DeltaCAD2CAM.bas version 6/24/2016 by AlwMVMO email: alwilliamsmo@yahoo.com

```
*****************************************************************************
*  Warranty Disclaimer:                                                     *
*  This program is for educational purposes.                                *
*  Not liable for damages, loss of profits, lost savings, or other incidental *
*  or consequential damages out of the use or inability to use the macro.   *
*****************************************************************************
```

This Macro creates a 2.5 axis milling machine G-Code file from DeltaCad geometry and Tool Cutting Information.

The G-Code file can be customized by making changes in the Post Processor Parameters section of the Macro.
The following Post Processor Information is based upon the MACH3 Milling Machine Controller.

```
*****************************************************************************************
*                                    Post Processor Parameters                          *
*****************************************************************************************
  NCFolder$ = "C:\Program Files\DeltaCAD\nc\"    '<- NC Files Folder, make sure to create this folder
  UseCADFilename = 1         '<- Use Cad file name, Tool No., and Layer Name in NC file name 0=No 1=Yes
  FileExt$  = ".nc"          '<- NC file extension
  ToolInfoText = .2          '<- Tool Info text height in inches
  NCodeDigits = 4            '<- Minimum Number of digits in N Code or 0=No N Code
  NCodeStep = 10             '<- Increment N code count and Starting N code default
  CompDigits = 2             '<- Minimum Number of digits in Diameter and Tool Length Compensation codes
  XYZDecPlaces = 4           '<- Maximum Number of digits after decimal point in X Y Z I J codes
  FeedDecPlaces = 2          '<- Maximum Number of digits after decimal point in Feed Rates
  ToolNumberDigits = 2       '<- Number of digits in Tool Number
  ArcMode = "INC"            '<- IJ Arc Center Mode  INC or ABS
'- - - - - - - - - - - - - - - - - - - - - - G & M Codes - - - - - - - - - - - - - - - - - - - - - -
  GCode$(0) = "G00"          '<- Rapid Move
  GCode$(1) = "G01"          '<- Cutting Move
  GCode$(2) = "G02"          '<- Arc CW Move
  GCode$(3) = "G03"          '<- Arc CCW Move
  GCode$(4) = "G80"          '<- Cancel Canned Cycles
'- - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
  CutterCompCode$(0) = "G40"   '<- None  Cutter Compensation Code
  CutterCompCode$(1) = "G41"   '<- Left  Cutter Compensation Code
  CutterCompCode$(2) = "G42"   '<- Right Cutter Compensation Code
'- - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
  SpindleDirType$(0) = "CW":    SpindleDirCode$(0) = "M03"     '<- Spindle Direction Clockwise
  SpindleDirType$(1) = "CCW":   SpindleDirCode$(1) = "M04"   '<- Spindle Direction Counter-Clockwise
'- - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
  CoolantType$(0) = "Mist":     CoolantCode$(0)    = "M07"         '<- Mist Coolant
  CoolantType$(1) = "Flood":    CoolantCode$(1)    = "M08"         '<- Flood Coolant
  CoolantType$(2) = "None":     CoolantCode$(2)    = "M09"         '<- No Coolant
' More Coolant Types can be added here, they will be automatically added in the Tool Cut Info Dialog
'- - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
  CycleList$(0)="Mill-Path"
  CycleList$(1)="G73 Drill w/HS peck"
  CycleList$(2)="G81 Drill"
  CycleList$(3)="G82 Drill w/dwell"
  CycleList$(4)="G83 Drill w/peck"
  CycleList$(5)="G84 Tap"
  CycleList$(6)="G85 Reaming w/Zret@F"
```

CycleList$(7)="G86 Boring w/Spindle stop"
CycleList$(8)="Mill-Hole CW"
CycleList$(9)="Mill-Hole CCW"
' More Cycles can be added here, they will be automatically added in the Tool Cut Info Dialog

- -

SpaceBetweenCodes = 1   '<- 1=space 0=no space
example: N0250 G00 X1.234 Y1.234 or N0250G00X1.234Y1.234

==

Machine Startup Codes

- -

Machine Startup Codes can be customized by changing the Machine Startup Codes section of the Macro.

* Each code starting with the character @ corresponds to information from the Tool Cutting Information list.
  The following @ codes are used in Machine Startup
    @ToolDescription
    @ToolNumber
    @SpindleDir
    @SpindleSpeed
    @Coolant
    @ToolLenComp

* Each line between quotes " " will create a line of code exactly as typed except for @ codes.
* Spaces in Machine Startup Codes will have to be editted to change. example: "G00G49G40G17G80G50G90"
* N codes will be inserted automatically if NCode parameter is set greater than 0 (zero).
* Keep the numbers between the parentheses in MachStartCodes$( ) in sequence and no more than (30) lines

MachStartCodes$(1) = "G00 G49 G40 G17 G80 G50 G90"
MachStartCodes$(2) = "@ToolDescription"
MachStartCodes$(3) = "M06 T@ToolNumber"
MachStartCodes$(4) = "G64"
MachStartCodes$(5) = "G20 (Inch)"
MachStartCodes$(6) = "M@SpindleDir S@SpindleSpeed"
MachStartCodes$(7) = "M@Coolant"
MachStartCodes$(8) = "G00 G43 H@ToolLenComp"
MachStartCodes$(9) = ""   ← The last line is double quotes

- -

Machine End Codes

- -

Machine End Codes can be customized by changing the Machine End Codes section of the Macro.

* There are no @ codes for Machine End Codes
* Each line between quotes " " will create a line of code exactly as typed.
* Spaces in Machine End Codes will have to be editted to change. example: "M5M9"
* N codes will be inserted automatically if NCode parameter is set greater than 0 (zero).
* Keep the numbers between the parentheses in MachEndCodes$( ) in sequence and no more than (30) lines

MachEndCodes$(1) = "M5 M9"
MachEndCodes$(2) = "M30"
MachEndCodes$(3) = ""   ← The last line is double quotes

When a Post Processor is customize to a machine, save the macro with a new file name.

****
*   Creating Tool Paths for Mill-Path and G## canned cycles   *
****

Start by creating a layer for your tool path and make it the current layer.
examples of layer names:--> Path01 MillLength DrillHoles
Do not use the following characters in the layer name < : > " \ / | ? *

- -

Tool Path Geometry are created by using Points, Lines and Arcs.
The following types are supported:

- Solid Lines
  Used for feed milling or in G canned cycles to position the tool to the next hole position at the APPROACH Z.

- Hidden Lines
  Used to position the tool to the RETRACT Z then rapid to the next position during milling and G canned cycles.

- Center Lines
  Used for cutter compensation Left move (G41) or cutter compensation off (G40).

- Phantom Lines
  Used for cutter compensation Right move (G42) or cutter compensation off (G40).

- Solid Arcs
  Used for milling radius.

- Point ■ Used for the starting position.

- Text ——— Used for Tool Cutting Information.

===

Rules for drawing Tool Paths for Mill-Path and G## canned cycles:

* Rules for Mill-Hole cycles are different and will be explained later.

* Put all objects for ONE tool path on ONE layer. Make sure there are no objects on that layer that isn't being used.

* Place a Point object at the beginning of a tool path and set it as Permanent.
Setting the width of a Point wider than the other objects will make it show up better.

* Draw Points, Lines and Arcs at the coordinates that the G-code file will run on the NC milling machine.

* Multiple Tool Cutting Information texts to a single tool path layer allows for milling or drilling a tool path with multiple tools. i.e. Center drill then drill then tap.

* A G code canned cycle (G73, G81-G86) tool path is drawn by connecting the hole centers with lines.
Solid lines will produce only X and Y coordinates for each hole with no Z retracts commands.
Hidden lines to retract the tool to the RETRACT Z position before positioning to the next hole.

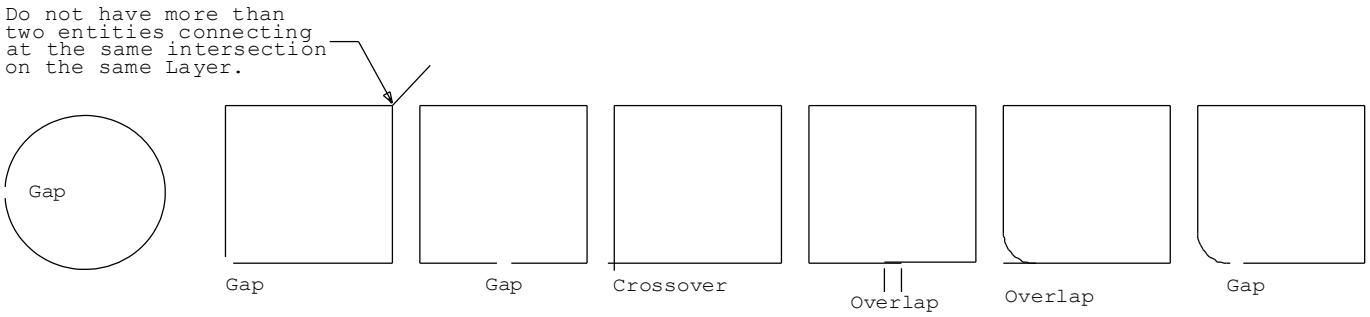
* A single tool path can machine multiple parts by connecting them with a Hidden line (Rapid move).
  Or you can create separate tool paths on separate layers and 'splice' the NC files together later.

* Do not have more than two objects with the same layer connecting at the same intersection.

* Each object in a tool path must precisely meet the next object by at least .0001

* Objects can be drawn in any order, width and color.

* The start and the end of a tool path can not be at the same X and Y.
  There needs to be a gap or overlap or crossover of at least .001
  Full Circles needs to have a gap or overlap at least .001
  See sample drawing below.



Examples of Gaps, Crossovers and Overlaps

* A Center line or Phantom line can be used to start and end a cutter compensated tool path.
  Use a Center line (3rd line type) for cutter compensation on the Left (G41)
  and use a Phantom line (4th line type) for cutter compensation on the Right (G42).
  Always use an ending compensation move before doing another starting compensation move.

* To drill a single hole just place a point where the hole is to be drilled.

* If you are going to use your part drawing to create your tool paths I would suggest making
  a copy of it first.

****
*   Creating Tool Paths for Mill-Holes cycle   *
****

Start by creating a layer for your tool path and make it the current layer.
examples of layer names:--> Path01 MillHoles 88mmHoles
Do not use the following characters in the layer name < : > " \ / | ? *

- -

Tool paths for Mill-Holes cycles are created by using a Point, Hidden Lines, and Circles.

* Points ——— Used to indicate the start of the tool path.
* Hidden lines — Used to position the hole centers.
* Circles ——— Used for the Hole size and Mill size and Mill Starting Position.
* Texts ——— Used for Tool Cutting Information.

==

Rules for drawing Tool Paths for Mill-Holes cycles:

* Put all objects for ONE tool path on ONE layer. Make sure there are no objects on that layer that isn't being used.

* Draw a Point object at the beginning of the Tool Path at the center of the first hole to be milled.
  Set it as Permanent, setting the width of a Point wider than the other objects will make it show up better.

* Draw Hidden Lines <u>ONLY</u> from the center of one hole to the center of the next hole and
  at the coordinates that will be in the G-code file. Solid lines will keep the mill down between holes.

* Draw a circle with the diameter of the hole to be milled <u>ONLY</u> at the beginning of the tool path
  on the same layer as the tool path.

* Draw another circle of the tool diameter at the center or anywhere within the hole to be milled.
  This is the starting position that the mill will plunge to the Bottom Z.

* All Holes on the tool path will be milled using these Hole Diameters and Mill Diameters and Positions.

* A tool path is drawn by connecting the hole centers with lines.
  Hidden lines will retract the tool to the RETRACT Z position before positioning to the next hole.

* Do not have more than two lines connecting at the same intersection. Do not use Arcs in the tool path.

* Each object in a tool path must precisely meet the next object by at least .0001

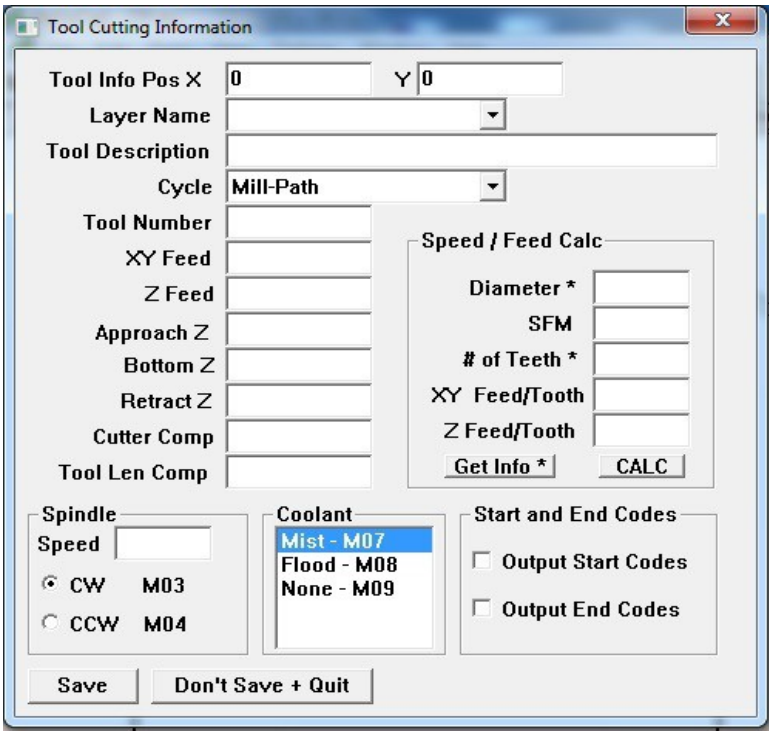
* Objects can be drawn in any order, width and color.

* The start and the end of a tool path can not be at the same X and Y.

* The direction of milling will be selected in the Tool Cutting Information dialog by selecting
  Mill-Hole CW or Mill-Hole CCW.

* To mill a single hole just place a point where the hole is to be milled, a circle for the hole to be milled and a
  circle for the mill diameter and starting position.

****
* Tool Cutting Information *
****



* After drawing the Tool Path Geometry, save the CAD file.
* If a Tool Cutting Information text is selected, the macro will edit the text information.
* If a Point, Line or Arc/Circle is selected, the macro will start a new Tool Cutting Information.
* If nothing is selected the macro will start a new Tool Cutting Information and a Layer will have to be selected in Tool Cutting Information dialog.

* Run the Macro

Enter the following information:
1) The X and Y position on the drawing that the Tool Cutting Information text will be placed.
2) Select the Layer Name of the tool path geometry if an object was not selected.
3) Tool Description of the tool or other information.
4) Select the type of cutting: Mill-Path, Mill-Hole CW,Mill-Hole CCW, G73, G81, G82, G83, G84, G85, G86, ect.
5) Enter the Tool Number.
6) Enter the X&Y Feed if it is a Mill cycle.
7) Enter the Z Feed.
8) Enter the Approach Z position.
9) Enter the Retract Z position.
10) Enter the Bottom Z position.
11) Enter the Cutter or Diameter Compensation number for this tool if it is a Mill cycle.
12) Enter the Tool Length Compensation number for this tool.
13) Enter the Spindle Speed.
   a) The Spindle Speed and Feedrates can be calculated and automatically entered with the Speed and Feed Calculator.
   b) If you already have the Spindle Speed and Feedrates then enter the Diameter* and # of Teeth* and click the [Get Info*] button to calculate the SFM (surface feet per minute) and Feed per tooth
14) Select the Spindle Direction CW or CCW.

15) Select the type of Coolant.
16) Check (on or off) if Start or End Codes are output.
This is to allow 'splicing' multiple tool paths to one tool.
Example: On the 1st tool path leave off the End Codes.
On the next tool path leave off both the Start and End Codes.
On the last tool path leave off the Start Codes.
You will have to 'splice' the NC files together in your favorite text editor.

Note: G73 and G83 Q step value is calculated using the following formula:
Qstep = (ApproachZ - BottomZ) / (Int((ApproachZ - BottomZ)/ .1))

- -
17) Click [Save] to save the Tool Cutting Information to the CAD file.
18) Then select a NC File Name or type in a new NC File Name.

note:
If the parameter UseCADFilename = 1 then a NC File Name with the
DeltaCAD file name-Tool Number-Layer Name will be put in NC file name entry box.
example: MyPart-T03-Path03
If it is 0 then the box will be empty.
The NC File will be placed in the folder specified by the parameter: NCFolder$
or SubFolderName\DeltaCAD file name-Tool Number-Layer to save it to a subfolder.

19) Enter the Starting N number

****
To Edit the Tool Cutting Information then do the following:
1) Click the [Select] tab.
2) Select the Tool Cutting Information text that you want to Edit.
3) Then run the MACRO
4) Edit the Tool Cutting Information.

****
The Tool Cutting Information text has the following information in it:

TOOL=
TOOLDESCRIPTION=
CYCLE=Mill-Path /G73/G81/G82/G83/G84/G85/G86 / Mill-Hole CW / Mill-Hole CCW <- Use only one cycle
XYFEED=
ZFEED=
SPINDLESPEED=
SPINDLEDIR=CW/CCW <- Use only one (CW or CCW)
COOLANT=FLOOD/MIST/NONE <- Use only one (FLOOD or MIST or NONE)
CUTTERCOMP=
TOOLLENCOMP=
APPROACHZ=
BOTTOMZ=
RETRACTZ=
OutputStart=Yes/No <- Use only one (Yes or No)
OutputEnd= Yes/No <- Use only one (Yes or No)

****