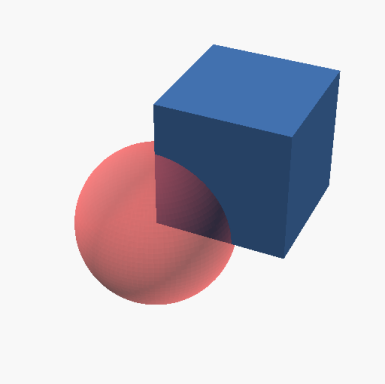

OpenScad examples

Table of Contents

1. Document information

Links to Document

Document online

Document Source

Document PDF

test repo for building openscad files into different outputs
This is still work in progress but can already build a png and stl of each scad file in the opescad directory.
See the online or pdf versions for the images as te readme is realy only the source and right now is not WYSIWYG!

2. Objects

+=== Object - Test

Just a simple example and also used as the logo for this repo.

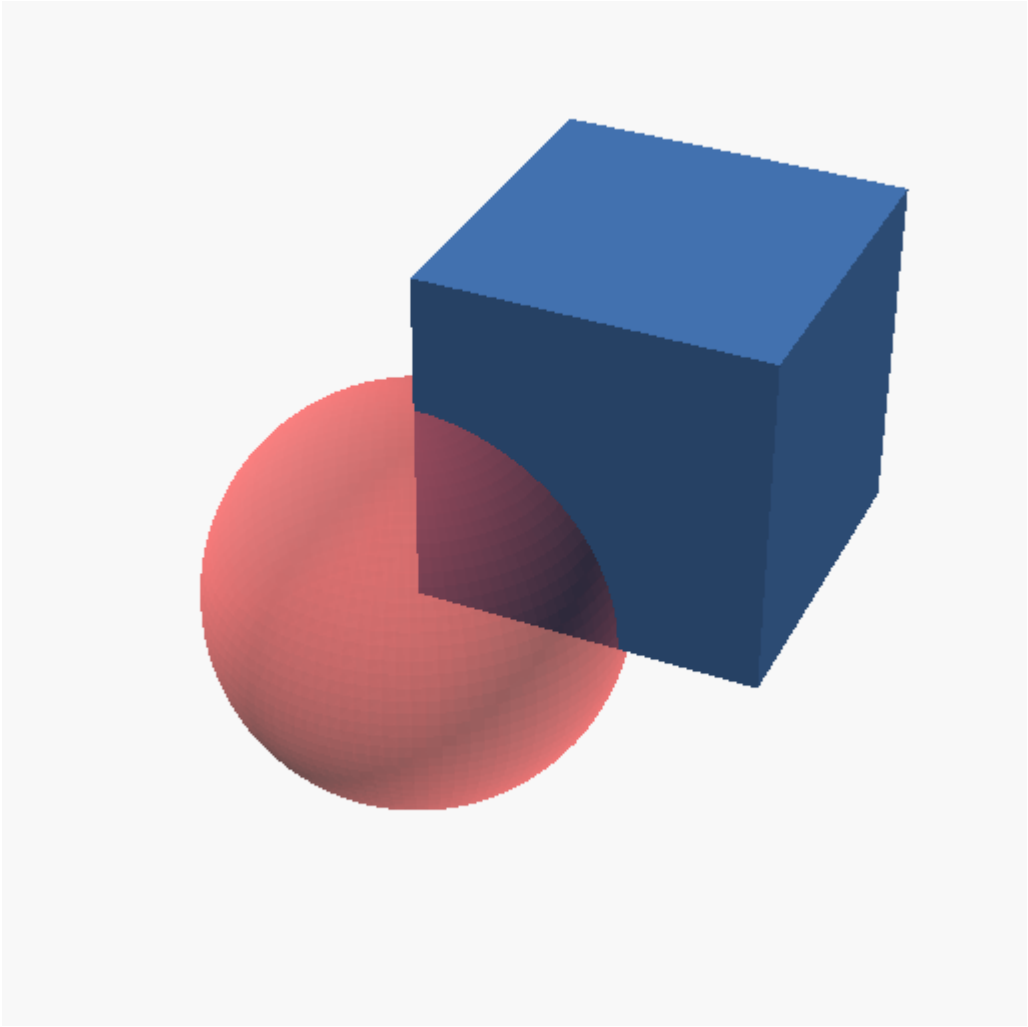

Figure 1. image

Listing 1. Openscad source

```
//just a simple test drawing
$fn=100;
cube([10,10,10]);
#sphere(d=12);
```

+=== Object - KitchenDoorHoleStopper

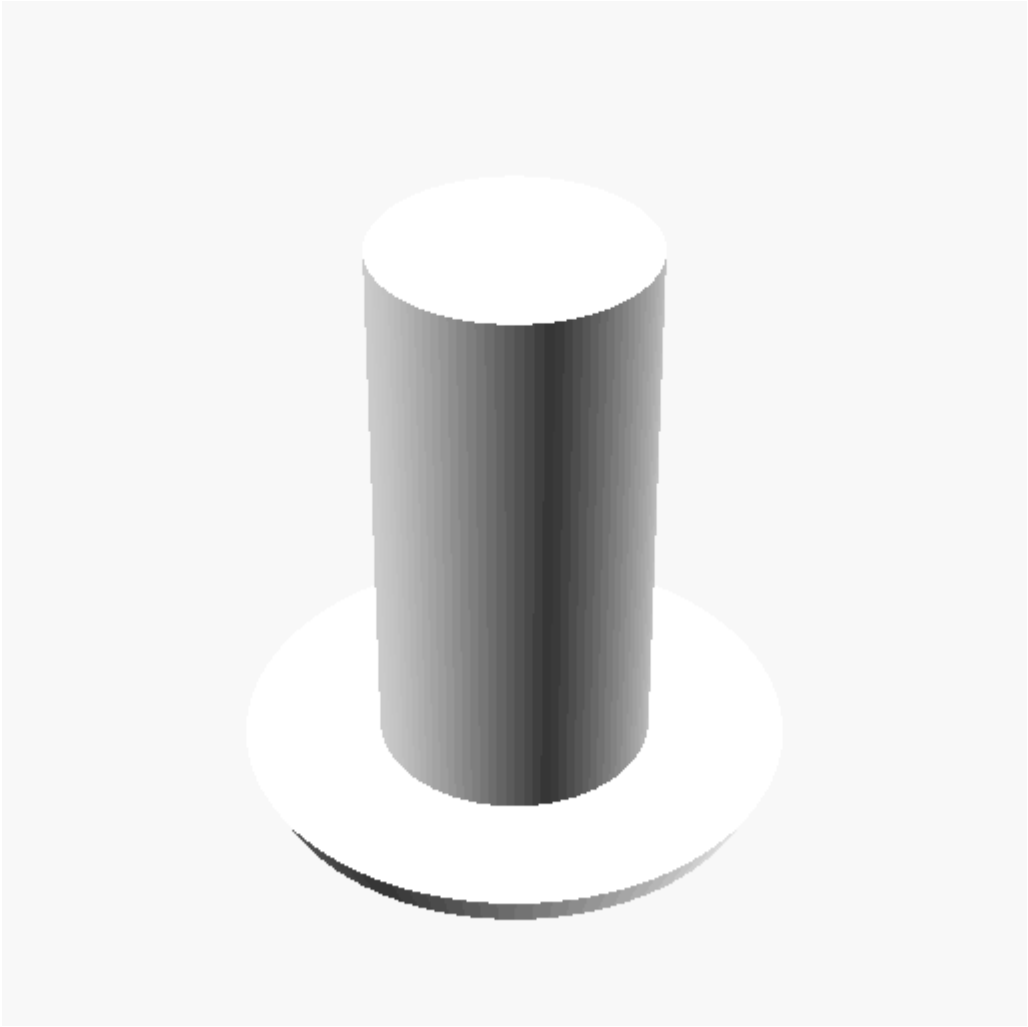

Figure 2. image

Listing 2. Openscad source

```
//plug for door hinge mounting hole (WHITE)
// door replaced by sliding glass door 27/11/2021
totDepth=15;
insertDiameter=7;
lidDiameter=14;
lidHeight=1;
$fn=100;
color ([1,1,1]) {
    cylinder(h=totDepth,d=insertDiameter);
    cylinder(h=lidHeight,d2=lidDiameter,d1=lidDiameter-lidHeight);
}
```

+=== Object - strikeplate

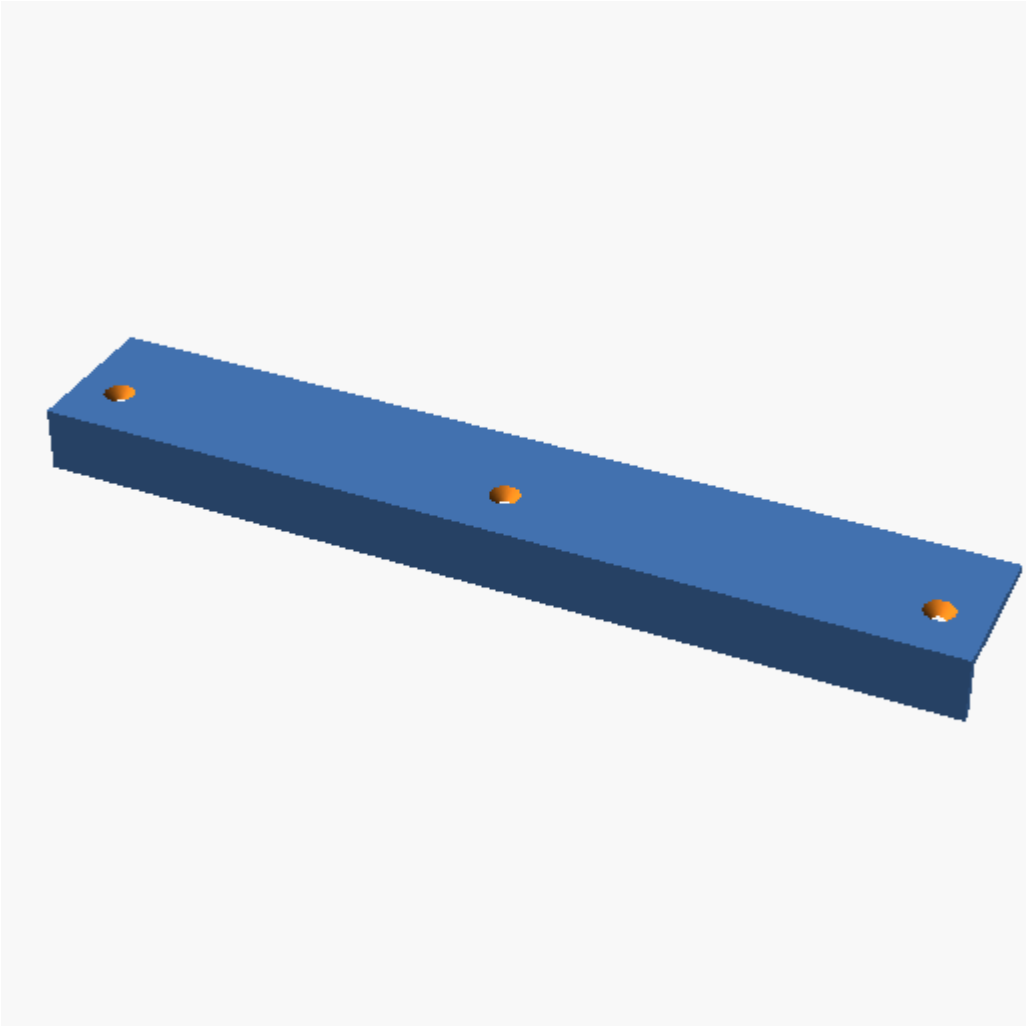

Figure 3. image

Listing 3. Openscad source

```
$fn=100;
SPlength=170;
SPwidth=28;
SPmaterialStrength=2;
;
module strikePlate () {
    cube ([SPlength,SPwidth,SPmaterialStrength]);
    translate ([0,0,-10])
        cube ([SPlength,SPmaterialStrength,10]);
}

module screw () {
    *cylinder(h=8,d=3);
    cylinder(h=3,d1=2,d2=7);
}

difference(){
    strikePlate();
```

```
    translate([8.5,10.5,-0.1]) screw();
    translate([85,10.5,-0.1]) screw();
    translate([SPlength-8.5,10.5,-0.1]) screw();
}
;
```

+=== Object - midletoninset

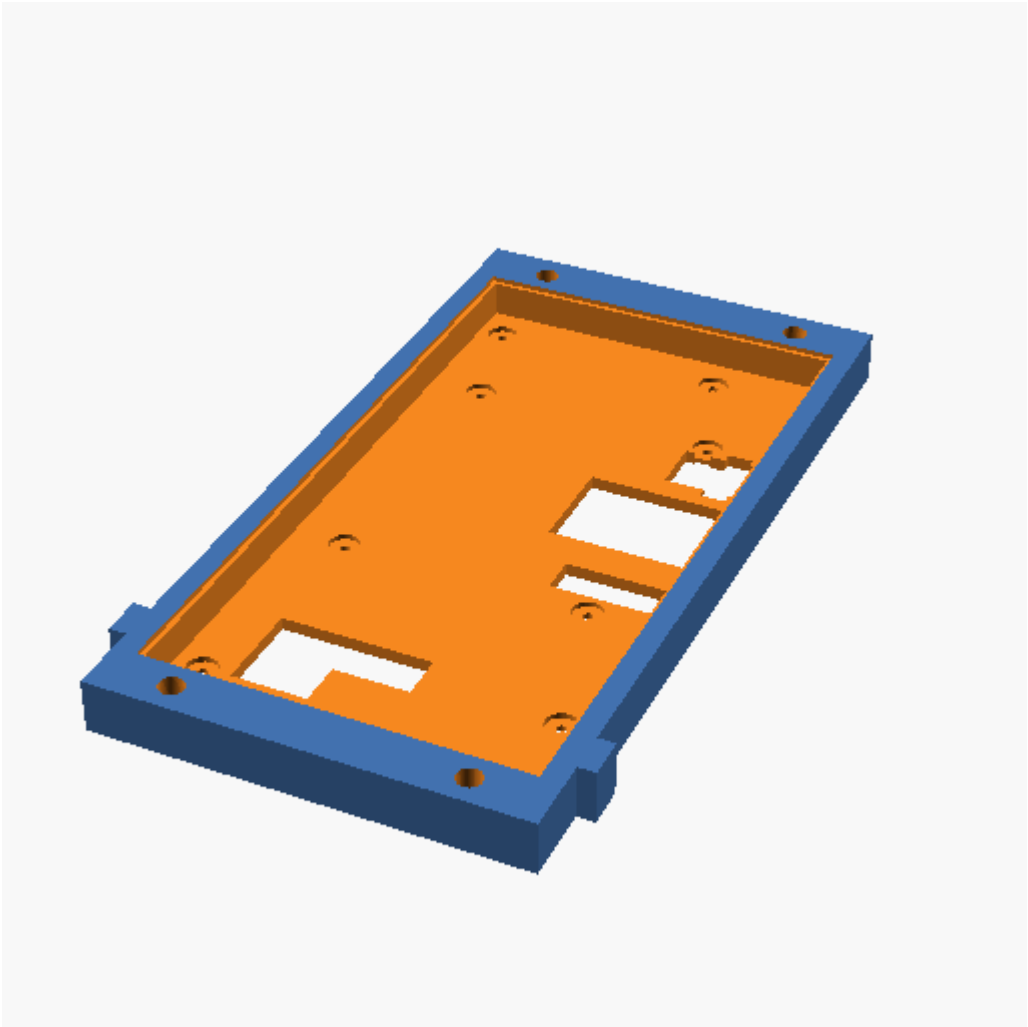

Figure 4. image

Listing 4. Openscad source

```
//This is an inlet for a whiskey presentation box from Midleton
$fn=50;
//Lower Notches
LowerNotchDepth=3.5;
LowerNotchLength=8;
LowerRNotchLengthOffset=15;
LowerLNotchLengthOffset=14.3;
module LLnotch(LowerLNotchLengthOffset){
    //Lower Left Notch
    translate([-LowerNotchDepth,LowerLNotchLengthOffset,0])
```

```
        cube([LowerNotchDepth,LowerNotchLength,BoxHeight]);
}
module LRnotch(LowerRNotchLengthOffset){
    //Lower Right Notch
    translate([BoxWidth,LowerRNotchLengthOffset,0])
        cube([LowerNotchDepth,LowerNotchLength,BoxHeight]);
}

//Variables for screen
ScreenTopY=75;
ScreenTopX=141;
ScreenTopZ=1;
ScreenEdge=1;
ScreenMaxDepth=7;
module waveshareHDMIscreen(wiggle){
    //full hd screen top face
    //and yes it has rounded corners but let's just start simple.
    union(){
        //Screen dimensions
        cube([ScreenTopY,ScreenTopX,ScreenTopZ+wiggle]);
        translate([ScreenEdge,ScreenEdge,-ScreenMaxDepth])
            cube([ScreenTopY-(2*ScreenEdge),ScreenTopX-
(2*ScreenEdge),ScreenMaxDepth+wiggle]);
        //connecting cable at the edge.
        translate([57,0,-ScreenMaxDepth]) cube([7,7,ScreenMaxDepth]);
        //USB for touch with offseted connector - wiggle through
        translate([12,19,-ScreenMaxDepth-10])
            cube([30,9,ScreenMaxDepth+10]);
        translate([12,10,-ScreenMaxDepth-10])
            cube([15,12,ScreenMaxDepth+10]);
        //USB for power - wriggle throrugh
        translate([65,95,-ScreenMaxDepth-10])
            cube([5,15,ScreenMaxDepth+10]);
        translate([57,97,-ScreenMaxDepth-10])
            cube([17,11,ScreenMaxDepth+10]);
        //HDMI connector - Wriggle through might not work... might have to make
hole larger
        translate([44,72,-ScreenMaxDepth-10])
            cube([30,20,ScreenMaxDepth+10]);
        //Audio?
        translate([51,56.75,-ScreenMaxDepth-4])
            cube([23,7.5,ScreenMaxDepth+4]);
        //The screw holes
        Standoffs();
        //The mounting holes for the displaycover
        translate([0+10,0-5,-ScreenMaxDepth-6])
            cylinder(h=20,d=4.8);
        translate([0+10,ScreenTopX+5,-ScreenMaxDepth-6])
            cylinder(h=20,d=4.8);
        translate([ScreenTopY-10,0-5,-ScreenMaxDepth-6])
```

```
            cylinder(h=20,d=4.8);
        translate([ScreenTopY-10,ScreenTopX+5,-ScreenMaxDepth-6])
            cylinder(h=20,d=4.8);
    }
}
StandoffDepth=9;
StandoffSpace=1;
StandoffScrewHead=2;
module HolePeg(offset1){
    //standoff
    translate([0,0,-StandoffDepth+1]+offset1)
        cylinder(h=StandoffDepth-1,r=3.05);
    //screwwshaft
    translate([0,0,-StandoffDepth-StandoffSpace+1]+offset1)
        cylinder(h=StandoffDepth+StandoffSpace-1,r=1);
    //Screw head
    translate([0,0,-StandoffDepth-StandoffSpace-StandoffScrewHead+1]+offset1)
        cylinder(h=StandoffScrewHead,r=3);
}
module Standoffs(){
    //Outside holes
    //one
    *HolePeg([6,9,0]);
    HolePeg([6.5,9.75,0]);
    //the rest
    HolePeg([69,22,0]);
    HolePeg([6,132.5,0]);
    HolePeg([53,132.5,0]);

    //inside holes
    HolePeg([11.5,52.5,0]);
    HolePeg([60.5,52.5,0]);
    HolePeg([60.5,110.5,0]);
    HolePeg([11.5,110.5,0]);
}

// Midleton box measurements
//Real total Height
//BoxHeight=61;
//Display inset Height
BoxHeight=10.5;
//testprint
//BoxHeight=8.5;
BoxWidth=83.8;
LowerPartLength=162.5;
//testing value
//LowerPartLength=50;
LowerPartWallThickness=1.5;
LowerPartFloorThickness=1.5;
module Displaymodule() {
```

```
    //Lower part of the box
    difference(){
        //Outercube
        cube([BoxWidth,LowerPartLength,BoxHeight]);
        //subtract for inner space

*translate([LowerPartWallThickness,LowerPartWallThickness,LowerPartFloorThicknes
s])
            cube([BoxWidth-2*LowerPartWallThickness,LowerPartLength,BoxHeight-
(2*LowerPartFloorThickness)]);
    }
    LLnotch(LowerLNotchLengthOffset);
    LRnotch(LowerRNotchLengthOffset);
}
//Displaymodule();
//Standoffs();
//waveshareHDMIscreen();

// put it all together
difference(){
    Displaymodule();
    //Screen
    translate([(BoxWidth-ScreenTopY)/2,(BoxWidth-ScreenTopY)/2+6,BoxHeight-
ScreenTopZ])
        waveshareHDMIscreen(.1);
    //for testprint only
    *translate([-10,10,2.5])cube([100,130,15]);
    *translate([10,-10,2.5])cube([65,160,15]);
}
//remove for print... only for animation
*translate([((BoxWidth-ScreenTopY)/2),(BoxWidth-ScreenTopY)/2+6,(BoxHeight-
ScreenTopZ)+30*(1-$t)]) waveshareHDMIscreen(0);
```

+=== Object - internal-volume

the internal Volume of a presentation box to test ideas on.

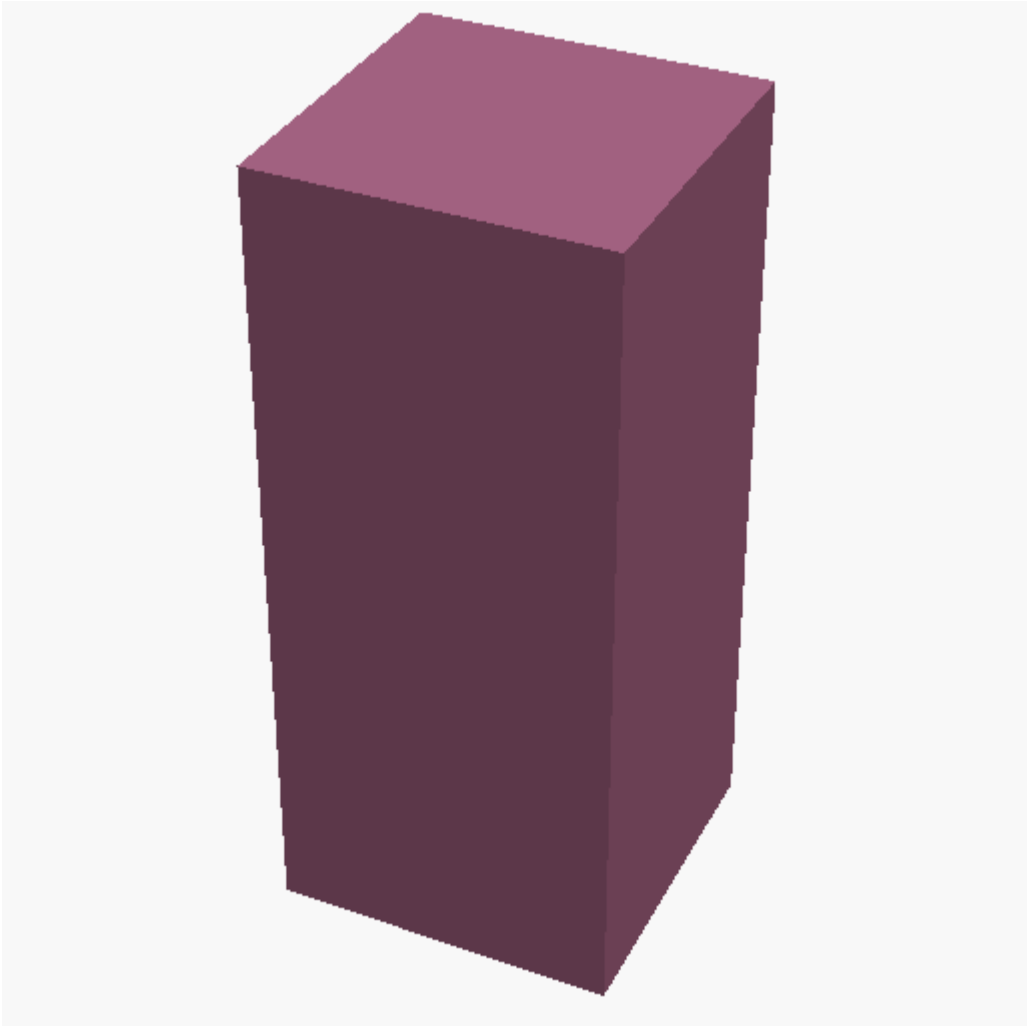

Figure 5. image

Listing 5. Openscad source

```
//inside of midleton wooden box with double doors
height=260;
width=111.1;
depth=108.5;

#cube([width,depth,height]);
```

+=== Object - steeringaxle

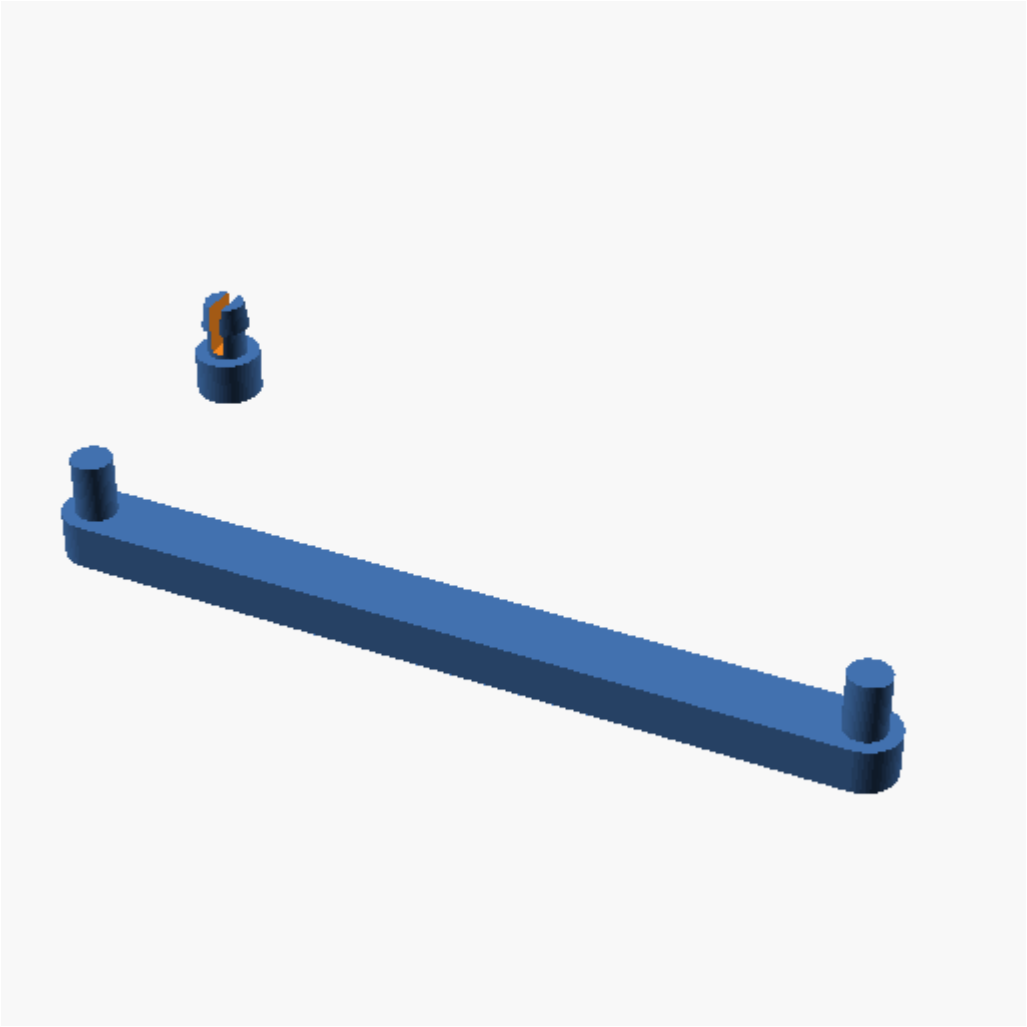

Figure 6. image

Listing 6. Openscad source

```
$fn=50;

module tabbedCylinder(){
    difference(){
        union (){
            cylinder(h=2,d1=3,d2=3);
            cylinder(h=4.6,d1=1.8,d2=1.8);
            translate ([0,0,3.4]) cylinder(h=1.2,d1=2.2,d2=1.8);
        }
        translate ([0,0,3.5]) cube([.6,2.5,2.5],center=true);
    }
}
module EndCylinder(){
        union (){
            cylinder(h=2,d1=3,d2=3);
            cylinder(h=4.6,d1=1.8,d2=1.8);
        }
}
```

```
module steeringAxle(){
    //axis
    translate ([1.5,0,0]) cube([34,3,2]);
    //connector
    translate ([1.5,1.5,0]) EndCylinder();
    //connector
    translate ([35.5,1.5,0]) EndCylinder();
}

steeringAxle();
translate([0,15,0]) tabbedCylinder();
```

+=== Object - screen_mounting_tabs

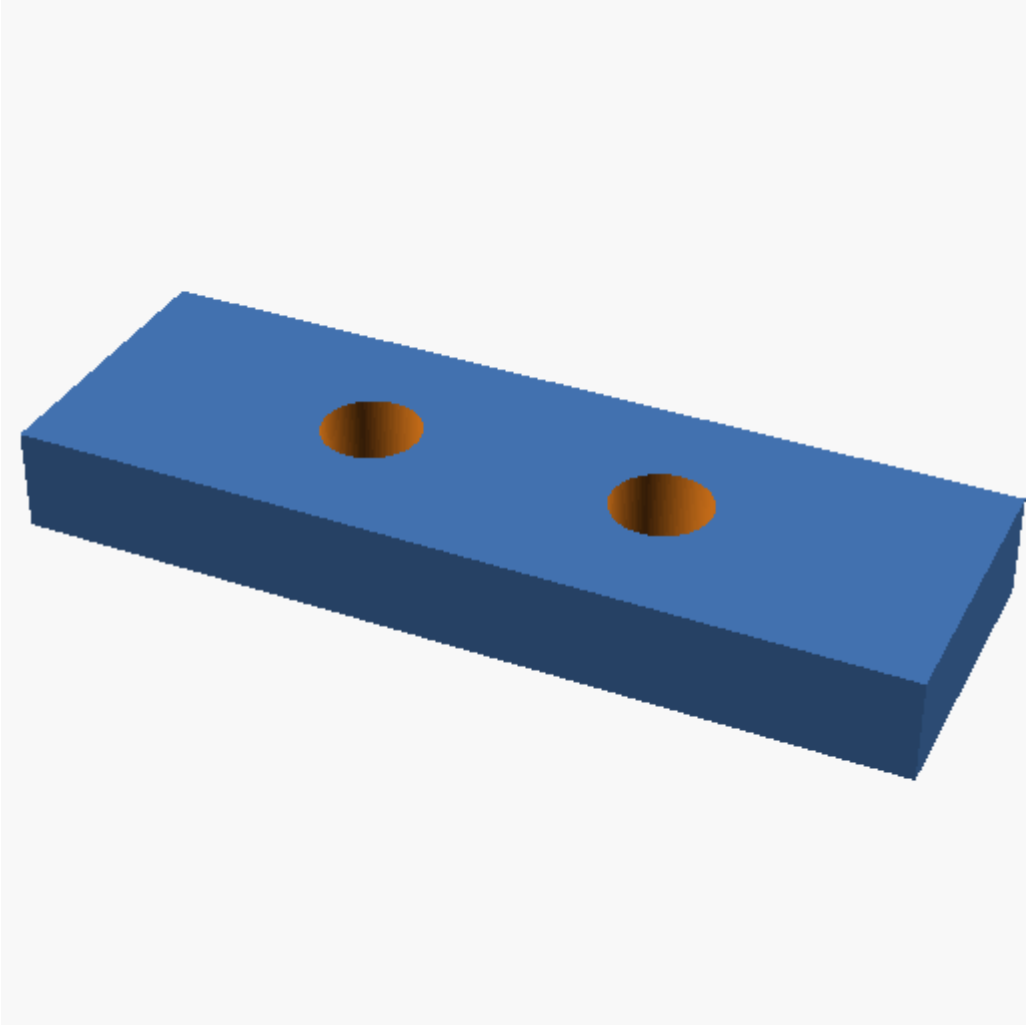

Figure 7. image

Listing 7. Openscad source

```
$fn=100;
tab_height=.3;
tab2bottom=2.4;
```

```
plus=.1; // this is to make parts larger than the hole they are to make
plusH=plus/2;

hole_d=2;
tab1_hole_spacing=8;
tab2_hole_spacing=6;
shim_height=tab2bottom-tab_height;
shim_depth=6;
shim_width=18;

module tab1(spacing){
    difference(){
        cube([shim_width,shim_depth,shim_height]);
        translate([shim_width/2-spacing/2,(shim_depth/2),-plusH])
            union() {
                cylinder(h=shim_height+plus,d=hole_d);
                translate([spacing,0,0]) cylinder(h=shim_height+plus,d=hole_d);
            }
    }
}

//for the bottom tabs
//tab1(tab1_hole_spacing);

//for the top tabs
tab1(tab2_hole_spacing);
```

+=== Object - RPi_zero_mount

This also is NOT one of mine but I've cleaned it up a bit as it wasn't displaying correctly. I needed it for a pi cluster and as it's quite good I didn't reinvent the wheel here.

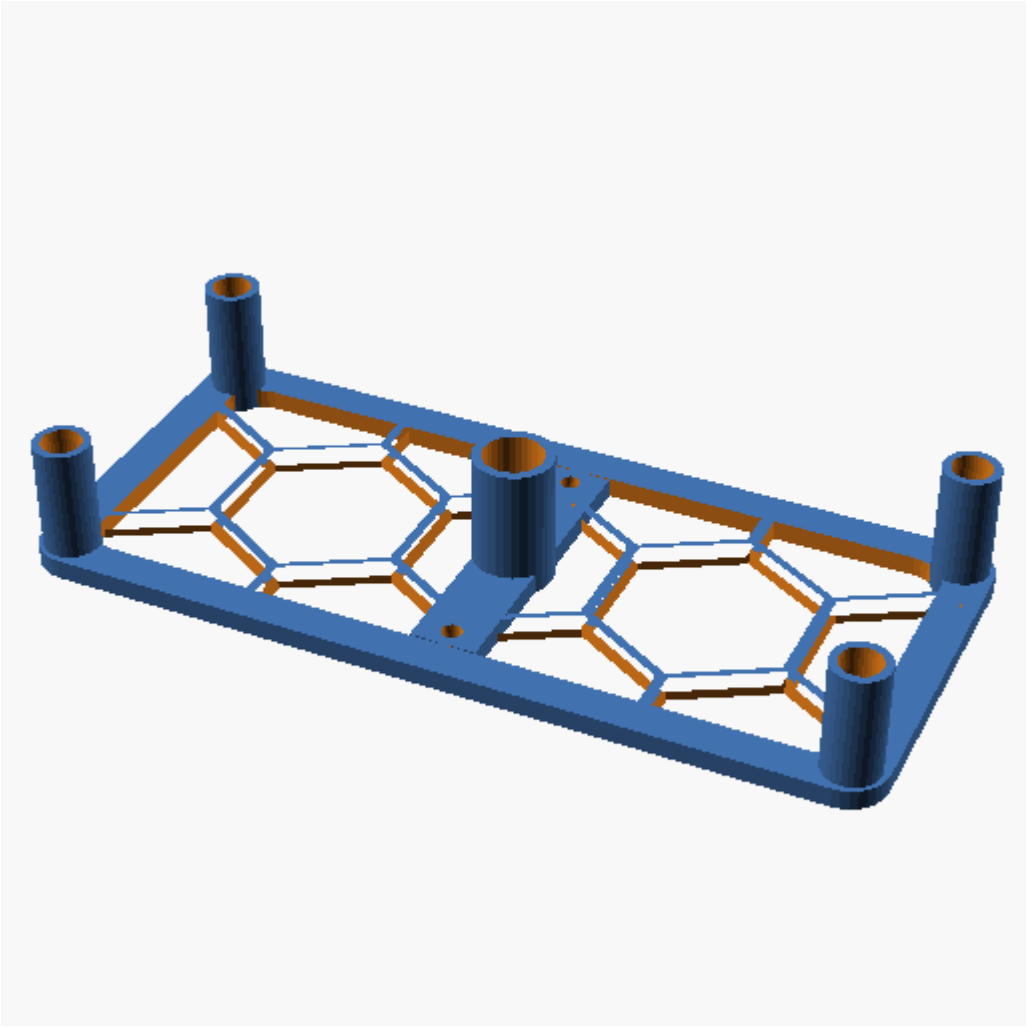

Figure 8. image

Listing 8. Openscad source

```
/* [Base] */
//type = 1; //[1:"Hexagon Grid",2:"Skeleton"]

/* [Hidden] */
$fn = 32;
zero_x = 64;
zero_y = 29;
zero_z = 1.5;

mounts_z = 8.5;
mounts_radius = 2.1;
screwholes = 2.6;
screwholes_radius = 1.5;
screwholes_depth = 10.7;

base_x = zero_x - 2*3.0;
base_y  = zero_y  - 2*3.0;
base_z = zero_z;
```

```
mount_x = zero_x/2 - screwholes;
mount_y = zero_y/2 - screwholes;
mount_z = zero_z + mounts_z;
screwhole_base_z = mount_z - screwholes_depth;

module baseplate(){
    translate([-zero_x/2+3,-zero_y/2+3,0])
        minkowski(){
            cube([base_x,base_y,base_z/2]);
            cylinder(r=3.0,h=base_z/2);
    }
}

module mounts(){
    translate([0,0,0]) cylinder(r=3.0,h=mount_z);
    translate([-mount_x, -mount_y, 0]) cylinder(r=mounts_radius,h=mount_z);
    translate([-mount_x, +mount_y, 0]) cylinder(r=mounts_radius,h=mount_z);
    translate([+mount_x, -mount_y, 0]) cylinder(r=mounts_radius,h=mount_z);
    translate([+mount_x, +mount_y, 0]) cylinder(r=mounts_radius,h=mount_z);
}

module hexagon (radius=8,latticeWidth=8,latticeLength=16,spacing=1,height=2){
    linear_extrude(height) {
        for(j = [0:latticeWidth-1]) {

translate([((sqrt(3)*radius)+spacing)/2*(j%2),sqrt((pow(((sqrt(3)*radius)+spacin
g),2))-(pow((((sqrt(3)*radius)+spacing))/2,2)))*j,0]) {
                for(i = [0:latticeLength-1]) {
                    translate([(sqrt(3)*radius*i)+spacing*i,0,0]) {
                        rotate([0,0,30]) {
                            circle(radius, $fn = 6);
                        }
                    }
                }
            }
        }
    }
}

module hex_border(){
    difference(){
        baseplate();
        holes();
        translate([0,0,-.01]) scale([0.9,0.8,1.02]) baseplate();
    }
}

module holes(){
    translate([0,0,screwhole_base_z+0.4]) {
        translate([0,0,0]) cylinder(r=screwholes_radius*1.5,h=screwholes_depth);
```

```
        translate([-mount_x,-mount_y,0])
cylinder(r=screwholes_radius,h=screwholes_depth);
        translate([-mount_x,+mount_y,0])
cylinder(r=screwholes_radius,h=screwholes_depth);
        translate([+mount_x,-mount_y,0])
cylinder(r=screwholes_radius,h=screwholes_depth);
        translate([+mount_x,+mount_y,0])
cylinder(r=screwholes_radius,h=screwholes_depth);
    };
}

module result(){
    difference(){
        translate([-2.5,-base_y/2,0]) cube([5,base_y,base_z]);
        translate([0,10,-3]) cylinder(d=1.5,h=10);
        translate([0,-10,-3]) cylinder(d=1.5,h=10);
        holes();
    }
    translate([0,0,0])
    hex_border();
    difference(){
        translate([0,0,0]) cylinder(r=3.0,h=mount_z);
        holes();
    }
    difference(){
        mounts();
        holes();
    }
    difference(){
        baseplate();
        holes();
        translate([-zero_x/2-5,-zero_y/2+1.5,-0.1]) hexagon();
    }
}

difference(){
    result();
    translate([0,10,-3]) cylinder(d=1.5,h=10);
    translate([0,-10,-3]) cylinder(d=1.5,h=10);
}
```

+=== Object - schuko

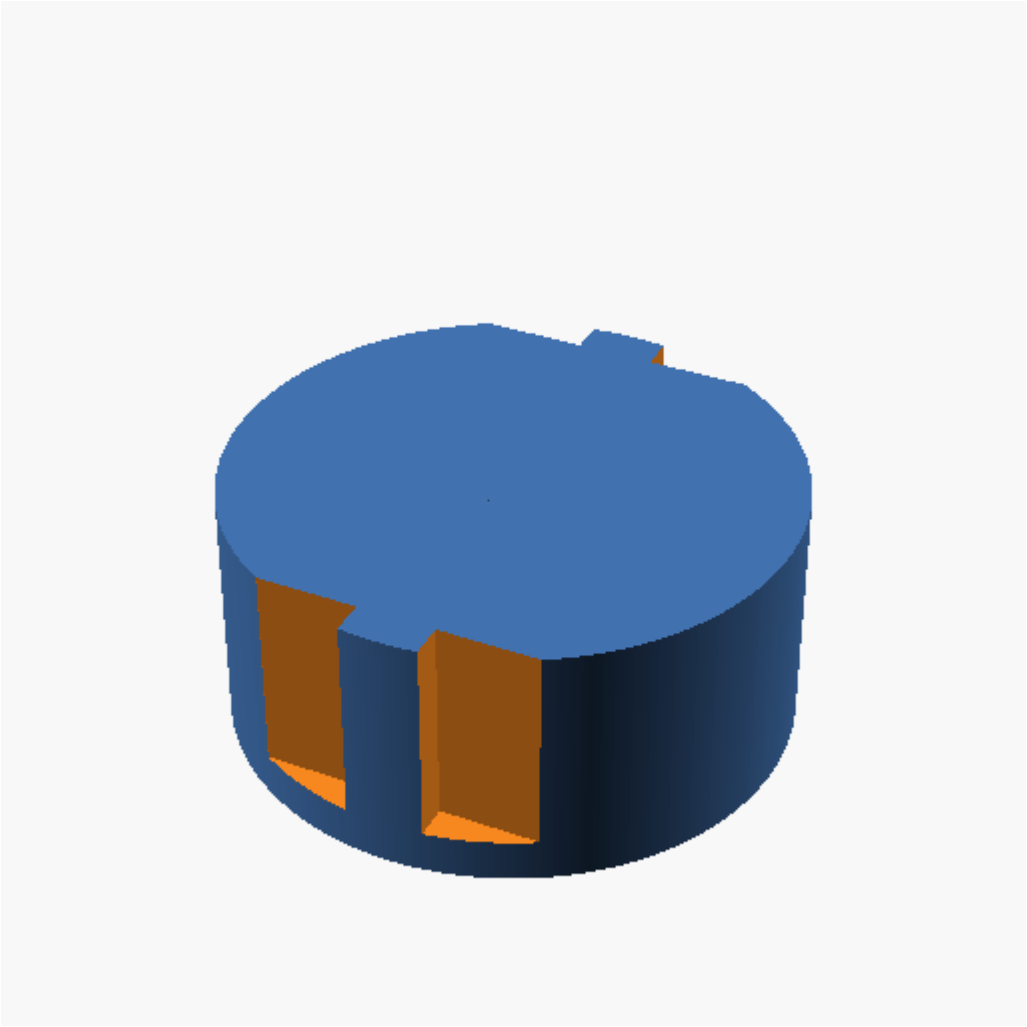

Figure 9. image

Listing 9. Openscad source

```
/*
 Parametric Schuko CEE 7/3 socket

 Copyright 2017 Anders Hammarquist <iko@iko.pp.se>
 Licensed under Creative Commons - Attribution - Share Alike

 Made using a negative "profile punch" that can be extracted
 and used to "punch" a schuko socket into any sufficiently large solid.
*/

// Diameter of cover
coverdiameter = 50; // [50:100]

// Thickness of cover
coverthickness = 4.8; // [2:0.2:15]

// Center screw offset (extreme values disables screw hole)
screwoffset = 0; // [-11:0.5:11]
```

```
// This is the socket punch. Includes cut-out for
// earthing contacts and holes for pins and center screw.
// Maximum screw offset from center is 10mm (use a larger
// value to remove the hole for the screw).
module schuko(screwoffset=0, screwdia=3.5, screwhead=6.5, screwsink=3)
{
    module earthing()
    {
        intersection() {
            union() {
                translate([-22,-2,3])
                    cube([6,4,20]);
                translate([-19,-2,17.5])
                    rotate([0,-30,0])
                        cube([15, 4, 4]);
            }
            translate([-22,-3,3])
                cube([22,6,20]);
        }
    }

    difference() {
        union() {
            translate([0,0,-1])
                cylinder(r=39/2, $fn=300, h=18.5);

            // Earthing cutouts
            color([1,1,1]) {
                earthing();

                rotate([0,0,180])
                    earthing();

            }

            // Power pins
            translate([0,10,0])
                cylinder(r=7/2, $fn=300, h=30);
            translate([0,-10,0])
                cylinder(r=7/2, $fn=300, h=30);

            if (abs(screwoffset) <= 10) {
                // Center screw
                translate([screwoffset,0,0])
                    cylinder(r=screwdia/2, $fn=300, h=30);
                translate([screwoffset,0,0])
                    cylinder(r=screwhead/2, $fn=300, h=17.5+screwsink);
            }
        }
```

```
        // Side key profile
        translate([5.4/2,16.9,3])
            cube([7,3,20]);
        translate([-5.4/2-7,16.9,3])
            cube([7,3,20]);
        translate([5.4/2,-20.4,3])
            cube([7,3.5,20]);
        translate([-5.4/2-7,-20.4,3])
            cube([7,3.5,20]);
    }
}
difference () {

difference () {
    cylinder(r=39/2, $fn=300, h=17.5);
translate ([-27.3/2,-27.8/2,0]) cube([27.3,27.8,10]);
rotate([0,0,0]){
    difference(){
        union() {
            translate([0,0,0])
                cylinder(r=44/2, $fn=300, h=21.5);
            // Lip
            rotate_extrude($fn=100) {
                polygon(points=[[0,0], [coverdiameter/2,0],
                [coverdiameter/2+0.2*coverthickness,coverthickness],
                [0,coverthickness]]);
            }

            // Pin guard: 9.5 x 28.5 x 3mm (rounded ends)
            translate([-4.75,-14.25,21.5])
                cube([9.5, 28.5, 3]);

            // center screw standoff: 6 x 2.5 (above pin guard) x 2 - 3
            // ( 8mm inside, 14 - 12.2 mm outside)
            translate([-7.25, -3, 21.5])
                cube([2.5, 6, 5.5]);
            translate([4.75, -3, 21.5])
                cube([2.5, 6, 5.5]);

        }
        schuko(screwoffset=screwoffset);
    }
}
}

}
```

+=== Object - tesa

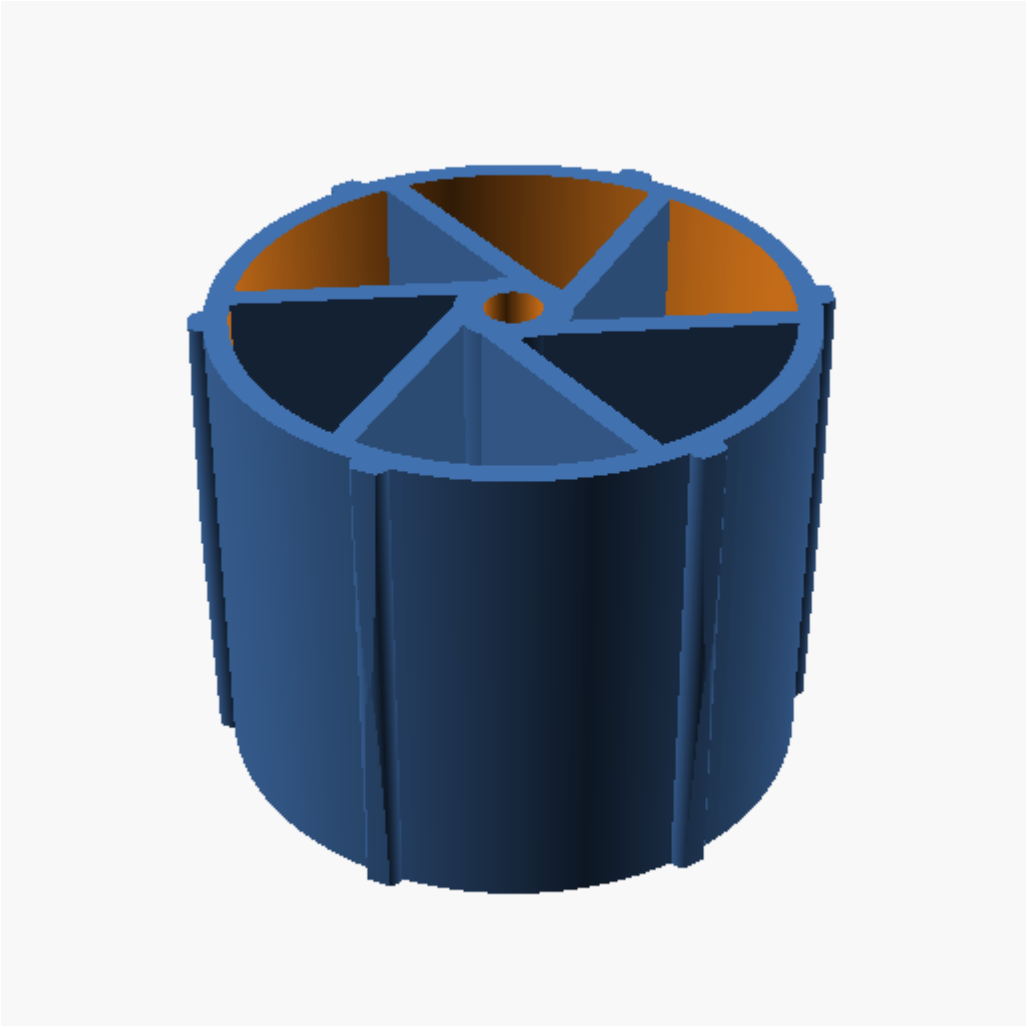

Figure 10. image

Listing 10. Openscad source

```
//Tesa roller ersatzroller
//celotape roller

$fn=360;
height=20;
outsideD=24.5;
outsideDepth=2;
axleD=2.4;
hubD=axleD*2;
nubR=.75;
module taper(){
difference(){
union(){
    translate([0,0,-.1]) cylinder(h=height+.2,d=outsideD+2+nubR);
}
union(){
    translate([0,0,-
.11])cylinder(h=height/2+.22,d1=outsideD+1.5*nubR,d2=outsideD+2*nubR);
```

```
    translate([0,0,height/2+.1])
cylinder(h=height/2+.1,d1=outsideD+2*nubR,d2=outsideD+1.5*nubR);
}
}
}
difference(){
union(){
    //outside
    difference(){
        cylinder(h=height,d=outsideD);
        translate([0,0,-.1])cylinder(h=height+.2,d=outsideD-outsideDepth);
    }
    //HUB
    difference(){
        cylinder(h=height,d=hubD);
        translate([0,0,-.1])cylinder(h=height+.2,d=axleD);
    }
    //nubs
    for (i = [0:5]) {
        translate([sin(360*i/6)*outsideD/2, cos(360*i/6)*outsideD/2, 0 ])
        rotate([0,0,0])cylinder(h = height/2, r=nubR);
    }
    for (i = [0:5]) {
        translate([sin(360*i/6)*outsideD/2, cos(360*i/6)*outsideD/2, height/2 ])
        cylinder(h = height/2, r=nubR);
    }
    //spokes
    for (i = [0:360/6:360]) {
        rotate([0,0,i])translate([1.2,0,0])cube([1,(outsideD/2)-
(axleD/2.4),height]);
    }
}
taper();
}
```

3. To do

Right now the github source is not perfect as the readme does not display the images when viewed in github.

Need to add further process steps for the images like meshlabserver to do furhter processing:

- glass rendering
- cleaning up the mesh
- Simplifying the mesh
- Stats

Need to add animation options.

☑ Need to add text display option for each item.

Need to add view parameters as options.