The gcodepreview OpenSCAD library*

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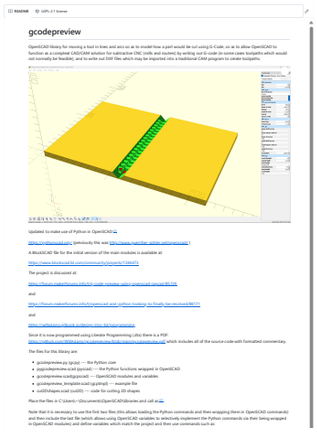
Abstract

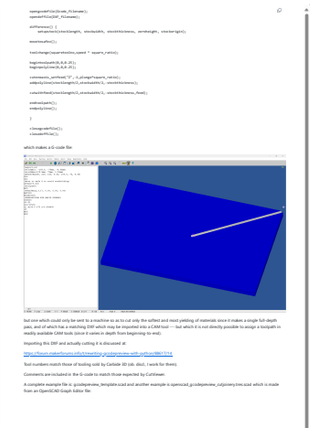
The gcodepreview library allows using PythonOpenSCAD to move a tool in lines and output dxf and G-code files so as to work as a CAD/CAM program for CNC.

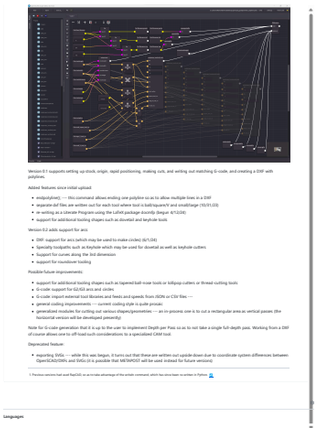
Contents

*This file (gcodepreview) has version number v0.4, last revised 2024/07/28.

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/
       openscad_cutjoinery.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
```

```
rdme    and variables
rdme  - gcodepreview_template.scad (gcptmpl) --- example file
rdme  - cut2Dshapes.scad (cut2D) --- code for cutting 2D shapes
rdme
rdme Place the files in C:\Users\\\~\Documents\OpenSCAD\libraries and call as:[^libraries]
rdme
rdme [^libraries]: C:\Users\\\~\Documents\RapCAD\libraries is deprecated since RapCAD is no longer needed since Python is now used for writing out files)
rdme
rdme     use <gcodepreview.py>;
rdme     use <pygcodepreview.scad>;
rdme     include <gcodepreview.scad>;
rdme
rdme Note that it is necessary to use the first two files
rdme (this allows loading the Python commands and then
rdme wrapping them in OpenSCAD commands) and then include
rdme the last file (which allows using OpenSCAD variables
rdme to selectively implement the Python commands via their
rdme being wrapped in OpenSCAD modules) and define
rdme variables which match the project and then use
rdme commands such as:
rdme
rdme     opengcodefile(Gcode_filename);
rdme     opendxffile(DXF_filename);
rdme
rdme     difference() {
rdme         setupstock(stocklength, stockwidth, stockthickness, zeroheight, stockorigin);
rdme
rdme     movetosafez();
rdme
rdme     toolchange(squaretoolno,speed * square_ratio);
rdme
rdme     begintoolpath(0,0,0.25);
rdme     beginpolyline(0,0,0.25);
rdme
rdme     cutoneaxis_setfeed("Z",-1,plunge*square_ratio);
rdme     addpolyline(stocklength/2,stockwidth/2,-stockthickness);
rdme
rdme     cutwithfeed(stocklength/2,stockwidth/2,-stockthickness,feed);
rdme
rdme     endtoolpath();
rdme     endpolyline();
rdme
rdme     }
rdme
rdme     closegcodefile();
rdme     closedxffile();
rdme
rdme which makes a G-code file:
rdme
rdme ![OpenSCAD template G-code file](https://raw.githubusercontent.com/WillAdams/gcodepreview/main/gcodepreview_template.png?raw=true)
rdme
rdme but one which could only be sent to a machine so as to
rdme cut only the softest and most yielding of materials
rdme since it makes a single full-depth pass, and of which
rdme has a matching DXF which may be imported into a
rdme CAM tool --- but which it is not directly possible
rdme to assign a toolpath in readily available CAM tools
rdme (since it varies in depth from beginning-to-end).
rdme
rdme Importing this DXF and actually cutting it
rdme is discussed at:
rdme
rdme https://forum.makerforums.info/t/rewriting-gcodepreview-with-python/88617/14
rdme
rdme Tool numbers match those of tooling sold by Carbide 3D
rdme (ob. discl., I work for them).
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)
122 rdme
123 rdme Version 0.1 supports setting up stock, origin, rapid
124 rdme positioning, making cuts, and writing out matching
125 rdme G-code, and creating a DXF with polylines.
126 rdme
127 rdme Added features since initial upload:
128 rdme
129 rdme  - endpolyline(); --- this command allows ending one
            polyline so as to allow multiple lines in a DXF
130 rdme  - separate dxf files are written out for each tool
            where tool is ball/square/V and small/large
            (10/31/23)
131 rdme  - re-writing as a Literate Program using the LaTeX
            package docmfp (begun 4/12/24)
132 rdme  - support for additional tooling shapes such as
            dovetail and keyhole tools
133 rdme
134 rdme Version 0.2 adds support for arcs
135 rdme
136 rdme  - DXF: support for arcs (which may be used to make
            circles) (6/1/24)
137 rdme  - Specialty toolpaths such as Keyhole which may be
            used for dovetail as well as keyhole cutters
138 rdme
139 rdme Version 0.3
140 rdme
141 rdme  - Support for curves along the 3rd dimension
142 rdme  - support for roundover tooling
143 rdme
144 rdme Version 0.4
145 rdme
146 rdme  - Rewrite using literati documentclass, suppression
            of SVG code
147 rdme
148 rdme Possible future improvements:
149 rdme
150 rdme  - support for additional tooling shapes such as
            tapered ball-nose tools or lollipop cutters or
            thread-cutting tools
151 rdme  - G-code: support for G2/G3 arcs and circles
152 rdme  - G-code: import external tool libraries and feeds
            and speeds from JSON or CSV files ---
153 rdme  - general coding improvements --- current coding
            style is quite prosaic
154 rdme  - generalized modules for cutting out various shapes/
            geometries --- an in-process one is to cut a
            rectangular area as vertical passes (the
            horizontal version will be developed presently)
155 rdme
156 rdme Note for G-code generation that it is up to the user
157 rdme to implement Depth per Pass so as to not take a
158 rdme single full-depth pass. Working from a DXF of course
159 rdme allows one to off-load such considerations to a
160 rdme specialized CAM tool.
161 rdme
162 rdme Deprecated feature:
163 rdme
164 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

As noted above, this library works by using Python code as a back-end so as to persistently store and access variables, and to write out files. 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 included allowing it to access OpenSCAD variables for branching

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

```
1 gcpy #!/usr/bin/env python
```

```
1 pyscad //!OpenSCAD
2 pyscad
3 pyscad //gcodepreview 0.3
```

```
1 gcpscad //!OpenSCAD
2 gcpscad
3 gcpscad //gcodepreview 0.3
4 gcpscad //
5 gcpscad //used via use <gcodepreview.py>;
6 gcpscad //          use <pygcodepreview.scad>;
7 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:

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

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

2.1 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, depth in toolpath, &c. 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.

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

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 used), while the other will be able to access and use OpenSCAD variables, as well as implement Customizer options (this will be included).

Note that as a convention, where it is necessary for a module to coordinate between Python and OpenSCAD, 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.

psetupstock The first such routine will be appropriately enough, to set up the stock, and perform other initializations.

```
10 gcpy def psetupstock(stocklength, stockwidth,
       stockthickness, zeroheight, stockorigin):
11 gcpy     global mpx
12 gcpy     mpx = float(0)
13 gcpy     global mpy
14 gcpy     mpy = float(0)
15 gcpy     global mpz
16 gcpy     mpz = float(0)
17 gcpy     global tpz
18 gcpy     tpz = float(0)
19 gcpy     global currenttool
20 gcpy     currenttool = 102
```

osetupstock The intermediary OpenSCAD code simply calls the Python version.

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

setupstock The OpenSCAD code which is called requires that the user set parameters and will create comments in the G-code which set the stock dimensions and its position relative to the zero as set relative to the stock.

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

```
            0.00mm)");
40 pyscad owriteeleven("(STOCK/BLOCK, ",str(stocklength),", ",
          str(stockwidth),", ",str(stockthickness),", 0.00,
          ",str(stockwidth),", ",str(stockthickness),")");
41 pyscad     }
42 pyscad     }
43 pyscad     }
44 pyscad         else if (stockorigin == "Center") {
45 pyscad //owritecomment("Center");
46 pyscad     translate([(-stocklength / 2), (-stockwidth / 2),
              -stockthickness]){
47 pyscad       cube([stocklength, stockwidth, stockthickness],
                 center=false);
48 pyscad if (generategcode == true) {
49 pyscad owriteseven("(stockMin: -",str(stocklength/2),", -",
          str(stockwidth/2),"mm, -",str(stockthickness),"mm)
          ");
50 pyscad owritefive("(stockMax:",str(stocklength/2),"mm, ",str(
          stockwidth/2),"mm, 0.00mm)");
51 pyscad owritethirteen("(STOCK/BLOCK, ",str(stocklength),", ",
          str(stockwidth),", ",str(stockthickness),", ",str(
          stocklength/2),", ", str(stockwidth/2),", ",str(
          stockthickness),")");
52 pyscad }
53 pyscad }
54 pyscad }
55 pyscad } else if (zeroheight == "Bottom") {
56 pyscad //owritecomment("Bottom");
57 pyscad     if (stockorigin == "Lower-Left") {
58 pyscad     cube([stocklength, stockwidth, stockthickness],
              center=false);
59 pyscad if (generategcode == true) {
60 pyscad owriteone("(stockMin:0.00mm, 0.00mm, 0.00mm)");
61 pyscad owriteseven("(stockMax:",str(stocklength),"mm, ",str(
          stockwidth),"mm, ",str(stockthickness),"mm)");
62 pyscad owriteseven("(STOCK/BLOCK, ",str(stocklength),", ",str
          (stockwidth),", ",str(stockthickness),",0.00,
          0.00, 0.00)");
63 pyscad     }
64 pyscad }        else if (stockorigin == "Center-Left") {
65 pyscad     translate([0, (-stockwidth / 2), 0]){
66 pyscad       cube([stocklength, stockwidth, stockthickness],
                 center=false);
67 pyscad if (generategcode == true) {
68 pyscad owritethree("(stockMin:0.00mm, -",str(stockwidth/2),"
          mm, 0.00mm)");
69 pyscad owriteseven("(stockMax:",str(stocklength),"mm, ",str(
          stockwidth/2),"mm, ",str(stockthickness),"mm)");
70 pyscad owritenine("(STOCK/BLOCK, ",str(stocklength),", ",str(
          stockwidth),", ",str(stockthickness),",0.00, ",str
          (stockwidth/2),", 0.00)");
71 pyscad     }
72 pyscad     }
73 pyscad         } else if (stockorigin == "Top-Left") {
74 pyscad     translate([0, (-stockwidth), 0]){
75 pyscad       cube([stocklength, stockwidth, stockthickness],
                 center=false);
76 pyscad     }
77 pyscad if (generategcode == true) {
78 pyscad owritethree("(stockMin:0.00mm, -",str(stockwidth),"mm,
            0.00mm)");
79 pyscad owritefive("(stockMax:",str(stocklength),"mm, 0.00mm,
          ",str(stockthickness),"mm)");
80 pyscad owritenine("(STOCK/BLOCK, ",str(stocklength),", ",str(
          stockwidth),", ",str(stockthickness),", 0.00, ",
          str(stockwidth),", 0.00)");
81 pyscad }
82 pyscad }        else if (stockorigin == "Center") {
83 pyscad     translate([(-stocklength / 2), (-stockwidth / 2),
              0]){
84 pyscad       cube([stocklength, stockwidth, stockthickness],
                 center=false);
85 pyscad     }
86 pyscad if (generategcode == true) {
87 pyscad owritefive("(stockMin:-",str(stocklength/2),", -",str(
          stockwidth/2),"mm, 0.00mm)");
88 pyscad owriteseven("(stockMax:",str(stocklength/2),"mm, ",str
          (stockwidth/2),"mm, ",str(stockthickness),"mm)");
```

```
 89 pyscad owriteeleven("(STOCK/BLOCK, ",str(stocklength),", ",
           str(stockwidth),", ",str(stockthickness),", ",str(
           stocklength/2),", ", str(stockwidth/2),", 0.00)");
 90 pyscad }
 91 pyscad }
 92 pyscad }
 93 pyscad if (generategcode == true) {
 94 pyscad         owriteone("G90");
 95 pyscad         owriteone("G21");
 96 pyscad //      owriteone("(Move to safe Z to avoid
           workholding)");
 97 pyscad //      owriteone("G53G0Z-5.000");
 98 pyscad }
 99 pyscad //owritecomment("ENDSETUP");
100 pyscad }
```

xpos ypos zpos

It will be necessary to have Python functions which return the current values of the machine position in Cartesian coordinates:

```
 22 gcpy def xpos():
 23 gcpy     global mpx
 24 gcpy     return mpx
 25 gcpy
 26 gcpy def ypos():
 27 gcpy     global mpy
 28 gcpy     return mpy
 29 gcpy
 30 gcpy def zpos():
 31 gcpy     global mpz
 32 gcpy     return mpz
 33 gcpy
 34 gcpy def tzpos():
 35 gcpy     global tpz
 36 gcpy     return tpz
```

psetxpos psetypos psetzpos psettzpos

and in turn, functions which set the positions:

```
 38 gcpy def psetxpos(newxpos):
 39 gcpy     global mpx
 40 gcpy     mpx = newxpos
 41 gcpy
 42 gcpy def psetypos(newypos):
 43 gcpy     global mpy
 44 gcpy     mpy = newypos
 45 gcpy
 46 gcpy def psetzpos(newzpos):
 47 gcpy     global mpz
 48 gcpy     mpz = newzpos
 49 gcpy
 50 gcpy def psettzpos(newtzpos):
 51 gcpy     global tpz
 52 gcpy     tpz = newtzpos
```

and as noted above, there will need to be matching OpenSCAD versions.

getxpos getypos getzpos 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 command.

setxpos setypos setzpos settzpos

```
102 pyscad function getxpos() = xpos();
103 pyscad function getypos() = ypos();
104 pyscad function getzpos() = zpos();
105 pyscad function gettzpos() = tzpos();
106 pyscad
107 pyscad module setxpos(newxpos) {
108 pyscad psetxpos(newxpos);
109 pyscad }
110 pyscad
111 pyscad module setypos(newypos) {
112 pyscad psetypos(newypos);
113 pyscad }
114 pyscad
115 pyscad module setzpos(newzpos) {
116 pyscad psetzpos(newzpos);
117 pyscad }
118 pyscad
119 pyscad module settzpos(newtzpos) {
```

```
120 pyscad psettzpos(newtzpos);
121 pyscad }
```

oset

```
10 gcpscad module oset(ex, ey, ez) {
11 gcpscad setxpos(ex);
12 gcpscad setypos(ey);
13 gcpscad setzpos(ez);
14 gcpscad }
```

osettz

```
16 gcpscad module osettz(tz) {
17 gcpscad settzpos(tz);
18 gcpscad }
```

2.2 Tools and Changes

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

```
54 gcpy def psettool(tn):
55 gcpy     global currenttool
56 gcpy     currenttool = tn
57 gcpy
58 gcpy def pcurrent_tool():
59 gcpy     global currenttool
60 gcpy     return currenttool
```

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

```
123 pyscad module osettool(tn){
124 pyscad psettool(tn);}
125 pyscad
126 pyscad function current_tool() = pcurrent_tool();
```

2.2.1 toolchange

toolchange and apply the appropriate commands for a `toolchange`. Note that it is expected that this subsubsection 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).

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 in the initial version of this project

2.2.1.1 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 rounded and organic shapes
- V tooling (#301, 302 and 390) — pointed at the tip, they are available in a variety 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, see #501 and 502)

```
20 gcpscad module toolchange(tool_number,speed) {
21 gcpscad    osettool(tool_number);
22 gcpscad if (generategcode == true) {
23 gcpscad          writecomment("Toolpath");
24 gcpscad          owriteone("M05");
25 gcpscad //       writecomment("Move to safe Z to avoid
             workholding");
```

```
26 gcpscad //       owriteone("G53G0Z-5.000");
27 gcpscad //     writecomment("Begin toolpath");
28 gcpscad   if (tool_number == 201) {
29 gcpscad         writecomment("TOOL/MILL,6.35, 0.00, 0.00,
                       0.00");
30 gcpscad   } else if (tool_number == 202) {
31 gcpscad         writecomment("TOOL/MILL,6.35, 3.17, 0.00,
                       0.00");
32 gcpscad   } else if (tool_number == 102) {
33 gcpscad         writecomment("TOOL/MILL,3.17, 0.00, 0.00,
                       0.00");
34 gcpscad   } else if (tool_number == 101) {
35 gcpscad         writecomment("TOOL/MILL,3.17, 1.58, 0.00,
                       0.00");
36 gcpscad   } else if (tool_number == 301) {
37 gcpscad         writecomment("TOOL/MILL,0.03, 0.00, 6.35,
                       45.00");
38 gcpscad   } else if (tool_number == 302) {
39 gcpscad         writecommment("TOOL/MILL,0.03, 0.00, 10.998,
                       30.00");
40 gcpscad   } else if (tool_number == 390) {
41 gcpscad         writecomment("TOOL/MILL,0.03, 0.00, 1.5875,
                       45.00");
```

2.2.1.2 Tooling for Keyhole Toolpaths Keyhole toolpaths (see: subsection 4.2 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 tools — intended to cut slots for retaining hardware used for picture hanging, they may be used to create slots for other purposes
- 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

```
42 gcpscad   } else if (tool_number == 375) {
43 gcpscad         writecomment("TOOL/MILL,9.53, 0.00, 3.17,
                       0.00");
```

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

2.2.1.3 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.

```
46 gcpscad   }
47 gcpscad    select_tool(tool_number);
48 gcpscad    owritetwo("M6T",str(tool_number));
49 gcpscad    owritetwo("M03S",str(speed));
50 gcpscad  }
51 gcpscad }
```

2.2.1.4 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.

2.2.1.5 Selecting Tools There must also be a module for selecting tools: `select_tool` which will select the matching module for 3D modeling and pass the appropriate parameters to that module:

`selecttool`
`tool number`

```
53 gcpscad module select_tool(tool_number) {
54 gcpscad //echo(tool_number);
55 gcpscad   if (tool_number == 201) {
56 gcpscad     gcp_endmill_square(6.35, 19.05);
57 gcpscad   } else if (tool_number == 202) {
58 gcpscad     gcp_endmill_ball(6.35, 19.05);
59 gcpscad   } else if (tool_number == 102) {
60 gcpscad     gcp_endmill_square(3.175, 19.05);
61 gcpscad   } else if (tool_number == 101) {
62 gcpscad     gcp_endmill_ball(3.175, 19.05);
63 gcpscad   } else if (tool_number == 301) {
64 gcpscad     gcp_endmill_v(90, 12.7);
65 gcpscad   } else if (tool_number == 302) {
66 gcpscad     gcp_endmill_v(60, 12.7);
67 gcpscad   } else if (tool_number == 390) {
68 gcpscad     gcp_endmill_v(90, 3.175);
```

For a keyhole tool:

```
69 gcpscad   } else if (tool_number == 375) {
70 gcpscad     gcp_keyhole(9.525, 3.175);
```

and dovetail tool:

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

```
73 gcpscad   }
74 gcpscad }
```

2.2.2 3D Shapes for Tools

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

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

gcp endmill square

```
76 gcpscad module gcp_endmill_square(es_diameter, es_flute_length
       ) {
77 gcpscad   cylinder(r1=(es_diameter / 2), r2=(es_diameter / 2),
           h=es_flute_length, center=false);
78 gcpscad }
```

gcp keyhole

```
80 gcpscad module gcp_keyhole(es_diameter, es_flute_length) {
81 gcpscad   cylinder(r1=(es_diameter / 2), r2=(es_diameter / 2),
           h=es_flute_length, center=false);
82 gcpscad }
```

gcp dovetail

```
84 gcpscad module gcp_dovetail(dt_bottomdiameter, dt_topdiameter,
       dt_height, dt_angle) {
85 gcpscad   cylinder(r1=(dt_bottomdiameter / 2), r2=(
           dt_topdiameter / 2), h= dt_height, center=false)
           ;
86 gcpscad }
```

gcp endmill ball

```
88 gcpscad module gcp_endmill_ball(es_diameter, es_flute_length)
       {
89 gcpscad   translate([0, 0, (es_diameter / 2)]){
90 gcpscad     union(){
91 gcpscad       sphere(r=(es_diameter / 2));
92 gcpscad       cylinder(r1=(es_diameter / 2), r2=(es_diameter /
               2), h=es_flute_length, center=false);
93 gcpscad     }
94 gcpscad   }
```

```
 95 gcpscad }
```

gcp endmill v

```
 97 gcpscad module gcp_endmill_v(es_v_angle, es_diameter) {
 98 gcpscad   union(){
 99 gcpscad     cylinder(r1=0, r2=(es_diameter / 2), h=((
                    es_diameter / 2) / tan((es_v_angle / 2))),
                    center=false);
100 gcpscad     translate([0, 0, ((es_diameter / 2) / tan((
                    es_v_angle / 2)))]){
101 gcpscad       cylinder(r1=(es_diameter / 2), r2=(es_diameter /
                        2), h=((es_diameter * 8) ), center=false)
                       ;/// tan((es_v_angle / 2))
102 gcpscad     }
103 gcpscad   }
104 gcpscad }
```

2.2.2.2 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 `hull`ed 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
radiuscut different modules, the public-facing version which includes the tool number:

```
106 gcpscad module radiuscut(bx, by, bz, ex, ey, ez, radiustn) {
107 gcpscad     if (radiustn == 56125) {
108 gcpscad         radiuscuttool(bx, by, bz, ex, ey, ez, 0.508/2,
                        1.531);
109 gcpscad     } else if (radiustn == 56142) {
110 gcpscad         radiuscuttool(bx, by, bz, ex, ey, ez, 0.508/2,
                        2.921);
111 gcpscad     } else if (radiustn == 312) {
112 gcpscad         radiuscuttool(bx, by, bz, ex, ey, ez, 1.524/2,
                        3.175);
113 gcpscad     } else if (radiustn == 1570) {
114 gcpscad         radiuscuttool(bx, by, bz, ex, ey, ez, 0.507/2,
                        4.509);
115 gcpscad     }
116 gcpscad }
```

which then calls the actual `radiuscuttool` 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`.

```
118 gcpscad module radiuscuttool(bx, by, bz, ex, ey, ez,
                tool_radius_tip, tool_radius_width) {
119 gcpscad n = 90 + $fn*3;
120 gcpscad step = 360/n;
121 gcpscad
122 gcpscad hull(){
123 gcpscad     translate([bx,by,bz])
124 gcpscad     cylinder(step,tool_radius_tip,tool_radius_tip);
125 gcpscad     translate([ex,ey,ez])
126 gcpscad     cylinder(step,tool_radius_tip,tool_radius_tip);
127 gcpscad     }
128 gcpscad
129 gcpscad hull(){
130 gcpscad translate([bx,by,bz+tool_radius_width])
131 gcpscad cylinder(tool_radius_width*2,tool_radius_tip+
                tool_radius_width,tool_radius_tip+
                tool_radius_width);
132 gcpscad
133 gcpscad translate([ex,ey,ez+tool_radius_width])
134 gcpscad   cylinder(tool_radius_width*2,tool_radius_tip+
                  tool_radius_width,tool_radius_tip+
                  tool_radius_width);
135 gcpscad     }
136 gcpscad
```

```
137 gcpscad for (i=[0:step:90]) {
138 gcpscad     angle = i;
139 gcpscad     dx = tool_radius_width*cos(angle);
140 gcpscad     dxx = tool_radius_width*cos(angle+step);
141 gcpscad     dzz = tool_radius_width*sin(angle);
142 gcpscad     dz = tool_radius_width*sin(angle+step);
143 gcpscad     dh = dz-dzz;
144 gcpscad     hull(){
145 gcpscad     translate([bx,by,bz+dz])
146 gcpscad        cylinder(dh,tool_radius_tip+tool_radius_width-
                    dx,tool_radius_tip+tool_radius_width-dxx);
147 gcpscad     translate([ex,ey,ez+dz])
148 gcpscad        cylinder(dh,tool_radius_tip+tool_radius_width-
                    dx,tool_radius_tip+tool_radius_width-dxx);
149 gcpscad      }
150 gcpscad     }
151 gcpscad }
```

2.2.3 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 simply calls the matching OpenSCAD module which wraps the Python code:

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

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

```
155 gcpscad function otool_diameter(td_tool, td_depth) =
            ptool_diameter(td_tool, td_depth);
```

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

```
 62 gcpy def ptool_diameter(ptd_tool, ptd_depth):
 63 gcpy     if ptd_tool == 201:
 64 gcpy         return 6.35
 65 gcpy     if ptd_tool == 202:
 66 gcpy         if ptd_depth > 3.175:
 67 gcpy             return 6.35
 68 gcpy         else:
 69 gcpy             return 0
 70 gcpy     if ptd_tool == 102:
 71 gcpy         return 3.175
 72 gcpy     if ptd_tool == 101:
 73 gcpy         if ptd_depth > 1.5875:
 74 gcpy             return 3.175
 75 gcpy         else:
 76 gcpy             return 0
 77 gcpy     if ptd_tool == 301:
 78 gcpy         return 0
 79 gcpy     if ptd_tool == 302:
 80 gcpy         return 0
 81 gcpy     if ptd_tool == 390:
 82 gcpy         return 0
 83 gcpy     if ptd_tool == 375:
 84 gcpy         if ptd_depth < 6.35:
 85 gcpy             return 9.525
 86 gcpy         else:
 87 gcpy             return 6.35
 88 gcpy     if ptd_tool == 814:
 89 gcpy         if ptd_depth > 12.7:
 90 gcpy             return 6.35
 91 gcpy         else:
 92 gcpy             return 12.7
```

(Note that zero (0) values will need to be replaced with appropriate code.)

2.3 File Handling

For writing to files it will be necessary to have commands for each step of working with the files.

popengcodefile
popendxffile
popendxlgblffile
popendxflgsqfile
popendxflgVfile
popendxfsmblfile
popendxfsmsqfile
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.

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

oopengcodefile
oopendxffile

There will need to be matching OpenSCAD modules for the Python functions.

```
128 pyscad module oopengcodefile(fn) {
129 pyscad          popengcodefile(fn);
130 pyscad }
131 pyscad
132 pyscad module oopendxffile(fn) {
133 pyscad     echo(fn);
134 pyscad          popendxffile(fn);
135 pyscad }
136 pyscad
137 pyscad module oopendxflgblfile(fn) {
138 pyscad          popendxflgblfile(fn);
139 pyscad }
140 pyscad
141 pyscad module oopendxflgsqfile(fn) {
142 pyscad          popendxflgsqfile(fn);
143 pyscad }
144 pyscad
145 pyscad module oopendxflgVfile(fn) {
146 pyscad          popendxflgVfile(fn);
147 pyscad }
148 pyscad
149 pyscad module oopendxfsmblfile(fn) {
150 pyscad          popendxfsmblfile(fn);
151 pyscad }
152 pyscad
153 pyscad module oopendxfsmsqfile(fn) {
154 pyscad     echo(fn);
155 pyscad          popendxfsmsqfile(fn);
156 pyscad }
```

```
157 pyscad
158 pyscad module oopendxfsmVfile(fn) {
159 pyscad          popendxfsmVfile(fn);
160 pyscad }
161 pyscad
162 pyscad module oopendxfKHfile(fn) {
163 pyscad          popendxfKHfile(fn);
164 pyscad }
165 pyscad
166 pyscad module oopendxfDTfile(fn) {
167 pyscad          popendxfDTfile(fn);
168 pyscad }
```

opengcodefile Which has matching OpenSCAD commands:

```
157 gcpscad module opengcodefile(fn) {
158 gcpscad if (generategcode == true) {
159 gcpscad          oopengcodefile(fn);
160 gcpscad     echo(fn);
161 gcpscad     owritecomment(fn);
162 gcpscad     }
163 gcpscad }
```

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

```
165 gcpscad module opendxffile(fn) {
166 gcpscad if (generatedxf == true) {
167 gcpscad          oopendxffile(str(fn,".dxf"));
168 gcpscad //     echo(fn);
169 gcpscad     dxfwriteone("0");
170 gcpscad     dxfwriteone("SECTION");
171 gcpscad     dxfwriteone("2");
172 gcpscad     dxfwriteone("ENTITIES");
173 gcpscad if (large_ball_tool_no >  0) {  oopendxflgblfile(str(
                fn,".",large_ball_tool_no,".dxf"));
174 gcpscad     dxfpreamble(large_ball_tool_no);
175 gcpscad }
176 gcpscad if (large_square_tool_no >  0) {
                oopendxflgsqfile(str(fn,".",large_square_tool_no
                ,".dxf"));
177 gcpscad     dxfpreamble(large_square_tool_no);
178 gcpscad }
179 gcpscad if (large_V_tool_no >  0) {     oopendxflgVfile(str(fn
                ,".",large_V_tool_no,".dxf"));
180 gcpscad     dxfpreamble(large_V_tool_no);
181 gcpscad }
182 gcpscad if (small_ball_tool_no >  0) { oopendxfsmblfile(str(fn
                ,".",small_ball_tool_no,".dxf"));
183 gcpscad     dxfpreamble(small_ball_tool_no);
184 gcpscad }
185 gcpscad if (small_square_tool_no >  0) {
                oopendxfsmsqfile(str(fn,".",small_square_tool_no
                ,".dxf"));
186 gcpscad //     echo(str("tool no",small_square_tool_no));
187 gcpscad     dxfpreamble(small_square_tool_no);
188 gcpscad }
189 gcpscad if (small_V_tool_no >  0) {     oopendxfsmVfile(str(fn
                ,".",small_V_tool_no,".dxf"));
190 gcpscad     dxfpreamble(small_V_tool_no);
191 gcpscad }
192 gcpscad if (KH_tool_no >  0) {  oopendxfKHfile(str(fn,".",
                KH_tool_no,".dxf"));
193 gcpscad     dxfpreamble(KH_tool_no);
194 gcpscad }
195 gcpscad if (DT_tool_no >  0) {  oopendxfDTfile(str(fn,".",
                DT_tool_no,".dxf"));
196 gcpscad     dxfpreamble(DT_tool_no);
197 gcpscad }
198 gcpscad }
199 gcpscad }
```

writedxf Once files have been opened they may be written to. There is a base command:

```
134 gcpy def writedxf(*arguments):
135 gcpy     line_to_write = ""
136 gcpy     for element in arguments:
```

```
137 gcpy         line_to_write += element
138 gcpy     dxf.write(line_to_write)
139 gcpy     dxf.write("\n")
```

writedxflgbl
writedxflgsq
writedxflgV
writedxfsmbl
writedxfsmsq
writedxfsmV
writedxfKH
writedxfDT

and for each tool/size combination, an appropriate command:

- Ball nose, large (lgbl)
- Ball nose, small (smbl)
- Square, large (lgsq)
- Square, small (smsq)
- V, large (lgV)
- V, small (smV)
- Keyhole (KH)
- Dovetail (DT)

```
141 gcpy def writedxflgbl(*arguments):
142 gcpy     line_to_write = ""
143 gcpy     for element in arguments:
144 gcpy         line_to_write += element
145 gcpy     dxflgbl.write(line_to_write)
146 gcpy     print(line_to_write)
147 gcpy     dxflgbl.write("\n")
148 gcpy
149 gcpy def writedxflgsq(*arguments):
150 gcpy     line_to_write = ""
151 gcpy     for element in arguments:
152 gcpy         line_to_write += element
153 gcpy     dxflgsq.write(line_to_write)
154 gcpy     print(line_to_write)
155 gcpy     dxflgsq.write("\n")
156 gcpy
157 gcpy def writedxflgV(*arguments):
158 gcpy     line_to_write = ""
159 gcpy     for element in arguments:
160 gcpy         line_to_write += element
161 gcpy     dxflgV.write(line_to_write)
162 gcpy     print(line_to_write)
163 gcpy     dxflgV.write("\n")
164 gcpy
165 gcpy def writedxfsmbl(*arguments):
166 gcpy     line_to_write = ""
167 gcpy     for element in arguments:
168 gcpy         line_to_write += element
169 gcpy     dxfsmbl.write(line_to_write)
170 gcpy     print(line_to_write)
171 gcpy     dxfsmbl.write("\n")
172 gcpy
173 gcpy def writedxfsmsq(*arguments):
174 gcpy     line_to_write = ""
175 gcpy     for element in arguments:
176 gcpy         line_to_write += element
177 gcpy     dxfsmsq.write(line_to_write)
178 gcpy     print(line_to_write)
179 gcpy     dxfsmsq.write("\n")
180 gcpy
181 gcpy def writedxfsmV(*arguments):
182 gcpy     line_to_write = ""
183 gcpy     for element in arguments:
184 gcpy         line_to_write += element
185 gcpy     dxfsmV.write(line_to_write)
186 gcpy     print(line_to_write)
187 gcpy     dxfsmV.write("\n")
188 gcpy
189 gcpy def writedxfKH(*arguments):
190 gcpy     line_to_write = ""
191 gcpy     for element in arguments:
192 gcpy         line_to_write += element
193 gcpy     dxfKH.write(line_to_write)
194 gcpy     print(line_to_write)
195 gcpy     dxfKH.write("\n")
196 gcpy
197 gcpy def writedxfDT(*arguments):
```

```
198 gcpy      line_to_write = ""
199 gcpy      for element in arguments:
200 gcpy          line_to_write += element
201 gcpy      dxfDT.write(line_to_write)
202 gcpy      print(line_to_write)
203 gcpy      dxfDT.write("\n")
```

owritecomment
dxfwriteone
dxfwritelgbl
dxfwritelgsq
dxfwritelgV
dxfwritesmbl
dxfwritesmsq
dxfwritesmV

Separate OpenSCAD modules 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.

```
170 pyscad module owritecomment(comment) {
171 pyscad         writeln("(",comment,")");
172 pyscad }
173 pyscad
174 pyscad module dxfwriteone(first) {
175 pyscad         writedxf(first);
176 pyscad //       writeln(first);
177 pyscad //       echo(first);
178 pyscad }
179 pyscad
180 pyscad module dxfwritelgbl(first) {
181 pyscad         writedxflgbl(first);
182 pyscad }
183 pyscad
184 pyscad module dxfwritelgsq(first) {
185 pyscad         writedxflgsq(first);
186 pyscad }
187 pyscad
188 pyscad module dxfwritelgV(first) {
189 pyscad         writedxflgV(first);
190 pyscad }
191 pyscad
192 pyscad module dxfwritesmbl(first) {
193 pyscad         writedxfsmbl(first);
194 pyscad }
195 pyscad
196 pyscad module dxfwritesmsq(first) {
197 pyscad         writedxfsmsq(first);
198 pyscad }
199 pyscad
200 pyscad module dxfwritesmV(first) {
201 pyscad         writedxfsmV(first);
202 pyscad }
203 pyscad
204 pyscad module dxfwriteKH(first) {
205 pyscad         writedxfKH(first);
206 pyscad }
207 pyscad
208 pyscad module dxfwriteDT(first) {
209 pyscad         writedxfDT(first);
210 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.

```
212 pyscad module owriteone(first) {
213 pyscad         writeln(first);
214 pyscad }
215 pyscad
216 pyscad module owritetwo(first, second) {
217 pyscad         writeln(first, second);
218 pyscad }
219 pyscad
220 pyscad module owritethree(first, second, third) {
221 pyscad         writeln(first, second, third);
222 pyscad }
223 pyscad
224 pyscad module owritefour(first, second, third, fourth) {
225 pyscad         writeln(first, second, third, fourth);
226 pyscad }
227 pyscad
228 pyscad module owritefive(first, second, third, fourth, fifth)
              {
229 pyscad         writeln(first, second, third, fourth, fifth);
230 pyscad }
```

```
231 pyscad
232 pyscad module owritesix(first, second, third, fourth, fifth,
        sixth) {
233 pyscad         writeln(first, second, third, fourth, fifth,
                    sixth);
234 pyscad }
235 pyscad
236 pyscad module owriteseven(first, second, third, fourth, fifth
        , sixth, seventh) {
237 pyscad         writeln(first, second, third, fourth, fifth,
                    sixth, seventh);
238 pyscad }
239 pyscad
240 pyscad module owriteeight(first, second, third, fourth, fifth
        , sixth, seventh,eighth) {
241 pyscad         writeln(first, second, third, fourth, fifth,
                    sixth, seventh,eighth);
242 pyscad }
243 pyscad
244 pyscad module owritenine(first, second, third, fourth, fifth,
         sixth, seventh, eighth, ninth) {
245 pyscad         writeln(first, second, third, fourth, fifth,
                    sixth, seventh, eighth, ninth);
246 pyscad }
247 pyscad
248 pyscad module owriteten(first, second, third, fourth, fifth,
        sixth, seventh, eighth, ninth, tenth) {
249 pyscad         writeln(first, second, third, fourth, fifth,
                    sixth, seventh, eighth, ninth, tenth);
250 pyscad }
251 pyscad
252 pyscad module owriteeleven(first, second, third, fourth,
        fifth, sixth, seventh, eighth, ninth, tenth,
        eleventh) {
253 pyscad         writeln(first, second, third, fourth, fifth,
                    sixth, seventh, eighth, ninth, tenth,
                    eleventh);
254 pyscad }
255 pyscad
256 pyscad module owritetwelve(first, second, third, fourth,
        fifth, sixth, seventh, eighth, ninth, tenth,
        eleventh, twelfth) {
257 pyscad         writeln(first, second, third, fourth, fifth,
                    sixth, seventh, eighth, ninth, tenth,
                    eleventh, twelfth);
258 pyscad }
259 pyscad
260 pyscad module owritethirteen(first, second, third, fourth,
        fifth, sixth, seventh, eighth, ninth, tenth,
        eleventh, twelfth, thirteenth) {
261 pyscad         writeln(first, second, third, fourth, fifth,
                    sixth, seventh, eighth, ninth, tenth,
                    eleventh, twelfth, thirteenth);
262 pyscad }
```

dxfwrite
dxfpreamble

The dxfwrite module requires that the tool number be passed in, and that value will be used to write out to the appropriate file.

```
201 gcpscad module dxfwrite(tn,arg) {
202 gcpscad if (tn == large_ball_tool_no) {
203 gcpscad     dxfwritelgbl(arg);}
204 gcpscad if (tn == large_square_tool_no) {
205 gcpscad     dxfwritelgsq(arg);}
206 gcpscad if (tn == large_V_tool_no) {
207 gcpscad     dxfwritelgV(arg);}
208 gcpscad if (tn == small_ball_tool_no) {
209 gcpscad     dxfwritesmbl(arg);}
210 gcpscad if (tn == small_square_tool_no) {
211 gcpscad     dxfwritesmsq(arg);}
212 gcpscad if (tn == small_V_tool_no) {
213 gcpscad     dxfwritesmV(arg);}
214 gcpscad if (tn == DT_tool_no) {
215 gcpscad     dxfwriteDT(arg);}
216 gcpscad if (tn == KH_tool_no) {
217 gcpscad     dxfwriteKH(arg);}
218 gcpscad }
219 gcpscad
220 gcpscad module dxfpreamble(tn) {
```

```
221 gcpscad //    echo(str("dxfpreamble",small_square_tool_no));
222 gcpscad      dxfwrite(tn,"0");
223 gcpscad      dxfwrite(tn,"SECTION");
224 gcpscad      dxfwrite(tn,"2");
225 gcpscad      dxfwrite(tn,"ENTITIES");
226 gcpscad }
```

beginpolyline
dxfbpl

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:

- a line: LWPOLYLINE is one possible implementation
- ARC — a notable option would be for the arc to close on itself, creating a circle

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(small\_square\_tool\_no,10,10,5,0,90);
dxfarc(small\_square\_tool\_no,10,10,5,90,180);
dxfarc(small\_square\_tool\_no,10,10,5,180,270);
dxfarc(small\_square\_tool\_no,10,10,5,270,360);
```

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.

```
228 gcpscad module dxfpl(tn,xbegin,ybegin,xend,yend) {
229 gcpscad      dxfwrite(tn,"0");
230 gcpscad      dxfwrite(tn,"LWPOLYLINE");
231 gcpscad      dxfwrite(tn,"90");
232 gcpscad      dxfwrite(tn,"2");
233 gcpscad      dxfwrite(tn,"70");
234 gcpscad      dxfwrite(tn,"0");
235 gcpscad      dxfwrite(tn,"43");
236 gcpscad      dxfwrite(tn,"0");
237 gcpscad      dxfwrite(tn,"10");
238 gcpscad      dxfwrite(tn,str(xbegin));
239 gcpscad      dxfwrite(tn,"20");
240 gcpscad      dxfwrite(tn,str(ybegin));
241 gcpscad      dxfwrite(tn,"10");
242 gcpscad      dxfwrite(tn,str(xend));
243 gcpscad      dxfwrite(tn,"20");
244 gcpscad      dxfwrite(tn,str(yend));
245 gcpscad }
246 gcpscad
247 gcpscad module dxfpolyline(tn,xbegin,ybegin,xend,yend) {
248 gcpscad if (generatedxf == true) {
249 gcpscad      dxfwriteone("0");
250 gcpscad      dxfwriteone("LWPOLYLINE");
251 gcpscad      dxfwriteone("90");
252 gcpscad      dxfwriteone("2");
253 gcpscad      dxfwriteone("70");
254 gcpscad      dxfwriteone("0");
255 gcpscad      dxfwriteone("43");
256 gcpscad      dxfwriteone("0");
257 gcpscad      dxfwriteone("10");
258 gcpscad      dxfwriteone(str(xbegin));
259 gcpscad      dxfwriteone("20");
260 gcpscad      dxfwriteone(str(ybegin));
261 gcpscad      dxfwriteone("10");
262 gcpscad      dxfwriteone(str(xend));
263 gcpscad      dxfwriteone("20");
264 gcpscad      dxfwriteone(str(yend));
265 gcpscad      dxfpl(tn,xbegin,ybegin,xend,yend);
266 gcpscad }
267 gcpscad }
```

```
269 gcpscad module dxfa(tn,xcenter,ycenter,radius,anglebegin,
        endangle) {
270 gcpscad     dxfwrite(tn,"0");
271 gcpscad     dxfwrite(tn,"ARC");
272 gcpscad     dxfwrite(tn,"10");
273 gcpscad     dxfwrite(tn,str(xcenter));
274 gcpscad     dxfwrite(tn,"20");
275 gcpscad     dxfwrite(tn,str(ycenter));
276 gcpscad     dxfwrite(tn,"40");
277 gcpscad     dxfwrite(tn,str(radius));
278 gcpscad     dxfwrite(tn,"50");
279 gcpscad     dxfwrite(tn,str(anglebegin));
280 gcpscad     dxfwrite(tn,"51");
281 gcpscad     dxfwrite(tn,str(endangle));
282 gcpscad }
283 gcpscad
284 gcpscad module dxfarc(tn,xcenter,ycenter,radius,anglebegin,
        endangle) {
285 gcpscad if (generatedxf == true) {
286 gcpscad     dxfwriteone("0");
287 gcpscad     dxfwriteone("ARC");
288 gcpscad     dxfwriteone("10");
289 gcpscad     dxfwriteone(str(xcenter));
290 gcpscad     dxfwriteone("20");
291 gcpscad     dxfwriteone(str(ycenter));
292 gcpscad     dxfwriteone("40");
293 gcpscad     dxfwriteone(str(radius));
294 gcpscad     dxfwriteone("50");
295 gcpscad     dxfwriteone(str(anglebegin));
296 gcpscad     dxfwriteone("51");
297 gcpscad     dxfwriteone(str(endangle));
298 gcpscad         dxfa(tn,xcenter,ycenter,radius,anglebegin,
                    endangle);
299 gcpscad }
300 gcpscad }
```

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

```
302 gcpscad module dxfbpl(tn,bx,by) {
303 gcpscad     dxfwrite(tn,"0");
304 gcpscad     dxfwrite(tn,"POLYLINE");
305 gcpscad     dxfwrite(tn,"8");
306 gcpscad     dxfwrite(tn,"default");
307 gcpscad     dxfwrite(tn,"66");
308 gcpscad     dxfwrite(tn,"1");
309 gcpscad     dxfwrite(tn,"70");
310 gcpscad     dxfwrite(tn,"0");
311 gcpscad     dxfwrite(tn,"0");
312 gcpscad     dxfwrite(tn,"VERTEX");
313 gcpscad     dxfwrite(tn,"8");
314 gcpscad     dxfwrite(tn,"default");
315 gcpscad     dxfwrite(tn,"70");
316 gcpscad     dxfwrite(tn,"32");
317 gcpscad     dxfwrite(tn,"10");
318 gcpscad     dxfwrite(tn,str(bx));
319 gcpscad     dxfwrite(tn,"20");
320 gcpscad     dxfwrite(tn,str(by));
321 gcpscad }
322 gcpscad
323 gcpscad module beginpolyline(bx,by,bz) {
324 gcpscad if (generatedxf == true) {
325 gcpscad     dxfwriteone("0");
326 gcpscad     dxfwriteone("POLYLINE");
327 gcpscad     dxfwriteone("8");
328 gcpscad     dxfwriteone("default");
329 gcpscad     dxfwriteone("66");
330 gcpscad     dxfwriteone("1");
331 gcpscad     dxfwriteone("70");
332 gcpscad     dxfwriteone("0");
333 gcpscad     dxfwriteone("0");
334 gcpscad     dxfwriteone("VERTEX");
335 gcpscad     dxfwriteone("8");
336 gcpscad     dxfwriteone("default");
337 gcpscad     dxfwriteone("70");
338 gcpscad     dxfwriteone("32");
339 gcpscad     dxfwriteone("10");
340 gcpscad     dxfwriteone(str(bx));
```

```
341 gcpscad     dxfwriteone("20");
342 gcpscad     dxfwriteone(str(by));
343 gcpscad     dxfbpl(current_tool(),bx,by);}
344 gcpscad }
345 gcpscad
346 gcpscad module dxfapl(tn,bx,by) {
347 gcpscad     dxfwriteone("0");
348 gcpscad     dxfwrite(tn,"VERTEX");
349 gcpscad     dxfwrite(tn,"8");
350 gcpscad     dxfwrite(tn,"default");
351 gcpscad     dxfwrite(tn,"70");
352 gcpscad     dxfwrite(tn,"32");
353 gcpscad     dxfwrite(tn,"10");
354 gcpscad     dxfwrite(tn,str(bx));
355 gcpscad     dxfwrite(tn,"20");
356 gcpscad     dxfwrite(tn,str(by));
357 gcpscad }
358 gcpscad
359 gcpscad module addpolyline(bx,by,bz) {
360 gcpscad if (generatedxf == true) {
361 gcpscad     dxfwrite(tn,"0");
362 gcpscad     dxfwriteone("VERTEX");
363 gcpscad     dxfwriteone("8");
364 gcpscad     dxfwriteone("default");
365 gcpscad     dxfwriteone("70");
366 gcpscad     dxfwriteone("32");
367 gcpscad     dxfwriteone("10");
368 gcpscad     dxfwriteone(str(bx));
369 gcpscad     dxfwriteone("20");
370 gcpscad     dxfwriteone(str(by));
371 gcpscad     dxfapl(current_tool(),bx,by);
372 gcpscad     }
373 gcpscad }
374 gcpscad
375 gcpscad module dxfcpl(tn) {
376 gcpscad     dxfwrite(tn,"0");
377 gcpscad     dxfwrite(tn,"SEQEND");
378 gcpscad }
379 gcpscad
380 gcpscad module closepolyline() {
381 gcpscad if (generatedxf == true) {
382 gcpscad     dxfwriteone("0");
383 gcpscad     dxfwriteone("SEQEND");
384 gcpscad     dxfcpl(current_tool());
385 gcpscad     }
386 gcpscad }
387 gcpscad
388 gcpscad module writecomment(comment) {
389 gcpscad if (generategcode == true) {
390 gcpscad         owritecomment(comment);
391 gcpscad }
392 gcpscad }
```

pclosegcodefile
pclosesvgfile
pclosedxffile

At the end of the project it will be necessary to close each file. In some instances it will be necessary to write additional information, depending on the file format.

```
205 gcpy def pclosegcodefile():
206 gcpy     f.close()
207 gcpy
208 gcpy def pclosesvgfile():
209 gcpy     svg.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()
```

oclosegcodefile
oclosedxffile
oclosedxflgblfile

```
264 pyscad module oclosegcodefile() {
265 pyscad          pclosegcodefile();
266 pyscad }
267 pyscad
268 pyscad module oclosedxffile() {
269 pyscad          pclosedxffile();
270 pyscad }
271 pyscad
272 pyscad module oclosedxflgblfile() {
273 pyscad          pclosedxflgblfile();
274 pyscad }
275 pyscad
276 pyscad module oclosedxflgsqfile() {
277 pyscad          pclosedxflgsqfile();
278 pyscad }
279 pyscad
280 pyscad module oclosedxflgVfile() {
281 pyscad          pclosedxflgVfile();
282 pyscad }
283 pyscad
284 pyscad module oclosedxfsmblfile() {
285 pyscad          pclosedxfsmblfile();
286 pyscad }
287 pyscad
288 pyscad module oclosedxfsmsqfile() {
289 pyscad          pclosedxfsmsqfile();
290 pyscad }
291 pyscad
292 pyscad module oclosedxfsmVfile() {
293 pyscad          pclosedxfsmVfile();
294 pyscad }
295 pyscad
296 pyscad module oclosedxfDTfile() {
297 pyscad          pclosedxfDTfile();
298 pyscad }
299 pyscad
300 pyscad module oclosedxfKHfile() {
301 pyscad          pclosedxfKHfile();
302 pyscad }
303 pyscad
304 pyscad module oclosesvgfile() {
305 pyscad          pclosesvgfile();
306 pyscad }
```

closegcodefile
dxfpostamble
closedxffile

```
394 gcpscad module closegcodefile() {
395 gcpscad if (generategcode == true) {
396 gcpscad     owriteone("M05");
397 gcpscad     owriteone("M02");
398 gcpscad         oclosegcodefile();
399 gcpscad }
400 gcpscad }
401 gcpscad
402 gcpscad module dxfpostamble(arg) {
403 gcpscad     dxfwrite(arg,"0");
404 gcpscad     dxfwrite(arg,"ENDSEC");
405 gcpscad     dxfwrite(arg,"0");
406 gcpscad     dxfwrite(arg,"EOF");
407 gcpscad }
408 gcpscad
409 gcpscad module closedxffile() {
410 gcpscad if (generatedxf == true) {
411 gcpscad     dxfwriteone("0");
```

```
412 gcpscad     dxfwriteone("ENDSEC");
413 gcpscad     dxfwriteone("0");
414 gcpscad     dxfwriteone("EOF");
415 gcpscad         oclosedxffile();
416 gcpscad     echo("CLOSING");
417 gcpscad if (large_ball_tool_no >  0) {  dxfpostamble(
                large_ball_tool_no);
418 gcpscad     oclosedxflgblfile();
419 gcpscad }
420 gcpscad if (large_square_tool_no >  0) {        dxfpostamble(
                large_square_tool_no);
421 gcpscad     oclosedxflgsqfile();
422 gcpscad }
423 gcpscad if (large_V_tool_no >  0) {     dxfpostamble(
                large_V_tool_no);
424 gcpscad     oclosedxflgVfile();
425 gcpscad }
426 gcpscad if (small_ball_tool_no >  0) {  dxfpostamble(
                small_ball_tool_no);
427 gcpscad     oclosedxfsmblfile();
428 gcpscad }
429 gcpscad if (small_square_tool_no >  0) {        dxfpostamble(
                small_square_tool_no);
430 gcpscad     oclosedxfsmsqfile();
431 gcpscad }
432 gcpscad if (small_V_tool_no >  0) {     dxfpostamble(
                small_V_tool_no);
433 gcpscad     oclosedxfsmVfile();
434 gcpscad }
435 gcpscad if (DT_tool_no >  0) {  dxfpostamble(DT_tool_no);
436 gcpscad     oclosedxfDTfile();
437 gcpscad }
438 gcpscad if (KH_tool_no >  0) {  dxfpostamble(KH_tool_no);
439 gcpscad     oclosedxfKHfile();
440 gcpscad }
441 gcpscad     }
442 gcpscad }
443 gcpscad
444 gcpscad module closesvgfile() {
445 gcpscad if (generatesvg == true) {
446 gcpscad     svgwriteone("</svg> ");
447 gcpscad         oclosesvgfile();
448 gcpscad     echo("CLOSING SVG");
449 gcpscad     }
450 gcpscad }
```

2.4 Movement and Cutting

otm ocut orapid

With all the scaffolding in place, it is possible to model tool movement and cutting and to write out files which represent the desired machine motions.

```
452 gcpscad module otm(ex, ey, ez, r,g,b) {
453 gcpscad color([r,g,b]) hull(){
454 gcpscad     translate([xpos(), ypos(), zpos()]){
455 gcpscad       select_tool(current_tool());
456 gcpscad     }
457 gcpscad     translate([ex, ey, ez]){
458 gcpscad       select_tool(current_tool());
459 gcpscad     }
460 gcpscad   }
461 gcpscad oset(ex, ey, ez);
462 gcpscad }
463 gcpscad
464 gcpscad module ocut(ex, ey, ez) {
465 gcpscad //color([0.2,1,0.2]) hull(){
466 gcpscad otm(ex, ey, ez, 0.2,1,0.2);
467 gcpscad }
468 gcpscad
469 gcpscad module orapid(ex, ey, ez) {
470 gcpscad //color([0.93,0,0]) hull(){
471 gcpscad otm(ex, ey, ez, 0.93,0,0);
472 gcpscad }
473 gcpscad
474 gcpscad module rapidbx(bx, by, bz, ex, ey, ez) {
475 gcpscad //      writeln("G0 X",bx," Y", by, "Z", bz);
476 gcpscad if (generategcode == true) {
477 gcpscad         writecomment("rapid");
```

```
478 gcpscad          owritesix("G0 X",str(ex)," Y", str(ey), " Z",
                        str(ez));
479 gcpscad }
480 gcpscad     orapid(ex, ey, ez);
481 gcpscad }
482 gcpscad
483 gcpscad module rapid(ex, ey, ez) {
484 gcpscad //      writeln("G0 X",bx," Y", by, "Z", bz);
485 gcpscad if (generategcode == true) {
486 gcpscad          writecomment("rapid");
487 gcpscad          owritesix("G0 X",str(ex)," Y", str(ey), " Z",
                        str(ez));
488 gcpscad }
489 gcpscad     orapid(ex, ey, ez);
490 gcpscad }
491 gcpscad
492 gcpscad module movetosafez() {
493 gcpscad //this should be move to retract height
494 gcpscad if (generategcode == true) {
495 gcpscad          writecomment("Move to safe Z to avoid
                        workholding");
496 gcpscad     owriteone("G53G0Z-5.000");
497 gcpscad }
498 gcpscad     orapid(getxpos(), getypos(), retractheight+55);
499 gcpscad }
500 gcpscad
501 gcpscad module begintoolpath(bx,by,bz) {
502 gcpscad if (generategcode == true) {
503 gcpscad          writecomment("PREPOSITION FOR RAPID PLUNGE");
504 gcpscad     owritefour("G0X", str(bx), "Y",str(by));
505 gcpscad     owritetwo("Z", str(bz));
506 gcpscad     }
507 gcpscad     orapid(bx,by,bz);
508 gcpscad }
509 gcpscad
510 gcpscad module movetosafeheight() {
511 gcpscad //this should be move to machine position
512 gcpscad if (generategcode == true) {
513 gcpscad //      writecomment("PREPOSITION FOR RAPID PLUNGE");
            Z25.650
514 gcpscad //G1Z24.663F381.0 ,"F",str(plunge)
515 gcpscad if (zeroheight == "Top") {
516 gcpscad     owritetwo("Z",str(retractheight));
517 gcpscad }
518 gcpscad }
519 gcpscad     orapid(getxpos(), getypos(), retractheight+55);
520 gcpscad }
521 gcpscad
522 gcpscad module cutoneaxis_setfeed(axis,depth,feed) {
523 gcpscad if (generategcode == true) {
524 gcpscad //      writecomment("PREPOSITION FOR RAPID PLUNGE");
            Z25.650
525 gcpscad //G1Z24.663F381.0 ,"F",str(plunge) G1Z7.612F381.0
526 gcpscad if (zeroheight == "Top") {
527 gcpscad     owritefive("G1",axis,str(depth),"F",str(feed));
528 gcpscad }
529 gcpscad }
530 gcpscad if (axis == "X") {setxpos(depth);
531 gcpscad ocut(depth, getypos(), getzpos());}
532 gcpscad if (axis == "Y") {setypos(depth);
533 gcpscad ocut(getxpos(), depth, getzpos());}
534 gcpscad if (axis == "Z") {setzpos(depth);
535 gcpscad ocut(getxpos(), getypos(), depth);}
536 gcpscad }
537 gcpscad
538 gcpscad module cut(ex, ey, ez) {
539 gcpscad //      writeln("G0 X",bx," Y", by, "Z", bz);
540 gcpscad if (generategcode == true) {
541 gcpscad          owritesix("G1 X",str(ex)," Y", str(ey), " Z",
                        str(ez));
542 gcpscad }
543 gcpscad //if (generatesvg == true) {
544 gcpscad //      owritesix("G1 X",str(ex)," Y", str(ey), " Z",
            str(ez));
545 gcpscad //    orapid(getxpos(), getypos(), retractheight+5);
546 gcpscad //    writesvgline(getxpos(),getypos(),ex,ey);
547 gcpscad //}
548 gcpscad ocut(ex, ey, ez);
```

```
549 gcpscad }
550 gcpscad
551 gcpscad module cutwithfeed(ex, ey, ez, feed) {
552 gcpscad //      writeln("G0 X",bx," Y", by, "Z", bz);
553 gcpscad if (generategcode == true) {
554 gcpscad //      writecomment("rapid");
555 gcpscad         owriteeight("G1 X",str(ex)," Y", str(ey), " Z
                        ", str(ez),"F",str(feed));
556 gcpscad }
557 gcpscad ocut(ex, ey, ez);
558 gcpscad }
559 gcpscad
560 gcpscad module endtoolpath() {
561 gcpscad if (generategcode == true) {
562 gcpscad //Z31.750
563 gcpscad //      owriteone("G53G0Z-5.000");
564 gcpscad     owritetwo("Z",str(retractheight));
565 gcpscad }
566 gcpscad     orapid(getxpos(),getypos(),retractheight);
567 gcpscad }
```

3 gcodepreviewtemplate.scad

The commands may then be put together using a template which will ensure that the various files are used/included as necessary, that files are opened before being written to, and that they are closed at the end.

```
 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
13 gcptmpl /* [Export] */
14 gcptmpl generatedxf = true;
15 gcptmpl
16 gcptmpl /* [Export] */
17 gcptmpl generategcode = true;
18 gcptmpl
19 gcptmpl ///* [Export] */
20 gcptmpl //generatesvg = false;
21 gcptmpl
22 gcptmpl /* [CAM] */
23 gcptmpl toolradius = 1.5875;
24 gcptmpl
25 gcptmpl /* [CAM] */
26 gcptmpl large_ball_tool_no = 0; //
            [0:0,111:111,101:101,202:202]
27 gcptmpl
28 gcptmpl /* [CAM] */
29 gcptmpl large_square_tool_no = 0; //
            [0:0,112:112,102:102,201:201]
30 gcptmpl
31 gcptmpl /* [CAM] */
32 gcptmpl large_V_tool_no = 0; // [0:0,301:301,690:690]
33 gcptmpl
34 gcptmpl /* [CAM] */
35 gcptmpl small_ball_tool_no = 0; //
            [0:0,121:121,111:111,101:101]
36 gcptmpl
37 gcptmpl /* [CAM] */
38 gcptmpl small_square_tool_no = 102; //
            [0:0,122:122,112:112,102:102]
39 gcptmpl
40 gcptmpl /* [CAM] */
41 gcptmpl small_V_tool_no = 0; // [0:0,390:390,301:301]
42 gcptmpl
43 gcptmpl /* [CAM] */
44 gcptmpl KH_tool_no = 0; // [0:0,375:375]
45 gcptmpl
```

```
gcptmpl /* [CAM] */
gcptmpl DT_tool_no = 0; // [0:0,814:814]
gcptmpl
gcptmpl /* [Feeds and Speeds] */
gcptmpl plunge = 100;
gcptmpl
gcptmpl /* [Feeds and Speeds] */
gcptmpl feed = 400;
gcptmpl
gcptmpl /* [Feeds and Speeds] */
gcptmpl speed = 16000;
gcptmpl
gcptmpl /* [Feeds and Speeds] */
gcptmpl square_ratio = 1.0; // [0.25:2]
gcptmpl
gcptmpl /* [Feeds and Speeds] */
gcptmpl small_V_ratio = 0.75; // [0.25:2]
gcptmpl
gcptmpl /* [Feeds and Speeds] */
gcptmpl large_V_ratio = 0.875; // [0.25:2]
gcptmpl
gcptmpl /* [Stock] */
gcptmpl stocklength = 219;
gcptmpl
gcptmpl /* [Stock] */
gcptmpl stockwidth = 150;
gcptmpl
gcptmpl /* [Stock] */
gcptmpl stockthickness = 8.35;
gcptmpl
gcptmpl /* [Stock] */
gcptmpl zeroheight = "Top"; // [Top, Bottom]
gcptmpl
gcptmpl /* [Stock] */
gcptmpl stockorigin = "Center"; // [Lower-Left, Center-Left,
            Top-Left, Center]
gcptmpl
gcptmpl /* [Stock] */
gcptmpl retractheight = 9;
gcptmpl
gcptmpl filename_gcode = str(Base_filename, ".nc");
gcptmpl filename_dxf = str(Base_filename);
gcptmpl //filename_svg = str(Base_filename, ".svg");
gcptmpl
gcptmpl opengcodefile(filename_gcode);
gcptmpl opendxffile(filename_dxf);
gcptmpl
gcptmpl difference() {
gcptmpl setupstock(stocklength, stockwidth, stockthickness,
            zeroheight, stockorigin);
gcptmpl
gcptmpl movetosafez();
gcptmpl
gcptmpl toolchange(small_square_tool_no,speed * square_ratio);
gcptmpl
gcptmpl begintoolpath(0,0,0.25);
gcptmpl beginpolyline(0,0,0.25);
gcptmpl
gcptmpl cutoneaxis_setfeed("Z",0,plunge*square_ratio);
gcptmpl
gcptmpl cutwithfeed(stocklength/2,stockwidth/2,-stockthickness
            ,feed);
gcptmpl addpolyline(stocklength/2,stockwidth/2,-stockthickness
            );
gcptmpl
gcptmpl endtoolpath();
gcptmpl closepolyline();
gcptmpl }
gcptmpl
gcptmpl closegcodefile();
gcptmpl closedxffile();
```

4 cut2Dshapes and expansion

New features will be tried out in a file such as `cut2Dshapes.scad` insofar as the file structures will allow (tool definitions for example will need to consolidated in 2.2.1 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 will be to define two-dimensional regions 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 use 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
- (Rectangular) Pocket — such toolpaths/geometry should include the rounding of the tool at the corners
- 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 dovetail 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)

4.1 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(tn,xcenter,ycenter,radius,anglebegin,endangle)`
- 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

Adding a simple loop to handle the processing of the `cut()` toolpaths 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):

```
1 cut2D //!OpenSCAD
2 cut2D
3 cut2D module arcloop(barc,earc, xcenter, ycenter, radius) {
4 cut2D   for (i = [barc : abs(1) : earc]) {
5 cut2D         cut(xcenter + radius * cos(i),
6 cut2D         ycenter + radius * sin(i),
7 cut2D         getzpos()-(gettzpos())
8 cut2D         );
9 cut2D     setxpos(xcenter + radius * cos(i));
10 cut2D    setypos(ycenter + radius * sin(i));
11 cut2D  }
12 cut2D }
13 cut2D
14 cut2D module narcloop(barc,earc, xcenter, ycenter, radius) {
15 cut2D   for (i = [barc : -1 : earc]) {
16 cut2D         cut(xcenter + radius * cos(i),
17 cut2D         ycenter + radius * sin(i),
18 cut2D         getzpos()-(gettzpos())
19 cut2D         );
20 cut2D    setxpos(xcenter + radius * cos(i));
21 cut2D    setypos(ycenter + radius * sin(i));
22 cut2D  }
23 cut2D }
```

The various textual versions are quite obvious:

```
25 cut2D module cutarcNECCdxf(tn, ex, ey, ez, xcenter, ycenter,
       radius) {
26 cut2D   dxfarc(tn,xcenter,ycenter,radius,0,90);
27 cut2D   settzpos((getzpos()-ez)/90);
```

```
28 cut2D     arcloop(1,90, xcenter, ycenter, radius);
29 cut2D }
30 cut2D
31 cut2D module cutarcNWCCdxf(tn, ex, ey, ez, xcenter, ycenter,
           radius) {
32 cut2D   dxfarc(tn,xcenter,ycenter,radius,90,180);
33 cut2D   settzpos((getzpos()-ez)/90);
34 cut2D     arcloop(91,180, xcenter, ycenter, radius);
35 cut2D }
36 cut2D
37 cut2D module cutarcSWCCdxf(tn, ex, ey, ez, xcenter, ycenter,
           radius) {
38 cut2D   dxfarc(tn,xcenter,ycenter,radius,180,270);
39 cut2D   settzpos((getzpos()-ez)/90);
40 cut2D     arcloop(181,270, xcenter, ycenter, radius);
41 cut2D }
42 cut2D
43 cut2D module cutarcSECCdxf(tn, ex, ey, ez, xcenter, ycenter,
           radius) {
44 cut2D   dxfarc(tn,xcenter,ycenter,radius,270,360);
45 cut2D   settzpos((getzpos()-ez)/90);
46 cut2D     arcloop(271,360, xcenter, ycenter, radius);
47 cut2D }
48 cut2D
49 cut2D module cutarcNECWdxf(tn, ex, ey, ez, xcenter, ycenter,
           radius) {
50 cut2D   dxfarc(tn,xcenter,ycenter,radius,0,90);
51 cut2D   settzpos((getzpos()-ez)/90);
52 cut2D     narcloop(89,0, xcenter, ycenter, radius);
53 cut2D }
54 cut2D
55 cut2D module cutarcSECWdxf(tn, ex, ey, ez, xcenter, ycenter,
           radius) {
56 cut2D   dxfarc(tn,xcenter,ycenter,radius,270,360);
57 cut2D   settzpos((getzpos()-ez)/90);
58 cut2D     narcloop(359,270, xcenter, ycenter, radius);
59 cut2D }
60 cut2D
61 cut2D module cutarcSWCWdxf(tn, ex, ey, ez, xcenter, ycenter,
           radius) {
62 cut2D   dxfarc(tn,xcenter,ycenter,radius,180,270);
63 cut2D   settzpos((getzpos()-ez)/90);
64 cut2D     narcloop(269,180, xcenter, ycenter, radius);
65 cut2D }
66 cut2D
67 cut2D module cutarcNWCWdxf(tn, ex, ey, ez, xcenter, ycenter,
           radius) {
68 cut2D   dxfarc(tn,xcenter,ycenter,radius,90,180);
69 cut2D   settzpos((getzpos()-ez)/90);
70 cut2D     narcloop(179,90, xcenter, ycenter, radius);
71 cut2D }
```

4.2 Keyhole toolpath and undercut tooling

The most topologically interesting toolpath is "Keyhole" — 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 2.2.1.2

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, W, S, and E will be supported in the initial version. This will be done by wrapping the command with a version which only accepts those options:

```
cut2D module keyhole_toolpath(kh_tool_no, kh_start_depth,
          kh_max_depth, kht_angle, kh_length) {
cut2D if (kht_angle == "N") {
cut2D   keyhole_toolpath_degrees(kh_tool_no, kh_start_depth,
            kh_max_depth, 90, kh_length);
cut2D    } else if (kht_angle == "S") {
cut2D   keyhole_toolpath_degrees(kh_tool_no, kh_start_depth,
            kh_max_depth, 270, kh_length);
cut2D    } else if (kht_angle == "E") {
cut2D   keyhole_toolpath_degrees(kh_tool_no, kh_start_depth,
            kh_max_depth, 0, kh_length);
cut2D    } else if (kht_angle == "W") {
cut2D   keyhole_toolpath_degrees(kh_tool_no, kh_start_depth,
            kh_max_depth, 180, kh_length);
cut2D    }
cut2D }
```

The original version of the command is renamed and called by that. 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:

```
cut2D module keyhole_toolpath_degrees(kh_tool_no,
          kh_start_depth, kh_max_depth, kh_angle, kh_length)
           {
cut2D dxfarc(KH_tool_no,getxpos(),getypos(),tool_diameter(
          KH_tool_no, (kh_max_depth+4.36))/2,0,90);
cut2D dxfarc(KH_tool_no,getxpos(),getypos(),tool_diameter(
          KH_tool_no, (kh_max_depth+4.36))/2,90,180);
cut2D dxfarc(KH_tool_no,getxpos(),getypos(),tool_diameter(
          KH_tool_no, (kh_max_depth+4.36))/2,180,270);
cut2D dxfarc(KH_tool_no,getxpos(),getypos(),tool_diameter(
          KH_tool_no, (kh_max_depth+4.36))/2,270,360);
```

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

```
cut2D   if (kh_angle == 0) {
cut2D dxfarc(KH_tool_no,getxpos(),getypos(),tool_diameter(
          KH_tool_no, (kh_max_depth))/2,180,270);
cut2D dxfarc(KH_tool_no,getxpos(),getypos(),tool_diameter(
          KH_tool_no, (kh_max_depth))/2,90,180);
cut2D dxfarc(KH_tool_no,getxpos(),getypos(),tool_diameter(
          KH_tool_no, (kh_max_depth))/2,asin((tool_diameter(
          KH_tool_no, (kh_max_depth+4.36))/2)/(tool_diameter
          (KH_tool_no, (kh_max_depth))/2)),90);
cut2D dxfarc(KH_tool_no,getxpos(),getypos(),tool_diameter(
          KH_tool_no, (kh_max_depth))/2,270,360-asin((
          tool_diameter(KH_tool_no, (kh_max_depth+4.36))/2)
          /(tool_diameter(KH_tool_no, (kh_max_depth))/2)));
cut2D dxfarc(KH_tool_no,getxpos()+kh_length,getypos(),
          tool_diameter(KH_tool_no, (kh_max_depth+4.36))
          /2,0,90);
cut2D dxfarc(KH_tool_no,getxpos()+kh_length,getypos(),
          tool_diameter(KH_tool_no, (kh_max_depth+4.36))
          /2,270,360);
cut2D dxfpolyline(KH_tool_no,
cut2D  getxpos()+sqrt((tool_diameter(KH_tool_no, (
           kh_max_depth))/2)^2-(tool_diameter(KH_tool_no, (
           kh_max_depth+4.36))/2)^2),
cut2D  getypos()+tool_diameter(KH_tool_no, (kh_max_depth
           +4.36))/2,
cut2D  getxpos()+kh_length,
cut2D  getypos()+tool_diameter(KH_tool_no, (kh_max_depth
           +4.36))/2);
cut2D dxfpolyline(KH_tool_no,
cut2D  getxpos()+sqrt((tool_diameter(KH_tool_no, (
           kh_max_depth))/2)^2-(tool_diameter(KH_tool_no, (
           kh_max_depth+4.36))/2)^2),
cut2D  getypos()-tool_diameter(KH_tool_no, (kh_max_depth
           +4.36))/2,
cut2D  getxpos()+kh_length,
cut2D  getypos()-tool_diameter(KH_tool_no, (kh_max_depth
           +4.36))/2);
cut2D dxfpolyline(KH_tool_no,getxpos(),getypos(),getxpos()+
          kh_length,getypos());
cut2D  cutwithfeed(getxpos()+kh_length,getypos(),-
           kh_max_depth,feed);
```

```
110 cut2D  setxpos(getxpos()-kh_length);
111 cut2D   } else if (kh_angle > 0 && kh_angle < 90) {
112 cut2D echo(kh_angle);
113 cut2D   dxfarc(KH_tool_no,getxpos(),getypos(),tool_diameter(
              KH_tool_no, (kh_max_depth))/2,90+kh_angle,180+
              kh_angle);
114 cut2D   dxfarc(KH_tool_no,getxpos(),getypos(),tool_diameter(
              KH_tool_no, (kh_max_depth))/2,180+kh_angle,270+
              kh_angle);
115 cut2D dxfarc(KH_tool_no,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);
116 cut2D dxfarc(KH_tool_no,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)));
117 cut2D dxfarc(KH_tool_no,
118 cut2D   getxpos()+(kh_length*cos(kh_angle)),
119 cut2D   getypos()+(kh_length*sin(kh_angle)),tool_diameter(
              KH_tool_no, (kh_max_depth+4.36))/2,0+kh_angle
              ,90+kh_angle);
120 cut2D dxfarc(KH_tool_no,getxpos()+(kh_length*cos(kh_angle)),
            getypos()+(kh_length*sin(kh_angle)),tool_diameter(
            KH_tool_no, (kh_max_depth+4.36))/2,270+kh_angle
            ,360+kh_angle);
121 cut2D dxfpolyline(KH_tool_no,
122 cut2D  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))),
123 cut2D  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))),
124 cut2D  getxpos()+(kh_length*cos(kh_angle))-((tool_diameter(
             KH_tool_no, (kh_max_depth+4.36))/2)*sin(kh_angle)
             ),
125 cut2D  getypos()+(kh_length*sin(kh_angle))+((tool_diameter(
             KH_tool_no, (kh_max_depth+4.36))/2)*cos(kh_angle)
             ));
126 cut2D echo("a",tool_diameter(KH_tool_no, (kh_max_depth+4.36)
            )/2);
127 cut2D echo("c",tool_diameter(KH_tool_no, (kh_max_depth))/2);
128 cut2D echo("Aangle",asin((tool_diameter(KH_tool_no, (
            kh_max_depth+4.36))/2)/(tool_diameter(KH_tool_no,
            (kh_max_depth))/2)));
129 cut2D echo(kh_angle);
130 cut2D  cutwithfeed(getxpos()+(kh_length*cos(kh_angle)),
             getypos()+(kh_length*sin(kh_angle)),-kh_max_depth
             ,feed);
131 cut2D  setxpos(getxpos()-(kh_length*cos(kh_angle)));
132 cut2D  setypos(getypos()-(kh_length*sin(kh_angle)));
133 cut2D   } else if (kh_angle == 90) {
134 cut2D dxfarc(KH_tool_no,getxpos(),getypos(),tool_diameter(
            KH_tool_no, (kh_max_depth))/2,180,270);
135 cut2D dxfarc(KH_tool_no,getxpos(),getypos(),tool_diameter(
            KH_tool_no, (kh_max_depth))/2,270,360);
136 cut2D dxfarc(KH_tool_no,getxpos(),getypos(),tool_diameter(
            KH_tool_no, (kh_max_depth))/2,0,90-asin(
137 cut2D     (tool_diameter(KH_tool_no, (kh_max_depth+4.36))/2)
                /(tool_diameter(KH_tool_no, (kh_max_depth))/2)
                ));
138 cut2D dxfarc(KH_tool_no,getxpos(),getypos(),tool_diameter(
            KH_tool_no, (kh_max_depth))/2,90+asin(
139 cut2D     (tool_diameter(KH_tool_no, (kh_max_depth+4.36))/2)
                /(tool_diameter(KH_tool_no, (kh_max_depth))/2)
                ),180);
140 cut2D  dxfpolyline(KH_tool_no,getxpos(),getypos(),getxpos(),
             getypos()+kh_length);
141 cut2D dxfarc(KH_tool_no,getxpos(),getypos()+kh_length,
            tool_diameter(KH_tool_no, (kh_max_depth+4.36))
            /2,0,90);
142 cut2D dxfarc(KH_tool_no,getxpos(),getypos()+kh_length,
            tool_diameter(KH_tool_no, (kh_max_depth+4.36))
            /2,90,180);
```

```
143 cut2D  dxfpolyline(KH_tool_no,getxpos()+tool_diameter(
               KH_tool_no, (kh_max_depth+4.36))/2,getypos()+sqrt
               ((tool_diameter(KH_tool_no, (kh_max_depth))/2)
               ^2-(tool_diameter(KH_tool_no, (kh_max_depth+4.36)
               )/2)^2),getxpos()+tool_diameter(KH_tool_no, (
               kh_max_depth+4.36))/2,getypos()+kh_length);
144 cut2D  dxfpolyline(KH_tool_no,getxpos()-tool_diameter(
               KH_tool_no, (kh_max_depth+4.36))/2,getypos()+sqrt
               ((tool_diameter(KH_tool_no, (kh_max_depth))/2)
               ^2-(tool_diameter(KH_tool_no, (kh_max_depth+4.36)
               )/2)^2),getxpos()-tool_diameter(KH_tool_no, (
               kh_max_depth+4.36))/2,getypos()+kh_length);
145 cut2D  cutwithfeed(getxpos(),getypos()+kh_length,-
               kh_max_depth,feed);
146 cut2D  setypos(getypos()-kh_length);
147 cut2D   } else if (kh_angle == 180) {
148 cut2D dxfarc(KH_tool_no,getxpos(),getypos(),tool_diameter(
            KH_tool_no, (kh_max_depth))/2,0,90);
149 cut2D dxfarc(KH_tool_no,getxpos(),getypos(),tool_diameter(
            KH_tool_no, (kh_max_depth))/2,270,360);
150 cut2D dxfarc(KH_tool_no,getxpos(),getypos(),tool_diameter(
            KH_tool_no, (kh_max_depth))/2,90,180-asin((
            tool_diameter(KH_tool_no, (kh_max_depth+4.36))/2)
            /(tool_diameter(KH_tool_no, (kh_max_depth))/2)));
151 cut2D dxfarc(KH_tool_no,getxpos(),getypos(),tool_diameter(
            KH_tool_no, (kh_max_depth))/2,180+asin((
            tool_diameter(KH_tool_no, (kh_max_depth+4.36))/2)
            /(tool_diameter(KH_tool_no, (kh_max_depth))/2))
            ,270);
152 cut2D dxfarc(KH_tool_no,getxpos()-kh_length,getypos(),
            tool_diameter(KH_tool_no, (kh_max_depth+4.36))
            /2,90,180);
153 cut2D dxfarc(KH_tool_no,getxpos()-kh_length,getypos(),
            tool_diameter(KH_tool_no, (kh_max_depth+4.36))
            /2,180,270);
154 cut2D dxfpolyline(KH_tool_no,
155 cut2D  getxpos()-sqrt((tool_diameter(KH_tool_no, (
             kh_max_depth))/2)^2-(tool_diameter(KH_tool_no, (
             kh_max_depth+4.36))/2)^2),
156 cut2D  getypos()+tool_diameter(KH_tool_no, (kh_max_depth
             +4.36))/2,
157 cut2D  getxpos()-kh_length,
158 cut2D  getypos()+tool_diameter(KH_tool_no, (kh_max_depth
             +4.36))/2);
159 cut2D dxfpolyline(KH_tool_no,
160 cut2D  getxpos()-sqrt((tool_diameter(KH_tool_no, (
             kh_max_depth))/2)^2-(tool_diameter(KH_tool_no, (
             kh_max_depth+4.36))/2)^2),
161 cut2D  getypos()-tool_diameter(KH_tool_no, (kh_max_depth
             +4.36))/2,
162 cut2D  getxpos()-kh_length,
163 cut2D  getypos()-tool_diameter(KH_tool_no, (kh_max_depth
             +4.36))/2);
164 cut2D  dxfpolyline(KH_tool_no,getxpos(),getypos(),getxpos()-
             kh_length,getypos());
165 cut2D  cutwithfeed(getxpos()-kh_length,getypos(),-
             kh_max_depth,feed);
166 cut2D  setxpos(getxpos()+kh_length);
167 cut2D   } else if (kh_angle == 270) {
168 cut2D dxfarc(KH_tool_no,getxpos(),getypos(),tool_diameter(
            KH_tool_no, (kh_max_depth))/2,0,90);
169 cut2D dxfarc(KH_tool_no,getxpos(),getypos(),tool_diameter(
            KH_tool_no, (kh_max_depth))/2,90,180);
170 cut2D dxfarc(KH_tool_no,getxpos(),getypos(),tool_diameter(
            KH_tool_no, (kh_max_depth))/2,270+asin((
            tool_diameter(KH_tool_no, (kh_max_depth+4.36))/2)
            /(tool_diameter(KH_tool_no, (kh_max_depth))/2))
            ,360);
171 cut2D dxfarc(KH_tool_no,getxpos(),getypos(),tool_diameter(
            KH_tool_no, (kh_max_depth))/2,180, 270-asin((
            tool_diameter(KH_tool_no, (kh_max_depth+4.36))/2)
            /(tool_diameter(KH_tool_no, (kh_max_depth))/2)));
172 cut2D dxfarc(KH_tool_no,getxpos(),getypos()-kh_length,
            tool_diameter(KH_tool_no, (kh_max_depth+4.36))
            /2,180,270);
173 cut2D dxfarc(KH_tool_no,getxpos(),getypos()-kh_length,
            tool_diameter(KH_tool_no, (kh_max_depth+4.36))
            /2,270,360);
```

```
174 cut2D  dxfpolyline(KH_tool_no,getxpos()+tool_diameter(
               KH_tool_no, (kh_max_depth+4.36))/2,getypos()-sqrt
               ((tool_diameter(KH_tool_no, (kh_max_depth))/2)
               ^2-(tool_diameter(KH_tool_no, (kh_max_depth+4.36)
               )/2)^2),getxpos()+tool_diameter(KH_tool_no, (
               kh_max_depth+4.36))/2,getypos()-kh_length);
175 cut2D  dxfpolyline(KH_tool_no,getxpos()-tool_diameter(
               KH_tool_no, (kh_max_depth+4.36))/2,getypos()-sqrt
               ((tool_diameter(KH_tool_no, (kh_max_depth))/2)
               ^2-(tool_diameter(KH_tool_no, (kh_max_depth+4.36)
               )/2)^2),getxpos()-tool_diameter(KH_tool_no, (
               kh_max_depth+4.36))/2,getypos()-kh_length);
176 cut2D  dxfpolyline(KH_tool_no,getxpos(),getypos(),getxpos(),
               getypos()-kh_length);
177 cut2D  cutwithfeed(getxpos(),getypos()-kh_length,-
               kh_max_depth,feed);
178 cut2D  setypos(getypos()+kh_length);
179 cut2D   }
180 cut2D }
```

4.3 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.

4.3.1 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.

4.3.1.1 begincutdxf The first command 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 `continuecutdxf`.

`begincutdxf`

```
182 cut2D module begincutdxf(rh, ex, ey, ez, fr) {
183 cut2D   rapid(getxpos(),getypos(),rh);
184 cut2D   cutwithfeed(ex,ey,ez,fr);
185 cut2D }
```

```
187 cut2D module continuecutdxf(ex, ey, ez, fr) {
188 cut2D   cutwithfeed(ex,ey,ez,fr);
189 cut2D }
```

4.4 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.

5 Other Resources

Holidays are from https://nationaltoday.com/

References

[ConstGeom] Walmsley, Brian. *Construction Geometry*. 2d ed., Centennial College Press, 1981.

[MkCalc] Horvath, Joan, and Rich Cameron. *Make: Calculus: Build models to learn, visualize, and explore*. First edition., Make: Community LLC, 2022.

[MkGeom] Horvath, Joan, and Rich Cameron. *Make: Geometry: Learn by 3D Printing, Coding and Exploring*. First edition., Make: Community LLC, 2021.

[MkTrig] Horvath, Joan, and Rich Cameron. *Make: Trigonometry: Build your way from triangles to analytic geometry*. First edition., Make: Community LLC, 2023.

[PractShopMath] Begnal, Tom. *Practical Shop Math: Simple Solutions to Workshop Fractions, Formulas + Geometric Shapes*. Updated edition, Spring House Press, 2018.

[RS274] Thomas R. Kramer, Frederick M. Proctor, Elena R. Messina.
https://tsapps.nist.gov/publication/get_pdf.cfm?pub_id=823374
https://www.nist.gov/publications/nist-rs274ngc-interpreter-version-3