# Quick Introduction to OpenSCAD

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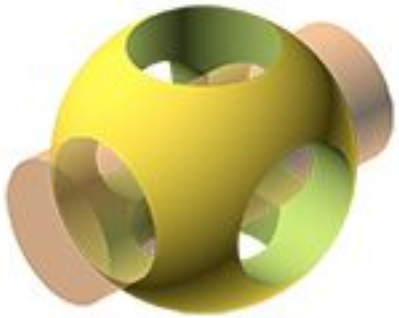
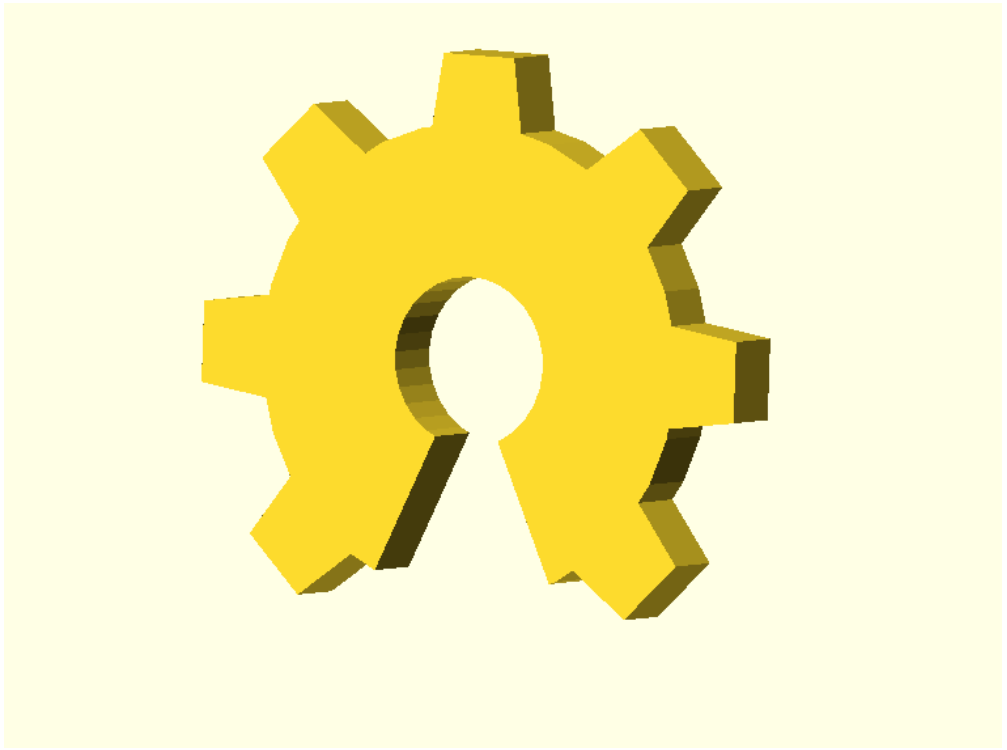


OpenSCAD
The Programmers Solid 3D CAD Modeller



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# Make Everything Parametric

**Allows later scaling, changing and newbie customization**

**All numbers should be made variables**

Can use letters for simple designs **// but comment**

-advantages: simple equations

-disadvantage: big memory for large projects

**Can use variable names describing it** // box_length

-advantages: no comments, can read the code in English

-disadvantage: big messy equations

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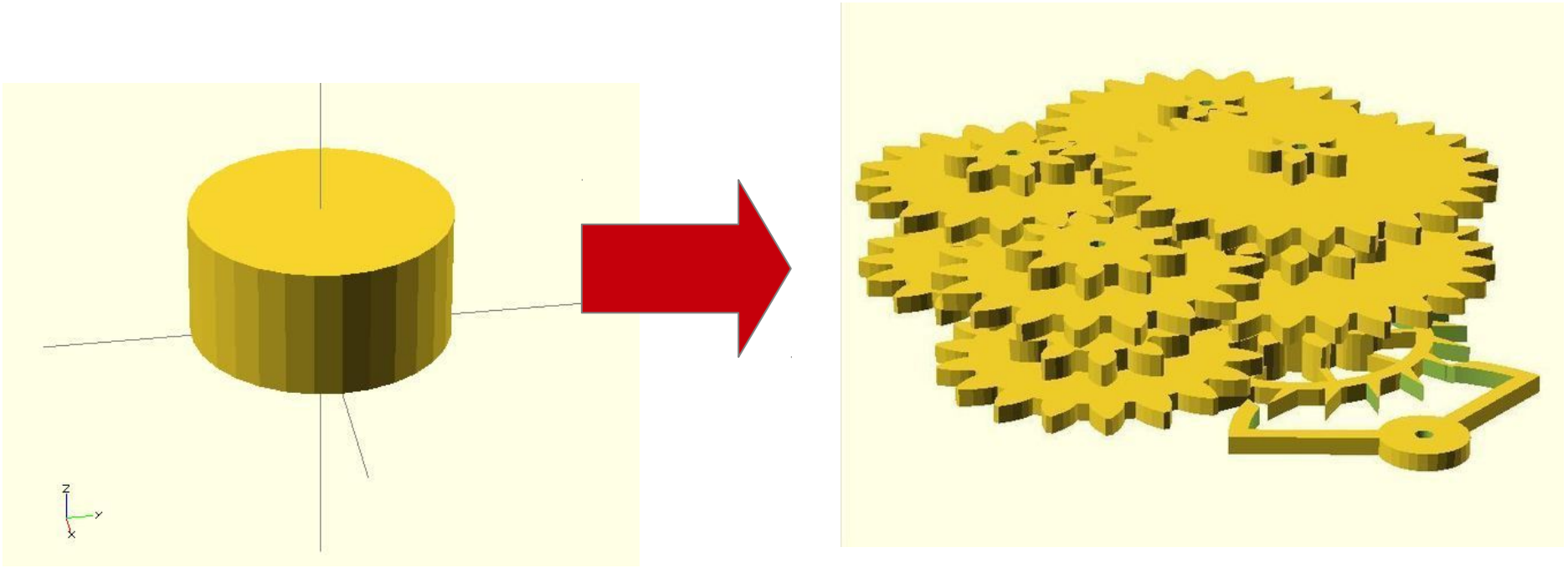
# Design Using Primitive Shapes and Collecting Together

Simple → Complex



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# When Designing: Show Axes


Helps Orient Primitives

Know which way is up for printing!

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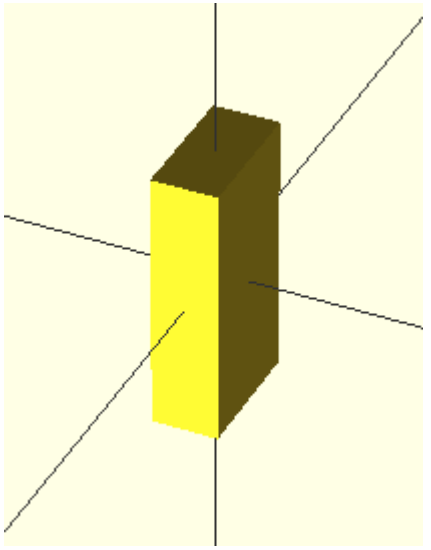
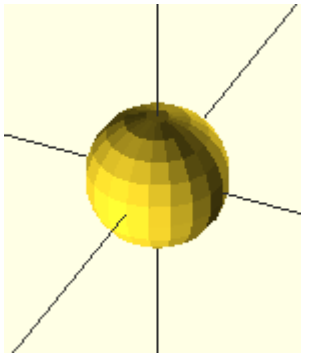
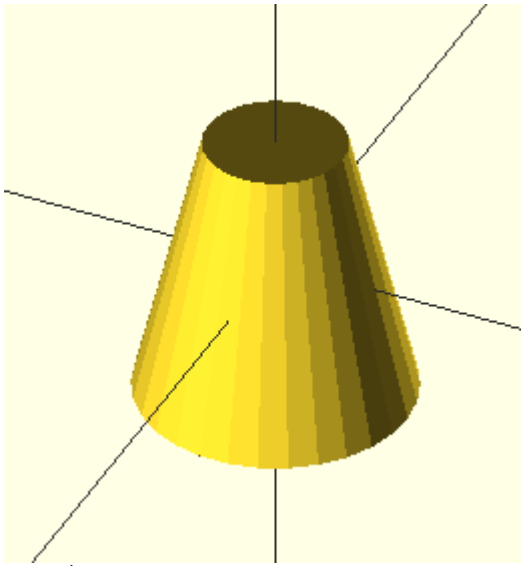
# Primitive Objects

```
a=5;
b=10;
c=20;
```







```
cube([a,b,c], center=true);
sphere(a, $fn=c);
//$fn is the resolution
cylinder(h = c, r1 = b, r2 = a, center = true);
```

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# Union Combining Primitives

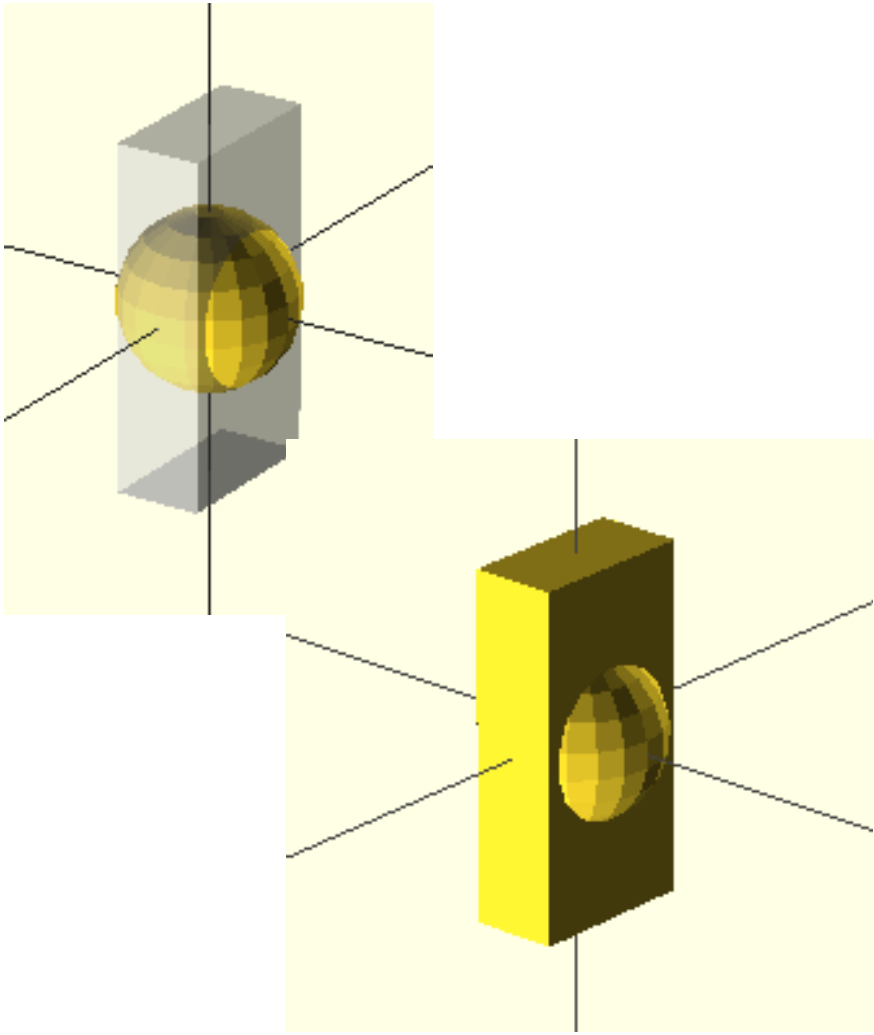
“Try before you Buy”=%

union(){

%cube([a,b,c], center=true);

sphere(a, $fn=c);

}



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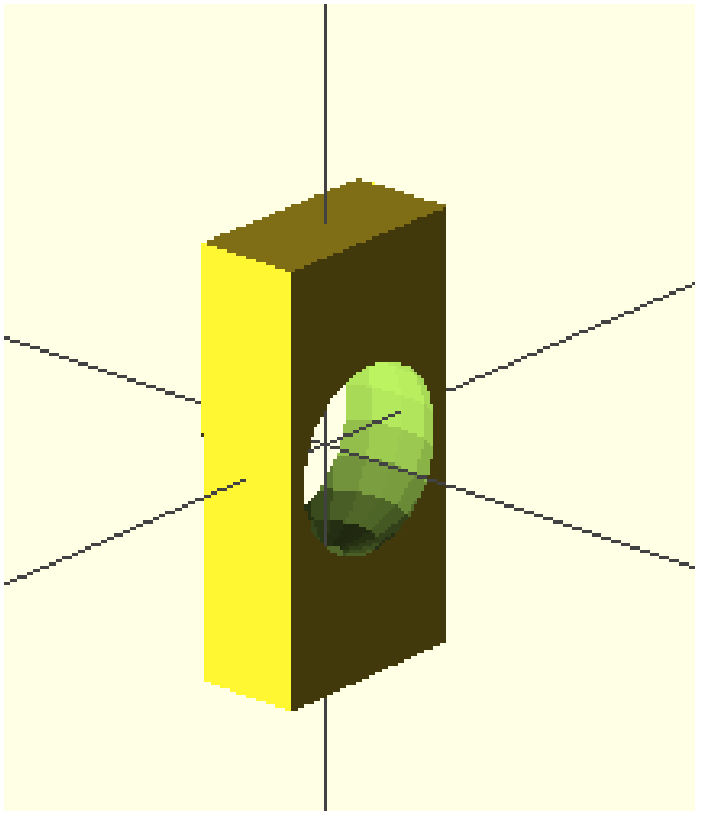
# Difference - Subtraction

```
difference(){
cube([a,b,c], center=true);
sphere(a, $fn=c);
}
```



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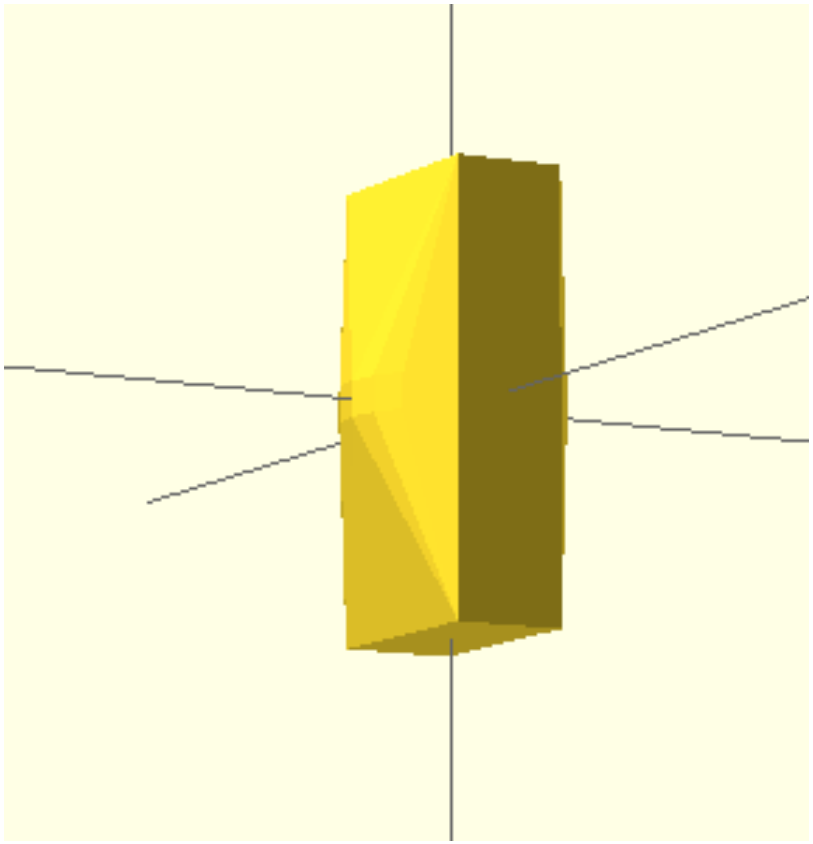
# Hull: Convex Hull of Child Nodes

```
hull(){
cube([a,b,c], center=true);
sphere(a, $fn=c);
}
```



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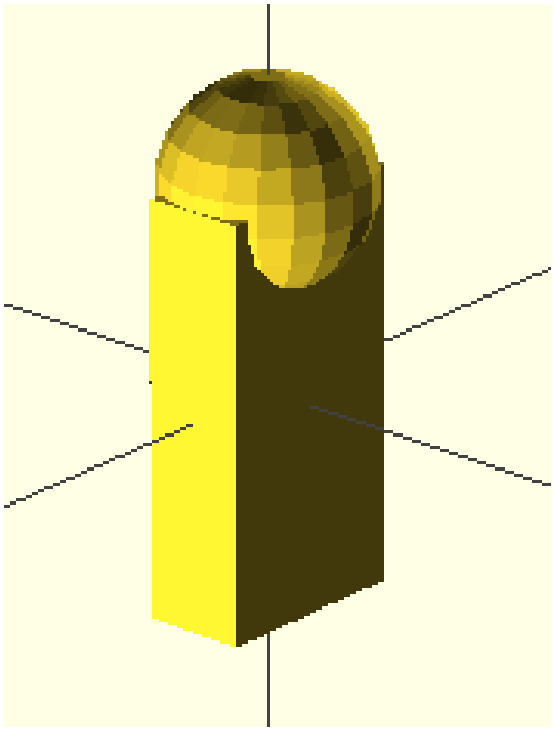
# Translate: Moving Stuff Around

```
union(){
cube([a,b,c], center=true);
translate([0,0,b])sphere(a, $fn=c);
   }
```



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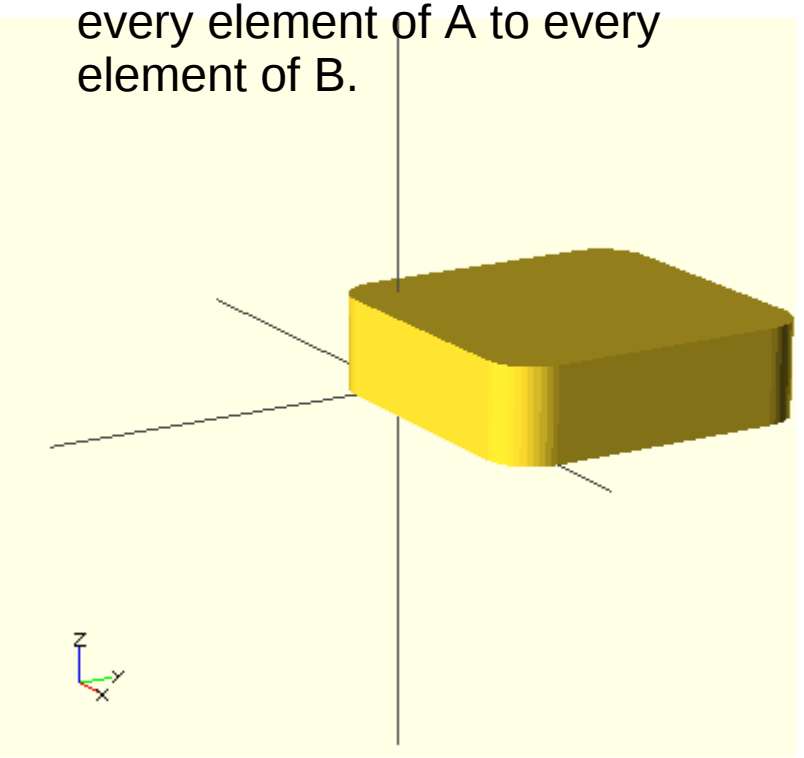
# Rounded Corners: Minkowski

```
$fn=50;
minkowski() {
    cube([10,10,2]);
    // rounded corners
    cylinder(r=2,h=2);
}
```

Minkowski sums allow to add every element of A to every element of B.

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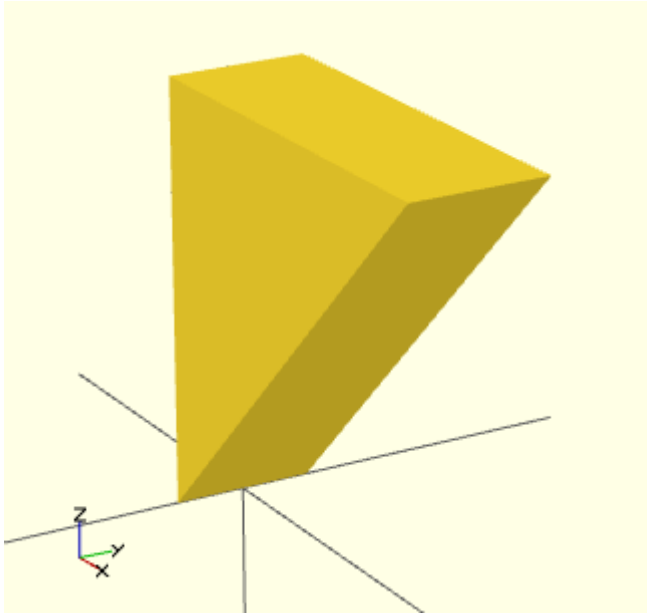
# Hand Crafting: Polyhydron

```
polyhedron ( points = [[0, -10,
    60], [0, 10, 60], [0, 10, 0], [0,
    -10, 0], [60, -10, 60], [60, 10,
    60]],

triangles = [[0,3,2], [0,2,1],
    [3,0,4], [1,2,5], [0,5,4],
    [0,1,5],  [5,2,4], [4,2,3], ]);
```

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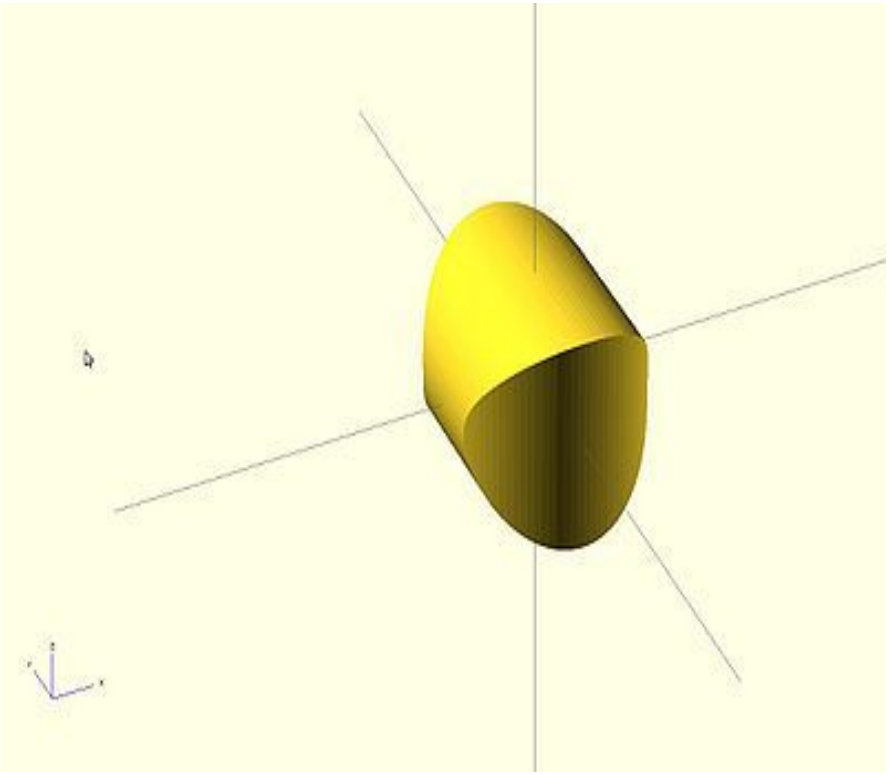
# Intersection : Keeps All Portions That Overlap

```
intersection() {
cylinder (h = 4, r=1, center
    = true, $fn=100);
rotate ([90,0,0]) cylinder (h
    = 4, r=0.9, center = true,
    $fn=100);
}
```



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# Make Each Completed Component a Module

**Allows for more complex design**

**Clears the work space as modules are not shown unless called**

**Syntax:**

module example(){ put your module scad here }

**Call it by:**

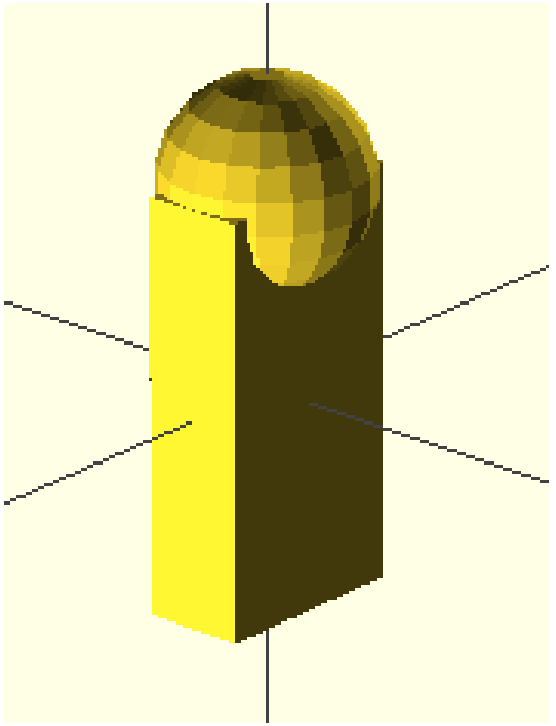
example();



# Modules

```
module example(){
union(){
cube([a,b,c], center=true);
translate([0,0,b])sphere(a,
   $fn=c);
}
}

example();
```



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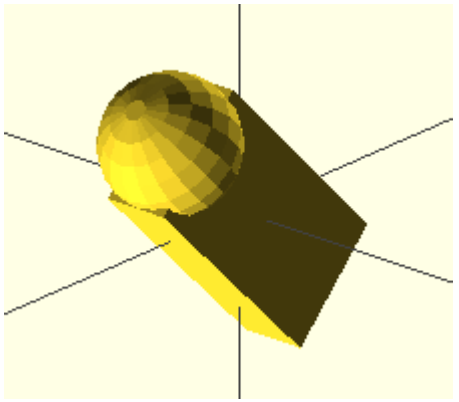
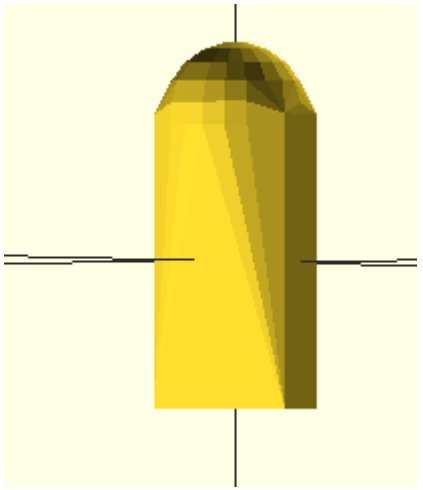
# Manipulate Your Module

```
rotate([45,0,0])example();
```



```
hull() {
example();
}
```



Add, subtract modules etc.

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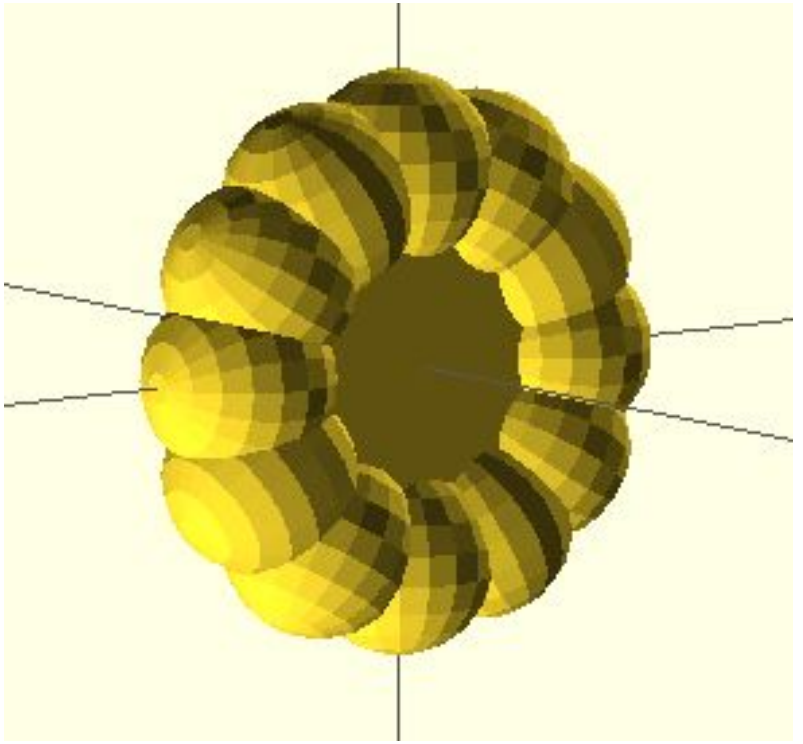
# For Repetitive Tasks Use Loops

```
for (i = [1:12])
{
    assign (angle = i*30)
    {
        rotate(angle, [1,0,0])
  example();
    }
}
```



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# Putting it All Together to Make Something Useful

Shelling corn is a chore done by hand in much of the rural developing world. Yet there are handy corn shellers that can save people hours of labor. **DIY shellers are a big chore to make...so you can print one.**

The finished, cement-filled corn sheller is on the right. A commercial aluminum corn sheller is on the left. The bottom sheller that was cut from a PVC pipe cap. It did not perform as well as the can.


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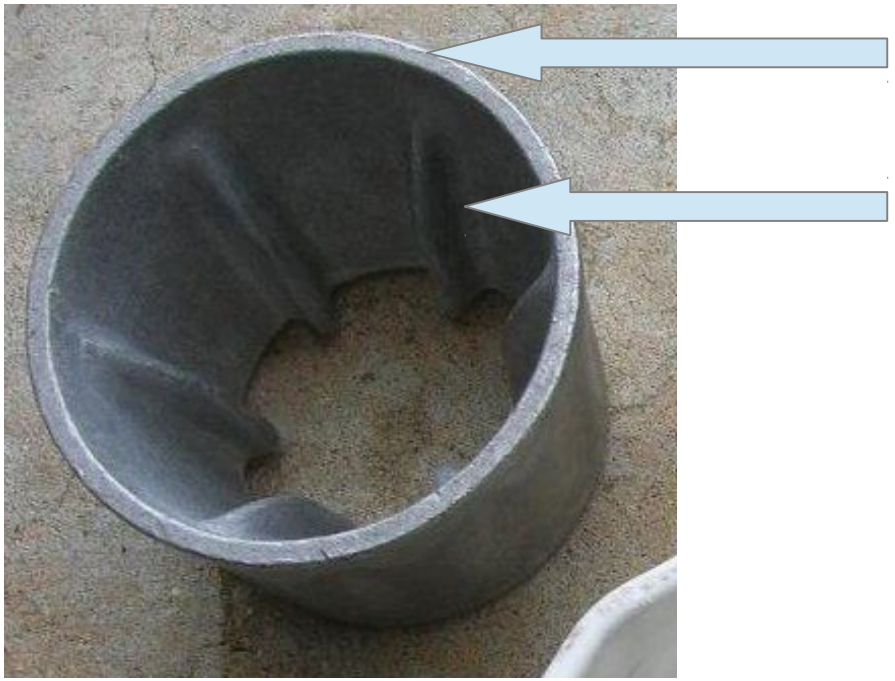
# Step 1: Break Complex Object Into Simple Parts



Bucket: 2 tapered cylinders

Fingers: 2 hulled cylinders

Fingers tapered in

Consider improvements:

Grips on outside – use fingers

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# Parametric – Design ALL of the Products at Once

Step 2: Lay out variables with comments to input to Customizer

```
//Open-source parametric hand corn sheller

// height of corn sheller
h=55;

// radius of top of corn sheller
rt=35; //[50:130]

rb=0.85*rt; //radius of bottom of corn sheller

//number of digits
d=6;

// digit radius
r=1.5;

// extra length of digit
l=1;

// thickness of sheller
t=3;
```

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# Using Modules

```
module sheller(){
union (){
for (z = [0:d]) // d iterations, z = 0 to d
{
    rotate([0,0, z*360/d])translate([rb,0,h*.1])finger();
}
difference(){
cylinder(h = h, r1 =rt, r2 =rb, center = true, $fn=100);
translate([0,0,0])cylinder(h = h+1, r1 =rt-t, r2 =rb-t, center = true, $fn=100);
}
}
}
module finger(){
rotate([0,(rb/rt)*-10,0])
hull(){
cylinder(h = h*.9, r1 =2*r, r2 =2*r, center = true, $fn=100);
translate([l-(rt-rb),0,0])cylinder(h = h*.9, r1 =r, r2 =r, center = true, $fn=10
0);
}
}
```

Many Fingers

Bucket Wall

Finger

Best practices:
Indent to see,
comment everything
$fn=100; one time

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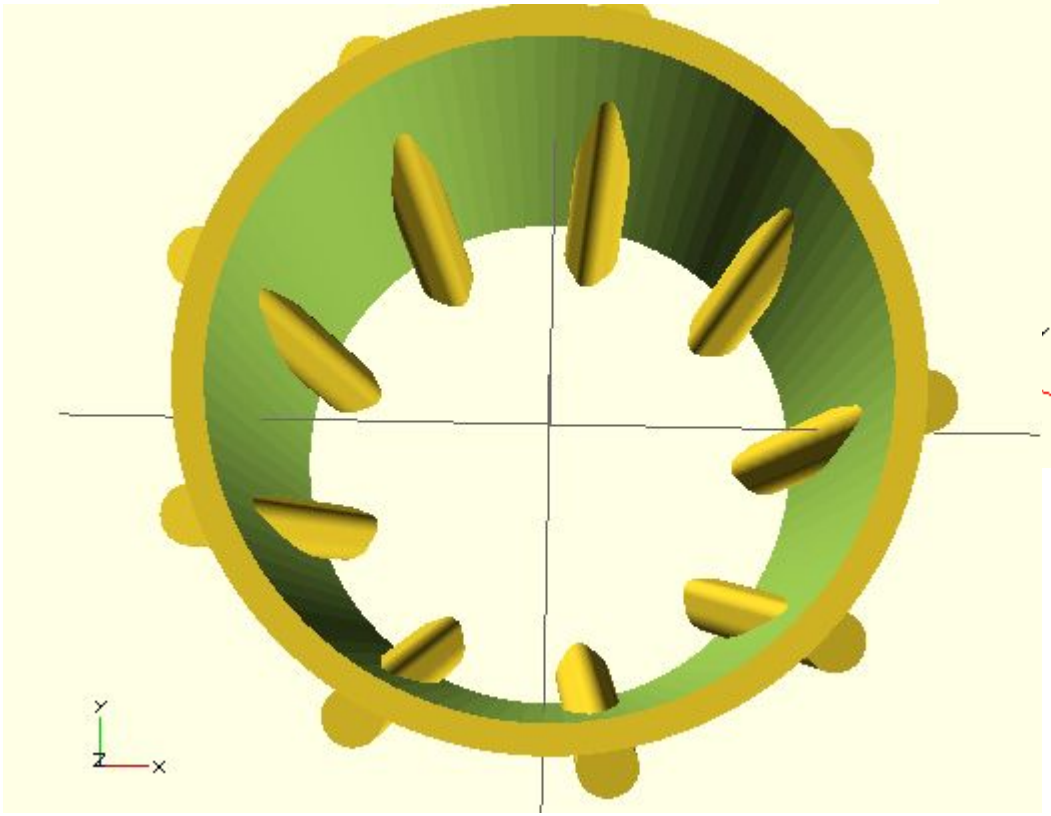
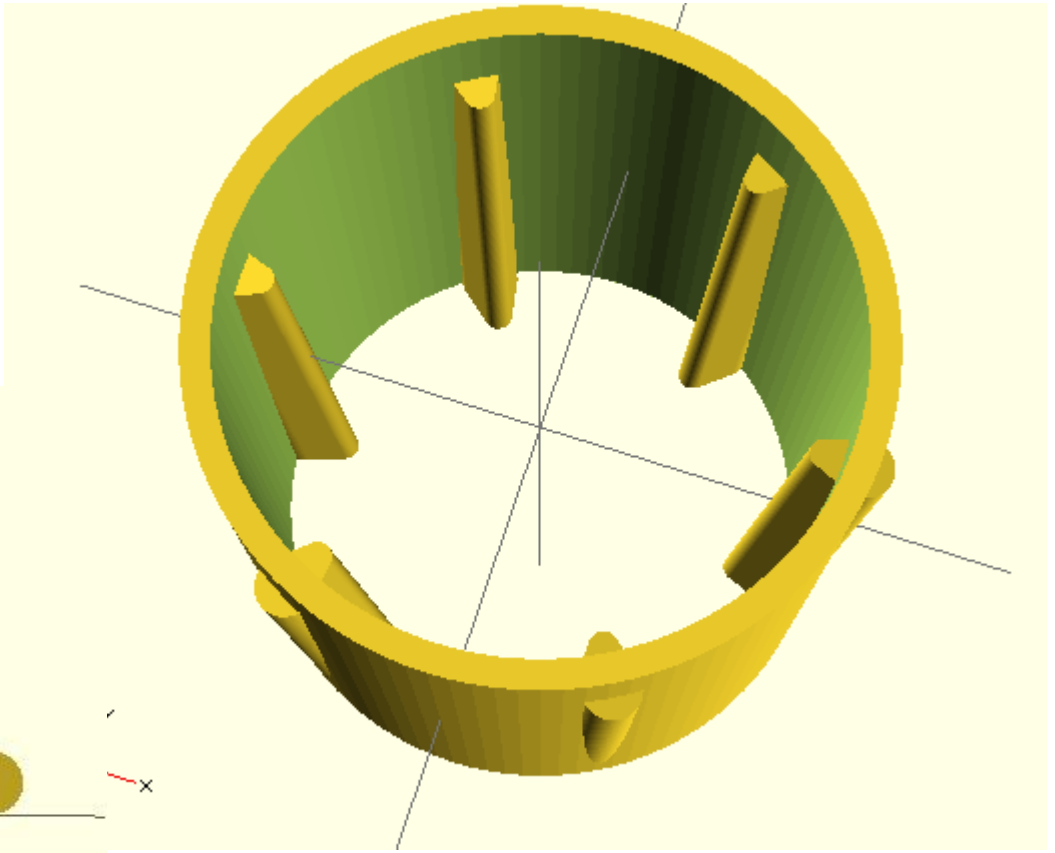
# Parametric Corn Sheller





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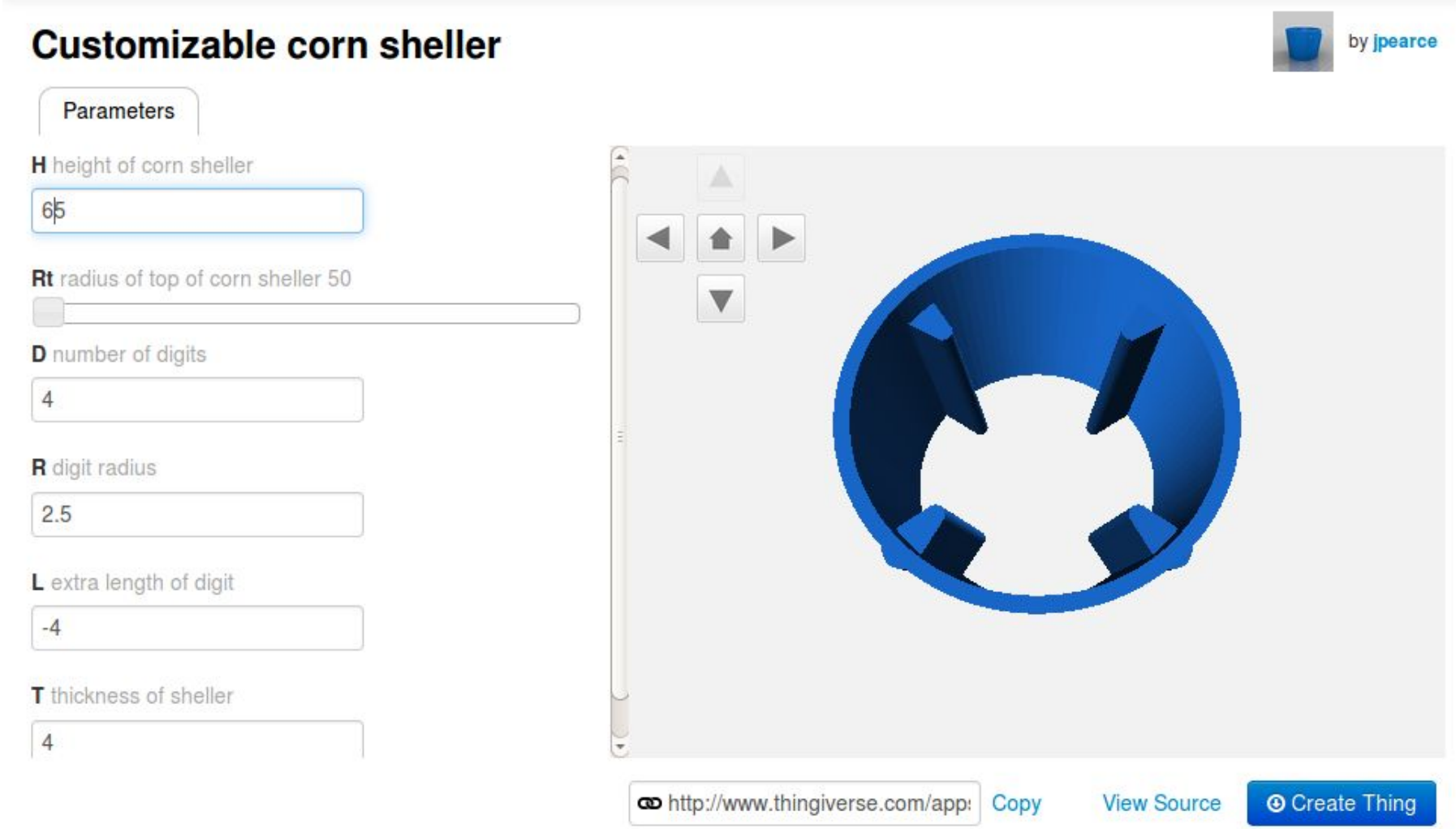
# Thingiverse Customizer




Anyone can make a corn sheller perfect for them with no coding.

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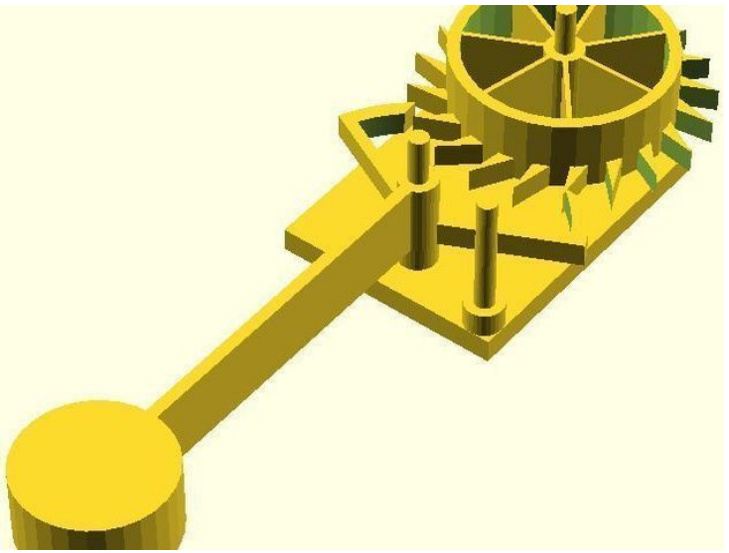
# Use Past Work

**Libraries:**

use <MCAD/involute_gears.scad>

include <escapementLibrary.scad>



You are using collections of

Modules written before...

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# Cheat Sheet

## Syntax

```
var = value;
module name(…) { … }
name();
function name(…) = …
name();
include <….scad>
use <….scad>
```

## 2D

```
circle(radius)
square(size,center)
square([width,height],center)
polygon([points])
polygon([points],[paths])
```

## 3D

```
sphere(radius)
cube(size)
cube([width,height,depth])
cylinder(h,r,center)
cylinder(h,r1,r2,center)
polyhedron(points, triangles, convexity)
```

## Transformations

```
translate([x,y,z])
rotate([x,y,z])
scale([x,y,z])
mirror([x,y,z])
multmatrix(m)
color("colorname")
color([r, g, b, a])
hull()
minkowski()
```

## Boolean operations

```
union()
difference()
intersection()
```

## Modifier Characters

```
*   disable
!   show only
#   highlight
%   transparent
```

## Mathematical

```
abs
sign
acos
asin
atan
atan2
sin
cos
floor
round
ceil
ln
len
log
lookup
min
max
pow
sqrt
exp
rands
```

## Other

```
echo(…)
str(…)
for (i = [start:end]) { … }
for (i = [start:step:end]) { … }
for (i = […,…,…]) { … }
intersection_for(i = [start:end]) { … }
intersection_for(i = [start:step:end]) { … }
intersection_for(i = […,…,…]) { … }
if (…) { … }
assign (…) { … }
search(…)
import("….stl")
linear_extrude(height,center,convexity,twist,slices)
rotate_extrude(convexity)
surface(file = "….dat",center,convexity)
projection(cut)
render(convexity)
```

## Special variables

```
$fa minimum angle
$fs minimum size
$fn number of fragments
$t  animation step
```

http://www.openscad.org/documentation.html

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# Thank you!

More information

- http://www.openscad.org/
- http://en.wikibooks.org/wiki/OpenSCAD_User_Manual
- http://www.appropedia.org/MOST
- http://reprap.org/
- pearce@mtu.edu

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