Parametrické modelování

KMI/3DT 3D tisk

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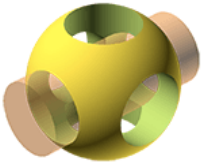
OpenSCAD



OpenSCAD

https://www.openscad.org

3D objekty

- `cube(size,center)`
- `sphere(r|d,$fa,$fs, $fn)`
- `cylinder(h,r|d,center)`
- `cylinder(h,r1|d1,r2|d2, center)`
- `polyhedron(points,faces,convexity)`

Transformace

- `scale([x, y, z])`
- `resize([x, y, z], auto)`
- `rotate(a, [x, y, z])`, `rotate([x, y, z])`
- `translate([x, y, z])`
- `mirror([x, y, z])`
- `multmatrix(m = [ ... ])`

Operace

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

Vykreslení

- F5 - náhled
- F6 - vyrenderování
- Export jako STL

Ladění

- `color("red")`, `color([r,g,b])`, `color([r,g,b,a])`
- #
- %
- !

2D objekty

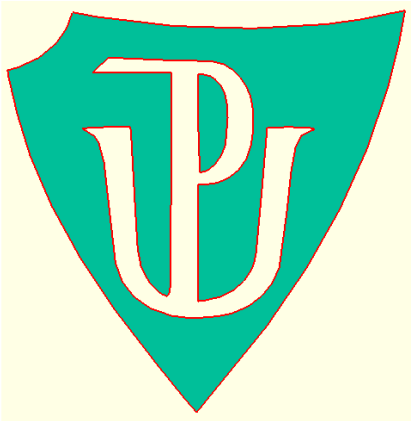
- `square(size,center)`
- `square([width,height],center)`
- `circle(r|d,$fa,$fs, $fn)`
- `polygon([points])`
- `polygon([points],[paths])`
- `text(t, size, font, halign, valign, spacing, direction, language, script)`
- `import("")`

2D objekty

- Jak vytvořit pravidelný šestiúhelník?

Import

- `import(file=".dfx")`
- Inkscape
- nutno objekt převést na sérii čar!



Operace

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

Example

```
union(){square(10);circle(10);}
difference(){square(10);circle(10);}
difference(){circle(10);square(10);}
intersection(){square(10);circle(10);}
```

Transformace

- `scale([x, y])`
- `resize([x, y], auto)`
- `rotate(a)`
- `translate([x, y])`

2D objekty

- Jak vytvořit elipsu?

Transformace

- `offset(r|delta,chamfer)`
- `hull()`
- `minkowski()`

Offset

r = x
chamfer = false

delta = x
chamfer = false

delta = x
chamfer = true

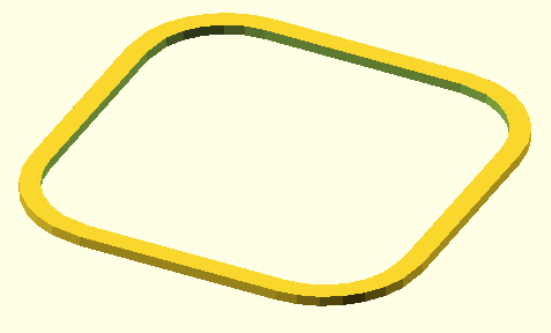
r = -x
chamfer = false

delta = -x
chamfer = false

delta = -x
chamfer = true

Offset

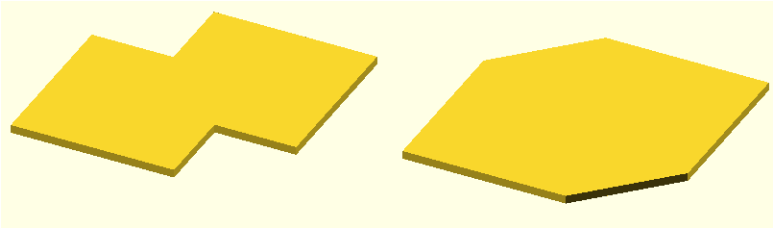
Vytvořte obdobný tvar.



Hull

Example

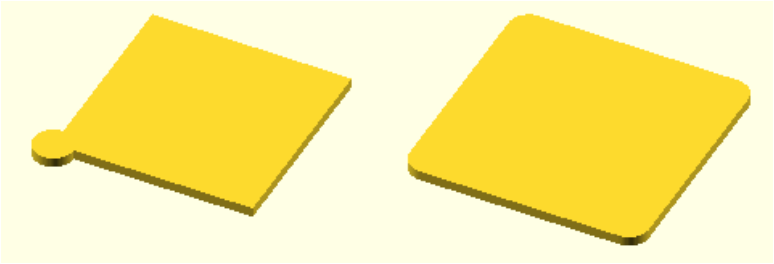
```
hull(){
   square(20);
   translate([10,10,0]) square(20);
}
```



Minkowski

Example

```
minkowski(){
   square(20);
   circle(2);
}
```



Linear extrude

- `linear_extrude(height) object();`

Linear extrude

Example

```
linear_extrude(height=20)
   difference(){
      square([12,3],center=true);
      circle(2);
   }
```

Linear extrude

Example

```
linear_extrude(height=20)
   rotate([45,0,0])
      circle(10);
```

Linear extrude - scale

- `linear_extrude(height, scale)`

Example

```
linear_extrude(height=20,scale=0)
   square(12,center=true);
```

Linear extrude - twist

- `linear_extrude(height, twist)`

Example

Zkuste různé úhly : 45, 90, 270, 360, 720

```
linear_extrude(height=20,twist=45)
   square(12,center=true);
```

Rotate extrude

- `rotate_extrude(angle, $fn) object();`

Example

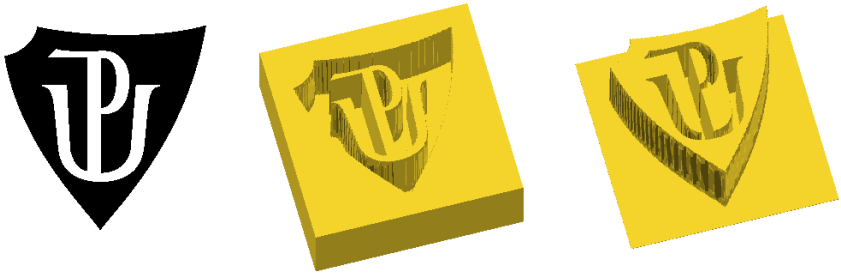
```
rotate_extrude($fn=100)
   translate([2,0,0])
      circle(1);
```

Vytvořte kouli pomocí `rotate_extrude()`.

Surface

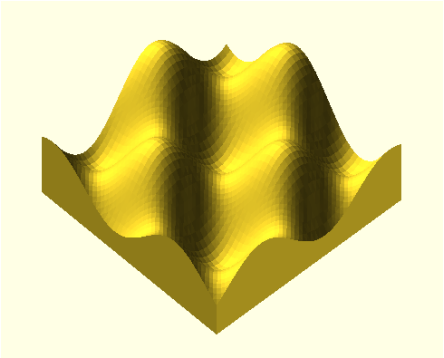
- `surface()`
- `file`
- `center`
- `invert`
- `convexity`

Surface

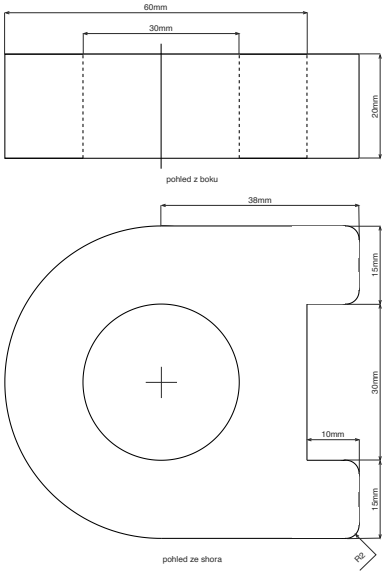


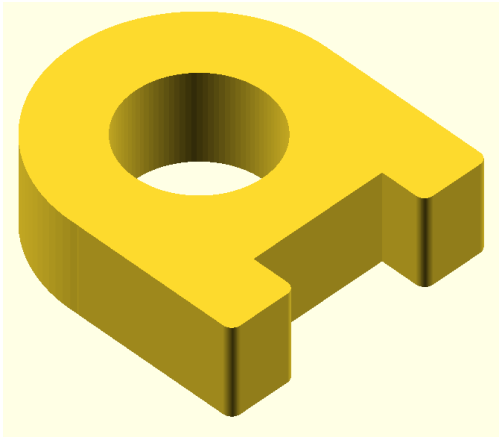
Surface

```
surf = (sin(1:0.2:10)' * cos(1:0.2:10)) * 10;
```



OpenSCAD - Příklad





2D projekce

- `projection()`
- `cut` false
- .dxf, .svg