# The gcodepreview OpenSCAD library*

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**Abstract**

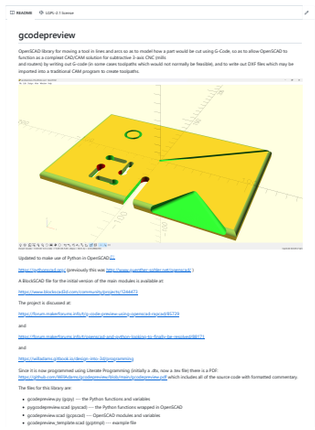
The gcodepreview library allows using OpenPythonSCAD to move a tool in lines and arcs and output dxf and G-code files so as to work as a CAD/CAM program for CNC.

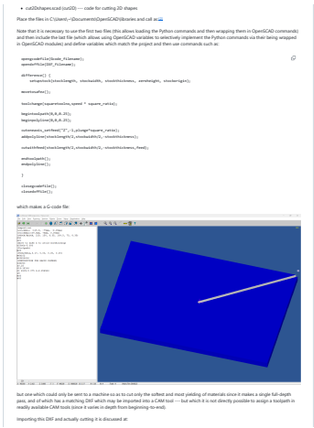
## Contents



*This file (gcodepreview) has version number v0.61, last revised 2024/09/08.

# 1 readme.md





```
 1 rdme # gcodepreview
 2 rdme
 3 rdme OpenSCAD library for moving a tool in lines and arcs
 4 rdme so as to model how a part would be cut using G-Code,
 5 rdme so as to allow OpenSCAD to function as a compleat
 6 rdme CAD/CAM solution for subtractive 3-axis CNC (mills
 7 rdme and routers) by writing out G-code (in some cases
 8 rdme toolpaths which would not normally be feasible),
 9 rdme and to write out DXF files which may be imported
10 rdme into a traditional CAM program to create toolpaths.
11 rdme
12 rdme ![OpenSCAD Cut Joinery Module](https://raw.githubusercontent.com/
       WillAdams/gcodepreview/main/gcodepreview_unittests.png?raw=true)
13 rdme
14 rdme Updated to make use of Python in OpenSCAD:[^rapcad]
15 rdme
16 rdme [^rapcad]: Previous versions had used RapCAD, so as to take
       advantage of the writeln command, which has since been re-
       written in Python.
17 rdme
18 rdme https://pythonscad.org/ (previously this was http://www.guenther-
       sohler.net/openscad/ )
19 rdme
20 rdme A BlockSCAD file for the initial version of the
21 rdme main modules is available at:
22 rdme
23 rdme https://www.blockscad3d.com/community/projects/1244473
24 rdme
25 rdme The project is discussed at:
26 rdme
27 rdme https://forum.makerforums.info/t/g-code-preview-using-openscad-
       rapcad/85729
28 rdme
29 rdme and
30 rdme
31 rdme https://forum.makerforums.info/t/openscad-and-python-looking-to-
       finally-be-resolved/88171
32 rdme
33 rdme and
34 rdme
35 rdme https://willadams.gitbook.io/design-into-3d/programming
36 rdme
37 rdme Since it is now programmed using Literate Programming
38 rdme (initially a .dtx, now a .tex file) there is a PDF:
39 rdme https://github.com/WillAdams/gcodepreview/blob/main/gcodepreview.
       pdf
40 rdme which includes all of the source code with formatted
41 rdme commentary.
42 rdme
43 rdme The files for this library are:
44 rdme
45 rdme  - gcodepreview.py (gcpy) --- the Python functions and variables
46 rdme  - pygcodepreview.scad (pyscad) --- the Python functions wrapped in
          OpenSCAD
47 rdme  - gcodepreview.scad (gcpscad) --- OpenSCAD modules and variables
48 rdme  - gcodepreview_template.scad (gcptmpl) --- example file
49 rdme  - cut2Dshapes.scad (cut2D) --- code for cutting 2D shapes
50 rdme
```

```
51 rdme Place the files in C:\Users\\\~\Documents\OpenSCAD\libraries and
        call as:[^libraries]
52 rdme
53 rdme [^libraries]: C:\Users\\\~\Documents\RapCAD\libraries is deprecated
         since RapCAD is no longer needed since Python is now used for
        writing out files)
54 rdme
55 rdme     use <gcodepreview.py>;
56 rdme     use <pygcodepreview.scad>;
57 rdme     include <gcodepreview.scad>;
58 rdme
59 rdme Note that it is necessary to use the first two files
60 rdme (this allows loading the Python commands and then
61 rdme wrapping them in OpenSCAD commands) and then include
62 rdme the last file (which allows using OpenSCAD variables
63 rdme to selectively implement the Python commands via their
64 rdme being wrapped in OpenSCAD modules) and define
65 rdme variables which match the project and then use
66 rdme commands such as:
67 rdme
68 rdme     opengcodefile(Gcode_filename);
69 rdme     opendxffile(DXF_filename);
70 rdme
71 rdme     difference() {
72 rdme         setupstock(stocklength, stockwidth, stockthickness,
                zeroheight, stockorigin);
73 rdme
74 rdme     movetosafez();
75 rdme
76 rdme     toolchange(squaretoolno,speed * square_ratio);
77 rdme
78 rdme     begintoolpath(0,0,0.25);
79 rdme     beginpolyline(0,0,0.25);
80 rdme
81 rdme     cutoneaxis_setfeed("Z",-1,plunge*square_ratio);
82 rdme     addpolyline(stocklength/2,stockwidth/2,-stockthickness);
83 rdme
84 rdme     cutwithfeed(stocklength/2,stockwidth/2,-stockthickness,feed);
85 rdme
86 rdme     endtoolpath();
87 rdme     endpolyline();
88 rdme
89 rdme     }
90 rdme
91 rdme     closegcodefile();
92 rdme     closedxffile();
93 rdme
94 rdme which makes a G-code file:
95 rdme
96 rdme ![OpenSCAD template G-code file](https://raw.githubusercontent.com/
        WillAdams/gcodepreview/main/gcodepreview_template.png?raw=true)
97 rdme
98 rdme but one which could only be sent to a machine so as to
99 rdme cut only the softest and most yielding of materials
100 rdme since it makes a single full-depth pass, and of which
101 rdme has a matching DXF which may be imported into a
102 rdme CAM tool --- but which it is not directly possible
103 rdme to assign a toolpath in readily available CAM tools
104 rdme (since it varies in depth from beginning-to-end).
105 rdme
106 rdme Importing this DXF and actually cutting it
107 rdme is discussed at:
108 rdme
109 rdme https://forum.makerforums.info/t/rewriting-gcodepreview-with-python
        /88617/14
110 rdme
111 rdme Tool numbers match those of tooling sold by Carbide 3D
112 rdme (ob. discl., I work for them).
113 rdme
114 rdme Comments are included in the G-code to match those
115 rdme expected by CutViewer.
116 rdme
117 rdme A complete example file is: gcodepreview_template.scad
118 rdme and another example is openscad_gcodepreview_cutjoinery.tres.scad
119 rdme which is made from an OpenSCAD Graph Editor file:
120 rdme
121 rdme ![OpenSCAD Graph Editor Cut Joinery File](https://raw.
        githubusercontent.com/WillAdams/gcodepreview/main/
```

```
          OSGE_cutjoinery.png?raw=true)
rdme
rdme Version 0.1 supports setting up stock, origin, rapid
rdme positioning, making cuts, and writing out matching
rdme G-code, and creating a DXF with polylines.
rdme
rdme Added features since initial upload:
rdme
rdme  - endpolyline(); --- this command allows ending one polyline so as
            to allow multiple lines in a DXF
rdme  - separate dxf files are written out for each tool where tool is
            ball/square/V and small/large (10/31/23)
rdme  - re-writing as a Literate Program using the LaTeX package docmfp
            (begun 4/12/24)
rdme  - support for additional tooling shapes such as dovetail and
            keyhole tools
rdme
rdme Version 0.2 adds support for arcs
rdme
rdme  - DXF: support for arcs (which may be used to make circles)
            (6/1/24)
rdme  - Specialty toolpaths such as Keyhole which may be used for
            dovetail as well as keyhole cutters
rdme
rdme Version 0.3
rdme
rdme  - Support for curves along the 3rd dimension
rdme  - support for roundover tooling
rdme
rdme Version 0.4
rdme
rdme  - Rewrite using literati documentclass, suppression of SVG code
rdme  - dxfrectangle (without G-code support)
rdme
rdme Version 0.5
rdme
rdme  - more shapes
rdme  - consolidate rectangles, arcs, and circles in gcodepreview.scad
rdme
rdme Version 0.6
rdme
rdme  - notes on modules
rdme  - change file for setupstock
rdme
rdme Version 0.7
rdme
rdme  - reduce usage of tool numbers
rdme  - validate all code so that it runs without errors from sample
            file
rdme
rdme Possible future improvements:
rdme
rdme  - support for additional tooling shapes such as tapered ball-nose
            tools or lollipop cutters or thread-cutting tools
rdme  - G-code: support for G2/G3 arcs and circles
rdme  - G-code: import external tool libraries and feeds and speeds from
            JSON or CSV files ---
rdme  - general coding improvements --- current coding style is quite
            prosaic
rdme  - additional generalized modules for cutting out various shapes/
            geometries
rdme
rdme Note for G-code generation that it is up to the user
rdme to implement Depth per Pass so as to not take a
rdme single full-depth pass. Working from a DXF of course
rdme allows one to off-load such considerations to a
rdme specialized CAM tool.
rdme
rdme Deprecated feature:
rdme
rdme  - exporting SVGs --- while this was begun, it turns out that these
            would be written out upside down due to coordinate system
            differences between OpenSCAD/DXFs and SVGs requiring managing
            the inversion of the coordinate system (it is possible that
            METAPOST, which shares the same orientation and which can write
            out SVGs will be used instead for future versions)
```

# 2 gcodepreview

This library for OpenPythonSCAD works by using Python code as a back-end so as to persistently store and access variables, and to write out files while both modeling the motion of a 3-axis CNC machine and if desired, writing out DXF and/or G-code files (as opposed to the normal technique of rendering to a 3D model and writing out an STL). Doing so requires a total of three files:

- A Python file: gcodepreview.py (`gcpy`) — this will have variables in the traditional sense which may be used for tracking machine position and so forth
- An OpenSCAD file: pygcodepreview.scad (`pyscad`) — which wraps the Python code in OpenSCAD
- An OpenSCAD file: gcodepreview.scad (`gcpscad`) — which uses the other two files and which is `include`d allowing it to access OpenSCAD variables for branching

Note that this architecture requires that many modules are essentially "Dispatchers" which pass information from one aspect of the environment to another.

## 2.1 gcodepreviewtemplate.scad

The various commands are shown all together in a template so as to provide examples of usage, and to ensure that the various files are used/included as necessary, all variables are set up with the correct names, and that files are opened before being written to, and that each is closed at the end.

Note that while the file seems overly verbose, it specifically incorporates variables for each tool shape, possibly in two different sizes, and a feed rate parameter or ratio for each, which may be used (by setting a tool #) or ignored (by leaving the variable at zero (0).

It should be that this section is all the documentation which some users will need. (and arguably is still too much). The balance of the document after this section shows all the code and implementation details.

```
1 gcptmpl //!OpenSCAD
2 gcptmpl
3 gcptmpl use <gcodepreview.py>;
4 gcptmpl use <pygcodepreview.scad>;
5 gcptmpl include <gcodepreview.scad>;
6 gcptmpl
7 gcptmpl $fa = 2;
8 gcptmpl $fs = 0.125;
9 gcptmpl
10 gcptmpl /* [Export] */
11 gcptmpl Base_filename = "export";
12 gcptmpl /* [Export] */
13 gcptmpl generatedxf = true;
14 gcptmpl /* [Export] */
15 gcptmpl generategcode = true;
16 gcptmpl ///* [Export] */
17 gcptmpl //generatesvg = false;
18 gcptmpl
19 gcptmpl /* [CAM] */
20 gcptmpl toolradius = 1.5875;
21 gcptmpl /* [CAM] */
22 gcptmpl large_ball_tool_no = 0; // [0:0,111:111,101:101,202:202]
23 gcptmpl /* [CAM] */
24 gcptmpl large_square_tool_no = 0; // [0:0,112:112,102:102,201:201]
25 gcptmpl /* [CAM] */
26 gcptmpl large_V_tool_no = 0; // [0:0,301:301,690:690]
27 gcptmpl /* [CAM] */
28 gcptmpl small_ball_tool_no = 0; // [0:0,121:121,111:111,101:101]
29 gcptmpl /* [CAM] */
30 gcptmpl small_square_tool_no = 102; // [0:0,122:122,112:112,102:102]
31 gcptmpl /* [CAM] */
32 gcptmpl small_V_tool_no = 0; // [0:0,390:390,301:301]
33 gcptmpl /* [CAM] */
34 gcptmpl KH_tool_no = 0; // [0:0,374:374,375:375,376:376,378]
35 gcptmpl /* [CAM] */
36 gcptmpl DT_tool_no = 0; // [0:0,814:814]
37 gcptmpl /* [CAM] */
38 gcptmpl Roundover_tool_no = 56125; // [56125:56125, 56142:56142,312:312,
          1570:1570]
39 gcptmpl /* [CAM] */
40 gcptmpl MISC_tool_no = 0; //
41 gcptmpl
42 gcptmpl /* [Feeds and Speeds] */
43 gcptmpl plunge = 100;
44 gcptmpl /* [Feeds and Speeds] */
```

```
45 gcptmpl feed = 400;
46 gcptmpl /* [Feeds and Speeds] */
47 gcptmpl speed = 16000;
48 gcptmpl /* [Feeds and Speeds] */
49 gcptmpl small_square_ratio = 0.75; // [0.25:2]
50 gcptmpl /* [Feeds and Speeds] */
51 gcptmpl small_ball_ratio = 0.75; // [0.25:2]
52 gcptmpl /* [Feeds and Speeds] */
53 gcptmpl large_ball_ratio = 1.0; // [0.25:2]
54 gcptmpl /* [Feeds and Speeds] */
55 gcptmpl small_V_ratio = 0.625; // [0.25:2]
56 gcptmpl /* [Feeds and Speeds] */
57 gcptmpl large_V_ratio = 0.875; // [0.25:2]
58 gcptmpl /* [Feeds and Speeds] */
59 gcptmpl KH_ratio = 0.75; // [0.25:2]
60 gcptmpl /* [Feeds and Speeds] */
61 gcptmpl DT_ratio = 0.75; // [0.25:2]
62 gcptmpl /* [Feeds and Speeds] */
63 gcptmpl RO_ratio = 0.5; // [0.25:2]
64 gcptmpl /* [Feeds and Speeds] */
65 gcptmpl MISC_ratio = 0.5; // [0.25:2]
66 gcptmpl
67 gcptmpl /* [Stock] */
68 gcptmpl stocklength = 219;
69 gcptmpl /* [Stock] */
70 gcptmpl stockwidth = 150;
71 gcptmpl /* [Stock] */
72 gcptmpl stockthickness = 8.35;
73 gcptmpl /* [Stock] */
74 gcptmpl zeroheight = "Top"; // [Top, Bottom]
75 gcptmpl /* [Stock] */
76 gcptmpl stockorigin = "Center"; // [Lower-Left, Center-Left, Top-Left,
        Center]
77 gcptmpl /* [Stock] */
78 gcptmpl retractheight = 9;
79 gcptmpl
80 gcptmpl filename_gcode = str(Base_filename, ".nc");
81 gcptmpl filename_dxf = str(Base_filename);
82 gcptmpl
83 gcptmpl opengcodefile(filename_gcode);
84 gcptmpl opendxffile(filename_dxf);
85 gcptmpl
86 gcptmpl difference() {
87 gcptmpl setupstock(stocklength, stockwidth, stockthickness, zeroheight,
        stockorigin);
88 gcptmpl
89 gcptmpl movetosafez();
90 gcptmpl
91 gcptmpl toolchange(small_square_tool_no,speed * small_square_ratio);
92 gcptmpl
93 gcptmpl begintoolpath(0,0,0.25);
94 gcptmpl
95 gcptmpl cutoneaxis_setfeed("Z",0,plunge*small_square_ratio);
96 gcptmpl
97 gcptmpl cutwithfeed(stocklength/2,stockwidth/2,-stockthickness,feed);
98 gcptmpl dxfpolyline(getxpos(),getypos(),stocklength/2,stockwidth/2,
        small_square_tool_no);
99 gcptmpl
100 gcptmpl endtoolpath();
101 gcptmpl rapid(-(stocklength/4-stockwidth/16),stockwidth/4,0);
102 gcptmpl cutoneaxis_setfeed("Z",-stockthickness,plunge*small_square_ratio);
103 gcptmpl
104 gcptmpl cutarcNECCdxf(-stocklength/4, stockwidth/4+stockwidth/16, -
        stockthickness, -stocklength/4, stockwidth/4, stockwidth/16,
        small_square_tool_no);
105 gcptmpl cutarcNWCCdxf(-(stocklength/4+stockwidth/16), stockwidth/4, -
        stockthickness, -stocklength/4, stockwidth/4, stockwidth/16,
        small_square_tool_no);
106 gcptmpl cutarcSWCCdxf(-stocklength/4, stockwidth/4-stockwidth/16, -
        stockthickness, -stocklength/4, stockwidth/4, stockwidth/16,
        small_square_tool_no);
107 gcptmpl cutarcSECCdxf(-(stocklength/4-stockwidth/16), stockwidth/4, -
        stockthickness, -stocklength/4, stockwidth/4, stockwidth/16,
        small_square_tool_no);
108 gcptmpl
109 gcptmpl rapid(getxpos(),getypos(),stockthickness);
110 gcptmpl toolchange(KH_tool_no,speed * KH_ratio);
111 gcptmpl rapid(-stocklength/8,-stockwidth/4,0);
```

```
gcptmpl
gcptmpl cutkeyhole_toolpath((stockthickness), (stockthickness), "N",
    stockwidth/8, KH_tool_no);
gcptmpl rapid(getxpos(),getypos(),stockthickness);
gcptmpl rapid(-stocklength/4,-stockwidth/4,0);
gcptmpl cutkeyhole_toolpath((stockthickness), (stockthickness), "S",
    stockwidth/8, KH_tool_no);
gcptmpl rapid(getxpos(),getypos(),stockthickness);
gcptmpl rapid(-stocklength/4,-stockwidth/8,0);
gcptmpl cutkeyhole_toolpath((stockthickness), (stockthickness), "E",
    stockwidth/8, KH_tool_no);
gcptmpl rapid(getxpos(),getypos(),stockthickness);
gcptmpl rapid(-stocklength/8,-stockwidth/8*3,0);
gcptmpl cutkeyhole_toolpath((stockthickness), (stockthickness), "W",
    stockwidth/8, KH_tool_no);
gcptmpl
gcptmpl rapid(getxpos(),getypos(),stockthickness);
gcptmpl toolchange(DT_tool_no,speed * DT_ratio);
gcptmpl rapid(0,-(stockwidth/2+tool_diameter(DT_tool_no,0)),0);
gcptmpl
gcptmpl cutoneaxis_setfeed("Z",-stockthickness,plunge*DT_ratio);
gcptmpl cutwithfeed(0,-(stockwidth/4),-stockthickness,feed*DT_ratio);
gcptmpl rapid(0,-(stockwidth/2+tool_diameter(DT_tool_no,0)),-stockthickness
    );
gcptmpl
gcptmpl rapid(getxpos(),getypos(),stockthickness);
gcptmpl toolchange(Roundover_tool_no, speed * RO_ratio);
gcptmpl rapid(-(stocklength/2),-(stockwidth/2),0);
gcptmpl cutoneaxis_setfeed("Z",-4.509,plunge*RO_ratio);
gcptmpl
gcptmpl cutroundovertool(-(stocklength/2++0.507/2), -(stockwidth/2+0.507/2)
    , -4.509, stocklength/2+0.507/2, -(stockwidth/2+0.507/2),
    -4.509, 0.507/2, 4.509);
gcptmpl
gcptmpl cutroundover(stocklength/2+0.507/2, -(stockwidth/2+0.507/2),
    -4.509, stocklength/2+0.507/2, stockwidth/2+0.507/2, -4.509,
    1570);
gcptmpl cutroundover(stocklength/2+0.507/2, stockwidth/2+0.507/2, -4.509,
    -(stocklength/2+0.507/2), stockwidth/2+0.507/2, -4.509, 1570);
gcptmpl cutroundover(-(stocklength/2+0.507/2), stockwidth/2+0.507/2,
    -4.509, -(stocklength/2+0.507/2), -(stockwidth/2+0.507/2),
    -4.509, 1570);
gcptmpl
gcptmpl for (i = [0 : abs(1) : 80]) {
gcptmpl cutwithfeed(stocklength/4,-stockwidth/4,-stockthickness/4,feed);
gcptmpl cutwithfeed(stocklength/8+(stocklength/256*i),-stockwidth/2,-
    stockthickness*3/4,feed);
gcptmpl
gcptmpl }
gcptmpl
gcptmpl closegcodefile();
gcptmpl closedxffile();
```

## 2.2 Implementation

Each file will begin with a suitable comment indicating the file type and suitable notes/comments:

```
gcpy #!/usr/bin/env python
gcpy #icon "C:\Program Files\PythonSCAD\bin\openscad.exe"  --trust-
    python
gcpy #Currently tested with 2024.09.03 and Python 3.11
gcpy #gcodepreview 0.61, see gcodepreview.scad
```

```
pyscad //!OpenSCAD
pyscad
pyscad //gcodepreview 0.61, see gcodepreview.scad
```

```
gcpscad //!OpenSCAD
gcpscad
gcpscad //gcodepreview 0.61
gcpscad //
gcpscad //used via use <gcodepreview.py>;
gcpscad //          use <pygcodepreview.scad>;
gcpscad //          include <gcodepreview.scad>;
```

```
 8 gcpscad //
```

writeln The original implementation in RapSCAD used a command `writeln` — fortunately, this command is easily re-created in Python:

```
 6 gcpy def writeln(*arguments):
 7 gcpy     line_to_write = ""
 8 gcpy     for element in arguments:
 9 gcpy         line_to_write += element
10 gcpy     f.write(line_to_write)
11 gcpy     f.write("\n")
```

which command will accept a series of arguments and then write them out to a file object.

## 2.3 Position and Variables

In modeling the machine motion and G-code it will be necessary to have the machine track several variables for machine position, current tool, and depth in toolpath. This will be done using paired functions (which will set and return the matching variable) and a matching (global) variable, as well as additional functions for setting the matching variable(s).

The first such variables are for XYZ position:

mpx • `mpx`

mpy • `mpy`

mpz • `mpz`

Similarly, for some toolpaths it will be necessary to track the depth along the Z-axis as the toolpath is cut out:

tpz • `tpz`

It will further be necessary to have a variable for the current tool:

currenttool • `currenttool`

Note that the `currenttool` variable should always be used for any specification of a tool, being read in whenever a tool is to be made use of, or a parameter or aspect of the tool needs to be used in a calculation.

For each intended command it will be necessary to implement an appropriate aspect in each file. The Python file will manage the Python variables and handle things which can only be done in Python, while there will be two OpenSCAD files as noted above, one which calls the Python code (this will be `use`d), while the other will be able to access and use OpenSCAD variables, as well as implement Customizer options (this will be `include`d).

## 2.4 Output files

### 2.4.1 G-code and modules and commands

Each module/command will write out certain G-code commands:

| Command/Module | G-code |
|---|---|
| `opengcodefile(...); setupstock(...)` | `(export.nc)`<br>`(stockMin:  -109.5, -75mm, -8.35mm)`<br>`(stockMax:109.5mm, 75mm, 0.00mm)`<br>`(STOCK/BLOCK, 219, 150, 8.35, 109.5, 75, 8.35)`<br>`G90`<br>`G21` |
| `movetosafez()` | `(Move to safe Z to avoid workholding)`<br>`G53G0Z-5.000` |
| `toolchange(...);` | `(TOOL/MILL,3.17, 0.00, 0.00, 0.00)`<br>`M6T102`<br>`M03S16000` |
| `cutoneaxis_setfeed(...);`<br><br><br><br><br>`cutwithfeed(...);` | `(PREPOSITION FOR RAPID PLUNGE)`<br>`G0X0Y0`<br>`Z0.25`<br>`G1Z0F100`<br>`G1 X109.5 Y75 Z-8.35F400`<br>`Z9` |
| `closegcodefile();` | `M05`<br>`M02` |

Conversely, the G-code commands which are supported are generated by the following modules:

| G-code | Command/Module |
|---|---|
| (Design File: )<br>(stockMin:0.00mm, -152.40mm, -34.92mm)<br>(stockMax:109.50mm, -77.40mm, 0.00mm)<br>(STOCK/BLOCK,109.50, 75.00, 34.92,0.00, 152.40, 34.92)<br>G90<br>G21 | opengcodefile(...); setupstock(...) |
| (Move to safe Z to avoid workholding)<br>G53G0Z-5.000 | movetosafez() |
| (Toolpath: Contour Toolpath 1)<br>M05<br>(TOOL/MILL,3.17, 0.00, 0.00, 0.00)<br>M6T102<br>M03S10000 | toolchange(...); |
| (PREPOSITION FOR RAPID PLUNGE) | writecomment(...) |
| G0X0.000Y-152.400<br>Z0.250 | rapid(...)<br>rapid(...) |
| G1Z-1.000F203.2<br>X109.500Y-77.400F508.0<br>X57.918Y16.302Z-0.726<br>Y22.023Z-1.023<br>X61.190Z-0.681<br>Y21.643<br>X57.681<br>Z12.700 | cutwithfeed(...);<br>cutwithfeed(...); |
| M05<br>M02 | closegcodefile(); |

The implication here is that it should be possible to read in a G-code file, and for each line/-command instantiate a matching command so as to create a 3D model/preview of the file.

### 2.4.2 DXF

Elements in DXFs are represented as lines or arcs. A minimal file showing both:

```
0
SECTION
2
ENTITIES
0
LWPOLYLINE
90
2
70
0
43
0
10
-31.375
20
-34.9152
10
-31.375
20
-18.75
0
ARC
10
-54.75
20
-37.5
40
4
50
0
51
90
0
ENDSEC
0
EOF
```

## 2.5 Modules

Note that as a convention, where it is necessary for a module to coordinate between Python and OpenSCAD, in certain cases it will be necessary for there to be three separate versions: a p<foo> Python definition for the manipulation of Python variables and any file routines, an o<foo> OpenSCAD module which will wrap up the Python function call, and lastly a <foo> OpenSCAD module which will be <include>d so as to be able to make use of OpenSCAD variables.

In natural languages such as English, there is an order to various parts of speech such as adjectives — since various prefixes and suffixes will be used for module names, having a consistent ordering/usage will help in consistency and make expression clearer. The ordering should be: sequence, action, function, parameter, filetype

- Both prefix and suffix
  - dxf (action (write out dxf file), filetype)
- Prefixes
  - begin (sequence)
  - continue (sequence)
  - end (sequence)
  - cut (action)
  - move (action)
  - rapid (action)
  - open (action)
  - close (action)
  - set (action/function)
- Suffixes
  - feed (parameter)
  - gcode (filetype)
  - polyline) (file (element))

For the sake of convenience, all user-facing modules will be listed here with their interface requirements/variables. Where appropriate, modules which interact will be listed together.

```
begincutdxf(rh, ex, ey, ez, fr); and continuecutdxf(ex, ey, ez, fr);

beginpolyline(bx,by,bz); and addpolyline(bx,by,bz); and closepolyline();

begintoolpath(bx,by,bz); and endtoolpath();

current_tool(); [function]

cut(ex, ey, ez);
cutoneaxis_setfeed(axis,depth,feed);
cutwithfeed(ex, ey, ez, feed);

cutarcNECCdxf(ex, ey, ez, xcenter, ycenter, radius);
cutarcNWCCdxf(ex, ey, ez, xcenter, ycenter, radius);
cutarcSWCCdxf(ex, ey, ez, xcenter, ycenter, radius);
cutarcSECCdxf(ex, ey, ez, xcenter, ycenter, radius);
cutarcNECWdxf(ex, ey, ez, xcenter, ycenter, radius);
cutarcSECWdxf(ex, ey, ez, xcenter, ycenter, radius);
cutarcSWCWdxf(ex, ey, ez, xcenter, ycenter, radius);

cutkeyhole_toolpath(kh_start_depth, kh_max_depth, kht_direction, kh_distance);

cutoutrectangledxf(bx, by, bz, rwidth, rheight, rdepth);
cutrectangledxf(bx, by, bz, rwidth, rheight, rdepth);
cutrectangleoutlinedxf(bx, by, bz, rwidth, rheight, rdepth);

cutroundover(bx, by, bz, ex, ey, ez);

dxfarc(xcenter, ycenter, radius, anglebegin, endangle);
dxfpolyline(xbegin,ybegin,xend,yend);

movetosafeheight();
movetosafez();

opendxffile(fn); and closedxffile();
opengcodefile(fn); and closegcodefile();

rapidbx(bx, by, bz, ex, ey, ez);
```

```
rapid(ex, ey, ez);

rectangleoutlinedxf(bx, by, bz, rwidth, rheight);

setupstock(stocklength, stockwidth, stockthickness, zeroheight, stockorigin);

setxpos(newxpos);
setypos(newypos);
setzpos(newzpos);

settzpos(newtzpos);

toolchange(tool_number,speed);
tool_diameter(td_tool, td_depth); [function]
tool_radius(td_tool, td_depth); [function]

writecomment(comment);
```

Principles for naming modules (and variables):

- minimize use of underscores (for convenience sake, underscores are not used for index entries)
- identify which aspect of the project structure is being worked with (`cut`(ting), `dxf`, `gcode`, `tool` management, etc.) and esp. note the use of `o`(penscad) and `p`(ython) as prefixes

Structurally, this will typically look like:

```
The user-facing module is \DescribeRoutine{FOOBAR}

\lstset{firstnumber=\thegcpscad}
\begin{writecode}{a}{gcodepreview.scad}{scad}
module FOOBAR(...) {
    oFOOBAR(...);
}

\end{writecode}
\addtocounter{gcpscad}{4}


which calls the internal OpenSCAD Module \DescribeSubroutine{FOOBAR}{oFOOBAR}

\begin{writecode}{a}{pygcodepreview.scad}{scad}
module oFOOBAR(...) {
    pFOOBAR(...);
 }

\end{writecode}
\addtocounter{pyscad}{4}

which in turn calls the internal Python definitioon \DescribeSubroutine{FOOBAR}{pFOOBAR}

\lstset{firstnumber=\thegcpy}
\begin{writecode}{a}{gcodepreview.py}{python}
def pFOOBAR (...)
    ...

\end{writecode}
\addtocounter{gcpy}{3}
```

Further note that this definition will not be necessary for some later modules since they are in turn calling internal modules which already use this structure.

### 2.5.1 Initial Modules

setupstock
psetupstock

The first such routine, (actually a subroutine, see `setupstock`) `psetupstock` will be appropriately enough, to set up the stock, and perform other initializations — in Python all that needs to be done is to set the value of the persistent (Python) variables:

```
13 gcpy def psetupstock(stocklength, stockwidth, stockthickness, zeroheight
       , stockorigin):
14 gcpy     global mpx
15 gcpy     mpx = float(0)
16 gcpy     global mpy
17 gcpy     mpy = float(0)
18 gcpy     global mpz
19 gcpy     mpz = float(0)
20 gcpy     global tpz
```

```
21 gcpy      tpz = float(0)
22 gcpy      global currenttool
23 gcpy      currenttool = 102
```

Note that while the #102 is declared as a default tool, while it was originally necessary to call a tool change after invoking `setupstock` in the 2024.09.03 version of PythonSCAD this requirement went away when a but which interfered with persistently setting a variable directly was fixed.

`osetupstock` The intermediary OpenSCAD code, `osetupstock` simply calls the Python version. Note that while the parameters are passed all the way down (for consistency) they are not used.

```
5 pyscad  module osetupstock(stocklength, stockwidth, stockthickness,
              zeroheight, stockorigin) {
6 pyscad      psetupstock(stocklength, stockwidth, stockthickness,
                  zeroheight, stockorigin);
7 pyscad  }
```

`setupstock` The OpenSCAD code, `setupstock` requires that the user set parameters for stock dimensions and so forth, and will create comments in the G-code which incorporate the stock dimensions and its position relative to the zero as set relative to the stock.

The internal variable `stockorigin` is used in an `<if then else>` structure to position the 3D model of the stock.

```
10 gcpscad module setupstock(stocklength, stockwidth, stockthickness,
               zeroheight, stockorigin) {
11 gcpscad   osetupstock(stocklength, stockwidth, stockthickness, zeroheight,
                 stockorigin);
12 gcpscad //initialize default tool and XYZ origin
13 gcpscad   osettool(102);
14 gcpscad   oset(0,0,0);
15 gcpscad   if (zeroheight == "Top") {
16 gcpscad     if (stockorigin == "Lower-Left") {
17 gcpscad     translate([0, 0, (-stockthickness)]){
18 gcpscad     cube([stocklength, stockwidth, stockthickness], center=false);
19 gcpscad       if (generategcode == true) {
20 gcpscad       owritethree("(stockMin:0.00mm, 0.00mm, -",str(stockthickness)
                     ,"mm)");
21 gcpscad       owritefive("(stockMax:",str(stocklength),"mm, ",str(
                     stockwidth),"mm, 0.00mm)");
22 gcpscad       owritenine("(STOCK/BLOCK, ",str(stocklength),", ",str(
                     stockwidth),", ",str(stockthickness),", 0.00, 0.00, ",str(
                     stockthickness),")");
23 gcpscad     }
24 gcpscad   }
25 gcpscad }
26 gcpscad      else if (stockorigin == "Center-Left") {
27 gcpscad     translate([0, (-stockwidth / 2), -stockthickness]){
28 gcpscad       cube([stocklength, stockwidth, stockthickness], center=false)
                     ;
29 gcpscad     if (generategcode == true) {
30 gcpscad owritefive("(stockMin:0.00mm, -",str(stockwidth/2),"mm, -",str(
               stockthickness),"mm)");
31 gcpscad owritefive("(stockMax:",str(stocklength),"mm, ",str(stockwidth/2),"
               mm, 0.00mm)");
32 gcpscad     owriteeleven("(STOCK/BLOCK, ",str(stocklength),", ",str(
                   stockwidth),", ",str(stockthickness),", 0.00, ",str(
                   stockwidth/2),", ",str(stockthickness),")");
33 gcpscad     }
34 gcpscad   }
35 gcpscad     } else if (stockorigin == "Top-Left") {
36 gcpscad     translate([0, (-stockwidth), -stockthickness]){
37 gcpscad       cube([stocklength, stockwidth, stockthickness], center=false)
                     ;
38 gcpscad if (generategcode == true) {
39 gcpscad owritefive("(stockMin:0.00mm, -",str(stockwidth),"mm, -",str(
               stockthickness),"mm)");
40 gcpscad owritethree("(stockMax:",str(stocklength),"mm, 0.00mm, 0.00mm)");
41 gcpscad owriteeleven("(STOCK/BLOCK, ",str(stocklength),", ",str(stockwidth)
               ,", ",str(stockthickness),", 0.00, ",str(stockwidth),", ",str(
               stockthickness),")");
42 gcpscad     }
43 gcpscad   }
44 gcpscad     }
45 gcpscad     else if (stockorigin == "Center") {
46 gcpscad //owritecomment("Center");
47 gcpscad     translate([(-stocklength / 2), (-stockwidth / 2), -
                   stockthickness]){
48 gcpscad       cube([stocklength, stockwidth, stockthickness], center=false)
```

```
                    ;
49 gcpscad if (generategcode == true) {
50 gcpscad owriteseven("(stockMin: -",str(stocklength/2),", -",str(stockwidth
           /2),"mm, -",str(stockthickness),"mm)");
51 gcpscad owritefive("(stockMax:",str(stocklength/2),"mm, ",str(stockwidth/2)
           ,"mm, 0.00mm)");
52 gcpscad owritethirteen("(STOCK/BLOCK, ",str(stocklength),", ",str(
           stockwidth),", ",str(stockthickness),", ",str(stocklength/2),",
           ", str(stockwidth/2),", ",str(stockthickness),")");
53 gcpscad       }
54 gcpscad     }
55 gcpscad   }
56 gcpscad } else if (zeroheight == "Bottom") {
57 gcpscad //owritecomment("Bottom");
58 gcpscad     if (stockorigin == "Lower-Left") {
59 gcpscad     cube([stocklength, stockwidth, stockthickness], center=false);
60 gcpscad if (generategcode == true) {
61 gcpscad owriteone("(stockMin:0.00mm, 0.00mm, 0.00mm)");
62 gcpscad owriteseven("(stockMax:",str(stocklength),"mm, ",str(stockwidth),"
           mm, ",str(stockthickness),"mm)");
63 gcpscad owriteseven("(STOCK/BLOCK, ",str(stocklength),", ",str(stockwidth)
           ,", ",str(stockthickness),",0.00, 0.00, 0.00)");
64 gcpscad     }
65 gcpscad }    else if (stockorigin == "Center-Left") {
66 gcpscad     translate([0, (-stockwidth / 2), 0]){
67 gcpscad       cube([stocklength, stockwidth, stockthickness], center=false)
                    ;
68 gcpscad if (generategcode == true) {
69 gcpscad owritethree("(stockMin:0.00mm, -",str(stockwidth/2),"mm, 0.00mm)");
70 gcpscad owriteseven("(stockMax:",str(stocklength),"mm, ",str(stockwidth/2)
           ,"mm, ",str(stockthickness),"mm)");
71 gcpscad owritenine("(STOCK/BLOCK, ",str(stocklength),", ",str(stockwidth)
           ,", ",str(stockthickness),",0.00, ",str(stockwidth/2),", 0.00)")
           ;
72 gcpscad     }
73 gcpscad   }
74 gcpscad     } else if (stockorigin == "Top-Left") {
75 gcpscad     translate([0, (-stockwidth), 0]){
76 gcpscad       cube([stocklength, stockwidth, stockthickness], center=false)
                    ;
77 gcpscad     }
78 gcpscad if (generategcode == true) {
79 gcpscad owritethree("(stockMin:0.00mm, -",str(stockwidth),"mm, 0.00mm)");
80 gcpscad owritefive("(stockMax:",str(stocklength),"mm, 0.00mm, ",str(
           stockthickness),"mm)");
81 gcpscad owritenine("(STOCK/BLOCK, ",str(stocklength),", ",str(stockwidth)
           ,", ",str(stockthickness),", 0.00, ", str(stockwidth),", 0.00)")
           ;
82 gcpscad   }
83 gcpscad }    else if (stockorigin == "Center") {
84 gcpscad     translate([(-stocklength / 2), (-stockwidth / 2), 0]){
85 gcpscad       cube([stocklength, stockwidth, stockthickness], center=false)
                    ;
86 gcpscad     }
87 gcpscad if (generategcode == true) {
88 gcpscad owritefive("(stockMin:-",str(stocklength/2),", -",str(stockwidth/2)
           ,"mm, 0.00mm)");
89 gcpscad owriteseven("(stockMax:",str(stocklength/2),"mm, ",str(stockwidth
           /2),"mm, ",str(stockthickness),"mm)");
90 gcpscad owriteeleven("(STOCK/BLOCK, ",str(stocklength),", ",str(stockwidth)
           ,", ",str(stockthickness),", ",str(stocklength/2),", ", str(
           stockwidth/2),", 0.00)");
91 gcpscad     }
92 gcpscad   }
93 gcpscad }
94 gcpscad if (generategcode == true) {
95 gcpscad     owriteone("G90");
96 gcpscad     owriteone("G21");
97 gcpscad //    owriteone("(Move to safe Z to avoid workholding)");
98 gcpscad //    owriteone("G53G0Z-5.000");
99 gcpscad   }
100 gcpscad //owritecomment("ENDSETUP");
101 gcpscad }
```

An example usage would be:

```
difference() {
setupstock(stocklength, stockwidth, stockthickness, zeroheight, stockorigin);
... // Cutting commands go here
```

```
}
```

xpos
ypos
zpos

It will be necessary to have Python functions (`xpos`, `ypos`, and `zpos`) which return the current values of the machine position in Cartesian coordinates:

```
def xpos():
    global mpx
    return mpx

def ypos():
    global mpy
    return mpy

def zpos():
    global mpz
    return mpz

def tzpos():
    global tpz
    return tpz
```

psetxpos
psetypos
psetzpos
psettzpos

and in turn, functions which set the positions: `psetxpos`, `psetypos`, `psetzpos`, and `psettzpos`

```
def psetxpos(newxpos):
    global mpx
    mpx = newxpos

def psetypos(newypos):
    global mpy
    mpy = newypos

def psetzpos(newzpos):
    global mpz
    mpz = newzpos

def psettzpos(newtzpos):
    global tpz
    tpz = newtzpos
```

setxpos
setypos
setzpos
settzpos
getxpos
getypos
getzpos
gettzpos

and as noted above, there will need to be matching OpenSCAD versions which will set: `setxpos`, `setypos`, `setzpos`, and `settzpos`; as well as return the value: `getxpos`, `getypos`, `getzpos`, and `gettzpos` Note that for routines where the variable is directly passed from OpenSCAD to Python it is possible to have OpenSCAD directly call the matching Python module with no need to use an intermediary OpenSCAD module.

```
function getxpos() = xpos();
function getypos() = ypos();
function getzpos() = zpos();
function gettzpos() = tzpos();

module setxpos(newxpos) {
    psetxpos(newxpos);
}

module setypos(newypos) {
    psetypos(newypos);
}

module setzpos(newzpos) {
    psetzpos(newzpos);
}

module settzpos(newtzpos) {
    psettzpos(newtzpos);
}
```

oset

`oset` while for setting all three of the variables, there is an internal OpenSCAD module:

```
module oset(ex, ey, ez) {
    setxpos(ex);
    setypos(ey);
    setzpos(ez);
}
```

osettz

and some toolpaths will require the storing and usage of an intermediate value via `osettz` for the Z-axis position during calculation:

```
109 gcpscad module osettz(tz) {
110 gcpscad     settzpos(tz);
111 gcpscad }
```

## 2.6 Tools and Changes

pcurrenttool
psettool

Similarly Python functions and variables will be used in: `pcurrenttool` and `psettool` to track and set and return the current tool

```
 57 gcpy def psettool(tn):
 58 gcpy     global currenttool
 59 gcpy     currenttool = tn
 60 gcpy
 61 gcpy def pcurrent_tool():
 62 gcpy     global currenttool
 63 gcpy     return currenttool
```

osettool
current tool

and matching OpenSCAD modules: `osettool` and `current tool` set and return the current tool:

```
 30 pyscad module osettool(tn){
 31 pyscad     psettool(tn);
 32 pyscad }
 33 pyscad
 34 pyscad function current_tool() = pcurrent_tool();
```

### 2.6.1 toolchange

toolchange

and apply the appropriate commands for a `toolchange`. Note that it is expected that this code will be updated as needed when new tooling is introduced as additional modules which require specific tooling are added below.

Note that the comments written out in G-code correspond to that used by the G-code previewing tool CutViewer (which is unfortunately, no longer readily available).

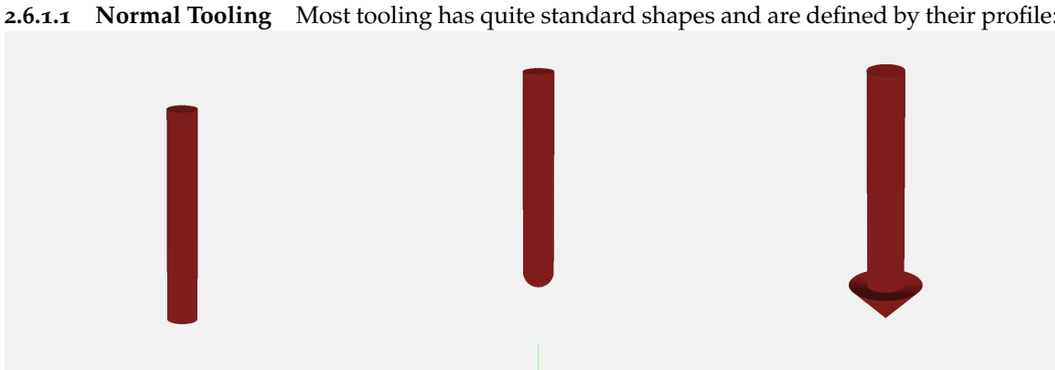
A further concern is that early versions often passed the tool into a module using a parameter. That ceased to be necessary in the 2024.09.03 version of PythonSCAD, and all modules should read the tool # from `currenttool()`.

It is possible that rather than hard-coding the tool definitions, a future update will instead read them in from an external file — the `.csv` format used for tool libraries in Carbide Create seems a likely candidate and worth exploring.

Note that there are many varieties of tooling and not all will be implemented, especially in the early versions of this project

#### 2.6.1.1 Normal Tooling

Most tooling has quite standard shapes and are defined by their profile:



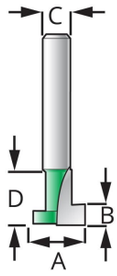
- Square (#201 and 102) — able to cut a flat bottom, perpendicular side and right angle their simple and easily understood geometry makes them a standard choice (a radiused form with a flat bottom, often described as a "bowl bit" is not implemented as-of-yet)
- Ballnose (#202 and 101) — rounded, they are the standard choice for concave and organic shapes
- V tooling (#301, 302 and 390) — pointed at the tip, they are available in a variety of angles and diameters and may be used for decorative V carving, or for chamfering or cutting specific angles (note that the commonly available radiused form is not implemented at this time, *e.g.*, #501 and 502)

```
113 gcpscad module toolchange(tool_number,speed) {
114 gcpscad     osettool(tool_number);
115 gcpscad if (generategcode == true) {
```

```
116 gcpscad     writecomment("Toolpath");
117 gcpscad     owriteone("M05");
118 gcpscad //     writecomment("Move to safe Z to avoid workholding");
119 gcpscad //     owriteone("G53G0Z-5.000");
120 gcpscad //     writecomment("Begin toolpath");
121 gcpscad     if (tool_number == 201) {
122 gcpscad       writecomment("TOOL/MILL,6.35, 0.00, 0.00, 0.00");
123 gcpscad     } else if (tool_number == 202) {
124 gcpscad       writecomment("TOOL/MILL,6.35, 3.17, 0.00, 0.00");
125 gcpscad     } else if (tool_number == 102) {
126 gcpscad       writecomment("TOOL/MILL,3.17, 0.00, 0.00, 0.00");
127 gcpscad     } else if (tool_number == 101) {
128 gcpscad       writecomment("TOOL/MILL,3.17, 1.58, 0.00, 0.00");
129 gcpscad     } else if (tool_number == 301) {
130 gcpscad       writecomment("TOOL/MILL,0.03, 0.00, 6.35, 45.00");
131 gcpscad     } else if (tool_number == 302) {
132 gcpscad       writecommment("TOOL/MILL,0.03, 0.00, 10.998, 30.00");
133 gcpscad     } else if (tool_number == 390) {
134 gcpscad       writecomment("TOOL/MILL,0.03, 0.00, 1.5875, 45.00");
```

**2.6.1.2 Tooling for Keyhole Toolpaths** Keyhole toolpaths (see: subsection 3.2 are intended for use with tooling which projects beyond the the narrower shaft and so will cut usefully underneath the visible surface. Also described as "undercut" tooling, but see below.

There are several notable candidates for such tooling:




*Keyhole Router Bits*

| # | A | B | C | D |
|---|---|---|---|---|
| 374 | 3/8″ | 1/8″ | 1/4″ | 3/8″ |
| 375 | 9.525mm | 3.175mm | 8mm | 9.525mm |
| 376 | 1/2″ | 3/16″ | 1/4″ | 1/2″ |
| 378 | 12.7mm | 4.7625mm | 8mm | 12.7mm |

- Keyhole tools — intended to cut slots for retaining hardware used for picture hanging, they may be used to create slots for other purposes Note that it will be necessary to model these twice, once for the shaft, the second time for the actual keyhole cutting https://assetssc.leevalley.com/en-gb/shop/tools/power-tool-accessories/router-bits/30113-keyhole-router-bits
- Dovetail cutters — used for the joinery of the same name, they cut a large area at the bottom which slants up to a narrower region at a defined angle
- Lollipop cutters — normally used for 3D work, as their name suggests they are essentially a (cutting) ball on a narrow stick (the tool shaft), they are mentioned here only for compleatness' sake and are not (at this time) implemented

```
135 gcpscad     } else if (tool_number == 374) {
136 gcpscad       writecomment("TOOL/MILL,9.53, 0.00, 3.17, 0.00");
137 gcpscad     } else if (tool_number == 375) {
138 gcpscad       writecomment("TOOL/MILL,9.53, 0.00, 3.17, 0.00");
139 gcpscad     } else if (tool_number == 376) {
140 gcpscad       writecomment("TOOL/MILL,12.7, 0.00, 4.77, 0.00");
141 gcpscad     } else if (tool_number == 378) {
142 gcpscad       writecomment("TOOL/MILL,12.7, 0.00, 4.77, 0.00");
```

```
143 gcpscad     } else if (tool_number == 814) {
144 gcpscad       writecomment("TOOL/MILL,12.7, 6.367, 12.7, 0.00");
```

**2.6.1.3 Thread mills** The implementation of arcs cutting along the Z-axis raises the possibility of cutting threads using "thread mills". See: https://community.carbide3d.com/t/thread-milling-in-metal-on-the-shapeoko-3/5332

Note that it will be necessary to to define modules (see below) for each tool shape.

With the tools delineated, the module is closed out and the tooling information written into the G-code.

```
145 gcpscad     }
146 gcpscad       select_tool(tool_number);
147 gcpscad       owritetwo("M6T",str(tool_number));
148 gcpscad       owritetwo("M03S",str(speed));
149 gcpscad   }
150 gcpscad }
```

For example:

```
toolchange(small_square_tool_no,speed * square_ratio);
```

**2.6.1.4 Roundover tooling** It is not possible to represent all tools using tool changes as coded above which require using a `hull` operation between 3D representations of the tools at the beginning and end points. Tooling which cannot be so represented will be implemented separately below, see paragraph 2.6.2.2.

`selecttool` `currenttool`

**2.6.1.5 Selecting Tools** There must also be a module for selecting tools: `selecttool` which will select the matching module for 3D modeling based on the `currenttool` (which is fed in to the module as `tool_number`, and pass the appropriate parameters to that module:

```
152 gcpscad module select_tool(tool_number) {
153 gcpscad //echo(tool_number);
154 gcpscad   if (tool_number == 201) {
155 gcpscad     gcp_endmill_square(6.35, 19.05);
156 gcpscad   } else if (tool_number == 202) {
157 gcpscad     gcp_endmill_ball(6.35, 19.05);
158 gcpscad   } else if (tool_number == 102) {
159 gcpscad     gcp_endmill_square(3.175, 19.05);
160 gcpscad   } else if (tool_number == 101) {
161 gcpscad     gcp_endmill_ball(3.175, 19.05);
162 gcpscad   } else if (tool_number == 301) {
163 gcpscad     gcp_endmill_v(90, 12.7);
164 gcpscad   } else if (tool_number == 302) {
165 gcpscad     gcp_endmill_v(60, 12.7);
166 gcpscad   } else if (tool_number == 390) {
167 gcpscad     gcp_endmill_v(90, 3.175);
```

For a keyhole tool:

```
168 gcpscad   } else if (tool_number == 374) {
169 gcpscad     gcp_keyhole(9.525, 3.175);
170 gcpscad   } else if (tool_number == 375) {
171 gcpscad     gcp_keyhole(9.525, 3.175);
172 gcpscad   } else if (tool_number == 376) {
173 gcpscad     gcp_keyhole(12.7, 4.7625);
174 gcpscad   } else if (tool_number == 378) {
175 gcpscad     gcp_keyhole(12.7, 4.7625);
```

and dovetail tool:

```
176 gcpscad   } else if (tool_number == 814) {
177 gcpscad     gcp_dovetail(12.7, 6.367, 12.7, 14);
```

Once all tools have been defined the `if` statement and module may be closed:

```
178 gcpscad   }
179 gcpscad }
```

### 2.6.2 3D Shapes for Tools

Each tool must be modeled in 3D using an OpenSCAD module.

**2.6.2.1 Normal toolshapes** Most tools are easily implemented with concise 3D descriptions which may be connected with a simple `hull` operation:

`gcp endmill square` The `gcp endmill square` is a simple cylinder:

```
181 gcpscad module gcp_endmill_square(es_diameter, es_flute_length) {
182 gcpscad   cylinder(r1=(es_diameter / 2), r2=(es_diameter / 2), h=
                es_flute_length, center=false);
```

```
183 gcpscad }
```

gcp keyhole The gcp keyhole is modeled by the cutting base:

```
185 gcpscad module gcp_keyhole(es_diameter, es_flute_length) {
186 gcpscad   cylinder(r1=(es_diameter / 2), r2=(es_diameter / 2), h=
               es_flute_length, center=false);
187 gcpscad }
```

and a second call for an additional cylinder for the shaft will be necessary:

```
189 gcpscad module gcp_keyhole_shaft(es_diameter, es_flute_length) {
190 gcpscad   cylinder(r1=(es_diameter / 2), r2=(es_diameter / 2), h=
               es_flute_length, center=false);
191 gcpscad }
```

gcp dovetail The gcp dovetail is modeled as a cylinder with the differing bottom and top diameters determining the angle (though dt_angle is still required as a parameter)

```
193 gcpscad module gcp_dovetail(dt_bottomdiameter, dt_topdiameter, dt_height,
             dt_angle) {
194 gcpscad   cylinder(r1=(dt_bottomdiameter / 2), r2=(dt_topdiameter / 2), h=
               dt_height, center=false);
195 gcpscad }
```

gcp endmill ball The gcp endmill ball is modeled as a hemisphere joined with a cylinder:

```
197 gcpscad module gcp_endmill_ball(es_diameter, es_flute_length) {
198 gcpscad   translate([0, 0, (es_diameter / 2)]){
199 gcpscad     union(){
200 gcpscad       sphere(r=(es_diameter / 2));
201 gcpscad       cylinder(r1=(es_diameter / 2), r2=(es_diameter / 2), h=
                   es_flute_length, center=false);
202 gcpscad     }
203 gcpscad   }
204 gcpscad }
```

gcp endmill v The gcp endmill v is modeled as a cylinder with a zero width base and a second cylinder for the shaft:

```
206 gcpscad module gcp_endmill_v(es_v_angle, es_diameter) {
207 gcpscad   union(){
208 gcpscad     cylinder(r1=0, r2=(es_diameter / 2), h=((es_diameter / 2) / tan
                 ((es_v_angle / 2))), center=false);
209 gcpscad     translate([0, 0, ((es_diameter / 2) / tan((es_v_angle / 2)))]){
210 gcpscad       cylinder(r1=(es_diameter / 2), r2=(es_diameter / 2), h=((
                   es_diameter * 8) ), center=false);/// tan((es_v_angle / 2)
                   )
211 gcpscad     }
212 gcpscad   }
213 gcpscad }
```

**2.6.2.2 Concave toolshapes** While normal tooling may be represented with a single hull operation betwixt two 3D toolshapes, concave tooling such as roundover/radius tooling require multiple slices of the tool shape which are then hulled together. Something of this can be seen in the manual work-around for previewing them: https://community.carbide3d.com/t/using-unsupported-tooling-in-carbide-create-roundover-cove-radius-bits/43723.

Ideally, it would be possible to simply identify such tooling using the tool # in the code used for normal toolshapes as above, but the most expedient option is to simply use a specific command for this. Since such tooling is quite limited in its use and normally only used at the surface of the part along an edge, this separation is easily justified.

Because it is necessary to divide the tooling into vertical slices and call the hull operation for each slice the tool definitions are tightly coupled with the module. Note that there are two different modules, the public-facing version which includes the tool number: cutroundover

cutroundover

```
215 gcpscad module cutroundover(bx, by, bz, ex, ey, ez, radiustn) {
216 gcpscad     if (radiustn == 56125) {
217 gcpscad         cutroundovertool(bx, by, bz, ex, ey, ez, 0.508/2, 1.531);
218 gcpscad     } else if (radiustn == 56142) {
219 gcpscad         cutroundovertool(bx, by, bz, ex, ey, ez, 0.508/2, 2.921);
220 gcpscad     } else if (radiustn == 312) {
221 gcpscad         cutroundovertool(bx, by, bz, ex, ey, ez, 1.524/2, 3.175);
```

```
222 gcpscad     } else if (radiustn == 1570) {
223 gcpscad         cutroundovertool(bx, by, bz, ex, ey, ez, 0.507/2, 4.509);
224 gcpscad     }
225 gcpscad }
```

which then calls the actual `cutroundovertool` module passing in the tip radius and the radius of the rounding. Note that this module sets its quality relative to the value of `$fn`.

```
227 gcpscad module cutroundovertool(bx, by, bz, ex, ey, ez, tool_radius_tip,
            tool_radius_width) {
228 gcpscad n = 90 + $fn*3;
229 gcpscad step = 360/n;
230 gcpscad
231 gcpscad hull(){
232 gcpscad     translate([bx,by,bz])
233 gcpscad     cylinder(step,tool_radius_tip,tool_radius_tip);
234 gcpscad     translate([ex,ey,ez])
235 gcpscad     cylinder(step,tool_radius_tip,tool_radius_tip);
236 gcpscad }
237 gcpscad
238 gcpscad hull(){
239 gcpscad translate([bx,by,bz+tool_radius_width])
240 gcpscad cylinder(tool_radius_width*2,tool_radius_tip+tool_radius_width,
            tool_radius_tip+tool_radius_width);
241 gcpscad
242 gcpscad translate([ex,ey,ez+tool_radius_width])
243 gcpscad   cylinder(tool_radius_width*2,tool_radius_tip+tool_radius_width,
              tool_radius_tip+tool_radius_width);
244 gcpscad }
245 gcpscad
246 gcpscad for (i=[0:step:90]) {
247 gcpscad     angle = i;
248 gcpscad     dx = tool_radius_width*cos(angle);
249 gcpscad     dxx = tool_radius_width*cos(angle+step);
250 gcpscad     dzz = tool_radius_width*sin(angle);
251 gcpscad     dz = tool_radius_width*sin(angle+step);
252 gcpscad     dh = dz-dzz;
253 gcpscad     hull(){
254 gcpscad         translate([bx,by,bz+dz])
255 gcpscad             cylinder(dh,tool_radius_tip+tool_radius_width-dx,
                        tool_radius_tip+tool_radius_width-dxx);
256 gcpscad         translate([ex,ey,ez+dz])
257 gcpscad             cylinder(dh,tool_radius_tip+tool_radius_width-dx,
                        tool_radius_tip+tool_radius_width-dxx);
258 gcpscad         }
259 gcpscad     }
260 gcpscad }
```

### 2.6.3 tooldiameter

It will also be necessary to be able to provide the diameter of the current tool. Arguably, this would be much easier using an object-oriented programming style/dot notation.

One aspect of tool parameters which will need to be supported is shapes which create different profiles based on how deeply the tool is cutting into the surface of the material at a given point. To accommodate this, it will be necessary to either track the thickness of uncut material at any given point, or, to specify the depth of cut as a parameter which is what the initial version will implement.

`tool diameter` The public-facing OpenSCAD code, `tool diameter` simply calls the matching OpenSCAD module which wraps the Python code:

```
262 gcpscad function tool_diameter(td_tool, td_depth) = otool_diameter(td_tool,
             td_depth);
```

`otool diameter` the matching OpenSCAD function, `otool diameter` calls the Python function:

```
 36 pyscad function otool_diameter(td_tool, td_depth) = ptool_diameter(td_tool
        , td_depth);
```

`ptool diameter` the Python code, `ptool diameter` returns appropriate values based on the specified tool number and depth:

```
 65 gcpy def ptool_diameter(ptd_tool, ptd_depth):
 66 gcpy # Square 122,112,102,201
 67 gcpy     if ptd_tool == 122:
```

```
68 gcpy          return 0.79375
69 gcpy      if ptd_tool == 112:
70 gcpy          return 1.5875
71 gcpy      if ptd_tool == 102:
72 gcpy          return 3.175
73 gcpy      if ptd_tool == 201:
74 gcpy          return 6.35
75 gcpy # Ball 121,111,101,202
76 gcpy      if ptd_tool == 122:
77 gcpy          return
78 gcpy          if ptd_depth > 0.396875:
79 gcpy              return 0.79375
80 gcpy          else:
81 gcpy              return 0
82 gcpy      if ptd_tool == 112:
83 gcpy          if ptd_depth > 0.79375:
84 gcpy              return 1.5875
85 gcpy          else:
86 gcpy              return 0
87 gcpy      if ptd_tool == 101:
88 gcpy          if ptd_depth > 1.5875:
89 gcpy              return 3.175
90 gcpy          else:
91 gcpy              return 0
92 gcpy      if ptd_tool == 202:
93 gcpy          if ptd_depth > 3.175:
94 gcpy              return 6.35
95 gcpy          else:
96 gcpy              return 0
97 gcpy # V 301, 302, 390
98 gcpy      if ptd_tool == 301:
99 gcpy          return 0
100 gcpy     if ptd_tool == 302:
101 gcpy         return 0
102 gcpy     if ptd_tool == 390:
103 gcpy         return 0
104 gcpy # Keyhole
105 gcpy     if ptd_tool == 374:
106 gcpy         if ptd_depth < 3.175:
107 gcpy             return 9.525
108 gcpy         else:
109 gcpy             return 6.35
110 gcpy     if ptd_tool == 375:
111 gcpy         if ptd_depth < 3.175:
112 gcpy             return 9.525
113 gcpy         else:
114 gcpy             return 8
115 gcpy     if ptd_tool == 376:
116 gcpy         if ptd_depth < 4.7625:
117 gcpy             return 12.7
118 gcpy         else:
119 gcpy             return 6.35
120 gcpy     if ptd_tool == 378:
121 gcpy         if ptd_depth < 4.7625:
122 gcpy             return 12.7
123 gcpy         else:
124 gcpy             return 8
125 gcpy # Dovetail
126 gcpy     if ptd_tool == 814:
127 gcpy         if ptd_depth > 12.7:
128 gcpy             return 6.35
129 gcpy         else:
130 gcpy             return 12.7
```

`tool radius` Since it is often necessary to utilise the radius of the tool, an additional command, `tool radius` to return this value is worthwhile:

```
264 gcpscad function tool_radius(td_tool, td_depth) = otool_diameter(td_tool,
        td_depth)/2;
```

(Note that zero (0) and other not fully calculated values will need to be replaced with code which calculates the appropriate values.)

### 2.6.4 Feeds and Speeds

`feed` `plunge` `speed` There are several possibilities for handling feeds and speeds. Currently, base values for `feed`, `plunge`, and `speed` are used, which may then be adjusted using various `<tooldescriptor>_ratio`

values, as an acknowledgement of the likelihood of a trim router being used as a spindle, the assumption is that the `speed` will remain unchanged.

One notable possibility for the future would be to load it from the .csv files used for User tool libraries in Carbide Create. Ideally, any use of such values in modules would be such that some other scheme could replace that usage with minimal editing and updating.

The tools which need to be calculated thus are those in addition to the `large_square` tool:

- `small_square_ratio`
- `small_ball_ratio`
- `large_ball_ratio`
- `small_V_ratio`
- `large_V_ratio`
- `KH_ratio`
- `DT_ratio`

## 2.7 File Handling

popengcodefile
popendxffile
popendxflgsqfile
popendxfsmsqfile
popendxflgblfile
popendxfsmblfile
popendxflgVfile
popendxfsmVfile

For writing to files it will be necessary to have commands: `popengcodefile`, `popendxffile`, `popendxflgsqfile`, `popendxfsmsqfile`, `popendxflgblfile`, `popendxfsmblfile`, `popendxflgVfile`, and `popendxfsmVfile`. There is a separate function for each type of file, and for DXFs, there are multiple file instances, one for each combination of different type and size of tool which it is expected a project will work with. Each such file will be suffixed with the tool number.

Integrating G-code and DXF generation with everything else would be ideal, but will require ensuring that each command which moves the tool creates a matching command for both files.

```
97 gcpy def popengcodefile(fn):
98 gcpy     global f
99 gcpy     f = open(fn, "w")
100 gcpy
101 gcpy def popendxffile(fn):
102 gcpy     global dxf
103 gcpy     dxf = open(fn, "w")
104 gcpy
105 gcpy def popendxflgblfile(fn):
106 gcpy     global dxflgbl
107 gcpy     dxflgbl = open(fn, "w")
108 gcpy
109 gcpy def popendxflgsqfile(fn):
110 gcpy     global dxflgsq
111 gcpy     dxflgsq = open(fn, "w")
112 gcpy
113 gcpy def popendxflgVfile(fn):
114 gcpy     global dxflgV
115 gcpy     dxflgV = open(fn, "w")
116 gcpy
117 gcpy def popendxfsmblfile(fn):
118 gcpy     global dxfsmbl
119 gcpy     dxfsmbl = open(fn, "w")
120 gcpy
121 gcpy def popendxfsmsqfile(fn):
122 gcpy     global dxfsmsq
123 gcpy     dxfsmsq = open(fn, "w")
124 gcpy
125 gcpy def popendxfsmVfile(fn):
126 gcpy     global dxfsmV
127 gcpy     dxfsmV = open(fn, "w")
128 gcpy
129 gcpy def popendxfKHfile(fn):
130 gcpy     global dxfKH
131 gcpy     dxfKH = open(fn, "w")
132 gcpy
133 gcpy def popendxfDTfile(fn):
134 gcpy     global dxfDT
135 gcpy     dxfDT = open(fn, "w")
```

oopengcodefile
oopendxffile

There will need to be matching OpenSCAD modules `oopengcodefile`, and `oopendxffile`, for the Python functions.

```
38 pyscad module oopengcodefile(fn) {
39 pyscad     popengcodefile(fn);
40 pyscad }
41 pyscad
```

```
42 pyscad module oopendxffile(fn) {
43 pyscad //    echo(fn);
44 pyscad     popendxffile(fn);
45 pyscad }
46 pyscad
47 pyscad module oopendxflgblfile(fn) {
48 pyscad     popendxflgblfile(fn);
49 pyscad }
50 pyscad
51 pyscad module oopendxflgsqfile(fn) {
52 pyscad     popendxflgsqfile(fn);
53 pyscad }
54 pyscad
55 pyscad module oopendxflgVfile(fn) {
56 pyscad     popendxflgVfile(fn);
57 pyscad }
58 pyscad
59 pyscad module oopendxfsmblfile(fn) {
60 pyscad     popendxfsmblfile(fn);
61 pyscad }
62 pyscad
63 pyscad module oopendxfsmsqfile(fn) {
64 pyscad //    echo(fn);
65 pyscad     popendxfsmsqfile(fn);
66 pyscad }
67 pyscad
68 pyscad module oopendxfsmVfile(fn) {
69 pyscad     popendxfsmVfile(fn);
70 pyscad }
71 pyscad
72 pyscad module oopendxfKHfile(fn) {
73 pyscad     popendxfKHfile(fn);
74 pyscad }
75 pyscad
76 pyscad module oopendxfDTfile(fn) {
77 pyscad     popendxfDTfile(fn);
78 pyscad }
```

opengcodefile With matching OpenSCAD commands: `opengcodefile`

```
266 gcpscad module opengcodefile(fn) {
267 gcpscad if (generategcode == true) {
268 gcpscad     oopengcodefile(fn);
269 gcpscad //    echo(fn);
270 gcpscad     owritecomment(fn);
271 gcpscad     }
272 gcpscad }
```

opendxffile For each DXF file, there will need to be a Preamble created by `opendxffile` in addition to opening the file in the file system:

```
274 gcpscad module opendxffile(fn) {
275 gcpscad   if (generatedxf == true) {
276 gcpscad       oopendxffile(str(fn,".dxf"));
277 gcpscad //    echo(fn);
278 gcpscad       dxfwriteone("0");
279 gcpscad       dxfwriteone("SECTION");
280 gcpscad       dxfwriteone("2");
281 gcpscad       dxfwriteone("ENTITIES");
282 gcpscad     if (large_ball_tool_no >  0) {    oopendxflgblfile(str(fn,".",
                    large_ball_tool_no,".dxf"));
283 gcpscad       dxfpreamble(large_ball_tool_no);
284 gcpscad     }
285 gcpscad     if (large_square_tool_no >  0) {    oopendxflgsqfile(str(fn
                    ,".",large_square_tool_no,".dxf"));
286 gcpscad       dxfpreamble(large_square_tool_no);
287 gcpscad     }
288 gcpscad     if (large_V_tool_no >  0) {    oopendxflgVfile(str(fn,".",
                    large_V_tool_no,".dxf"));
289 gcpscad       dxfpreamble(large_V_tool_no);
290 gcpscad     }
291 gcpscad     if (small_ball_tool_no >  0) { oopendxfsmblfile(str(fn,".",
                    small_ball_tool_no,".dxf"));
292 gcpscad       dxfpreamble(small_ball_tool_no);
293 gcpscad     }
294 gcpscad     if (small_square_tool_no >  0) {    oopendxfsmsqfile(str(fn
                    ,".",small_square_tool_no,".dxf"));
```

```
295 gcpscad //     echo(str("tool no",small_square_tool_no));
296 gcpscad        dxfpreamble(small_square_tool_no);
297 gcpscad      }
298 gcpscad      if (small_V_tool_no >  0) {    oopendxfsmVfile(str(fn,".",
                     small_V_tool_no,".dxf"));
299 gcpscad        dxfpreamble(small_V_tool_no);
300 gcpscad      }
301 gcpscad      if (KH_tool_no >  0) {    oopendxfKHfile(str(fn,".",KH_tool_no
                     ,".dxf"));
302 gcpscad        dxfpreamble(KH_tool_no);
303 gcpscad      }
304 gcpscad      if (DT_tool_no >  0) {    oopendxfDTfile(str(fn,".",DT_tool_no
                     ,".dxf"));
305 gcpscad        dxfpreamble(DT_tool_no);
306 gcpscad      }
307 gcpscad    }
308 gcpscad }
```

### 2.7.1 Writing to files

writedxf Once files have been opened they may be written to. The base command: `writedxf`

```
137 gcpy def writedxf(*arguments):
138 gcpy     line_to_write = ""
139 gcpy     for element in arguments:
140 gcpy         line_to_write += element
141 gcpy     dxf.write(line_to_write)
142 gcpy     dxf.write("\n")
```

has a matching command each tool/size combination:

writedxflgbl • Ball nose, large (lgbl) `writedxflgbl`

writedxfsmbl • Ball nose, small (smbl) `writedxfsmbl`

writedxflgsq • Square, large (lgsq) `writedxflgsq`

writedxfsmsq • Square, small (smsq) `writedxfsmsq`

writedxflgV • V, large (lgV) `writedxflgV`

writedxfsmV • V, small (smV) `writedxfsmV`

writedxfKH • Keyhole (KH) `writedxfKH`

writedxfDT • Dovetail (DT) `writedxfDT`

```
144 gcpy def writedxflgbl(*arguments):
145 gcpy     line_to_write = ""
146 gcpy     for element in arguments:
147 gcpy         line_to_write += element
148 gcpy     dxflgbl.write(line_to_write)
149 gcpy     print(line_to_write)
150 gcpy     dxflgbl.write("\n")
151 gcpy
152 gcpy def writedxflgsq(*arguments):
153 gcpy     line_to_write = ""
154 gcpy     for element in arguments:
155 gcpy         line_to_write += element
156 gcpy     dxflgsq.write(line_to_write)
157 gcpy     print(line_to_write)
158 gcpy     dxflgsq.write("\n")
159 gcpy
160 gcpy def writedxflgV(*arguments):
161 gcpy     line_to_write = ""
162 gcpy     for element in arguments:
163 gcpy         line_to_write += element
164 gcpy     dxflgV.write(line_to_write)
165 gcpy     print(line_to_write)
166 gcpy     dxflgV.write("\n")
167 gcpy
168 gcpy def writedxfsmbl(*arguments):
169 gcpy     line_to_write = ""
170 gcpy     for element in arguments:
171 gcpy         line_to_write += element
172 gcpy     dxfsmbl.write(line_to_write)
173 gcpy     print(line_to_write)
```

```
174 gcpy      dxfsmbl.write("\n")
175 gcpy
176 gcpy  def writedxfsmsq(*arguments):
177 gcpy      line_to_write = ""
178 gcpy      for element in arguments:
179 gcpy          line_to_write += element
180 gcpy      dxfsmsq.write(line_to_write)
181 gcpy      print(line_to_write)
182 gcpy      dxfsmsq.write("\n")
183 gcpy
184 gcpy  def writedxfsmV(*arguments):
185 gcpy      line_to_write = ""
186 gcpy      for element in arguments:
187 gcpy          line_to_write += element
188 gcpy      dxfsmV.write(line_to_write)
189 gcpy      print(line_to_write)
190 gcpy      dxfsmV.write("\n")
191 gcpy
192 gcpy  def writedxfKH(*arguments):
193 gcpy      line_to_write = ""
194 gcpy      for element in arguments:
195 gcpy          line_to_write += element
196 gcpy      dxfKH.write(line_to_write)
197 gcpy      print(line_to_write)
198 gcpy      dxfKH.write("\n")
199 gcpy
200 gcpy  def writedxfDT(*arguments):
201 gcpy      line_to_write = ""
202 gcpy      for element in arguments:
203 gcpy          line_to_write += element
204 gcpy      dxfDT.write(line_to_write)
205 gcpy      print(line_to_write)
206 gcpy      dxfDT.write("\n")
```

owritecomment
dxfwriteone
dxfwritelgbl
dxfwritelgsq
dxfwritelgV
dxfwritesmbl
dxfwritesmsq
dxfwritesmV

Separate OpenSCAD modules, `owritecomment`, `dxfwriteone`, `dxfwritelgbl`, `dxfwritelgsq`, `dxfwritelgV`, `dxfwritesmbl`, `dxfwritesmsq`, and `dxfwritesmV` will be used for either writing out comments in G-code (.nc) files or adding to a DXF file — for each different tool in a file there will be a matching module to write to it.

```
 80 pyscad module owritecomment(comment) {
 81 pyscad     writeln("(",comment,")");
 82 pyscad }
 83 pyscad
 84 pyscad module dxfwriteone(first) {
 85 pyscad     writedxf(first);
 86 pyscad //    writeln(first);
 87 pyscad //    echo(first);
 88 pyscad }
 89 pyscad
 90 pyscad module dxfwritelgbl(first) {
 91 pyscad     writedxflgbl(first);
 92 pyscad }
 93 pyscad
 94 pyscad module dxfwritelgsq(first) {
 95 pyscad     writedxflgsq(first);
 96 pyscad }
 97 pyscad
 98 pyscad module dxfwritelgV(first) {
 99 pyscad     writedxflgV(first);
100 pyscad }
101 pyscad
102 pyscad module dxfwritesmbl(first) {
103 pyscad     writedxfsmbl(first);
104 pyscad }
105 pyscad
106 pyscad module dxfwritesmsq(first) {
107 pyscad     writedxfsmsq(first);
108 pyscad }
109 pyscad
110 pyscad module dxfwritesmV(first) {
111 pyscad     writedxfsmV(first);
112 pyscad }
113 pyscad
114 pyscad module dxfwriteKH(first) {
115 pyscad     writedxfKH(first);
116 pyscad }
117 pyscad
118 pyscad module dxfwriteDT(first) {
```

```
119 pyscad      writedxfDT(first);
120 pyscad }
```

Since it is not convenient to stitch together and then write out multiple elements, the most expedient thing to do is to have discrete commands for each possible number of arguments, one through thirteen, `owrite...`

owrite...

```
122 pyscad module owriteone(first) {
123 pyscad     writeln(first);
124 pyscad }
125 pyscad
126 pyscad module owritetwo(first, second) {
127 pyscad     writeln(first, second);
128 pyscad }
129 pyscad
130 pyscad module owritethree(first, second, third) {
131 pyscad     writeln(first, second, third);
132 pyscad }
133 pyscad
134 pyscad module owritefour(first, second, third, fourth) {
135 pyscad     writeln(first, second, third, fourth);
136 pyscad }
137 pyscad
138 pyscad module owritefive(first, second, third, fourth, fifth) {
139 pyscad     writeln(first, second, third, fourth, fifth);
140 pyscad }
141 pyscad
142 pyscad module owritesix(first, second, third, fourth, fifth, sixth) {
143 pyscad     writeln(first, second, third, fourth, fifth, sixth);
144 pyscad }
145 pyscad
146 pyscad module owriteseven(first, second, third, fourth, fifth, sixth,
            seventh) {
147 pyscad     writeln(first, second, third, fourth, fifth, sixth, seventh);
148 pyscad }
149 pyscad
150 pyscad module owriteeight(first, second, third, fourth, fifth, sixth,
            seventh,eighth) {
151 pyscad     writeln(first, second, third, fourth, fifth, sixth, seventh,
                eighth);
152 pyscad }
153 pyscad
154 pyscad module owritenine(first, second, third, fourth, fifth, sixth,
            seventh, eighth, ninth) {
155 pyscad     writeln(first, second, third, fourth, fifth, sixth, seventh,
                eighth, ninth);
156 pyscad }
157 pyscad
158 pyscad module owriteten(first, second, third, fourth, fifth, sixth,
            seventh, eighth, ninth, tenth) {
159 pyscad     writeln(first, second, third, fourth, fifth, sixth, seventh,
                eighth, ninth, tenth);
160 pyscad }
161 pyscad
162 pyscad module owriteeleven(first, second, third, fourth, fifth, sixth,
            seventh, eighth, ninth, tenth, eleventh) {
163 pyscad     writeln(first, second, third, fourth, fifth, sixth, seventh,
                eighth, ninth, tenth, eleventh);
164 pyscad }
165 pyscad
166 pyscad module owritetwelve(first, second, third, fourth, fifth, sixth,
            seventh, eighth, ninth, tenth, eleventh, twelfth) {
167 pyscad     writeln(first, second, third, fourth, fifth, sixth, seventh,
                eighth, ninth, tenth, eleventh, twelfth);
168 pyscad }
169 pyscad
170 pyscad module owritethirteen(first, second, third, fourth, fifth, sixth,
            seventh, eighth, ninth, tenth, eleventh, twelfth, thirteenth) {
171 pyscad     writeln(first, second, third, fourth, fifth, sixth, seventh,
                eighth, ninth, tenth, eleventh, twelfth, thirteenth);
172 pyscad }
```

dxfwrite
dxfpreamble

**2.7.1.1 Beginning Writing to DXFs** The `dxfwrite` module requires that the tool number be passed in, and after writing out `dxfpreamble`, that value will be used to write out to the appropriate file with a series of `if` statements.

```
310 gcpscad module dxfwrite(tn,arg) {
311 gcpscad if (tn == large_ball_tool_no) {
312 gcpscad     dxfwritelgbl(arg);}
313 gcpscad if (tn == large_square_tool_no) {
314 gcpscad     dxfwritelgsq(arg);}
315 gcpscad if (tn == large_V_tool_no) {
316 gcpscad     dxfwritelgV(arg);}
317 gcpscad if (tn == small_ball_tool_no) {
318 gcpscad     dxfwritesmbl(arg);}
319 gcpscad if (tn == small_square_tool_no) {
320 gcpscad     dxfwritesmsq(arg);}
321 gcpscad if (tn == small_V_tool_no) {
322 gcpscad     dxfwritesmV(arg);}
323 gcpscad if (tn == DT_tool_no) {
324 gcpscad     dxfwriteDT(arg);}
325 gcpscad if (tn == KH_tool_no) {
326 gcpscad     dxfwriteKH(arg);}
327 gcpscad }
328 gcpscad
329 gcpscad module dxfpreamble(tn) {
330 gcpscad //    echo(str("dxfpreamble",small_square_tool_no));
331 gcpscad     dxfwrite(tn,"0");
332 gcpscad     dxfwrite(tn,"SECTION");
333 gcpscad     dxfwrite(tn,"2");
334 gcpscad     dxfwrite(tn,"ENTITIES");
335 gcpscad }
```

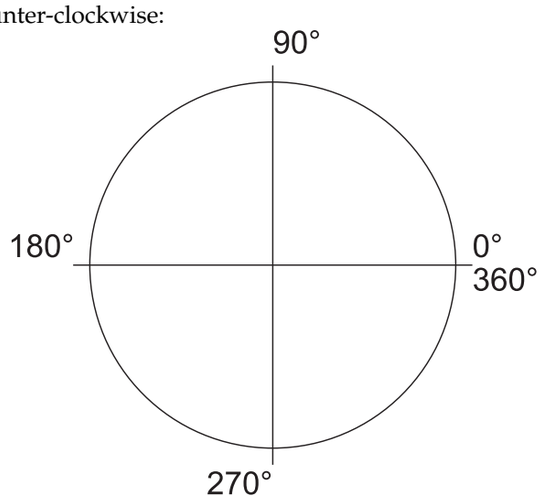
**2.7.1.2 DXF Lines and Arcs** Similarly, each each element which may be written to a DXF file will have a user module as well as an internal module which will be called by it so as to write to the file for the current tool.

There are two notable elements which may be written to a DXF:

dxfbpl
- a line: LWPOLYLINE is one possible implementation: `dxfbpl`

dxfarc
- ARC — a notable option would be for the arc to close on itself, creating a circle: `dxfarc`

DXF orders arcs counter-clockwise:




Note that arcs of greater than 90 degrees are not rendered accurately, so, for the sake of precision, they should be limited to a swing of 90 degrees or less. Further note that 4 arcs may be stitched together to make a circle:

```
dxfarc(10, 10, 5,   0,  90, small_square_tool_no);
dxfarc(10, 10, 5,  90, 180, small_square_tool_no);
dxfarc(10, 10, 5, 180, 270, small_square_tool_no);
dxfarc(10, 10, 5, 270, 360, small_square_tool_no);
```

A further refinement would be to connect multiple line segments/arcs into a larger polyline, but since most CAM tools implicitly join elements on import, that is not necessary.

There are three possible interactions for DXF elements and toolpaths:

- describe the motion of the tool
- define a perimeter of an area which will be cut by a tool
- define a centerpoint for a specialty toolpath such as Drill or Keyhhole

and it is possible that multiple such elements could be instantiated for a given toolpath.

```
337 gcpscad module dxfpl(tn,xbegin,ybegin,xend,yend) {
338 gcpscad     dxfwrite(tn,"0");
339 gcpscad     dxfwrite(tn,"LWPOLYLINE");
340 gcpscad     dxfwrite(tn,"90");
341 gcpscad     dxfwrite(tn,"2");
342 gcpscad     dxfwrite(tn,"70");
343 gcpscad     dxfwrite(tn,"0");
344 gcpscad     dxfwrite(tn,"43");
345 gcpscad     dxfwrite(tn,"0");
346 gcpscad     dxfwrite(tn,"10");
346 gcpscad     dxfwrite(tn,str(xbegin));
347 gcpscad     dxfwrite(tn,"20");
348 gcpscad     dxfwrite(tn,str(ybegin));
349 gcpscad     dxfwrite(tn,"10");
350 gcpscad     dxfwrite(tn,str(xend));
351 gcpscad     dxfwrite(tn,"20");
352 gcpscad     dxfwrite(tn,str(yend));
353 gcpscad }
354 gcpscad
355 gcpscad module dxfpolyline(xbegin,ybegin,xend,yend, tn) {
356 gcpscad if (generatedxf == true) {
357 gcpscad     dxfwriteone("0");
358 gcpscad     dxfwriteone("LWPOLYLINE");
359 gcpscad     dxfwriteone("90");
360 gcpscad     dxfwriteone("2");
361 gcpscad     dxfwriteone("70");
362 gcpscad     dxfwriteone("0");
363 gcpscad     dxfwriteone("43");
364 gcpscad     dxfwriteone("0");
365 gcpscad     dxfwriteone("10");
366 gcpscad     dxfwriteone(str(xbegin));
367 gcpscad     dxfwriteone("20");
368 gcpscad     dxfwriteone(str(ybegin));
369 gcpscad     dxfwriteone("10");
370 gcpscad     dxfwriteone(str(xend));
371 gcpscad     dxfwriteone("20");
372 gcpscad     dxfwriteone(str(yend));
373 gcpscad     dxfpl(tn,xbegin,ybegin,xend,yend);
374 gcpscad     }
375 gcpscad }
```

`dxfa` `dxfarc` As for other files, we have two versions, `dxfa` and `dxfarc`, one which accepts a `tn` (tool number), writing only to it, while a publicly facing version writes to the main DXF file *and* writes to the specific DXF file for the specified tool.

```
376 gcpscad module dxfa(tn,xcenter,ycenter,radius,anglebegin,endangle) {
377 gcpscad     dxfwrite(tn,"0");
378 gcpscad     dxfwrite(tn,"ARC");
379 gcpscad     dxfwrite(tn,"10");
380 gcpscad     dxfwrite(tn,str(xcenter));
381 gcpscad     dxfwrite(tn,"20");
382 gcpscad     dxfwrite(tn,str(ycenter));
383 gcpscad     dxfwrite(tn,"40");
384 gcpscad     dxfwrite(tn,str(radius));
385 gcpscad     dxfwrite(tn,"50");
386 gcpscad     dxfwrite(tn,str(anglebegin));
387 gcpscad     dxfwrite(tn,"51");
388 gcpscad     dxfwrite(tn,str(endangle));
389 gcpscad }
390 gcpscad
391 gcpscad module dxfarc(xcenter, ycenter, radius, anglebegin, endangle, tn) {
392 gcpscad if (generatedxf == true) {
393 gcpscad     dxfwriteone("0");
394 gcpscad     dxfwriteone("ARC");
395 gcpscad     dxfwriteone("10");
396 gcpscad     dxfwriteone(str(xcenter));
397 gcpscad     dxfwriteone("20");
398 gcpscad     dxfwriteone(str(ycenter));
399 gcpscad     dxfwriteone("40");
400 gcpscad     dxfwriteone(str(radius));
401 gcpscad     dxfwriteone("50");
402 gcpscad     dxfwriteone(str(anglebegin));
403 gcpscad     dxfwriteone("51");
404 gcpscad     dxfwriteone(str(endangle));
405 gcpscad     dxfa(tn,xcenter,ycenter,radius,anglebegin,endangle);
406 gcpscad     }
407 gcpscad }
```

The original implementation of polylines worked, but may be removed.

```
411 gcpscad module dxfbpl(tn,bx,by) {
412 gcpscad     dxfwrite(tn,"0");
413 gcpscad     dxfwrite(tn,"POLYLINE");
414 gcpscad     dxfwrite(tn,"8");
415 gcpscad     dxfwrite(tn,"default");
416 gcpscad     dxfwrite(tn,"66");
417 gcpscad     dxfwrite(tn,"1");
418 gcpscad     dxfwrite(tn,"70");
419 gcpscad     dxfwrite(tn,"0");
420 gcpscad     dxfwrite(tn,"0");
421 gcpscad     dxfwrite(tn,"VERTEX");
422 gcpscad     dxfwrite(tn,"8");
423 gcpscad     dxfwrite(tn,"default");
424 gcpscad     dxfwrite(tn,"70");
425 gcpscad     dxfwrite(tn,"32");
426 gcpscad     dxfwrite(tn,"10");
427 gcpscad     dxfwrite(tn,str(bx));
428 gcpscad     dxfwrite(tn,"20");
429 gcpscad     dxfwrite(tn,str(by));
430 gcpscad }
431 gcpscad
432 gcpscad module beginpolyline(bx,by,bz) {
433 gcpscad if (generatedxf == true) {
434 gcpscad     dxfwriteone("0");
435 gcpscad     dxfwriteone("POLYLINE");
436 gcpscad     dxfwriteone("8");
437 gcpscad     dxfwriteone("default");
438 gcpscad     dxfwriteone("66");
439 gcpscad     dxfwriteone("1");
440 gcpscad     dxfwriteone("70");
441 gcpscad     dxfwriteone("0");
442 gcpscad     dxfwriteone("0");
443 gcpscad     dxfwriteone("VERTEX");
444 gcpscad     dxfwriteone("8");
445 gcpscad     dxfwriteone("default");
446 gcpscad     dxfwriteone("70");
447 gcpscad     dxfwriteone("32");
448 gcpscad     dxfwriteone("10");
449 gcpscad     dxfwriteone(str(bx));
450 gcpscad     dxfwriteone("20");
451 gcpscad     dxfwriteone(str(by));
452 gcpscad     dxfbpl(current_tool(),bx,by);}
453 gcpscad }
454 gcpscad
455 gcpscad module dxfapl(tn,bx,by) {
456 gcpscad     dxfwrite(tn,"0");
457 gcpscad     dxfwrite(tn,"VERTEX");
458 gcpscad     dxfwrite(tn,"8");
459 gcpscad     dxfwrite(tn,"default");
460 gcpscad     dxfwrite(tn,"70");
461 gcpscad     dxfwrite(tn,"32");
462 gcpscad     dxfwrite(tn,"10");
463 gcpscad     dxfwrite(tn,str(bx));
464 gcpscad     dxfwrite(tn,"20");
465 gcpscad     dxfwrite(tn,str(by));
466 gcpscad }
467 gcpscad
468 gcpscad module addpolyline(bx,by,bz) {
469 gcpscad if (generatedxf == true) {
470 gcpscad     dxfwriteone("0");
471 gcpscad     dxfwriteone("VERTEX");
472 gcpscad     dxfwriteone("8");
473 gcpscad     dxfwriteone("default");
474 gcpscad     dxfwriteone("70");
475 gcpscad     dxfwriteone("32");
476 gcpscad     dxfwriteone("10");
477 gcpscad     dxfwriteone(str(bx));
478 gcpscad     dxfwriteone("20");
479 gcpscad     dxfwriteone(str(by));
480 gcpscad     dxfapl(current_tool(),bx,by);
481 gcpscad     }
482 gcpscad }
483 gcpscad
484 gcpscad module dxfcpl(tn) {
485 gcpscad     dxfwrite(tn,"0");
486 gcpscad     dxfwrite(tn,"SEQEND");
```

```
487 gcpscad }
488 gcpscad
489 gcpscad module closepolyline() {
490 gcpscad   if (generatedxf == true) {
491 gcpscad     dxfwriteone("0");
492 gcpscad     dxfwriteone("SEQEND");
493 gcpscad     dxfcpl(current_tool());
494 gcpscad   }
495 gcpscad }
496 gcpscad
497 gcpscad module writecomment(comment) {
498 gcpscad   if (generategcode == true) {
499 gcpscad     owritecomment(comment);
500 gcpscad   }
501 gcpscad }
```

At the end of the project it will be necessary to close each file using the commands: `pclosegcodefile` `pclosegcodefile`, and `closedxffile`. In some instances it will be necessary to write additional information, depending on the file format.

`closedxffile`

```
208 gcpy def pclosegcodefile():
209 gcpy     f.close()
210 gcpy
211 gcpy def pclosedxffile():
212 gcpy     dxf.close()
213 gcpy
214 gcpy def pclosedxflgblfile():
215 gcpy     dxflgbl.close()
216 gcpy
217 gcpy def pclosedxflgsqfile():
218 gcpy     dxflgsq.close()
219 gcpy
220 gcpy def pclosedxflgVfile():
221 gcpy     dxflgV.close()
222 gcpy
223 gcpy def pclosedxfsmblfile():
224 gcpy     dxfsmbl.close()
225 gcpy
226 gcpy def pclosedxfsmsqfile():
227 gcpy     dxfsmsq.close()
228 gcpy
229 gcpy def pclosedxfsmVfile():
230 gcpy     dxfsmV.close()
231 gcpy
232 gcpy def pclosedxfDTfile():
233 gcpy     dxfDT.close()
234 gcpy
235 gcpy def pclosedxfKHfile():
236 gcpy     dxfKH.close()
```

In addition to the Python forms, there will need to be matching OpenSCAD commands to call them: `oclosegcodefile`, and `oclosedxffile`.

`oclosegcodefile` `oclosedxffile`

```
174 pyscad module oclosegcodefile() {
175 pyscad     pclosegcodefile();
176 pyscad }
177 pyscad
178 pyscad module oclosedxffile() {
179 pyscad     pclosedxffile();
180 pyscad }
181 pyscad
182 pyscad module oclosedxflgblfile() {
183 pyscad     pclosedxflgblfile();
184 pyscad }
185 pyscad
186 pyscad module oclosedxflgsqfile() {
187 pyscad     pclosedxflgsqfile();
188 pyscad }
189 pyscad
190 pyscad module oclosedxflgVfile() {
191 pyscad     pclosedxflgVfile();
192 pyscad }
193 pyscad
194 pyscad module oclosedxfsmblfile() {
195 pyscad     pclosedxfsmblfile();
196 pyscad }
197 pyscad
```

```
198 pyscad module oclosedxfsmsqfile() {
199 pyscad     pclosedxfsmsqfile();
200 pyscad }
201 pyscad
202 pyscad module oclosedxfsmVfile() {
203 pyscad     pclosedxfsmVfile();
204 pyscad }
205 pyscad
206 pyscad module oclosedxfDTfile() {
207 pyscad     pclosedxfDTfile();
208 pyscad }
209 pyscad
210 pyscad module oclosedxfKHfile() {
211 pyscad     pclosedxfKHfile();
212 pyscad }
```

closegcodefile
closedxffile
dxfpostamble

The commands: `closegcodefile`, and `closedxffile` are used to close the files at the end of a program. For efficiency, each references the command: `dxfpostamble` which when called provides the boilerplate needed at the end of their respective files.

```
503 gcpscad module closegcodefile() {
504 gcpscad   if (generategcode == true) {
505 gcpscad     owriteone("M05");
506 gcpscad     owriteone("M02");
507 gcpscad     oclosegcodefile();
508 gcpscad   }
509 gcpscad }
510 gcpscad
511 gcpscad module dxfpostamble(arg) {
512 gcpscad     dxfwrite(arg,"0");
513 gcpscad     dxfwrite(arg,"ENDSEC");
514 gcpscad     dxfwrite(arg,"0");
515 gcpscad     dxfwrite(arg,"EOF");
516 gcpscad }
517 gcpscad
518 gcpscad module closedxffile() {
519 gcpscad   if (generatedxf == true) {
520 gcpscad     dxfwriteone("0");
521 gcpscad     dxfwriteone("ENDSEC");
522 gcpscad     dxfwriteone("0");
523 gcpscad     dxfwriteone("EOF");
524 gcpscad     oclosedxffile();
525 gcpscad //    echo("CLOSING");
526 gcpscad     if (large_ball_tool_no >  0) {    dxfpostamble(
                    large_ball_tool_no);
527 gcpscad       oclosedxflgblfile();
528 gcpscad     }
529 gcpscad     if (large_square_tool_no >  0) {    dxfpostamble(
                    large_square_tool_no);
530 gcpscad       oclosedxflgsqfile();
531 gcpscad     }
532 gcpscad     if (large_V_tool_no >  0) {    dxfpostamble(large_V_tool_no);
533 gcpscad       oclosedxflgVfile();
534 gcpscad     }
535 gcpscad     if (small_ball_tool_no >  0) {    dxfpostamble(
                    small_ball_tool_no);
536 gcpscad       oclosedxfsmblfile();
537 gcpscad     }
538 gcpscad     if (small_square_tool_no >  0) {    dxfpostamble(
                    small_square_tool_no);
539 gcpscad       oclosedxfsmsqfile();
540 gcpscad     }
541 gcpscad     if (small_V_tool_no >  0) {    dxfpostamble(small_V_tool_no);
542 gcpscad       oclosedxfsmVfile();
543 gcpscad     }
544 gcpscad     if (DT_tool_no >  0) {    dxfpostamble(DT_tool_no);
545 gcpscad       oclosedxfDTfile();
546 gcpscad     }
547 gcpscad     if (KH_tool_no >  0) {    dxfpostamble(KH_tool_no);
548 gcpscad       oclosedxfKHfile();
549 gcpscad     }
550 gcpscad   }
551 gcpscad }
```

## 2.8 Movement and Cutting

otm
ocut
orapid
rapid
rapidbx

With all the scaffolding in place, it is possible to model the tool: `otm`, (colors the tool model so as to differentiate cut areas) and cutting: `ocut`, as well as Rapid movements to position the tool to begin a cut: `orapid`, `rapid`, and `rapidbx` which will also need to write out files which represent the desired machine motions.

```
553 gcpscad module otm(ex, ey, ez, r,g,b) {
554 gcpscad color([r,g,b]) hull(){
555 gcpscad     translate([xpos(), ypos(), zpos()]){
556 gcpscad       select_tool(current_tool());
557 gcpscad     }
558 gcpscad     translate([ex, ey, ez]){
559 gcpscad       select_tool(current_tool());
560 gcpscad     }
561 gcpscad   }
562 gcpscad oset(ex, ey, ez);
563 gcpscad }
564 gcpscad
565 gcpscad module ocut(ex, ey, ez) {
566 gcpscad   //color([0.2,1,0.2]) hull(){
567 gcpscad   otm(ex, ey, ez, 0.2,1,0.2);
568 gcpscad }
569 gcpscad
570 gcpscad module orapid(ex, ey, ez) {
571 gcpscad   //color([0.93,0,0]) hull(){
572 gcpscad   otm(ex, ey, ez, 0.93,0,0);
573 gcpscad }
574 gcpscad
575 gcpscad module rapidbx(bx, by, bz, ex, ey, ez) {
576 gcpscad   //    writeln("G0 X",bx," Y", by, "Z", bz);
577 gcpscad   if (generategcode == true) {
578 gcpscad     writecomment("rapid");
579 gcpscad     owritesix("G0 X",str(ex)," Y", str(ey), " Z", str(ez));
580 gcpscad   }
581 gcpscad     orapid(ex, ey, ez);
582 gcpscad }
583 gcpscad
584 gcpscad module rapid(ex, ey, ez) {
585 gcpscad   //    writeln("G0 X",bx," Y", by, "Z", bz);
586 gcpscad   if (generategcode == true) {
587 gcpscad       writecomment("rapid");
588 gcpscad       owritesix("G0 X",str(ex)," Y", str(ey), " Z", str(ez));
589 gcpscad   }
590 gcpscad   orapid(ex, ey, ez);
591 gcpscad }
592 gcpscad
593 gcpscad module movetosafez() {
594 gcpscad   //this should be move to retract height
595 gcpscad   if (generategcode == true) {
596 gcpscad       writecomment("Move to safe Z to avoid workholding");
597 gcpscad       owriteone("G53G0Z-5.000");
598 gcpscad   }
599 gcpscad   orapid(getxpos(), getypos(), retractheight+55);
600 gcpscad }
601 gcpscad
602 gcpscad module begintoolpath(bx,by,bz) {
603 gcpscad   if (generategcode == true) {
604 gcpscad     writecomment("PREPOSITION FOR RAPID PLUNGE");
605 gcpscad     owritefour("G0X", str(bx), "Y",str(by));
606 gcpscad     owritetwo("Z", str(bz));
607 gcpscad   }
608 gcpscad   orapid(bx,by,bz);
609 gcpscad }
610 gcpscad
611 gcpscad module movetosafeheight() {
612 gcpscad   //this should be move to machine position
613 gcpscad   if (generategcode == true) {
614 gcpscad   //    writecomment("PREPOSITION FOR RAPID PLUNGE");Z25.650
615 gcpscad   //G1Z24.663F381.0 ,"F",str(plunge)
616 gcpscad     if (zeroheight == "Top") {
617 gcpscad       owritetwo("Z",str(retractheight));
618 gcpscad     }
619 gcpscad   }
620 gcpscad     orapid(getxpos(), getypos(), retractheight+55);
621 gcpscad }
622 gcpscad
623 gcpscad module cutoneaxis_setfeed(axis,depth,feed) {
```

```
624 gcpscad   if (generategcode == true) {
625 gcpscad   //    writecomment("PREPOSITION FOR RAPID PLUNGE");Z25.650
626 gcpscad   //G1Z24.663F381.0 ,"F",str(plunge) G1Z7.612F381.0
627 gcpscad     if (zeroheight == "Top") {
628 gcpscad       owritefive("G1",axis,str(depth),"F",str(feed));
629 gcpscad     }
630 gcpscad   }
631 gcpscad   if (axis == "X") {setxpos(depth);
632 gcpscad     ocut(depth, getypos(), getzpos());}
633 gcpscad     if (axis == "Y") {setypos(depth);
634 gcpscad       ocut(getxpos(), depth, getzpos());
635 gcpscad     }
636 gcpscad       if (axis == "Z") {setzpos(depth);
637 gcpscad         ocut(getxpos(), getypos(), depth);
638 gcpscad       }
639 gcpscad }
640 gcpscad
641 gcpscad module cut(ex, ey, ez) {
642 gcpscad   //    writeln("G0 X",bx," Y", by, "Z", bz);
643 gcpscad   if (generategcode == true) {
644 gcpscad      owritesix("G1 X",str(ex)," Y", str(ey), " Z", str(ez));
645 gcpscad   }
646 gcpscad   //if (generatesvg == true) {
647 gcpscad   //    owritesix("G1 X",str(ex)," Y", str(ey), " Z", str(ez));
648 gcpscad   //    orapid(getxpos(), getypos(), retractheight+5);
649 gcpscad   //    writesvgline(getxpos(),getypos(),ex,ey);
650 gcpscad   //}
651 gcpscad   ocut(ex, ey, ez);
652 gcpscad }
653 gcpscad
654 gcpscad module cutwithfeed(ex, ey, ez, feed) {
655 gcpscad   //    writeln("G0 X",bx," Y", by, "Z", bz);
656 gcpscad   if (generategcode == true) {
657 gcpscad   //    writecomment("rapid");
658 gcpscad     owriteeight("G1 X",str(ex)," Y", str(ey), " Z", str(ez),"F",str
                  (feed));
659 gcpscad   }
660 gcpscad   ocut(ex, ey, ez);
661 gcpscad }
662 gcpscad
663 gcpscad module endtoolpath() {
664 gcpscad   if (generategcode == true) {
665 gcpscad   //Z31.750
666 gcpscad   //    owriteone("G53G0Z-5.000");
667 gcpscad     owritetwo("Z",str(retractheight));
668 gcpscad   }
669 gcpscad   orapid(getxpos(),getypos(),retractheight);
670 gcpscad }
```

# 3 Cutting shapes, cut2Dshapes, and expansion

Certain basic shapes (arcs, circles, rectangles), will be incorporated in the main code. Other shapes will be added to the additional/optional file, `cut2Dshapes.scad` as they are developed, and of course the user is free to develop their own systems.

It is most expedient to test out new features in a new/separate file insofar as the file structures will allow (tool definitions for example will need to consolidated in 2.6.1) which will need to be included in the projects which will make use of said features until such time as they are added into the main `gcodepreview.scad` file.

A basic requirement for two-dimensional regions will be to define them so as to cut them out. Two different geometric treatments will be necessary: modeling the geometry which defines the region to be cut out (output as a DXF); and modeling the movement of the tool, the toolpath which will be used in creating the 3D model and outputting the G-code.

In the TUG presentation/paper: http://tug.org/TUGboat/tb40-2/tb125adams-3d.pdf a list of 2D shapes was put forward — which of these will need to be created, or if some more general solution will be put forward is uncertain. For the time being, shapes will be implemented on an as-needed basis, as modified by the interaction with the requirements of toolpaths.

The program Carbide Create has toolpath types and options which are as follows:

- Contour — No Offset — the default, this is already supported in the existing code
- Contour — Outside Offset
- Contour — Inside Offset
- Pocket — such toolpaths/geometry should include the rounding of the tool at the corners, c.f., cutrectangledxf

- 0
  - circle
  - ellipse (oval) (requires some sort of non-arc curve)
    - egg-shaped
  - annulus (one circle within another, forming a ring)
  - superellipse (see astroid below)
- 1
  - cone with rounded end (arc)see also "sector" under 3 below
- 2
  - semicircle/circular/half-circle segment (arc and a straight line); see also sector below
  - arch—curve possibly smoothly joining a pair of straight lines with a flat bottom
  - lens/vesica piscis (two convex curves)
  - lune/crescent (one convex, one concave curve)
  - heart (two curves)
  - tomoe (comma shape)—non-arc curves
- 3
  - triangle
    - equilateral
    - isosceles
    - right triangle
    - scalene
  - (circular) sector (two straight edges, one convex arc)
    - quadrant (90°)
    - sextants (60°)
    - octants (45°)
  - deltoid curve (three concave arcs)
  - Reuleaux triangle (three convex arcs)
  - arbelos (one convex, two concave arcs)
  - two straight edges, one concave arc—an example is the hyperbolic sector[1]
  - two convex, one concave arc
- 4
  - rectangle (including square) — cutrectangledxf, cutoutrectangledxf, rectangleoutlinedxf
  - parallelogram
  - rhombus
  - trapezoid/trapezium
  - kite
  - ring/annulus segment (straight line, concave arc, straight line, convex arc)
  - astroid (four concave arcs)
  - salinon (four semicircles)
  - three straight lines and one concave arc

Is the list of shapes for which there are not widely known names interesting for its lack of notoriety?

- two straight edges, one concave arcoddly, an asymmetric form (hyperbolic sector) has a name, but not the symmetrical—while the colloquial/prosaic arrowhead was considered, it was rejected as being better applied to the shape below. (Its also the shape used for the spaceship in the game Asteroids (or Hyperspace), but that is potentially confusing with astroid.) At the conference, Dr. Knuth suggested dart as a suitable term.
- two convex, one concave arcwith the above named, the term arrowhead is freed up to use as the name for this shape.
- three straight lines and one concave arc.

The first in particular is sorely needed for this project (its the result of inscribing a circle in a square or other regular geometric shape). Do these shapes have names in any other languages which might be used instead?

- Drill — note that this is implemented as the plunging of a tool centered on a circle and normally that circle is the same diameter as the tool which is used.
- Keyhole — also beginning from a circle, a nice feature for this would be to include/model the areas which should be cleared for the sake of reducing wear on the tool and ensuring chip clearance

Some further considerations:

- relationship of geometry to toolpath — arguably there should be an option for each toolpath (we will use Carbide Create as a reference implementation) which is to be supported. Note that there are several possibilities: modeling the tool movement, describing the outline which the tool will cut, modeling a reference shape for the toolpath
- tool geometry — it should be possible to include support for specialty tooling such as dove-tail cutters and to get an accurate 3D model, esp. for tooling which undercuts since they cannot be modeled in Carbide Create.
- feeds and speeds — if outputting G-code it would be nice to be able to import feeds and speeds from external files such as the .csv files used for user tool libraries in Carbide Create
- Starting and Max Depth — are there CAD programs which will make use of Z-axis information in a DXF? — would it be possible/necessary to further differentiate the DXF geometry? (currently written out separately for each toolpath in addition to one combined file)

## 3.1 Arcs for toolpaths and DXFs

A further consideration here is that G-code supports arcs in addition to the lines and polylines already implemented.

Implementing arcs wants at least the following options for quadrant and direction:

- cutarcNWCW — cut the upper-left quadrant of a circle moving clockwise
- cutarcNWCC — upper-left quadrant counter-clockwise
- cutarcNECW
- cutarcNECC
- cutarcSECW
- cutarcSECC
- cutarcNECW
- cutarcNECC
- cutcircleCW — while it wont matter for generating a DXF, when G-code is implemented direction of cut will be a consideration for that
- cutcircleCCdxf

It will be necessary to have two separate representations of arcs — the DXF may be easily and directly supported with a single command, but representing the matching tool movement in OpenSCAD will require a series of short line movements which approximate the arc. At this time, the current version of Carbide Create only imports circles in DXF as curves, any other example is converted into polylines — unfortunately, the implementation of this is not such as would allow directly matching that representation. A work-around to import a DXF as curves is to convert the arc into a reasonable number of line segments so as to approximate the arc.

Note that there are the following representations/interfaces for representing an arc:

- G-code — G2 (clockwise) and G3 (counter-clockwise) arcs may be specified, and since the endpoint is the positional requirement, it is most likely best to use the offset to the center (I and J), rather than the radius parameter (K) `G2/3 ...`
- DXF — `dxfarc(xcenter, ycenter, radius, anglebegin, endangle, tn)`
- approximation of arc using lines (OpenSCAD) — note that this may also be used in DXF so as to sidestep the question of how many line segments there would be for a given arc representation

Cutting the quadrant arcs will greatly simplify the calculation and interface for the modules. A full set of 8 will be necessary, then circles may either be stitched together manually or a pair of modules made for them.

At this time, despite what the module names imply (`cutarcNWCWdxf`, &c.), only cutting and DXF generation is supported. Adding support for G-code will be done at a later time. Since these modules will ultimately support G-code, the interface will assume the stored `xpos` and `ypos` as the origin. Parameters which will need to be passed in are:

- tn
- ex
- ey
- ez — allowing a different Z position will make possible threading and similar helical toolpaths
- xcenter — the center position will be specified as an absolute position which will require calculating the offset when it is used for G-code's IJ, for which xctr/yctr are suggested
- ycenter
- radius — while this could be calculated, passing it in as a parameter is both convenient and acts as a check on the other parameters

Since OpenSCAD does not have an arc movement command it is necessary to iterate through a loop: arcloop (clockwise), narcloop (counterclockwise) to handle the drawing and processing of the cut() toolpaths as short line segments which additionally affords a single point of control for adding additional features such as allowing the depth to vary as one cuts along an arc (two when the need to have a version which steps down):

arcloop
narcloop

```
672 gcpscad module arcloop(barc,earc, xcenter, ycenter, radius) {
673 gcpscad   for (i = [barc : abs(1) : earc]) {
674 gcpscad         cut(xcenter + radius * cos(i),
675 gcpscad         ycenter + radius * sin(i),
676 gcpscad         getzpos()-(gettzpos())
677 gcpscad         );
678 gcpscad     setxpos(xcenter + radius * cos(i));
679 gcpscad     setypos(ycenter + radius * sin(i));
680 gcpscad   }
681 gcpscad }
682 gcpscad
683 gcpscad module narcloop(barc,earc, xcenter, ycenter, radius) {
684 gcpscad   for (i = [barc : -1 : earc]) {
685 gcpscad         cut(xcenter + radius * cos(i),
686 gcpscad         ycenter + radius * sin(i),
687 gcpscad         getzpos()-(gettzpos())
688 gcpscad         );
689 gcpscad     setxpos(xcenter + radius * cos(i));
690 gcpscad     setypos(ycenter + radius * sin(i));
691 gcpscad   }
692 gcpscad }
```

The various textual versions are quite obvious:

```
694 gcpscad module cutarcNECCdxf(ex, ey, ez, xcenter, ycenter, radius, tn) {
695 gcpscad   dxfarc(xcenter,ycenter,radius,0,90, tn);
696 gcpscad   settzpos((getzpos()-ez)/90);
697 gcpscad     arcloop(1,90, xcenter, ycenter, radius);
698 gcpscad }
699 gcpscad
700 gcpscad module cutarcNWCCdxf(ex, ey, ez, xcenter, ycenter, radius, tn) {
701 gcpscad   dxfarc(xcenter,ycenter,radius,90,180, tn);
702 gcpscad   settzpos((getzpos()-ez)/90);
703 gcpscad     arcloop(91,180, xcenter, ycenter, radius);
704 gcpscad }
705 gcpscad
706 gcpscad module cutarcSWCCdxf(ex, ey, ez, xcenter, ycenter, radius, tn) {
707 gcpscad   dxfarc(xcenter,ycenter,radius,180,270, tn);
708 gcpscad   settzpos((getzpos()-ez)/90);
709 gcpscad     arcloop(181,270, xcenter, ycenter, radius);
710 gcpscad }
711 gcpscad
712 gcpscad module cutarcSECCdxf(ex, ey, ez, xcenter, ycenter, radius, tn) {
713 gcpscad   dxfarc(xcenter,ycenter,radius,270,360, tn);
714 gcpscad   settzpos((getzpos()-ez)/90);
715 gcpscad     arcloop(271,360, xcenter, ycenter, radius);
716 gcpscad }
717 gcpscad
718 gcpscad module cutarcNECWdxf(ex, ey, ez, xcenter, ycenter, radius, tn) {
719 gcpscad   dxfarc(xcenter,ycenter,radius,0,90, tn);
720 gcpscad   settzpos((getzpos()-ez)/90);
721 gcpscad     narcloop(89,0, xcenter, ycenter, radius);
722 gcpscad }
723 gcpscad
724 gcpscad module cutarcSECWdxf(ex, ey, ez, xcenter, ycenter, radius, tn) {
```

```
725 gcpscad    dxfarc(xcenter,ycenter,radius,270,360, tn);
726 gcpscad    settzpos((getzpos()-ez)/90);
727 gcpscad      narcloop(359,270, xcenter, ycenter, radius);
728 gcpscad }
729 gcpscad
730 gcpscad module cutarcSWCWdxf(ex, ey, ez, xcenter, ycenter, radius, tn) {
731 gcpscad    dxfarc(xcenter,ycenter,radius,180,270, tn);
732 gcpscad    settzpos((getzpos()-ez)/90);
733 gcpscad      narcloop(269,180, xcenter, ycenter, radius);
734 gcpscad }
735 gcpscad
736 gcpscad module cutarcNWCWdxf(ex, ey, ez, xcenter, ycenter, radius, tn) {
737 gcpscad    dxfarc(xcenter,ycenter,radius,90,180, tn);
738 gcpscad    settzpos((getzpos()-ez)/90);
739 gcpscad      narcloop(179,90, xcenter, ycenter, radius);
740 gcpscad }
```

Using such commands to create a circle is quite straight-forward:

```
cutarcNECCdxf(-stocklength/4, stockwidth/4+stockwidth/16, -stockthickness, -stocklength/4, stockwidth/4
cutarcNWCCdxf(-(stocklength/4+stockwidth/16), stockwidth/4, -stockthickness, -stocklength/4, stockwidth,
cutarcSWCCdxf(-stocklength/4, stockwidth/4-stockwidth/16, -stockthickness, -stocklength/4, stockwidth/4
cutarcSECCdxf(-(stocklength/4-stockwidth/16), stockwidth/4, -stockthickness, -stocklength/4, stockwidth,
```

## 3.2 Keyhole toolpath and undercut tooling

cutkeyhole toolpath The first topologically unusual toolpath is `cutkeyhole toolpath` — where other toolpaths have a direct correspondence between the associated geometry and the area cut, that Keyhole toolpaths may be used with tooling which undercuts will result in the creation of two different physical physical regions: the visible surface matching the union of the tool perimeter at the entry point and the linear movement of the shaft and the larger region of the tool perimeter at the depth which the tool is plunged to and moved along.

Tooling for such toolpaths is defined at paragraph 2.6.1.2

Due to the possibility of rotation, for the in-between positions there are more cases than one would think for each quadrant there are the following possibilities:

- one node on the clockwise side is outside of the quadrant
- two nodes on the clockwise side are outside of the quadrant
- all nodes are w/in the quadrant
- one node on the counter-clockwise side is outside of the quadrant
- two nodes on the counter-clockwise side are outside of the quadrant

Supporting all of these would require trigonometric comparisons in the If else blocks, so only the 4 quadrants, N, S, E, and W will be supported in the initial version. This will be done by wrapping the command with a version which only accepts those options:

```
742 gcpscad module cutkeyhole_toolpath(kh_start_depth, kh_max_depth,
              kht_direction, kh_distance, kh_tool_no) {
743 gcpscad if (kht_direction == "N") {
744 gcpscad    cutKH_toolpath_degrees(kh_start_depth, kh_max_depth, 90,
                 kh_distance, kh_tool_no);
745 gcpscad      } else if (kht_direction == "S") {
746 gcpscad    cutKH_toolpath_degrees(kh_start_depth, kh_max_depth, 270,
                 kh_distance, kh_tool_no);
747 gcpscad      } else if (kht_direction == "E") {
748 gcpscad    cutKH_toolpath_degrees(kh_start_depth, kh_max_depth, 0,
                 kh_distance, kh_tool_no);
749 gcpscad      } else if (kht_direction == "W") {
750 gcpscad    cutKH_toolpath_degrees(kh_start_depth, kh_max_depth, 180,
                 kh_distance, kh_tool_no);
751 gcpscad      }
752 gcpscad }
```

cutKH toolpath degrees The original version of the command, `cutKH toolpath degrees` retains an interface which allows calling it for arbitrary beginning and ending points of an arc. Note that code is still present for the partial calculation of one quadrant (for the case of all nodes within the quadrant).

The first task is to place a circle at the origin which is invariant of angle:

```
754 gcpscad module cutKH_toolpath_degrees(kh_start_depth, kh_max_depth,
              kh_angle, kh_distance, kh_tool_no) {
755 gcpscad //Circle at entry hole
756 gcpscad dxfarc(getxpos(),getypos(),tool_diameter(KH_tool_no, (7))/2,0,90,
              KH_tool_no);
```

```
757 gcpscad dxfarc(getxpos(),getypos(),tool_diameter(KH_tool_no, (7))/2,90,180,
            KH_tool_no);
758 gcpscad dxfarc(getxpos(),getypos(),tool_diameter(KH_tool_no, (7))
           /2,180,270, KH_tool_no);
759 gcpscad dxfarc(getxpos(),getypos(),tool_diameter(KH_tool_no, (7))
           /2,270,360, KH_tool_no);
```

Then it will be necessary to test for each possible case in a series of If Else blocks:

```
 1 gcpscad //Outlines of entry hole and slot
 2 gcpscad   if (kh_angle == 0) {
 3 gcpscad     //Lower left of entry hole
 4 gcpscad     dxfarc(getxpos(),getypos(),9.525/2,180,270, KH_tool_no);
 5 gcpscad     //Upper left of entry hole
 6 gcpscad     dxfarc(getxpos(),getypos(),9.525/2,90,180, KH_tool_no);
 7 gcpscad     //Upper right of entry hole
 8 gcpscad     dxfarc(getxpos(),getypos(),9.525/2,90-acos(tool_diameter(
                   KH_tool_no, 5)/tool_diameter(KH_tool_no, 1)), 90, KH_tool_no
                   );
 9 gcpscad     //Lower right of entry hole
10 gcpscad     dxfarc(getxpos(),getypos(),9.525/2,270, 270+acos(tool_diameter(
                   KH_tool_no, 5)/tool_diameter(KH_tool_no, 1)), KH_tool_no);
11 gcpscad     //Actual line of cut
12 gcpscad     dxfpolyline(getxpos(),getypos(),getxpos()+kh_distance,getypos()
                   );
13 gcpscad     //upper right of slot
14 gcpscad     dxfarc(getxpos()+kh_distance,getypos(),tool_diameter(KH_tool_no
                   , (kh_max_depth+4.36))/2,0,90, KH_tool_no);
15 gcpscad     //lower right of slot
16 gcpscad     dxfarc(getxpos()+kh_distance,getypos(),tool_diameter(KH_tool_no
                   , (kh_max_depth+4.36))/2,270,360, KH_tool_no);
17 gcpscad     //upper right slot
18 gcpscad     dxfpolyline(
19 gcpscad         getxpos()+(sqrt((tool_diameter(KH_tool_no,1)^2)-(
                       tool_diameter(KH_tool_no,5)^2))/2),
20 gcpscad         getypos()+tool_diameter(KH_tool_no, (kh_max_depth))/2,//( (
                       kh_max_depth-6.34))/2)^2-(tool_diameter(KH_tool_no, (
                       kh_max_depth-6.34))/2)^2,
21 gcpscad         getxpos()+kh_distance,
22 gcpscad     //end position at top of slot
23 gcpscad         getypos()+tool_diameter(KH_tool_no, (kh_max_depth))/2,
24 gcpscad         KH_tool_no);
25 gcpscad     //lower right slot
26 gcpscad     dxfpolyline(
27 gcpscad         getxpos()+(sqrt((tool_diameter(KH_tool_no,1)^2)-(
                       tool_diameter(KH_tool_no,5)^2))/2),
28 gcpscad         getypos()-tool_diameter(KH_tool_no, (kh_max_depth))/2,//( (
                       kh_max_depth-6.34))/2)^2-(tool_diameter(KH_tool_no, (
                       kh_max_depth-6.34))/2)^2,
29 gcpscad         getxpos()+kh_distance,
30 gcpscad     //end position at top of slot
31 gcpscad         getypos()-tool_diameter(KH_tool_no, (kh_max_depth))/2,
32 gcpscad         KH_tool_no);
33 gcpscad     hull(){
34 gcpscad       translate([xpos(), ypos(), zpos()]){
35 gcpscad         gcp_keyhole_shaft(6.35, 9.525);
36 gcpscad       }
37 gcpscad       translate([xpos(), ypos(), zpos()-kh_max_depth]){
38 gcpscad         gcp_keyhole_shaft(6.35, 9.525);
39 gcpscad       }
40 gcpscad     }
41 gcpscad     hull(){
42 gcpscad       translate([xpos(), ypos(), zpos()-kh_max_depth]){
43 gcpscad         gcp_keyhole_shaft(6.35, 9.525);
44 gcpscad       }
45 gcpscad       translate([xpos()+kh_distance, ypos(), zpos()-kh_max_depth]){
46 gcpscad         gcp_keyhole_shaft(6.35, 9.525);
47 gcpscad       }
48 gcpscad     }
49 gcpscad     cutwithfeed(getxpos(),getypos(),-kh_max_depth,feed);
50 gcpscad     cutwithfeed(getxpos()+kh_distance,getypos(),-kh_max_depth,feed)
                   ;
51 gcpscad     setxpos(getxpos()-kh_distance);
52 gcpscad   } else if (kh_angle > 0 && kh_angle < 90) {
53 gcpscad //echo(kh_angle);
54 gcpscad   dxfarc(getxpos(),getypos(),tool_diameter(KH_tool_no, (
                 kh_max_depth))/2,90+kh_angle,180+kh_angle, KH_tool_no);
55 gcpscad   dxfarc(getxpos(),getypos(),tool_diameter(KH_tool_no, (
```

```
              kh_max_depth))/2,180+kh_angle,270+kh_angle, KH_tool_no);
56 gcpscad dxfarc(getxpos(),getypos(),tool_diameter(KH_tool_no, (kh_max_depth)
            )/2,kh_angle+asin((tool_diameter(KH_tool_no, (kh_max_depth+4.36)
            )/2)/(tool_diameter(KH_tool_no, (kh_max_depth))/2)),90+kh_angle,
             KH_tool_no);
57 gcpscad dxfarc(getxpos(),getypos(),tool_diameter(KH_tool_no, (kh_max_depth)
            )/2,270+kh_angle,360+kh_angle-asin((tool_diameter(KH_tool_no, (
            kh_max_depth+4.36))/2)/(tool_diameter(KH_tool_no, (kh_max_depth)
            )/2)), KH_tool_no);
58 gcpscad dxfarc(getxpos()+(kh_distance*cos(kh_angle)),
59 gcpscad   getypos()+(kh_distance*sin(kh_angle)),tool_diameter(KH_tool_no, (
              kh_max_depth+4.36))/2,0+kh_angle,90+kh_angle, KH_tool_no);
60 gcpscad dxfarc(getxpos()+(kh_distance*cos(kh_angle)),getypos()+(kh_distance
            *sin(kh_angle)),tool_diameter(KH_tool_no, (kh_max_depth+4.36))
            /2,270+kh_angle,360+kh_angle, KH_tool_no);
61 gcpscad dxfpolyline( getxpos()+tool_diameter(KH_tool_no, (kh_max_depth))/2*
            cos(kh_angle+asin((tool_diameter(KH_tool_no, (kh_max_depth+4.36)
            )/2)/(tool_diameter(KH_tool_no, (kh_max_depth))/2))),
62 gcpscad  getypos()+tool_diameter(KH_tool_no, (kh_max_depth))/2*sin(kh_angle
             +asin((tool_diameter(KH_tool_no, (kh_max_depth+4.36))/2)/(
             tool_diameter(KH_tool_no, (kh_max_depth))/2))),
63 gcpscad  getxpos()+(kh_distance*cos(kh_angle))-((tool_diameter(KH_tool_no,
             (kh_max_depth+4.36))/2)*sin(kh_angle)),
64 gcpscad  getypos()+(kh_distance*sin(kh_angle))+((tool_diameter(KH_tool_no,
             (kh_max_depth+4.36))/2)*cos(kh_angle)), KH_tool_no);
65 gcpscad //echo("a",tool_diameter(KH_tool_no, (kh_max_depth+4.36))/2);
66 gcpscad //echo("c",tool_diameter(KH_tool_no, (kh_max_depth))/2);
67 gcpscad echo("Aangle",asin((tool_diameter(KH_tool_no, (kh_max_depth+4.36))
            /2)/(tool_diameter(KH_tool_no, (kh_max_depth))/2)));
68 gcpscad //echo(kh_angle);
69 gcpscad  cutwithfeed(getxpos()+(kh_distance*cos(kh_angle)),getypos()+(
             kh_distance*sin(kh_angle)),-kh_max_depth,feed);
70 gcpscad  setxpos(getxpos()-(kh_distance*cos(kh_angle)));
71 gcpscad  setypos(getypos()-(kh_distance*sin(kh_angle)));
72 gcpscad   } else if (kh_angle == 90) {
73 gcpscad     //Lower left of entry hole
74 gcpscad     dxfarc(getxpos(),getypos(),9.525/2,180,270, KH_tool_no);
75 gcpscad     //Lower right of entry hole
76 gcpscad     dxfarc(getxpos(),getypos(),9.525/2,270,360, KH_tool_no);
77 gcpscad     //Upper right of entry hole
78 gcpscad     dxfarc(getxpos(),getypos(),9.525/2,0,acos(tool_diameter(
                 KH_tool_no, 5)/tool_diameter(KH_tool_no, 1)), KH_tool_no);
79 gcpscad     //Upper left of entry hole
80 gcpscad     dxfarc(getxpos(),getypos(),9.525/2,180-acos(tool_diameter(
                 KH_tool_no, 5)/tool_diameter(KH_tool_no, 1)), 180,KH_tool_no
                 );
81 gcpscad     //Actual line of cut
82 gcpscad     dxfpolyline(getxpos(),getypos(),getxpos(),getypos()+kh_distance
                 );
83 gcpscad     //upper right of slot
84 gcpscad     dxfarc(getxpos(),getypos()+kh_distance,tool_diameter(KH_tool_no
                 , (kh_max_depth+4.36))/2,0,90, KH_tool_no);
85 gcpscad     //upper left of slot
86 gcpscad     dxfarc(getxpos(),getypos()+kh_distance,tool_diameter(KH_tool_no
                 , (kh_max_depth+6.35))/2,90,180, KH_tool_no);
87 gcpscad     //right of slot
88 gcpscad     dxfpolyline(
89 gcpscad         getxpos()+tool_diameter(KH_tool_no, (kh_max_depth))/2,
90 gcpscad         getypos()+(sqrt((tool_diameter(KH_tool_no,1)^2)-(
                     tool_diameter(KH_tool_no,5)^2))/2),//( (kh_max_depth
                     -6.34))/2)^2-(tool_diameter(KH_tool_no, (kh_max_depth
                     -6.34))/2)^2,
91 gcpscad         getxpos()+tool_diameter(KH_tool_no, (kh_max_depth))/2,
92 gcpscad     //end position at top of slot
93 gcpscad         getypos()+kh_distance,
94 gcpscad         KH_tool_no);
95 gcpscad     dxfpolyline(getxpos()-tool_diameter(KH_tool_no, (kh_max_depth))
                 /2, getypos()+(sqrt((tool_diameter(KH_tool_no,1)^2)-(
                 tool_diameter(KH_tool_no,5)^2))/2), getxpos()-tool_diameter(
                 KH_tool_no, (kh_max_depth+6.35))/2,getypos()+kh_distance,
                 KH_tool_no);
96 gcpscad     hull(){
97 gcpscad       translate([xpos(), ypos(), zpos()]){
98 gcpscad         gcp_keyhole_shaft(6.35, 9.525);
99 gcpscad       }
100 gcpscad      translate([xpos(), ypos(), zpos()-kh_max_depth]){
101 gcpscad        gcp_keyhole_shaft(6.35, 9.525);
102 gcpscad      }
```

```
103 gcpscad     }
104 gcpscad     hull(){
105 gcpscad       translate([xpos(), ypos(), zpos()-kh_max_depth]){
106 gcpscad         gcp_keyhole_shaft(6.35, 9.525);
107 gcpscad       }
108 gcpscad       translate([xpos(), ypos()+kh_distance, zpos()-kh_max_depth]){
109 gcpscad         gcp_keyhole_shaft(6.35, 9.525);
110 gcpscad       }
111 gcpscad     }
112 gcpscad     cutwithfeed(getxpos(),getypos(),-kh_max_depth,feed);
113 gcpscad     cutwithfeed(getxpos(),getypos()+kh_distance,-kh_max_depth,feed)
                ;
114 gcpscad     setypos(getypos()-kh_distance);
115 gcpscad   } else if (kh_angle == 180) {
116 gcpscad     //Lower right of entry hole
117 gcpscad     dxfarc(getxpos(),getypos(),9.525/2,270,360, KH_tool_no);
118 gcpscad     //Upper right of entry hole
119 gcpscad     dxfarc(getxpos(),getypos(),9.525/2,0,90, KH_tool_no);
120 gcpscad     //Upper left of entry hole
121 gcpscad     dxfarc(getxpos(),getypos(),9.525/2,90, 90+acos(tool_diameter(
                KH_tool_no, 5)/tool_diameter(KH_tool_no, 1)), KH_tool_no);
122 gcpscad     //Lower left of entry hole
123 gcpscad     dxfarc(getxpos(),getypos(),9.525/2, 270-acos(tool_diameter(
                KH_tool_no, 5)/tool_diameter(KH_tool_no, 1)), 270,
                KH_tool_no);
124 gcpscad     //upper left of slot
125 gcpscad     dxfarc(getxpos()-kh_distance,getypos(),tool_diameter(KH_tool_no
                , (kh_max_depth+6.35))/2,90,180, KH_tool_no);
126 gcpscad     //lower left of slot
127 gcpscad     dxfarc(getxpos()-kh_distance,getypos(),tool_diameter(KH_tool_no
                , (kh_max_depth+6.35))/2,180,270, KH_tool_no);
128 gcpscad     //Actual line of cut
129 gcpscad     dxfpolyline(getxpos(),getypos(),getxpos()-kh_distance,getypos()
                );
130 gcpscad     //upper left slot
131 gcpscad     dxfpolyline(
132 gcpscad         getxpos()-(sqrt((tool_diameter(KH_tool_no,1)^2)-(
                    tool_diameter(KH_tool_no,5)^2))/2),
133 gcpscad         getypos()+tool_diameter(KH_tool_no, (kh_max_depth))/2,//( (
                    kh_max_depth-6.34))/2)^2-(tool_diameter(KH_tool_no, (
                    kh_max_depth-6.34))/2)^2,
134 gcpscad         getxpos()-kh_distance,
135 gcpscad     //end position at top of slot
136 gcpscad         getypos()+tool_diameter(KH_tool_no, (kh_max_depth))/2,
137 gcpscad         KH_tool_no);
138 gcpscad     //lower right slot
139 gcpscad     dxfpolyline(
140 gcpscad         getxpos()-(sqrt((tool_diameter(KH_tool_no,1)^2)-(
                    tool_diameter(KH_tool_no,5)^2))/2),
141 gcpscad         getypos()-tool_diameter(KH_tool_no, (kh_max_depth))/2,//( (
                    kh_max_depth-6.34))/2)^2-(tool_diameter(KH_tool_no, (
                    kh_max_depth-6.34))/2)^2,
142 gcpscad         getxpos()-kh_distance,
143 gcpscad     //end position at top of slot
144 gcpscad         getypos()-tool_diameter(KH_tool_no, (kh_max_depth))/2,
145 gcpscad         KH_tool_no);
146 gcpscad     hull(){
147 gcpscad       translate([xpos(), ypos(), zpos()]){
148 gcpscad         gcp_keyhole_shaft(6.35, 9.525);
149 gcpscad       }
150 gcpscad       translate([xpos(), ypos(), zpos()-kh_max_depth]){
151 gcpscad         gcp_keyhole_shaft(6.35, 9.525);
152 gcpscad       }
153 gcpscad     }
154 gcpscad     hull(){
155 gcpscad       translate([xpos(), ypos(), zpos()-kh_max_depth]){
156 gcpscad         gcp_keyhole_shaft(6.35, 9.525);
157 gcpscad       }
158 gcpscad       translate([xpos()-kh_distance, ypos(), zpos()-kh_max_depth]){
159 gcpscad         gcp_keyhole_shaft(6.35, 9.525);
160 gcpscad       }
161 gcpscad     }
162 gcpscad     cutwithfeed(getxpos(),getypos(),-kh_max_depth,feed);
163 gcpscad     cutwithfeed(getxpos()-kh_distance,getypos(),-kh_max_depth,feed)
                ;
164 gcpscad     setxpos(getxpos()+kh_distance);
165 gcpscad   } else if (kh_angle == 270) {
166 gcpscad     //Upper right of entry hole
```

```
167 gcpscad     dxfarc(getxpos(),getypos(),9.525/2,0,90, KH_tool_no);
168 gcpscad     //Upper left of entry hole
169 gcpscad     dxfarc(getxpos(),getypos(),9.525/2,90,180, KH_tool_no);
170 gcpscad     //lower right of slot
171 gcpscad     dxfarc(getxpos(),getypos()-kh_distance,tool_diameter(KH_tool_no
                    , (kh_max_depth+4.36))/2,270,360, KH_tool_no);
172 gcpscad     //lower left of slot
173 gcpscad     dxfarc(getxpos(),getypos()-kh_distance,tool_diameter(KH_tool_no
                    , (kh_max_depth+4.36))/2,180,270, KH_tool_no);
174 gcpscad     //Actual line of cut
175 gcpscad     dxfpolyline(getxpos(),getypos(),getxpos(),getypos()-kh_distance
                    );
176 gcpscad     //right of slot
177 gcpscad     dxfpolyline(
178 gcpscad         getxpos()+tool_diameter(KH_tool_no, (kh_max_depth))/2,
179 gcpscad         getypos()-(sqrt((tool_diameter(KH_tool_no,1)^2)-(
                        tool_diameter(KH_tool_no,5)^2))/2),//( (kh_max_depth
                        -6.34))/2)^2-(tool_diameter(KH_tool_no, (kh_max_depth
                        -6.34))/2)^2,
180 gcpscad         getxpos()+tool_diameter(KH_tool_no, (kh_max_depth))/2,
181 gcpscad     //end position at top of slot
182 gcpscad         getypos()-kh_distance,
183 gcpscad         KH_tool_no);
184 gcpscad     //left of slot
185 gcpscad     dxfpolyline(
186 gcpscad         getxpos()-tool_diameter(KH_tool_no, (kh_max_depth))/2,
187 gcpscad         getypos()-(sqrt((tool_diameter(KH_tool_no,1)^2)-(
                        tool_diameter(KH_tool_no,5)^2))/2),//( (kh_max_depth
                        -6.34))/2)^2-(tool_diameter(KH_tool_no, (kh_max_depth
                        -6.34))/2)^2,
188 gcpscad         getxpos()-tool_diameter(KH_tool_no, (kh_max_depth))/2,
189 gcpscad     //end position at top of slot
190 gcpscad         getypos()-kh_distance,
191 gcpscad         KH_tool_no);
192 gcpscad     //Lower right of entry hole
193 gcpscad     dxfarc(getxpos(),getypos(),9.525/2,360-acos(tool_diameter(
                    KH_tool_no, 5)/tool_diameter(KH_tool_no, 1)), 360,
                    KH_tool_no);
194 gcpscad     //Lower left of entry hole
195 gcpscad     dxfarc(getxpos(),getypos(),9.525/2,180, 180+acos(tool_diameter(
                    KH_tool_no, 5)/tool_diameter(KH_tool_no, 1)), KH_tool_no);
196 gcpscad     hull(){
197 gcpscad       translate([xpos(), ypos(), zpos()]){
198 gcpscad         gcp_keyhole_shaft(6.35, 9.525);
199 gcpscad       }
200 gcpscad       translate([xpos(), ypos(), zpos()-kh_max_depth]){
201 gcpscad         gcp_keyhole_shaft(6.35, 9.525);
202 gcpscad       }
203 gcpscad     }
204 gcpscad     hull(){
205 gcpscad       translate([xpos(), ypos(), zpos()-kh_max_depth]){
206 gcpscad         gcp_keyhole_shaft(6.35, 9.525);
207 gcpscad       }
208 gcpscad       translate([xpos(), ypos()-kh_distance, zpos()-kh_max_depth]){
209 gcpscad         gcp_keyhole_shaft(6.35, 9.525);
210 gcpscad       }
211 gcpscad     }
212 gcpscad     cutwithfeed(getxpos(),getypos(),-kh_max_depth,feed);
213 gcpscad     cutwithfeed(getxpos(),getypos()-kh_distance,-kh_max_depth,feed)
                    ;
214 gcpscad     setypos(getypos()+kh_distance);
215 gcpscad   }
216 gcpscad }
```

## 3.3 Shapes and tool movement

The majority of commands will be more general, focusing on tooling which is generally supported by this library, moving in lines and arcs so as to describe shapes which lend themselves to representation with those tool and which match up with both toolpaths and supported geometry in Carbide Create, and the usage requirements of the typical user.

### 3.3.1 Generalized commands and cuts

The first consideration is a naming convention which will allow a generalized set of associated commands to be defined. The initial version will only create OpenSCAD commands for 3D modeling and write out matching DXF files. At a later time this will be extended with G-code support.

begincutdxf **3.3.1.1 begincutdxf** The first command, `begincutdxf` will need to allow the machine to rapid to the beginning point of the cut and then rapid down to the surface of the stock, and then plunge down to the depth of the cut. The implementation will need to allow for a hook where the Depth per Pass is applied to the plunge operation so that multiple passes are made.

The first module will ensure that the tool is safely up above the stock and will rapid to the position specified at the retract height (moving to that position as an initial step, then will `cutwithfeed` to the specified position at the specified feed rate. Despite `dxf` being included in the filename no change is made to the dxf file at this time, this simply indicates that this file is preparatory to the
continuecutdxf use of `continuecutdxf`.

```
867 gcpscad module begincutdxf(rh, ex, ey, ez, fr) {
868 gcpscad   rapid(getxpos(),getypos(),rh);
869 gcpscad   cutwithfeed(ex,ey,ez,fr);
870 gcpscad }
```

```
872 gcpscad module continuecutdxf(ex, ey, ez, fr) {
873 gcpscad   cutwithfeed(ex,ey,ez,fr);
874 gcpscad }
```

**3.3.1.2 Rectangles** Cutting rectangles while writing out their perimeter in the DXF files (so that they may be assigned a matching toolpath in a traditional CAM program upon import) will require the origin coordinates, height and width and depth of the pocket, and the tool # so that the corners may have a radius equal to the tool which is used. Whether a given module is an interior pocket or an outline (interior or exterior) will be determined by the specifics of the module and its usage/positioning, with `outline` being added to those modules which cut perimeter.

A further consideration is that cut orientation as an option should be accounted for if writing out G-code, as well as stepover, and the nature of initial entry (whether ramping in would be implemented, and if so, at what angle). Advanced toolpath strategies such as trochoidal milling could also be implemented.

cutrectangledxf Th routine `cutrectangledxf`cuts the outline of a rectangle creating sharp corners. Note that the initial version would work as a beginning point for vertical cutting if the `hull()` operation was removed and the loop was uncommented:

```
876 gcpscad module cutrectangledxf(bx, by, bz, rwidth, rheight, rdepth, rtn)
        {//passes
877 gcpscad   movetosafez();
878 gcpscad   hull(){
879 gcpscad     //  for (i = [0 : abs(1) : passes]) {
880 gcpscad     //      rapid(bx+tool_radius(rtn)+i*(rwidth-tool_diameter(
                current_tool()))/passes,bx+tool_radius(rtn),1);
881 gcpscad     //      cutwithfeed(bx+tool_radius(rtn)+i*(rwidth-tool_diameter
                (current_tool()))/passes,by+tool_radius(rtn),bz-rdepth,feed)
                ;
882 gcpscad     //      cutwithfeed(bx+tool_radius(rtn)+i*(rwidth-tool_diameter
                (current_tool()))/passes,by+rheight-tool_radius(rtn),bz-
                rdepth,feed);
883 gcpscad
884 gcpscad     cutwithfeed(bx+tool_radius(rtn),by+tool_radius(rtn),bz-rdepth,
                feed);
885 gcpscad     cutwithfeed(bx+rwidth-tool_radius(rtn),by+tool_radius(rtn),bz-
                rdepth,feed);
886 gcpscad     cutwithfeed(bx+rwidth-tool_radius(rtn),by+rheight-tool_radius(
                rtn),bz-rdepth,feed);
887 gcpscad     cutwithfeed(bx+tool_radius(rtn),by+rheight-tool_radius(rtn),bz-
                rdepth,feed);
888 gcpscad   }
889 gcpscad   //dxfarc(xcenter,ycenter,radius,anglebegin,endangle, tn)
890 gcpscad   dxfarc(bx+tool_radius(rtn),by+tool_radius(rtn),tool_radius(rtn)
                ,180,270, rtn);
891 gcpscad   //dxfpolyline(xbegin,ybegin,xend,yend, tn)
892 gcpscad   dxfpolyline(bx,by+tool_radius(rtn),bx,by+rheight-tool_radius(rtn)
                , rtn);
893 gcpscad   dxfarc(bx+tool_radius(rtn),by+rheight-tool_radius(rtn),
                tool_radius(rtn),90,180, rtn);
894 gcpscad   dxfpolyline(bx+tool_radius(rtn),by+rheight,bx+rwidth-tool_radius(
                rtn),by+rheight, rtn);
895 gcpscad   dxfarc(bx+rwidth-tool_radius(rtn),by+rheight-tool_radius(rtn),
                tool_radius(rtn),0,90, rtn);
896 gcpscad   dxfpolyline(bx+rwidth,by+rheight-tool_radius(rtn),bx+rwidth,by+
                tool_radius(rtn), rtn);
897 gcpscad   dxfarc(bx+rwidth-tool_radius(rtn),by+tool_radius(rtn),tool_radius
                (rtn),270,360, rtn);
898 gcpscad   dxfpolyline(bx+rwidth-tool_radius(rtn),by,bx+tool_radius(rtn),by,
```

```
                rtn);
899 gcpscad }
```

cutrectangleoutlinedxf A matching command: `cutrectangleoutlinedxf` cuts the outline of a rounded rectangle and is a simplification of the above:

```
901 gcpscad module cutrectangleoutlinedxf(bx, by, bz, rwidth, rheight, rdepth,
            rtn) {//passes
902 gcpscad   movetosafez();
903 gcpscad   cutwithfeed(bx+tool_radius(rtn),by+tool_radius(rtn),bz-rdepth,
              feed);
904 gcpscad   cutwithfeed(bx+rwidth-tool_radius(rtn),by+tool_radius(rtn),bz-
              rdepth,feed);
905 gcpscad   cutwithfeed(bx+rwidth-tool_radius(rtn),by+rheight-tool_radius(rtn
              ),bz-rdepth,feed);
906 gcpscad   cutwithfeed(bx+tool_radius(rtn),by+rheight-tool_radius(rtn),bz-
              rdepth,feed);
907 gcpscad   dxfarc(bx+tool_radius(rtn),by+tool_radius(rtn),tool_radius(rtn)
              ,180,270, rtn);
908 gcpscad   dxfpolyline(bx,by+tool_radius(rtn),bx,by+rheight-tool_radius(rtn)
              , rtn);
909 gcpscad   dxfarc(bx+tool_radius(rtn),by+rheight-tool_radius(rtn),
              tool_radius(rtn),90,180, rtn);
910 gcpscad   dxfpolyline(bx+tool_radius(rtn),by+rheight,bx+rwidth-tool_radius(
              rtn),by+rheight, rtn);
911 gcpscad   dxfarc(bx+rwidth-tool_radius(rtn),by+rheight-tool_radius(rtn),
              tool_radius(rtn),0,90, rtn);
912 gcpscad   dxfpolyline(bx+rwidth,by+rheight-tool_radius(rtn),bx+rwidth,by+
              tool_radius(rtn), rtn);
913 gcpscad   dxfarc(bx+rwidth-tool_radius(rtn),by+tool_radius(rtn),tool_radius
              (rtn),270,360, rtn);
914 gcpscad   dxfpolyline(bx+rwidth-tool_radius(rtn),by,bx+tool_radius(rtn),by,
               rtn);
915 gcpscad }
```

rectangleoutlinedxf Which suggests a further command, `rectangleoutlinedxf` for simply adding a rectangle (a potential use of which would be in Job Setup to add the stock outline to DXFs to assist in registration of jobs with multiple tools):

```
917 gcpscad module rectangleoutlinedxf(bx, by, bz, rwidth, rheight, rtn) {
918 gcpscad   dxfpolyline(bx,by,bx,by+rheight, rtn);
919 gcpscad   dxfpolyline(bx,by+rheight,bx+rwidth,by+rheight, rtn);
920 gcpscad   dxfpolyline(bx+rwidth,by+rheight,bx+rwidth,by, rtn);
921 gcpscad   dxfpolyline(bx+rwidth,by,bx,by, rtn);
922 gcpscad }
```

the initial section performs the cutting operation for the 3D preview while the latter section writes out the outline to the DXF files.

cutoutrectangledxf A variant of the cutting version of that file, `cutoutrectangledxf` will cut to the outside:

```
924 gcpscad module cutoutrectangledxf(bx, by, bz, rwidth, rheight, rdepth, rtn)
              {
925 gcpscad   movetosafez();
926 gcpscad   cutwithfeed(bx-tool_radius(rtn),by-tool_radius(rtn),bz-rdepth,
              feed);
927 gcpscad   cutwithfeed(bx+rwidth+tool_radius(rtn),by-tool_radius(rtn),bz-
              rdepth,feed);
928 gcpscad   cutwithfeed(bx+rwidth+tool_radius(rtn),by+rheight+tool_radius(rtn
              ),bz-rdepth,feed);
929 gcpscad   cutwithfeed(bx-tool_radius(rtn),by+rheight+tool_radius(rtn),bz-
              rdepth,feed);
930 gcpscad   cutwithfeed(bx-tool_radius(rtn),by-tool_radius(rtn),bz-rdepth,
              feed);
931 gcpscad   dxfpolyline(bx,by,bx,by+rheight, rtn);
932 gcpscad   dxfpolyline(bx,by+rheight,bx+rwidth,by+rheight, rtn);
933 gcpscad   dxfpolyline(bx+rwidth,by+rheight,bx+rwidth,by, rtn);
934 gcpscad   dxfpolyline(bx+rwidth,by,bx,by, rtn);
935 gcpscad }
```

## 3.4 Expansion

The balance of shapes will go into `cut2Dshapes.scad` and of course it will be possible to create additional files for specific purposes.

```
1 cut2D //!OpenSCAD
```

# 4 Future

## 4.1 Images

Would it be helpful to re-create code algorithms/sections using OpenSCAD Graph Editor so as to represent/illustrate the program?

## 4.2 Generalized DXF creation

Generalize the creation of DXFs based on the projection() of a toolpath?

## 4.3 Import G-code

Use a tool to read in a G-code file, then create a 3D model which would serve as a preview of the cut?

- https://stackoverflow.com/questions/34638372/simple-python-program-to-read-gcode-file
- https://pypi.org/project/gcodeparser/
- https://github.com/fragmuffin/pygcode/wiki

## 4.4 Bézier curves in 2 dimensions

Take a Bézier curve definition and approximate it as arcs and write them into a DXF?
https://pomax.github.io/bezierinfo/
c.f., https://linuxcnc.org/docs/html/gcode/g-code.html#gcode:g5

## 4.5 Bézier curves in 3 dimensions

One question is how many Bézier curves would it be necessary to have to define a surface in 3 dimensions. Attributes for this which are desirable/necessary:

- concise — a given Bézier curve should be represented by just the point coordinates, so two on-curve points, two off-curve points, each with a pair of coordinates
- For a given shape/region it will need to be possible to have a matching definition exactly match up with it so that one could piece together a larger more complex shape from smaller/simpler regions
- similarly it will be necessary for it to be possible to sub-divide a defined region — for example it should be possible if one had 4 adjacent regions, then the four quadrants at the intersection of the four regions could be used to construct a new region — is it possible to derive a new Bézier curve from half of two other curves?

For the three planes:

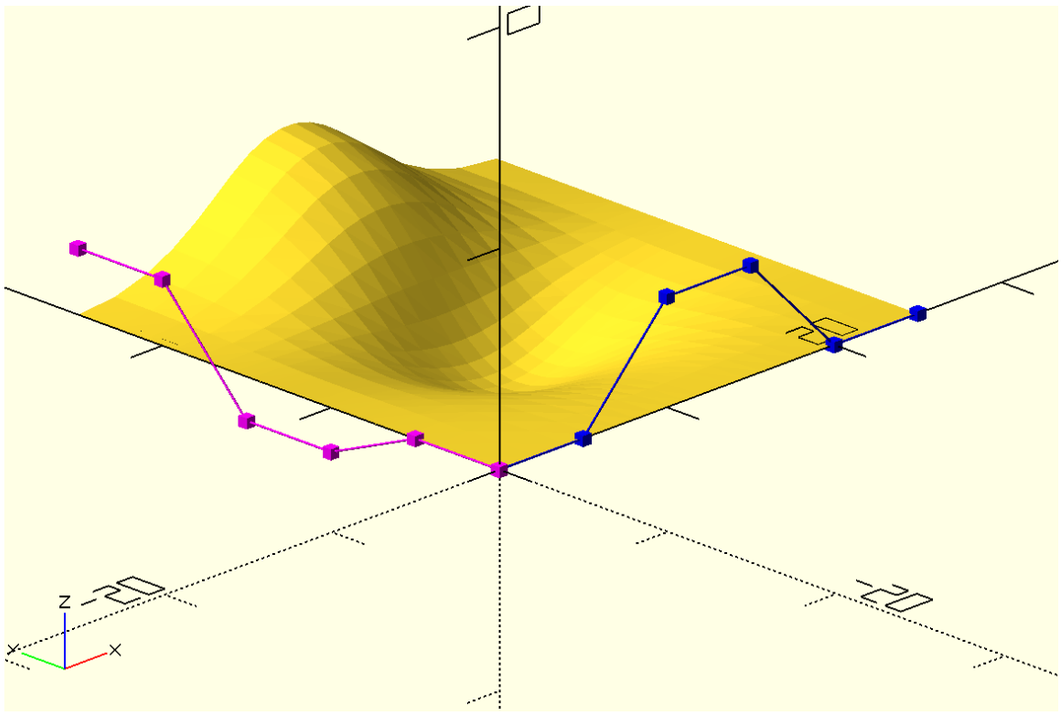
- XY
- XZ
- ZY

it should be possible to have three Bézier curves (left-most/right-most or front-back or top/bottom for two, and a mid-line for the third), so a region which can be so represented would be definable by:

```
3 planes * 3 Béziers * (2 on-curve + 2 off-curve points) == 36 coordinate pairs
```

which is a marked contrast to representations such as:
https://github.com/DavidPhillipOster/Teapot
and regions which could not be so represented could be sub-divided until the representation is workable.

Or, it may be that fewer (only two?) curves are needed:




https://pages.mtu.edu/~shene/COURSES/cs3621/NOTES/notes.html

# 5 Other Resources

Holidays are from https://nationaltoday.com/

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# Index

# Routines

## Variables