example009.scad

Editor

STL

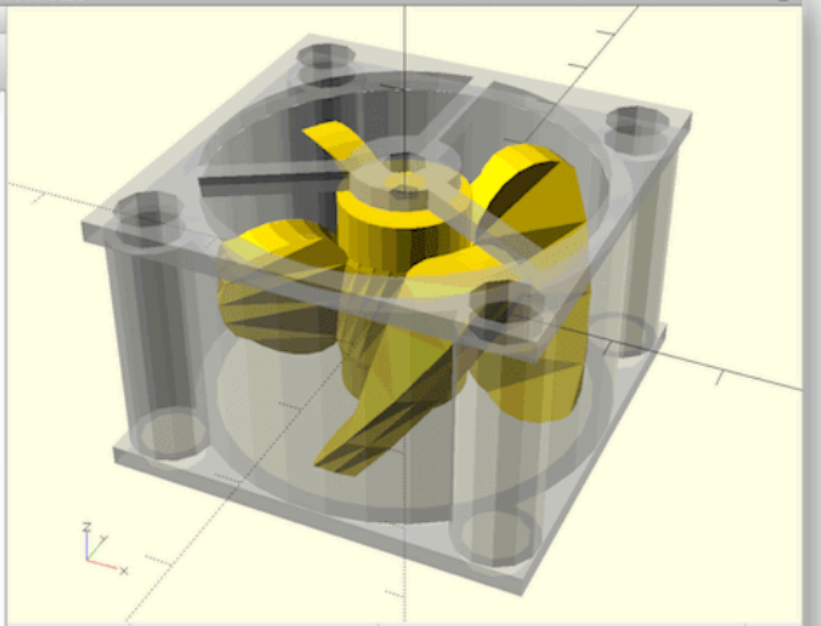
```
bodywidth = dxf_dim(file = "example009.dxf", name = "bodywidth");
fanwidth = dxf_dim(file = "example009.dxf", name = "fanwidth");
platewidth = dxf_dim(file = "example009.dxf", name = "platewidth");
fan_side_center = dxf_cross(file = "example009.dxf", layer = "fan_side_center");
fanrot = dxf_dim(file = "example009.dxf", name = "fanrot");

% linear_extrude(height = bodywidth, center = true, convexity = 10)
  import(file = "example009.dxf", layer = "body");

% for (z = [+(bodywidth/2 + platewidth/2),
    -(bodywidth/2 + platewidth/2)]) {
  translate([0, 0, z])
    linear_extrude(height = platewidth, center = true, convexity = 10)
      import(file = "example009.dxf", layer = "plate");
}

intersection() {
  linear_extrude(height = fanwidth, center = true, convexity = 10, twist = -fanrot)
    import(file = "example009.dxf", layer = "fan_top");

  // NB! We have to use the deprecated module here since the "fan_side"
  // layer contains an open polyline, which is not yet supported
  // by the import() module.
  rotate_extrude(file = "example009.dxf", layer = "fan_side",
                 origin = fan_side_center, convexity = 10);
}
```



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

OpenSCAD 2015.03