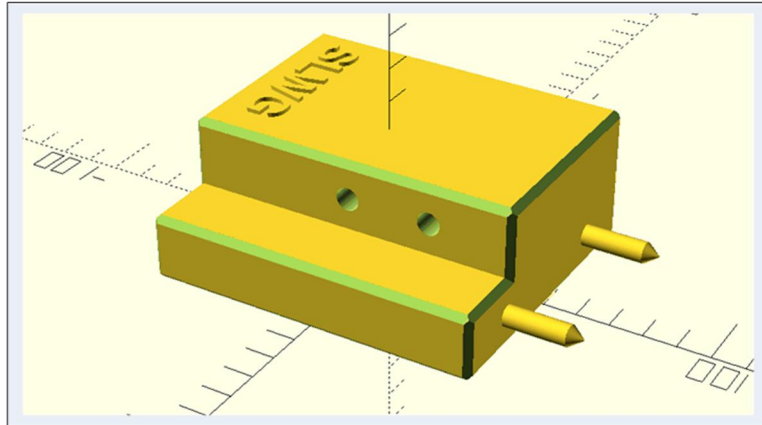


3D PRINTED WOODWORKING AIDS

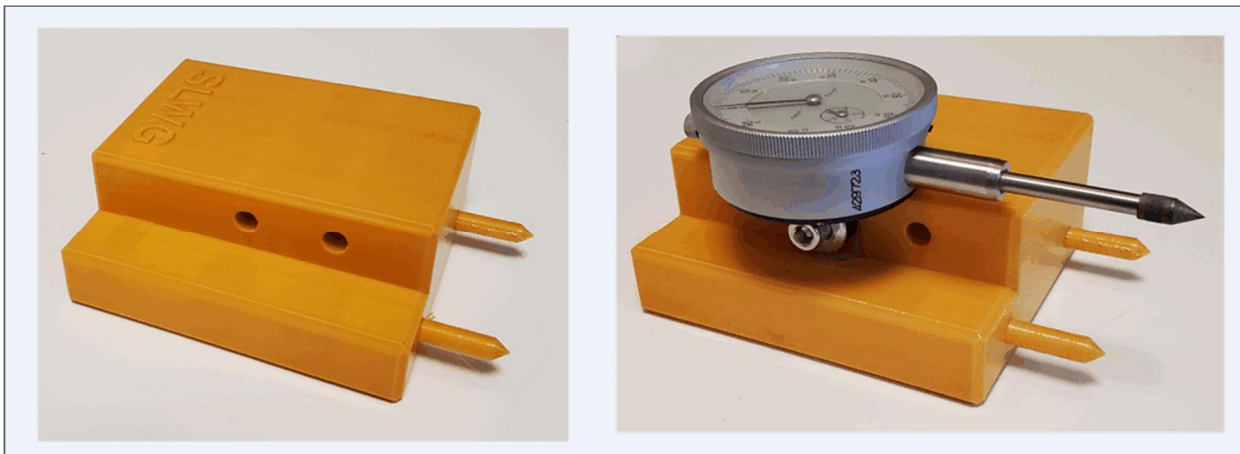
Don Snyder

Sept. 2020

Below is a drawing of a woodworking aid I designed in August 2020 for 3D printing. With a dial indicator mounted on it, the aid is used to set saw blades accurately and repeatably to 90° from the saw's surface [1]. The aid can also be used to set fences on table saws and jointers to 90° . Setting 45° and other angles is also possible.



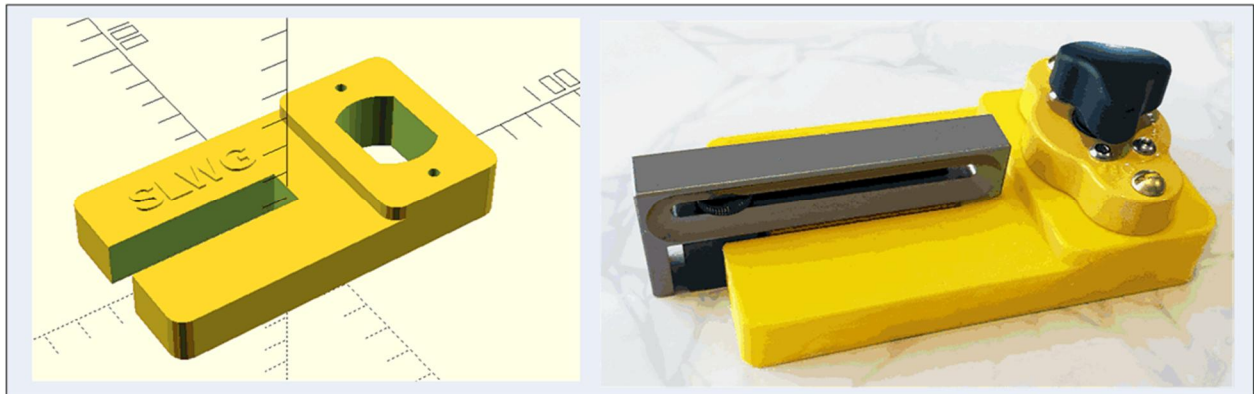
The picture below is of the 3D printed aid without and with a dial indicator mounted on it.



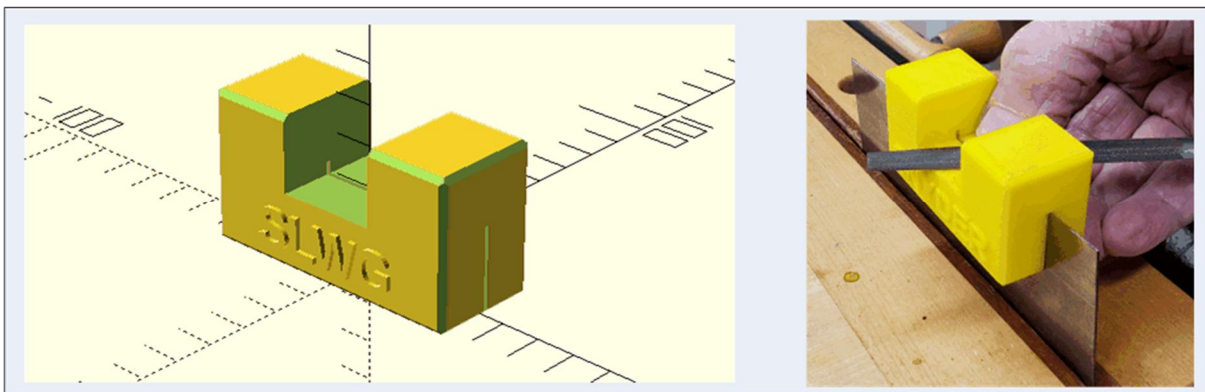
This aid was designed using OpenScad, which is readily available, free software for doing 3D designs and producing STL files used by 3D printers [2]. Minimal coding skills are needed to use OpenScad, but it helps to remember a little about the order of doing operations with vectors and matrices. Using this guide to set the angle of a saw blade or fence requires presetting the dial indicator to zero at the desired angle. 3D printed angle guides can be used for this calibration. Shown below are three guides I designed for 45° , 22.5° (from vertical) and 90° angles using OpenScad and then 3D printed. Guides for other angles can be made without requiring any additional OpenScad coding; all that is needed is to set the angle parameter in the existing code. This kind of parameterization is one of the attractive features of OpenScad. There are two mounting holes for the dial indicator. The hole near the center of the aid is for mounting the indicator for blade angles near 90° , and the second hole is for use mounting the indicator for blades tilted near 45° when the plunger of the indicator needs to reach the more distant blade surface.



Two other woodworking aids I designed using OpenScad and then 3D printed are shown below. One is made to hold the KM-1 Kerfmaker device of Bridge City Tools, which is used for making accurate kerfs, grooves, dados and cross-lap joints [3, 4]. Shown below is an OpenScad drawing of the Kerfmaker holder and a printed holder with the KM-1 Kerfmaker and a magnetic switch in the holder; the magnetic switch permits the holder to be placed in position on a saw's surface and then firmly keeps it in place while using the Kerfmaker to cut accurate kerfs.



Another woodworking aid designed using OpenScad is used to simplify sharpening rectangular and curved scrapers using Peter Galbert's method [5].



3D printers can be used to make aids for woodworking, and OpenScad is a free software program that can be useful for accomplishing the design of the aids.

1. <https://www.bing.com/videos/search?q=dial+indicator+sawblade+angle&docid=607998065924834424&mid=7B0D688152BEE2E766827B0D688152BEE2E76682&view=detail&FORM=VIRE>
2. <http://www.openscad.org/>
3. <https://bridgecitytools.com/products/km-2v2-kerfmaker-pro>
4. <https://www.rockler.com/bridge-city-tools-km-1-kerfmaker>
5. Peter Galbert, "Sharpen and use a curved card scraper," *Fine Woodworking*, pp. 20-25, Nov/Dec 2018.

Appendix 1. OpenScad procedure for the dial-indicator mount

/*****

DIAL-INDICATOR MOUNT

Use: aid for using a dial indicator to set a saw blade angle to 90° or 45°

OpenScad procedure written by Don Snyder, 31AUG2020 St. Louis Woodworkers Guild

Notes:

1. "text_on.scad" is used. This is a library of OpenScad modules for putting text on 3D shapes that is available from GitHub. See "https://github.com/brodykenrick/text_on_OpenSCAD"
2. "Chamfer.scad" is used. This is a library to create primitives with 45° chamfers in OpenSCAD that is available from GitHub. See "<https://github.com/SebiTimeWaster/Chamfers-for-OpenSCAD/releases>"
3. Base units are millimeters for length and degrees for angles.

*****/

```
include <text_on/text_on.scad>;
include <Chamfer.scad>;
//fixed parameters:
in2mm = 25.4; //for converting inches to millimeters
//dial-indicator parameters:
diameter = 2.08*in2mm;
tab_thickness = 0.2*in2mm;
//selectable parameters:
BaseWidth = 3.5*in2mm; //y width of base
BaseLength = 4.00*in2mm; //x length of the base
BaseHeight = (13/16)*in2mm; //thickness of the base
StepWidth = (7/8)*in2mm;
MountWidth = BaseWidth-StepWidth;
MountLength = BaseLength;
MountHeight = BaseHeight;
ChamferWidth = 0.08*in2mm;
ProbeRadius = (1/8)*in2mm;
ProbeLength = (9/8)*in2mm;
BoltDiameter = 0.275*in2mm;
HexNutDiameter = 0.56*in2mm;
LetterHeight = 1;
//start .....
translate([0,BaseHeight,BaseLength/2])
```

```

rotate([90,0,0]){
MakeBase(BaseWidth,BaseLength,BaseHeight);
difference(){
MakeMount(MountWidth,MountLength,MountHeight,BaseWidth,ChamferWidth);
translate([BaseWidth/2-(7/8)*in2mm,0,0]);
MakeBoltHole(BoltDiameter,HexNutDiameter,MountWidth,BaseWidth); //90° hole
translate([0,1*in2mm,0])
MakeBoltHole(BoltDiameter,HexNutDiameter,MountWidth,BaseWidth); //45° hole
}
translate([BaseWidth/2-(0.75)*in2mm,0,0])
MakeProbe(ProbeLength,ProbeRadius,BaseWidth,BaseHeight);
  translate([-BaseWidth/2+(0.75)*in2mm,0,0])
MakeProbe(ProbeLength,ProbeRadius,BaseWidth,BaseHeight);
translate([-BaseWidth/2+8.5,-BaseLength/2+9,BaseHeight+MountHeight])
  text_extrude("SLWG",font="ariel:style=bold",
  extrusion_height=1.2*LetterHeight,size=12,spacing=1.02,
  center=false);
}
//end .....
module MakeBase(xwidth,ylength,zheight){
  translate([-xwidth/2,-ylength/2,0])
  chamferCube(xwidth,ylength,zheight,ChamferWidth,[1,1,1,1]);
}
module MakeProbe(PL,PR,BW,BH){
  translate([0,BW/2+PL*(7/8),BH/2])
  rotate([90,0,0])
  hull(){
    cylinder(h=PL,r=PR,$fn=100);
    cylinder(h=PL/3.5,r1=0.2
    ,r2=PR,center=true,$fn=100);
  }
}
module MakeMount(xwidth,ylength,zheight,BW,CW){
  translate([-BW/2,-ylength/2,zheight-2*CW])
  chamferCube(xwidth,ylength,zheight+2*CW,CW,[1,1,1,1]);
}
module MakeBoltHole(BD,HND,MW,BW){
  translate([MW*0.1,0,(5.5+MW-HND*cos(30))/2])
  rotate([0,-90,0])
  union(){
    rotate([0,0,30])
    cylinder(MW*0.8,d=HND,$fn=6);
    translate([0,0,-MW*0.25])
    cylinder(MW*0.25,d=BD,$fn=100);
  }
}
}

```

Appendix 2. OpenScad procedure for angle-reference guides for use with the dial-indicator mount

/*

SAW-TILT GUIDE

(for use with a dial indicator)

OpenScad procedure

written by Donald Snyder May 2019 St. Louis Woodworkers Guild

1. "text_on.scad" is used. This is a library of OpenScad modules for putting text on 3D shapes that is available from GitHub. See "https://github.com/brodykenrick/text_on_OpenSCAD"
2. "Chamfer.scad" is used. This is a library to create primitives with 45° chamfers in OpenSCAD that is available from GitHub. See "<https://github.com/SebiTimeWaster/Chamfers-for-OpenSCAD/releases>"
4. Base units are millimeters for length and degrees for angles.

*/

```
include <text_on/text_on.scad>;
```

```
include <Chamfer.scad>;
```

```
in2mm = 25.4; //for converting inches to millimeters
```

```
BWidth = 2.75*in2mm; //y width of base
```

```
BLength = 3.0*in2mm; //x length of the base
```

```
BHeight = 0.5*in2mm; //thickness of the base
```

```
Angle = 0; //TiltAngle, degrees, measured from vertical
```

```
RLength = 2.5*in2mm; //x length of reference piece
```

```
RWidth = 0.5*in2mm; //height if reference
```

```
LetterHeight = BHeight/20;
```

```
//start .....
```

```
//position for printing
```

```
translate([0,-RLength/2,0])
```

```
rotate([-90,0,0]){
```

```
translate([0,-BWidth/2,0]){
```

```
//done position for printing
```

```
MakePart(BWidth,BLength,BHeight,Angle,RLength,RWidth);
```

```
//MakeReferencePart(RWidth,RLength,RHeight,RAngle);
```

```
//MakeShoulderPart(SWidth,SLength,SHeight,SAngle);
```

```
//add text front
```

```
//name=str("DLS ", Angle);
```

```
//name = "DLS 90°";
```

```
name = " SLWG 90°";
```

```
//name = "SNYDER";
```

```
translate([-BLength/1.75,-BWidth/2-0.1,BHeight/2])
```

```

rotate([90,0,0])
text_extrude(name,font="ariel:style=bold",
  extrusion_height=2*LetterHeight,size=9,spacing=1.02,
  center=true);
}
}
//..... end
module MakePart(BW,BL,BH,A,RL,RW)
{
  translate([-BL,0,0])
  rotate([90,0,0])
  linear_extrude(height=BW,center=true)
  polygon([
    [0,0],
    [BL,0],
    [BL-RL*sin(A),RL*cos(A)],
    [BL-RL*sin(A)-RW/cos(A),RL*cos(A)],
    [BL-BH*tan(A)-RW/cos(A),BH],
    [0,BH]]);
}

```

Appendix 3. OpenScad procedure for a Kerfmaker holder
 /*****

Holder for
 Bridge City kerfmaker and a magnetic hold-down switch
 Don Snyder 7 April 2018
 St. Louis Woodworkers Guild

"text_on.scad" is used. This is a library of OpenScad modules for putting text on 3D shapes that is available from GitHub. See "https://github.com/brodykenrick/text_on_OpenSCAD"
 "morphology.scad" is used. This is a library of OpenScad modules for smoothing edges in various ways. It is available from GitHub. See "<https://github.com/openscad/scad-utils/blob/master/morphology.scad>"
 *****/

```

use <text_on/text_on.scad>;
use <morphology.scad>
LetterHeight=2;
difference()
{
  difference()
  {
    difference()
    {
      {union()
      {
        linear_extrude(14.4)

```

```

    rounding(5)
    square([110,60],center=true);
    translate([35,0,13.5])
    linear_extrude(5.9)
    rounding(5)
    square([40,60],center=true);
}
}
{translate([35,0,0])
intersection()
{
    cube([21.5,35,50],center=true);
    cylinder(h=50,d=30.5,center=true,$fa=1);
}
}
}

union()
{
    for(y=[-23,23])
    {
        translate([35,y,10])
        cylinder(r=1.73,h=20,$fn=30);}
        translate([-27.6,0,6])
        cube([55,13,20],center=true);
    }
}

//make a logo on bottom of plate
rotate([180,0,90])
translate([0,11.5,0])

text_extrude("DLS",font="abdiel:style=bold",extrusion_height=1.3*LetterHeight,size=9,spacing=1.25,center=true);
}

//add text on top arm
name = "SLWG"; xlocation = -45;
translate([xlocation,13,13])
text_extrude(name,font="ariel:style=bold",
    extrusion_height=1.15*LetterHeight,size=10,spacing=1.02,
    center=false);

```

Appendix 4. OpenScad procedure for a card-scraper holder

/*****

CARD-SCRAPER HOLDER

(for sharpening a card scraper)

OpenScad procedure written by Donald Snyder

March 2019 St. Louis Woodworkers Guild

1. The holder and its use are based on:
Peter Galbert, "Sharpen and use a curved scraper," Fine Woodworking Magazine, No. 271, pp. 20-25, Nov/Dec 2019.
2. "text_on.scad" is used. This is a library of OpenScad modules for putting text on 3D shapes that is available from GitHub. See "https://github.com/brodykenrick/text_on_OpenSCAD"
3. "Chamfer.scad" is used. This is a library to create primitives with 45° chamfers in OpenSCAD that is available from GitHub. See "<https://github.com/SebiTimeWaster/Chamfers-for-OpenSCAD/releases>"
4. Base units are millimeters for length and degrees for angles.

*****/

```
include <text_on/text_on.scad>;
```

```
include <Chamfer.scad>;
```

```
in2mm = 25.4; //for converting inches to millimeters
```

```
xlength = 2.75*in2mm;
```

```
ylength = 1.125*in2mm;
```

```
zlength = 1.5*in2mm;
```

```
kerfwidth = 1.06; //mm
```

```
LetterHeight = zlength/3;
```

```
//start .....
```

```
difference(){
```

```
difference(){
```

```
translate([-xlength/2, -ylength/2, 0]) chamferCube(xlength, ylength, zlength, 1.5, [1, 1, 1, 1]);
```

```
translate([-0.5*in2mm, -ylength/2-0.1, zlength/2]) cube([1*in2mm, ylength+0.5, zlength/2+0.1]);
```

```
}
```

```
//make kerf for scraper
```

```
translate([-xlength/2-0.1, -kerfwidth/2, -0.1]) cube([xlength+0.2, kerfwidth, zlength/2+0.125*in2mm]);
```

```
}
```

```
//add text front
```

```
name = "SLWG";
```

```
translate([0, -ylength/2-0.01, zlength/4])
```

```
rotate([90, 0, 0])
```

```
text_extrude(name, font="arial:style=bold",
```

```
extrusion_height=0.18*LetterHeight, size=10, spacing=1.02,
```

```
center=true);
```

```
//..... end
```