# 04 - Running a Job

	Edit G-Code	Rewind Ctrl-W
Cycle Start <alt-r></alt-r>	Recent File	Single BLK Alt-N
SAILTRA		Reverse Run
Feed Hold <spc></spc>	Load G-Code	Block Delete
Stop <alt-s></alt-s>	Set Next Line Line: 0 Run From Here	M1 Optional Stop Flood Ctrl-F
Res	et G-Codes	

## Step 1 — Load GCode File

	Open	×
File: No File Loaded.	Look in: SOS (C:)	▼ 🖶 🖆 🎟 ▼
Cycle Start   Edit G-Code   Rewind Ctrl-W <alt-r>   Recent File   Single BLK Alt-N     Feed Hold   Load G-Code   Block Delete     Stop   Set Next Line   M1 Optional Stop     Line:   0   Flood Ctrl-F     Run From Here   U   CVMode</alt-r>	Quick access   Name     Desktop   Apps     Libraries   Autodesk     This PC   Dell     Network   Drivers     EAGLE 8.0   EAGLE 9.1.0     HP CLJ3600 Driver   Intel	Date modified        4/5/2016 3:27 PM     3/24/2017 3:02 PM       3/24/2017 3:02 PM     4/18/2016 9:10 PM       4/5/2016 9:10 PM     4/5/2016 3:25 PM       2/5/2017 7:33 PM     8/2/2018 6:58 PM       9/20/2018 7:14 PM     6/15/2017 1:56 PM
Reset G.Codes Color History Clear Status:	Mach3 MSOCache File name: Files of type: All Files (".") Open as read-only	9/22/2018 2:38 PM 4x12/2016 8:00 AM

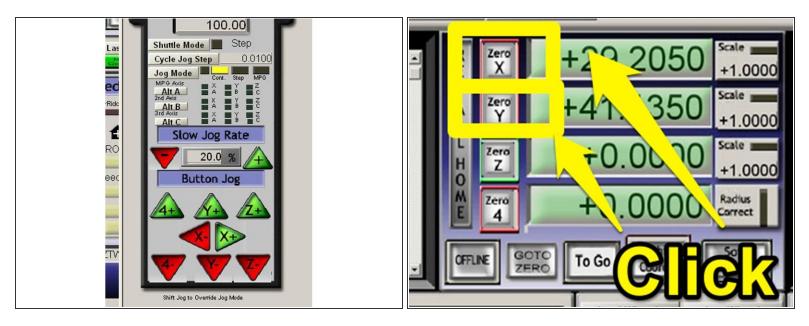
- Click "Load GCode"
- Locate your Gcode / TAP file and click Open

## Step 2 — Preview GCode

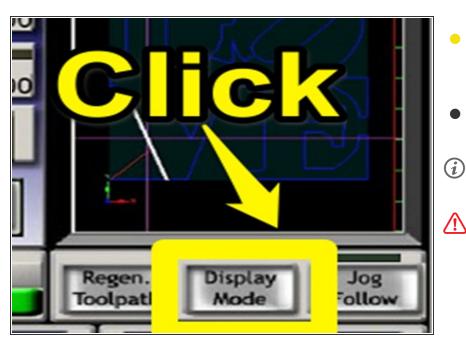
Program Run Alt.1     MDI Alt2     TeelPath Alt     Offsets Alt6     Diagnostics Alt7     M(1>G15     G1     G17     G40     G21     G90     G64     G97
N180(Toolpath - Profile 1) N170() N1800504 N10004 00000 5300 0 N2006000 00010 01220 000 N20061044 17210 024 N20061142 55165 823 N24061141 251768 787 N28061141 251788 787 N28061141 150788 9812 N28061141 551788 863 UTUE PERC To Go Machine Soft
File: DICNCVove_fourn_cut_200x2u3 bd
Feed Rate Spindle Speed
Feed Hold Set Noxt Line Line: 15 Bick Delete M1 Optional Stop M1 Optional Stop
CALLS Run From Here Dwoll CV Mode Jog Rate 85.0 00 007 Zinnet
G.codes M.Codes +0.000 UnitsRev 0.00 Initialization Macro Called on reset. Profile: Mach3Mill
P10     P12     P15     Jeg Step     O.0010     Jeg Mode     MPG Axis       P11     P13     Velocity Oniy     Multi Step     Cont. Step for 0     2     2     2     2     2     2     2     3     3

- Verify that your GCode has loaded and the preview looks correct.
- You may need to double-click in the tool path preview to regenerate the display after loading your gcode.

## Step 3 — Zero X and Y



- Using the JOG controls, move the machine over the desired origin position you choose during the CAM process.
- Once the machine is positioned correctly over your X-Y origin, click the X and Y axis buttons to zero the axis.



# Step 4 — Inspect Toolpath

- Click "Display Mode" under the toolpath preview window until the display shows the outline of the machine along with the toolpath.
- The crosshair indicates the current position of the tool bit in relation to the toolpath.
- You may have to click "Regen Toolpath" after zeroing to update the display.
  - Make sure the toolpath and its orientation to the machine and material affixed to the bed all seem reasonable before continuing.

## Step 5 — Preparing to Run



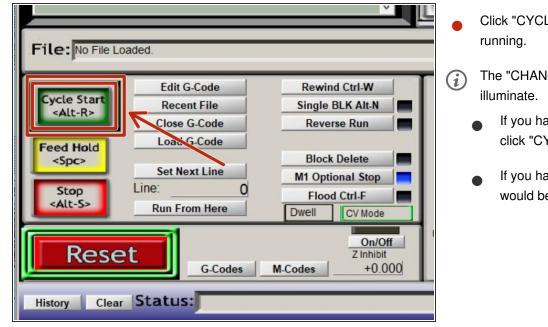
- Follow guide 03 Changing Tools to install the first bit used in your program.
- As your program runs, when your program indicates a tool change is required the "CHANGE TOOL" LED on the Tool Information panel will blink and the program execution will pause.
  - Follow <u>03 Changing Tools</u> to change to the next tool in your program. The tool number will be indicated in the "Tool Information" panel.
- One you complete the changing tools SOP, the spindle should be armed and the Yellow light on the light stalk should be illuminated.

#### Step 6 — Turn on dust collector



- When you are ready to run your job, turn on the dust collection system.
- Ensure that the blast gate for the spindle in use is pulled OPEN and all other un-used gates are pushed CLOSED.

## Step 7 — Run your program



- Click "CYCLE START" to start your program running.
- The "CHANGE TOOL" LED will likely illuminate.

If you have already loaded your first tool, click "CYCLE START" to continue.

 If you have not loaded your first tool, now would be a good time.