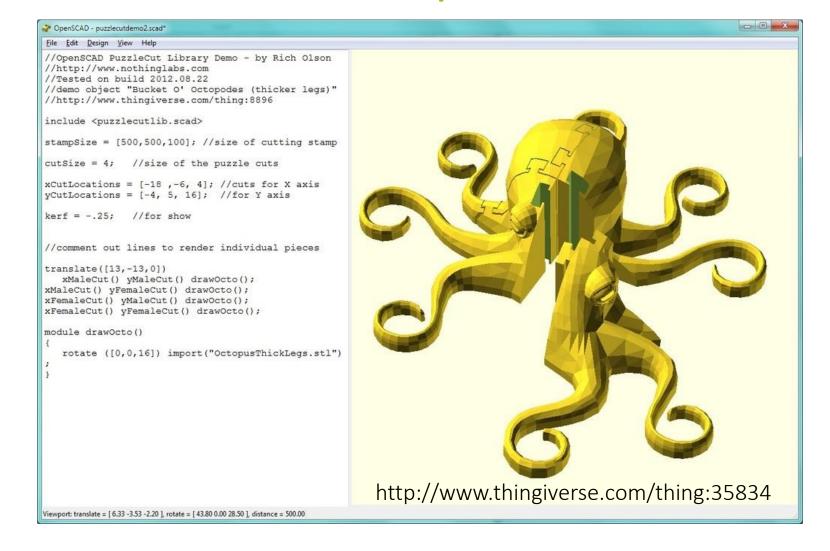
PRUSA 13



MENDEL 90

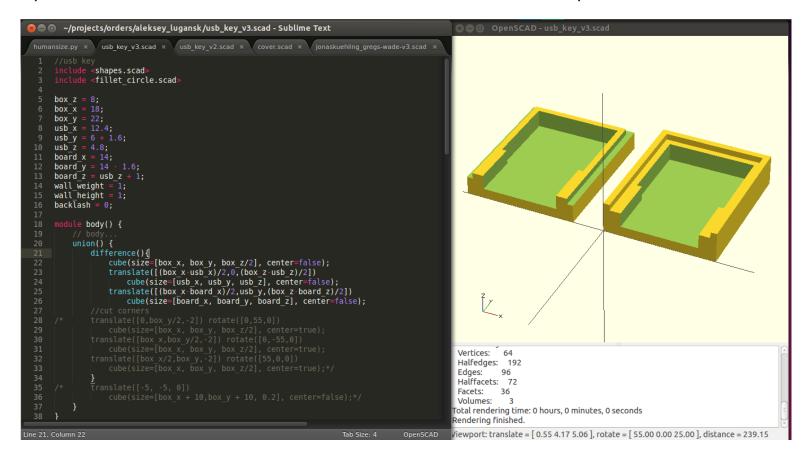


Combinaison STL + OpenSCAD



Astuces

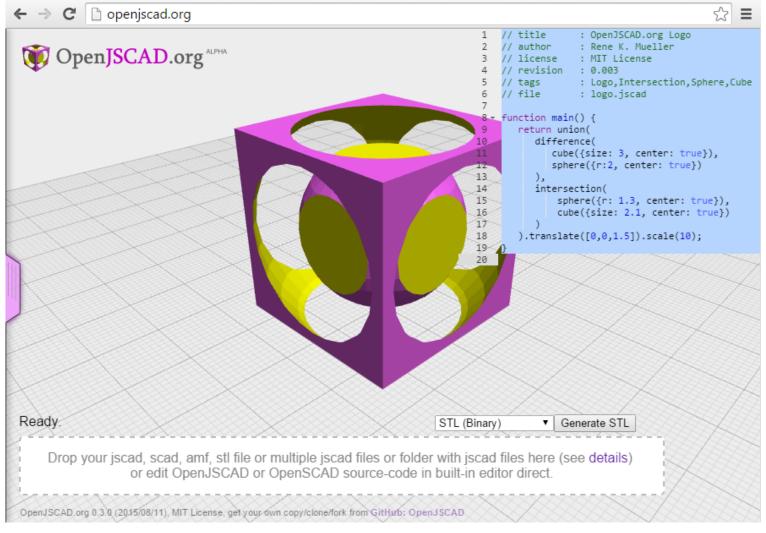
Mode viewer pur + édition dans votre éditeur favori tel que sublime Text



DSL OpenSCAD

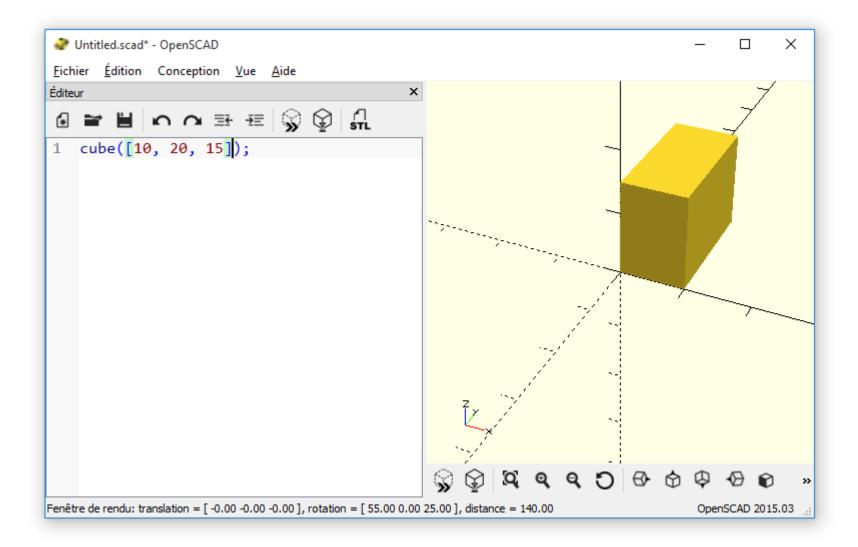
```
Clojure
scad-clj
Python
SolidPython
```

OpenJSCAD (très similaire mais différent)

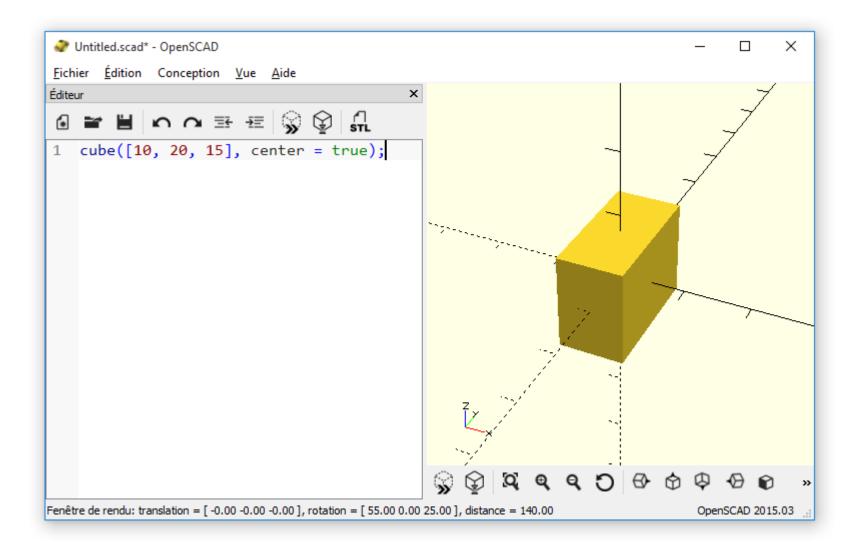


Premier pas

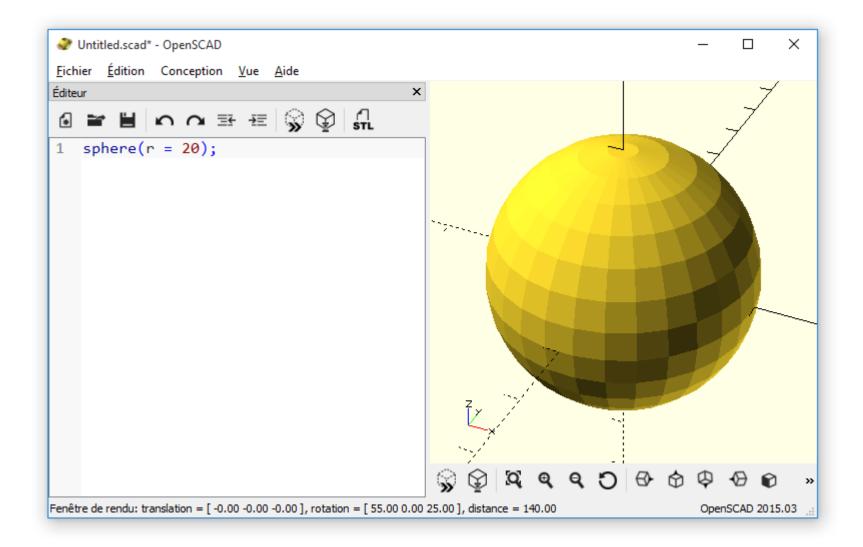
http://blog.cubehero.com/2013/11/19/know-only-10-things-to-be-dangerous-in-openscad



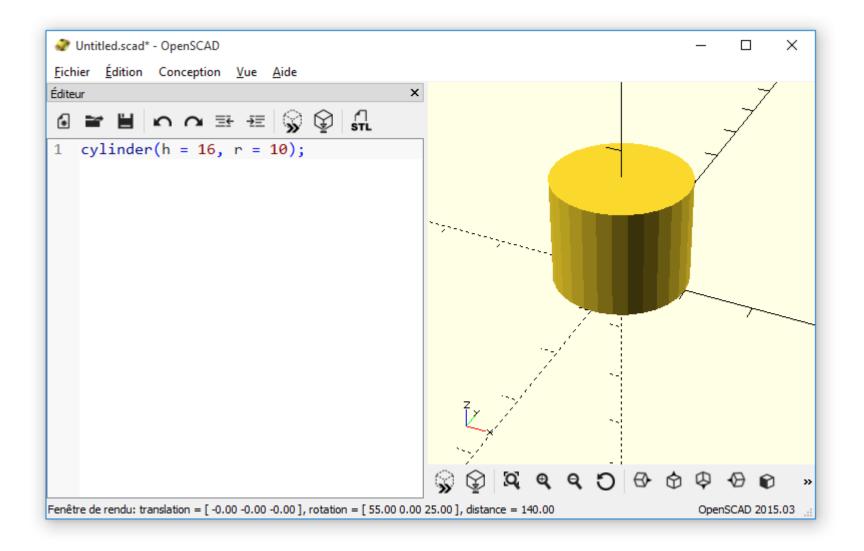
Cube



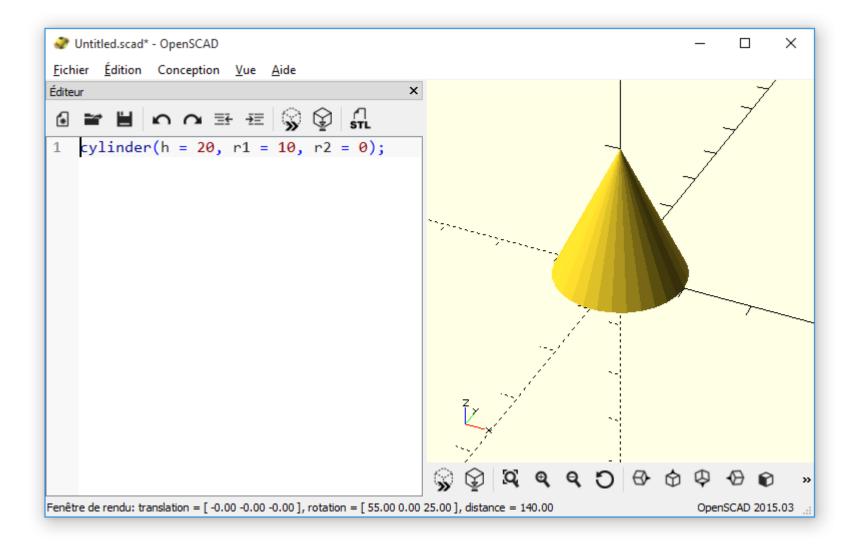
Cube centré



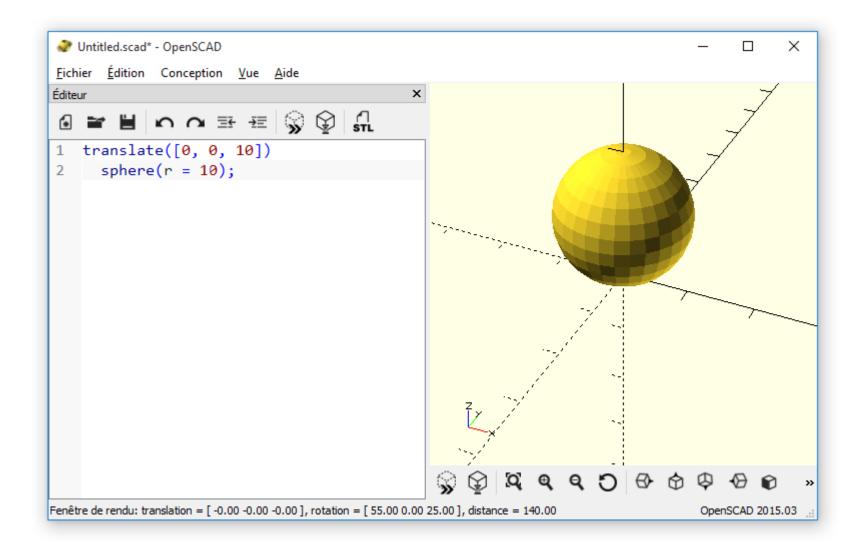
Sphère



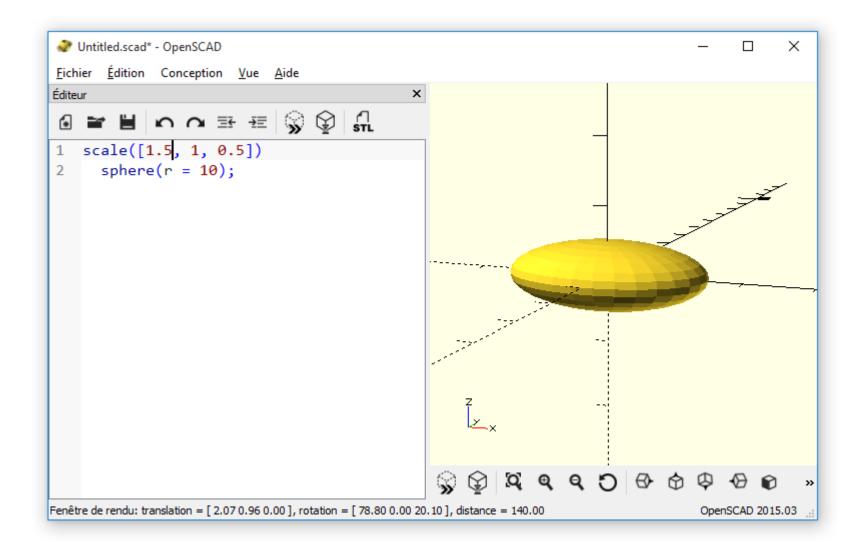
Cylindre



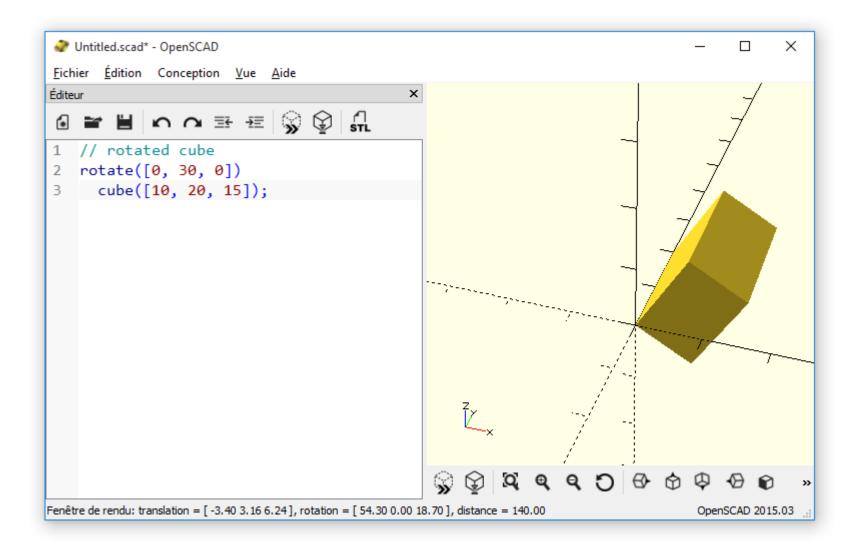
Cône



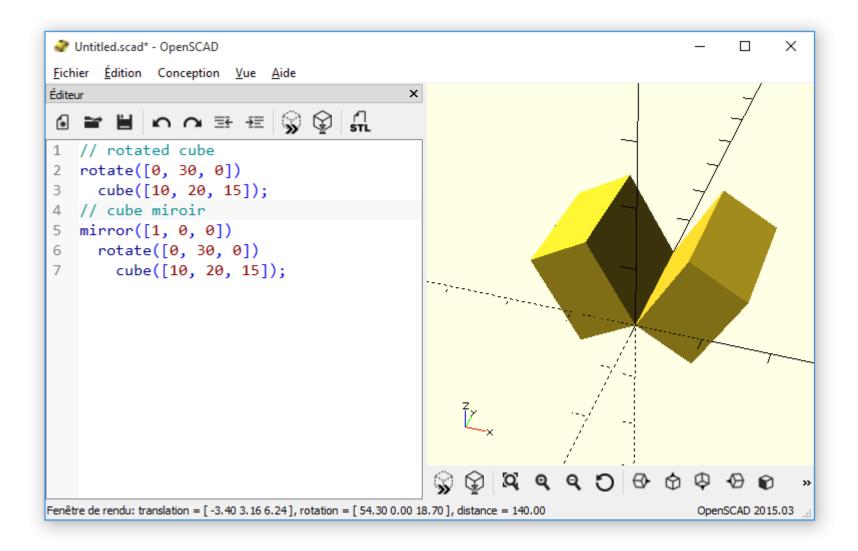
Translation



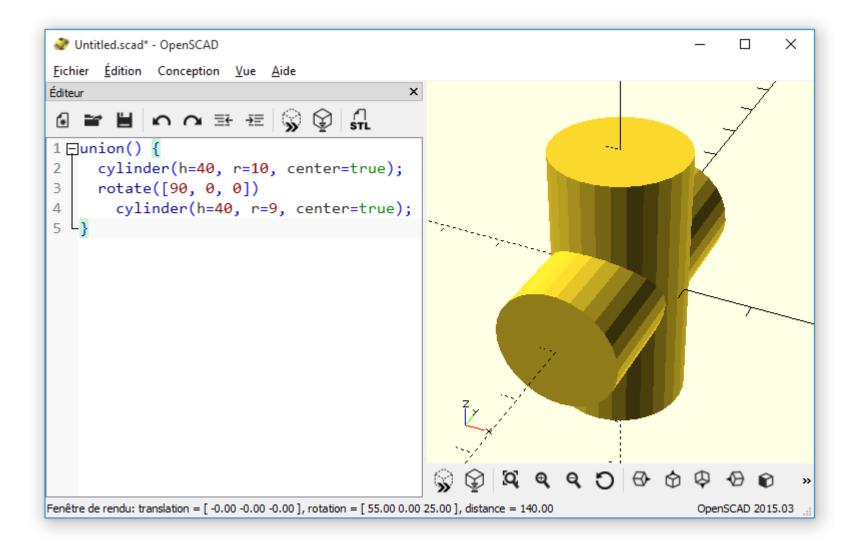
Mise à l'échelle



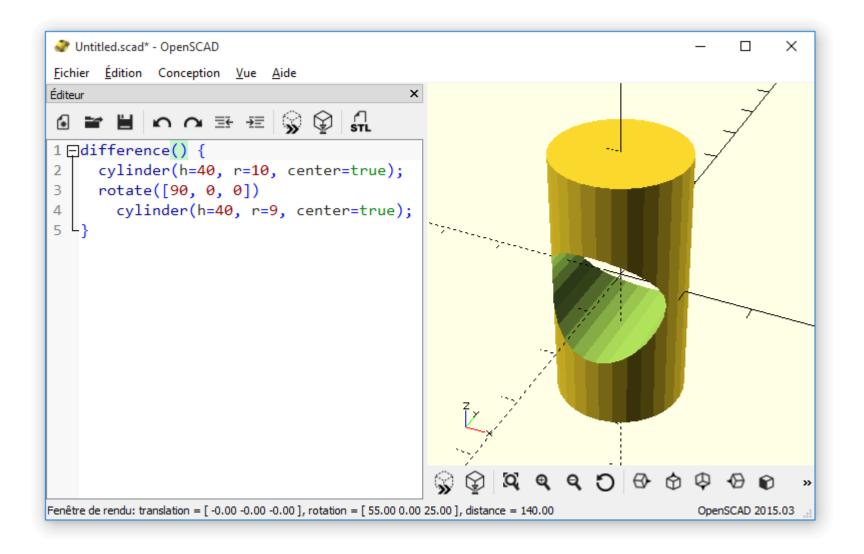
Rotation



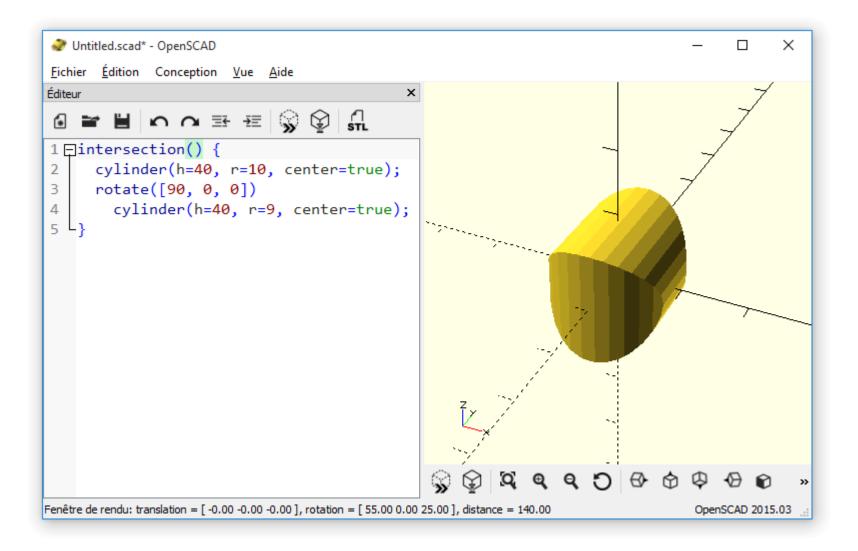
Miroir



Union



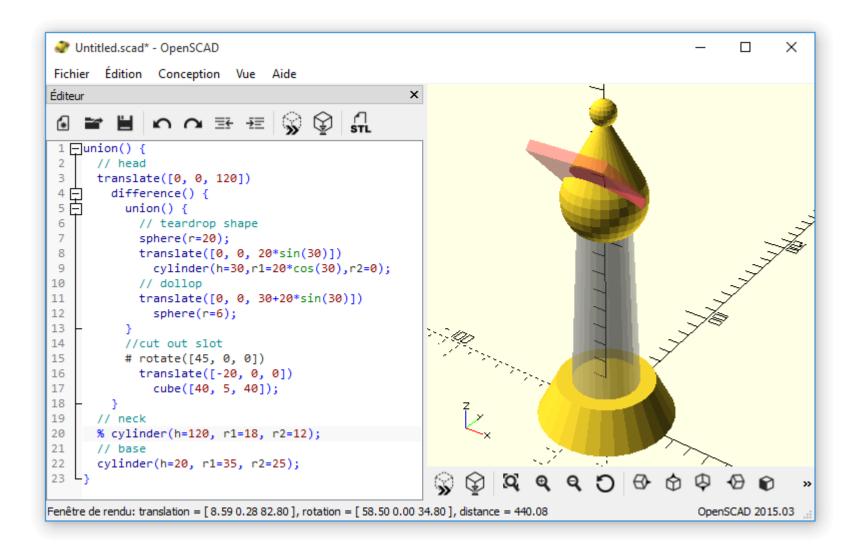
Différence



Intersection

Untitled.scad* - OpenSCAD Édition Conception Vue Aide Éditeur **₽ ॐ** 🕸 1 [union() { // head translate([0, 0, 120]) 4 白 5 白 difference() { union() { // teardrop shape 6 sphere(r=20); 8 translate([0, 0, 20*sin(30)]) cylinder(h=30,r1=20*cos(30),r2=0); 10 // dollop 11 translate([0, 0, 30+20*sin(30)]) 12 sphere(r=6); 13 14 //cut out slot rotate([45, 0, 0]) 16 translate([-20, 0, 0]) 17 cube([40, 5, 40]); 18 19 // neck 20 cylinder(h=120, r1=18, r2=12); 21 // base cylinder(h=20, r1=35, r2=25); 23 L3 Fenêtre de rendu: translation = [8.59 0.28 82.80], rotation = [58.50 0.00 34.80], distance = 440.08 OpenSCAD 2015.03

Exemple plus complexe



Visualisation d'éléments

Conclusion

The end

Plus d'informations

http://www.openscad.org

http://www.openscad.org/cheatsheet

http://blog.cubehero.com/2013/11/19/know-only-10-things-to-be-dangerous-in-openscad