

# Parametrické modelování

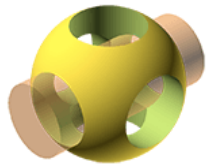
KMI/3DT 3D tisk

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OpenSCAD

<https://www.openscad.org>



- `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)`



- `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 = [ ... ])`



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



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



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



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



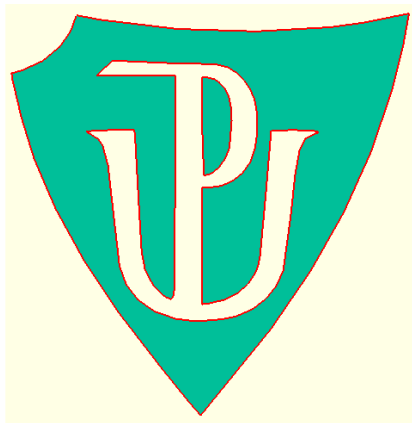


- Jak vytvořit pravidelný šestiúhelník?

# Import



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



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

## Example

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



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



- Jak vytvořit elipsu?



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



$r = x$   
chamfer = false



$\text{delta} = x$   
chamfer = false



$\text{delta} = x$   
chamfer = true



$r = -x$   
chamfer = false

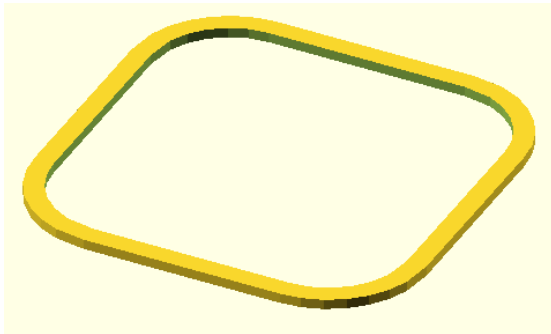


$\text{delta} = -x$   
chamfer = false



$\text{delta} = -x$   
chamfer = true

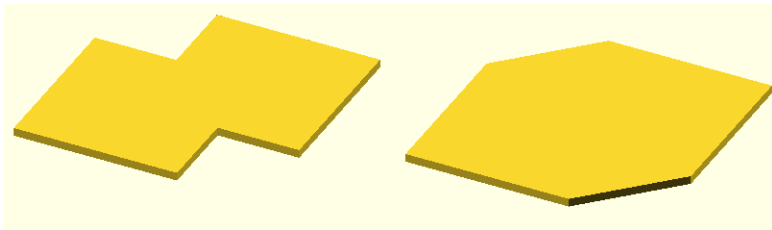
Vytvořte obdobný tvar.





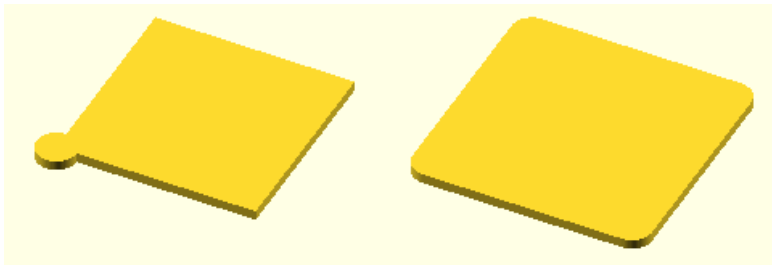
## Example

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



## Example

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



# Linear extrude



- `linear_extrude(height) object();`





## Example

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



## Example

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



- `linear_extrude`(height, scale)

## Example

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



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

## Example

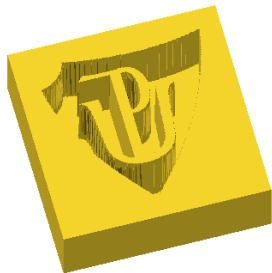
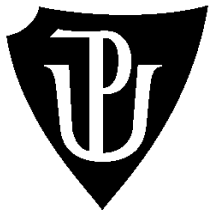
```
rotate_extrude($fn=100)
  translate([2,0,0])
    circle(1);
```

Vytvořte kouli pomocí `rotate_extrude()`.

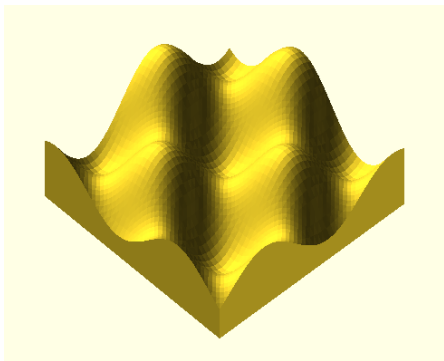


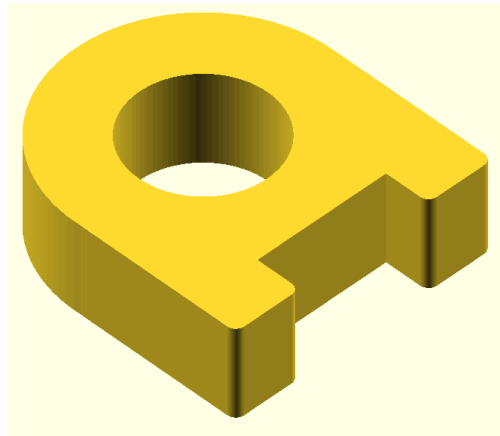
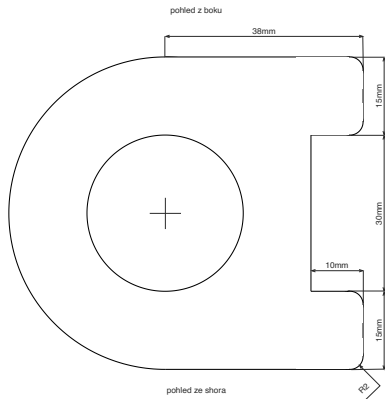
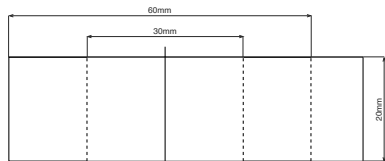


- `surface()`
- `file`
- `center`
- `invert`
- `convexity`



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





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

