AutoCAD 2 G-Code beta v0.45 (10-17-09)

*** Warning***
All g-code produced by this program should be verified prior to running on any machine.
As this program is under development, unknown errors may occur.

Note: screen resolution should be at least 1024 x 768. The main window may not fit with lower resolutions.
Edit: As of V0.44, screen size requirements have been reduced. 1280 x 1024 is no longer needed.

AutoCAD 2 G-Code is an AutoCAD macro that will export a g-code file for Polylines, 3Dpolylines, and Circles that are drawn in model space only. Regular Lines and Arcs are NOT supported, but can be converted to polylines using the PEDIT command in AutoCAD. All references to "polylines" imply lwpolylines. Old style 2d polylines are not yet supported.

G2/G3 moves always use Incremental I and J.

Circles are always coded as holes. If you need to cut out round parts, draw the tool paths using polylines with arcs. If Circles in the drawing are the same size as the tool, then a G83 drill cycle will be used. G83 uses the ENTRY feed rate. The format should work in both TurboCNC v4 and Mach2/3.

Ex: G83 X2 Y3 Z-.5 R0.05 Q0.1

Where R is the retract height and Q is the depth increment.

G41/G42 are formatted for Mach2/3, using the P word for tool radius. This may be changed in the future to using tool #'s, as this will be needed if and when tool change ability is added.

You can email me with any questions at cncwoodworker <at> comcast.net

Operation

Before running, the user must create a folder named "gcode" on their C: drive. This is the default folder where g-code files will be saved. In this folder there must be another folder named "config". This is where the program settings will be saved, in a file named ac2gc.ini. There is a sample file included in the download, but if it does not exist, it will be created by using the Save Settings button. Note: v0.45 will not recognize older settings files.
Again, before using, create

C:\gcode, and C:\gcode\config.

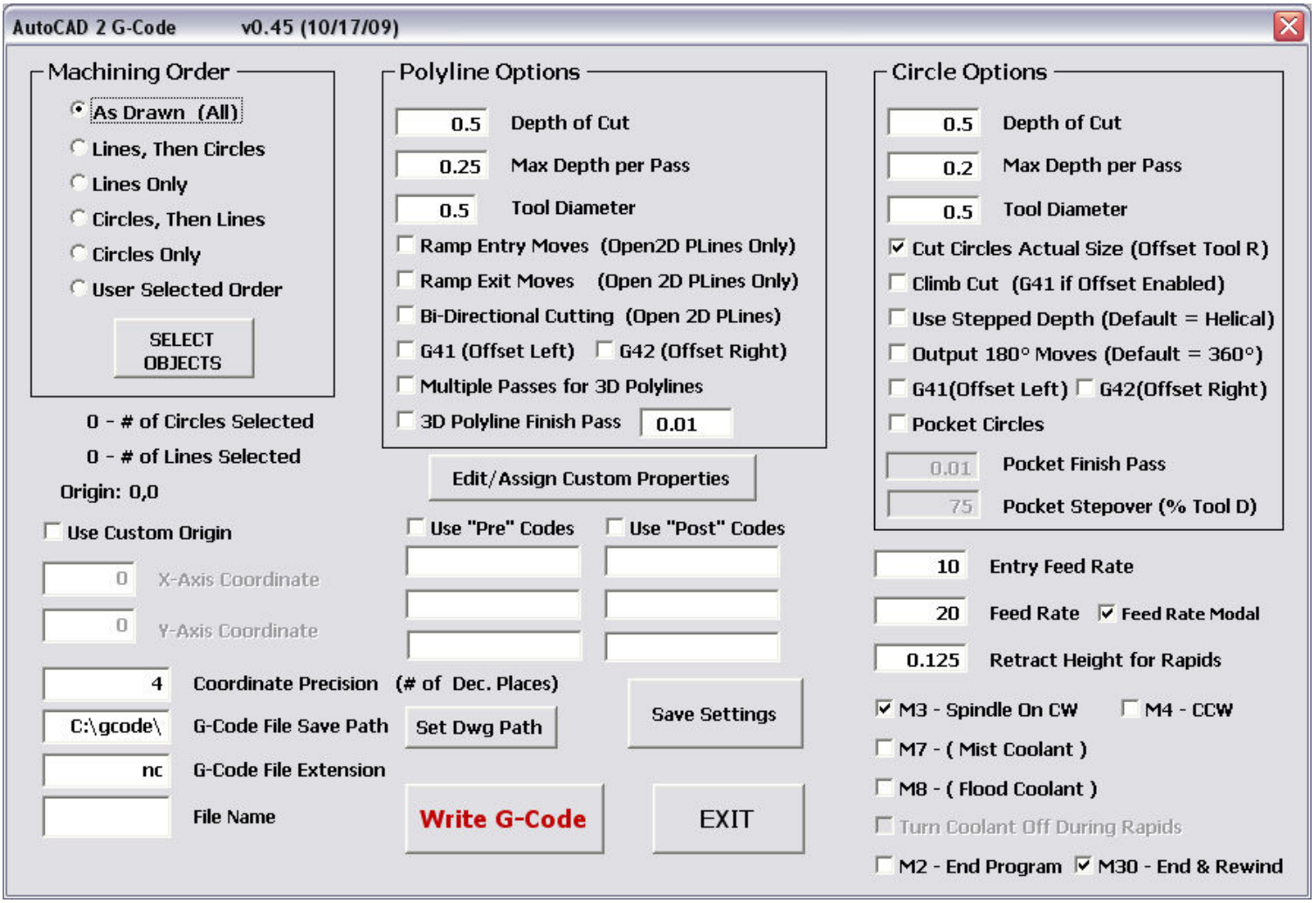
The file AC2GC.dvb can be placed anywhere, but ideally should be located in a folder included in the AutoCAD support files search path. (This is mandatory if you intend to run the macro from a toolbar button) I create a folder in my AutoCAD directory named VBA, and place it there. Then, in AutoCAD, in the options dialog box, go to the File Paths section and add the folder to the search paths.

Once this is done, the program can be run from a toolbar button. Right click on a toolbar, and choose customize. Under commands, click user defined. There should be a "user defined button". Drag this into the toolbar you'd like to add it to. Right click on the blank toolbutton, and choose "properties". In the macro section, enter the following:
^C^C-vbarun AC2GC.dvb!AC2GC.AC2GC

Note: In newer versions of AutoCAD, toolbars and other interface changes are done using the CUI command. The procedure has changed, but the macro is still the same:
^C^C-vbarun AC2GC.dvb!AC2GC.AC2GC

You can also run the program by going to the tools menu, and choosing macro> load project. Load the file, and then to run the program, again from the tools menu, choose macro>macros. (or use alt+F8)

You should see this.



Prior to selecting objects, the user must select the appropriate options. Following is a list of options and information on each.

Use Custom Origin – When not checked, the gcode coordinates will match the coordinates in the drawing. By checking this box, the user can specify an origin, corresponding to the lower left corner of a box containing all selected objects. For example, if you have a circle with radius of 2, and center point at 7,3. If you enter an X coordinate of 2, and a Y coordinate of 2, the gcode will place the center of the circle @ 3,3.

Entry Feed Rate – Feedrate at which all entry moves are performed. An F word will be placed in the line of code of the entry move. The Entry feedrate is also used on any G83 holes.

Feed Rate – Feedrate of all G1,G2, and G3 moves. If the Feed Rate Modal box is checked (default), only G1,G2, or G3 moves following an entry move(above) or rapid (G0) move will contain an F word. If Feed Rate Modal is NOT checked, all G1,G2, and G3 moves will contain an F word.

Retract Height for Rapids – Z value at which all rapid moves (G0) are performed.

M3 (Spindle on Clockwise) – When checked, an M3 will be placed at the beginning of the code, prior to any moves. An M5 (spindle off) will also be added at the end of the code.

M4 (Spindle on Clockwise) – When checked, an M4 will be placed at the beginning of the code, prior to any moves. An M5 (spindle off) will also be added at the end of the code.

M7 (mist coolant) – When checked, an M7 will be placed at the beginning of the code, prior to any moves. An M9 (coolant off) will be added to the end of the code.

M8 (flood coolant) – When checked, an M8 will be placed at the beginning of the code, prior to any moves. An M9 (coolant off) will be added to the end of the code.

Turn Coolant Off During Rapids – When checked, both M7 and M8 will be turned off (M9) during all rapid moves.

M2 – When checked, an M2 will be added as the last line of code.

M30 – When checked, an M30 will be added as the last line of code.

Use Pre Codes – When checked, up to 3 boxes of user input will be placed prior to any moves. Each box will be written as 1 line of code. Example: G16 G20 Note: added one box for three each pre and post in v0.45.

Use Post Codes – Same as above, but code will be placed after all movement.

Coordinate Precision - # of decimal places for gcode coordinates.

G-Code File Save Path – The full path where g-code files will be written to. By clicking the “Set DWG Path” button, the path of the drawing will be used. If the drawing has not been saved, the AutoCAD directory will be used. Warning: Invalid paths will result in an error.

G-Code File Extension – the file extension to be used. Do NOT include the “.”

File Name – Name of the g-code file to be saved. Extension above will be added to the file name, and the file will be saved to the above path. If left blank, file name will be saved as “noname”.

Polyline Options – The following options will apply to all selected polylines.

Depth of Cut – Final desired depth of cut. If the polyline's elevation is less than 0, then the elevation will override the Depth of Cut box and be used as the Depth. Note: This has no effect on 3D Polylines. A 3D Polyline's depth is the z value @ each vertex.

Max Depth per Pass – Maximum desired depth for each pass. Passes will not be greater than this depth, but may be less. Does not affect 3D Polylines.

Tool Diameter – this value is only used for the G41/G42 options above. No actual offsetting takes place in the code

Ramp Entry Moves – When checked, tool will plunge to start of line @ Z=0, and the move to the second vertex of the line will ramp down to the depth/pass. This option has no effect on 3D Polylines.

Ramp Exit Moves – When checked, at vertex *n*-1, the tool will go from current depth/pass to Z=0 at the last vertex. This option has no effect on 3D Polylines.

Bi-Directional Cutting – When checked, the tool will plunge to the depth/pass at the first point of the polyline. When the tool reaches the end of the line, it will plunge down to the next Z level and return in the opposite direction, continuing back and forth until the desired depth is reached. This should work really well for cutting slots. Both entry ramping and exit ramping will be unchecked, when BiDirectional Cutting is checked. They may be able to be used in future versions. If G41/G42 are checked, they will be ignored.
Warning If this option is chosen, and the polyline is closed, no code will be written for that polyline. There is no error message to warn the user, it just ignores the line

G41/G42, Cutter Compensation – Use these to have Mach2/3 do the tool offsets, instead of offsetting all your lines in AutoCAD. To use this, the polyline should be open and include a lead-in and lead-out segment where the comp will take place. See the Mach2 manual for more info. Mach3 uses an advanced comp method that must be enabled. This method is highly recommended, but undocumented. When used, the center of the tool will start at the start of the polyline. The tool will then move the tool radius to the right (G42) or left (G41) while moving to the second vertex of the polyline. This first segment is known as a lead-in move, and must be at least as long as the tool radius, preferably a little longer. The lead-in can be ramped using the ramped entry option above. A lead-out move is not required, but is good practice.

G41 – Checking this option will place a G41 in the code, between the 1st and 2nd vertices of the polyline. The format will be G41Px.xxx where x.xxx is the specified tool radius. This will make the first move of the polyline a lead-in move. A G40 will also be added to the code prior to the last vertex, creating a lead-out move. Be sure to add the lead-in and lead-out moves to your polylines when using this option.

G42 – see above.

Multiple Passes for 3D Polyline's – This option will create multiple toolpaths for 3D Polylines, using the "Max Depth per Pass" depth. Currently, each pass will be independent, and the same direction, with a rapid move back to the start.

3D Polyline Finish Pass – When used in conjunction with 3D Polyline Multiple passes, will create a finish toolpath removing the specified amount of material. Currently, the finish pass only works with Multiple Passes. I haven't tested it, but by setting the Max Depth per Pass to the full depth, you should be able to get a single rough pass + finish pass, if so desired. The next version should remove this restriction.

Circle Options – The following options will apply to all selected circles

Depth of Cut – Final desired depth of cut. If the circle's center has a z-value less than 0, the z-value will override the Depth of Cut box value and be used as the Depth. This makes it relatively easy to have circles with different depths. Just move them to the required depth before writing the g-code.

Max Depth per Pass – Maximum desired depth for each pass. Passes will not be greater than this depth, but may be less.

Tool Diameter – This is used for G41/G42 (G41Px.xxx), as well as to offset the toolpath if "Actual Size" is selected.

Cut Circles Actual Size – When checked, if G41/G42 are not selected, then a offset will be applied based on the specified tool diameter, to cut the circles the size they were drawn. If not checked, the toolpath will follow the drawn circle, resulting in a circle biiger by the tool diameter. G41/G42 will always cut actual size, whether checked or not.

Climb Cut – When checked, circle will be cut CCW.

Use Stepped Depth – Default G2/G3 moves are helical. By checking this box, the tool will plunge straight down to the current depth/pass, and then make the circular move.
Note: Helical moves will require 1 more pass than the stepped depth moves, in order to create a flat bottom.

Output 180° Moves – When checked, gcode will be written as 180° cuts, versus the default moves of 360°

G41 – When this box is checked, the circle will be cut the actual size of the drawing. The format will be G41Px.xxx where x.xxx is the specifies tool radius A lead-in move will be performed to allow the tool offset. Because of this, the circle must be AT LEAST 4x the tool diameter to use G41. At the completion of the cut, the tool will retract along the circle for 90°, after which G40 will be applied and then a small lead out move performed at retract height.

G42 – see above. (4x rule listed above applies here as well)

Pocket Circles – When this box is checked, a spiral pocketing routine will be added to mill out the entire circle. New in V.43 – The polygonal spiral has been replaced by an actual spiral. Utilizing a Mach3 *Trick,* a spiral consisting of G2 or G3 moves is created by offsetting the endpoint of an arc. NOTE: The stepover can not be more than 0.5inches or Mach3 may give an error and not run the code. Anything larger than 0.5inches will be set to 0.5inches internally. A ramped lead in move, up to 3x the tool diameter long, will start just below the circle center. The depth of the pocketing pass will be the same depth as the circle pass, determined by the Max Depth Per Pass. At the end of each pass, the tool will ramp down, to the center of the circle for the next pass. The pocket will leave the amount entered in the Finish Pass box below, to be removed during the actual circle cutting pass(es). If Climb Cutting is selected, the spiral will be CCW, otherwise it will be CW.

Note: If the tool diameter is greater than the circle size minus 2x the finish pass size, then no pocketing will be done. There is no error message to notify the user.

Pocket Finish Pass – Amount of material left after pocketing, to be removed during the actual circle cutting passes. See Pocket Circles above.

Pocket Stepover – Amount of material to remove during the spiral pocketing passes. This is a percentage of the tool diameter. The minimum value is 5%, max is 99%. As noted above, a stepover greater than 0.5 inches will be reduced to 0.5inches.

In addition to the above options, there are 2 additional buttons.

Save Settings – As the name implies. This will save all current settings. These settings will be reloaded the next time you start the program. The settings are ALWAYS saved to C:\gcode\config\ac2gc.ini

Edit / Assign Custom Properties – By Clicking this button, the user can assign custom options to individual Circles and/or Polylines. Only ONE object can be selected at a time. You will be warned only once. The second time an error will result. When custom properties are assigned to an object, the object will be moved to a new layer named "Custom_Prop", and the color will be changed to "by layer" (if not already). This lets you know which objects have been modified.

Custom properties will be saved with the object, if the drawing is saved. The objects can be moved to another layer, but the properties will still remain.

A cool thing about the custom properties, is that once assigned to an object, you can copy the object, or use the ARRAY or MIRROR command, and all copied objects will have the same properties as the original. Say for instance, you have a bolt circle, but you want to counterbore all the holes. Draw 2 concentric circles, 1 the size of the hole, 1 the size of the counterbore. Using the custom properties, assign the depth of the counterbore to the counterbore circle. Use the main settings for the hole depth. Exit the macro, and create a polar array of both circles in AutoCAD. When you create the gcode, all the counterbores will have the custom depth, and all the holes will have the depth from the main circle settings.

Once all options have been chosen, you need to select objects. Currently, there are 6 options. These can be found under Machining Order.

As Drawn – gcode will be created for all polylines and Circles in model space, in the order they were drawn.

Lines, Then Circles – gcode will be created for all polylines in the drawing first, followed by all the Circles in the drawing. These again will be in the order they were drawn.

Lines Only – gcode will be created for all polylines in model space, in the order they were drawn. Circles will be ignored.

Circles, Then Lines. – gcode will be created for all Circles in model space first, followed by all the polylines in model space. These again will be in the order they were drawn.

Circles Only – gcode will be created for all Circles in model space, in the order they were drawn. Polylines will be ignored.

User Selected Order – gcode will be created for all Circles and Polylines in model space selected by the user, in the order they were selected.

Choose the appropriate option, and click the "Select Objects" button. The number of objects selected will be displayed below the button, as well as the coordinates of the lower left corner of a bounding box of the selected objects. (Origin)

Once the objects are selected, and all options set, click the "Write G-Code" button. That's it.
You can make changes to the settings and write code without having to re-select objects, but if custom properties are applied to objects, re-selecting is recommended.

Bug Fixes / Changes

1-12-05
Circular lead-in move for G41/G42 was calculated incorrectly. Also G41/G42 P word was using tool diameter instead of radius. Both fixed.

1-13-05
Removed comma separator for values over 1000. New version uses 1001.25, versus 1,001.25

5-21-05
If tool was bigger than circle, and "Cut Actual Size" was NOT checked, a warning would be displayed and no code would be written. Related issue, if tool diameter was equal to circle diameter, and "Cut Actual Size" wan NOT checked, no code would be written. Both Fixed.

8-30-05
NOTE : The following fixes have not been well tested, but appear to work. Let me know if there are any problems

1) If Bidirectional cutting was assigned using Custom Properties, it wouldn't work unless it was checked in the main screen. Fixed, but the fix was a bit of a hack. A major rewrite would have been required to fix it. It seems to be OK, though.

2) If a circle was smaller than the tool, the macro would stop reading entities after the offending circle. Fixed.

Also, I noticed a slight problem. I tend to scale my drawings a lot from metric to imperial and back. Scaling by 25.4 and then by 1/25.4 will change the size of circles by something like .00000000001. Less than you can see in AutoCAD, but my macro noticed. I couldn't drill a .49999999999" hole with a .5" bit. I added a small "fudge factor" of -.0001 in case you encounter this error. I don't think it will cause any problems.

2-14-09
Mach3's comp code changed sometime during version 3, which caused circle code output using G41/G42 to not work correctly, due to the lead-in arc move that I was using. This version fixes this issue by doing a small leadin move from retract height down to part Z zero, then arcing into the circle cut. At the end of the cut, the tool travels around the circle 90° while lifting to retract height, then doing a small leadout move to turn comp off. It appears to be working correctly.

6-9-09
1) Fixed bug in Cutom Properties dialog. When assigning custom properties to multiple circles, if a circle had any "circle options" checked, then any circle selected after would also display the boxes as checked, even though they may not have been assigned. It didn't create wrong code, but the user could be led to believe that they may. This should be fixed.
2) Circle Pockets. Polygonal segmented pocket paths have been replaced by G2/G3 spirals.

9-19-09 (v0.44)
1) Cleaned up circle pocket code for small circles. For circles only slightly larger than the tool, pocketing would result in a duplicate circular move. The duplicate moves have been removed. Also, added the F word for the pocketing to the first line of the spiral move. These changes result in at least two fewer lines of g-code per depth pass for small pockets.
2) Added options for multiple passes on 3D polyline toolpaths. Currently inactive in this version.

10/17/09 (v0.45)

1) Added an additional Pre Code and Post Code Input box.
2) Implemented multiple passes for 3D polylines.
3) Modified Custom Properties window to only show available options for the selected entities. Window size and contents now change as needed.