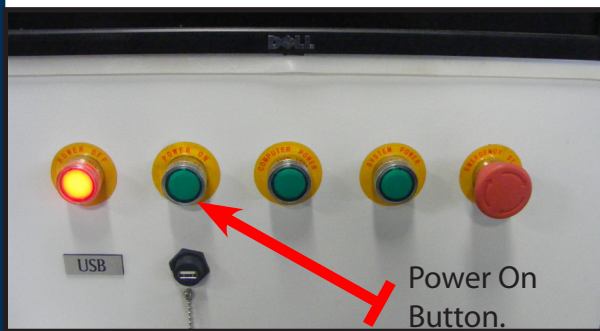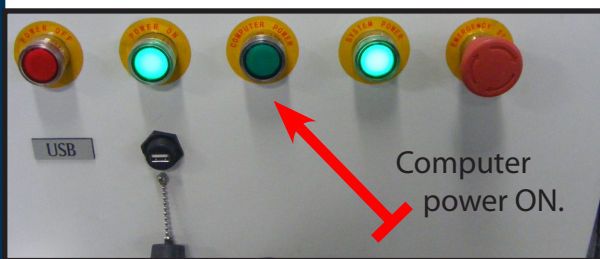## Techno-Osai Start Up Sequence

Turn the Main power switch to the ON Position.

220 volts should have been attached to this switch by an electrician.

The red light on the front of the machine will light up, this indicates that 220 volts is coming into the machine.
Press the Power On button to start the system.

Power On Button.

The Power on and System power light will light up indicating that power has been applied to the controller system and the motors. Press the Computer Power button to start the PC. This will only light temporarily.

Computer power ON.

The PC will start to boot.

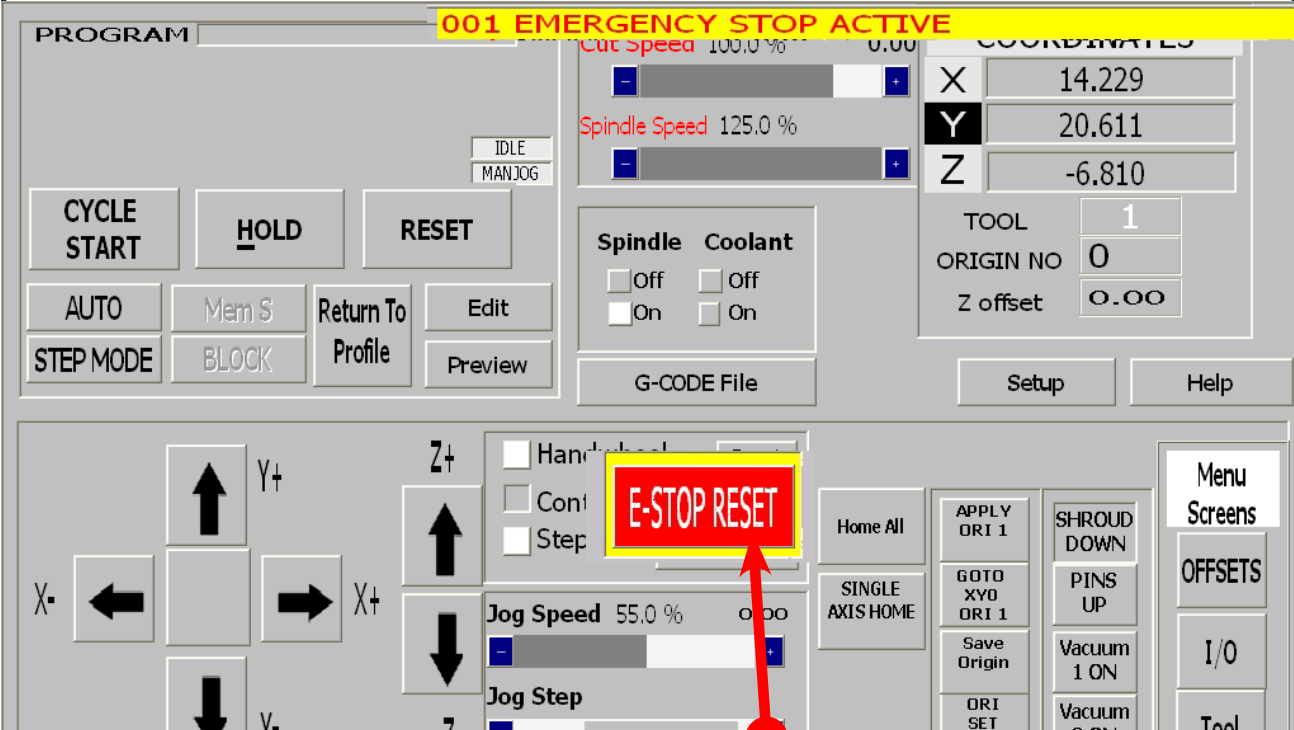Currently the Techno-Osai controller will only operate in Window XP mode.
If the PC has a later version of windows installed the system will have to boot up and then start running virtual Windows XP. This will happen automatically, please be patient as this occurs.

**Techno** *Inc.* *SINCE 1986*
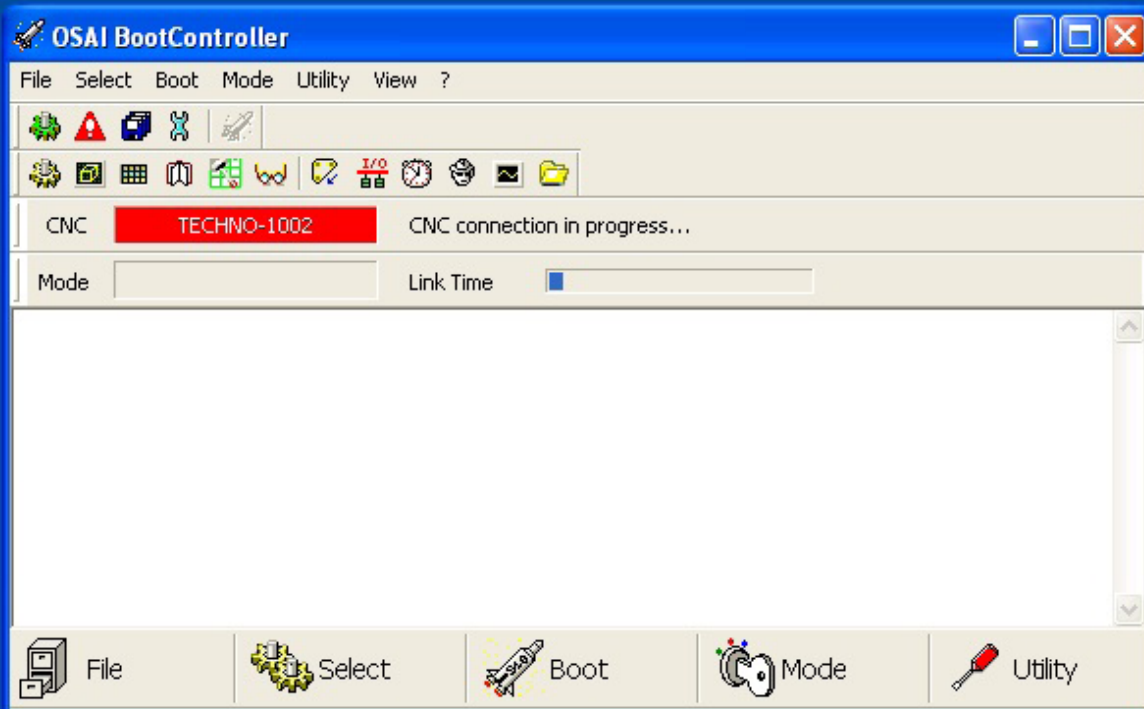


The Boot Controller software will start automatically. If it does not double click on the Boot Controller Icon on the desktop. If no text appears in the box, after 30 seconds, check that the light for the controller is on and that the network cable from PC to Osai controller is connected.
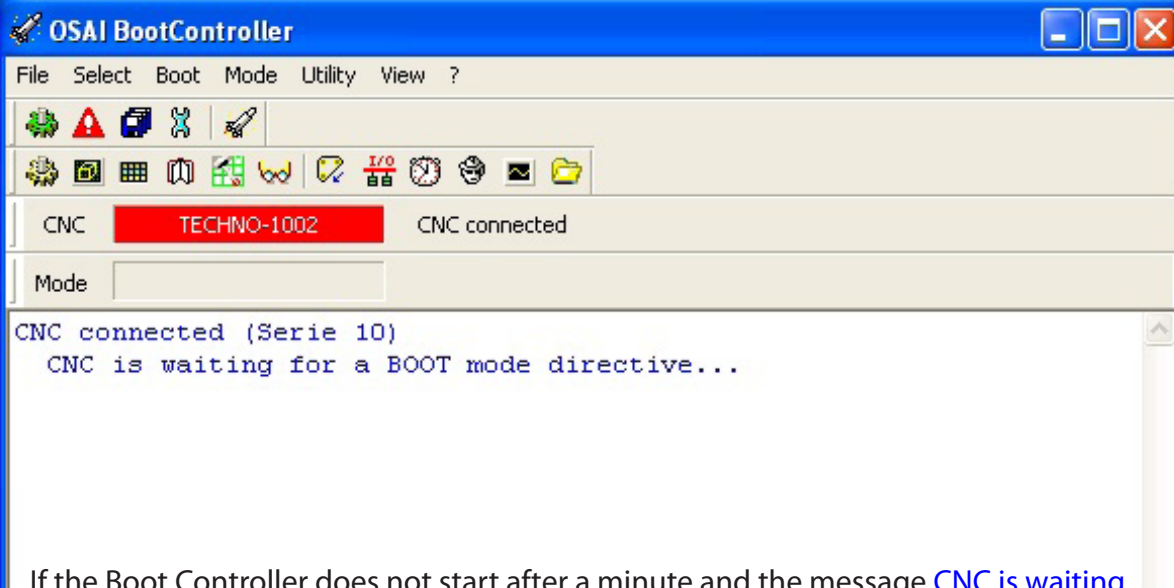


If the Boot Controller does not start after a minute and the message CNC is waiting for a BOOT mode directive... appears on the screen then normal mode needs to be selected. Click on the Mode Icon and select Normal.

**Techno** *Inc.* SINCE 1986

---

**001 EMERGENCY STOP ACTIVE**

PROGRAM

Cut Speed 100.0 % 0.00    COORDINATES

| − | | + |

Spindle Speed 125.0 %

| − | | + |

IDLE / MANJOG

CYCLE START    HOLD    RESET

AUTO    Mem S    Return To Profile    Edit

STEP MODE    BLOCK    Preview

Spindle    Coolant
☐ Off    ☐ Off
☐ On    ☐ On

G-CODE File

X    14.229
Y    20.611
Z    -6.810

TOOL    1
ORIGIN NO    0
Z offset    0.00

Setup    Help

Y+    Z+    ☐ Handwheel    **E-STOP RESET**    Home All    APPLY ORI 1    SHROUD DOWN    Menu Screens

☐ Cont

X-    X+    ☐ Step    SINGLE AXIS HOME    GOTO XYO ORI 1    PINS UP    OFFSETS

Jog Speed 55.0 %    0.00    Save Origin    Vacuum 1 ON    I/O

| − | | + |

Jog Step    ORI SET    Vacuum    Tool

The Techno interface screen will now open. There will be a warning message saying
Emergency Stop Active. Click on the E-stop Reset Button to remove this warning.

---

**011 AXES NOT REFERENCED**

PROGRAM

Cut Speed 100.0 % 0.00    COORDINATES

| − | | + |

Spindle Speed 125.0 %

| − | | + |

IDLE / MANJOG

CYCLE START    HOLD    RESET

AUTO    Mem S    Return T Reset Errors

STEP MODE    BLOCK    Profile    Preview

Spindle    Coolant
☐ Off    ☐ Off
☐ On    ☐ On

G-CODE File

X    14.229
Y    20.611
Z    -6.810

TOOL    1
ORIGIN NO    0
Z offset    0.00

Setup    Help

Y+    Z+    ☐ Handwheel    Reset Errors    APPLY ORI 1    SHROUD DOWN    Menu Screens

☐ Continuous

X-    X+    ☐ Step    MDI    Home All    GOTO XYO ORI 1    PINS UP    OFFSETS

HOME    Jog Speed    0.00    SINGLE AXIS HOME    Save    Vacuum    I/O

The Axis not referenced error will appear. Click on Home and all the axes will move to their home
position. If any errors remain on the screen (i.e. Low Air Pressure,) rectify the problem and click
on Reset Errors to remove the message.
The machine is now ready to be jogged.

## Jogging the machine.

The machine has three jog modes:
- Handwheel: The machine is controlled by the handwheel on the side of the machine.
- Continuous: The machine is controlled by placing the mouse over the arrows on the screen and left clicking. Speed is controlled by the Jog Speed Bar.
- Step: Similar to continuous, however with every mouse click the machine moves a distance determined by the Jog Step Bar.

A mode MUST be selected in order for the machine to move. The selection box to the left of the text indicates what mode is selected.
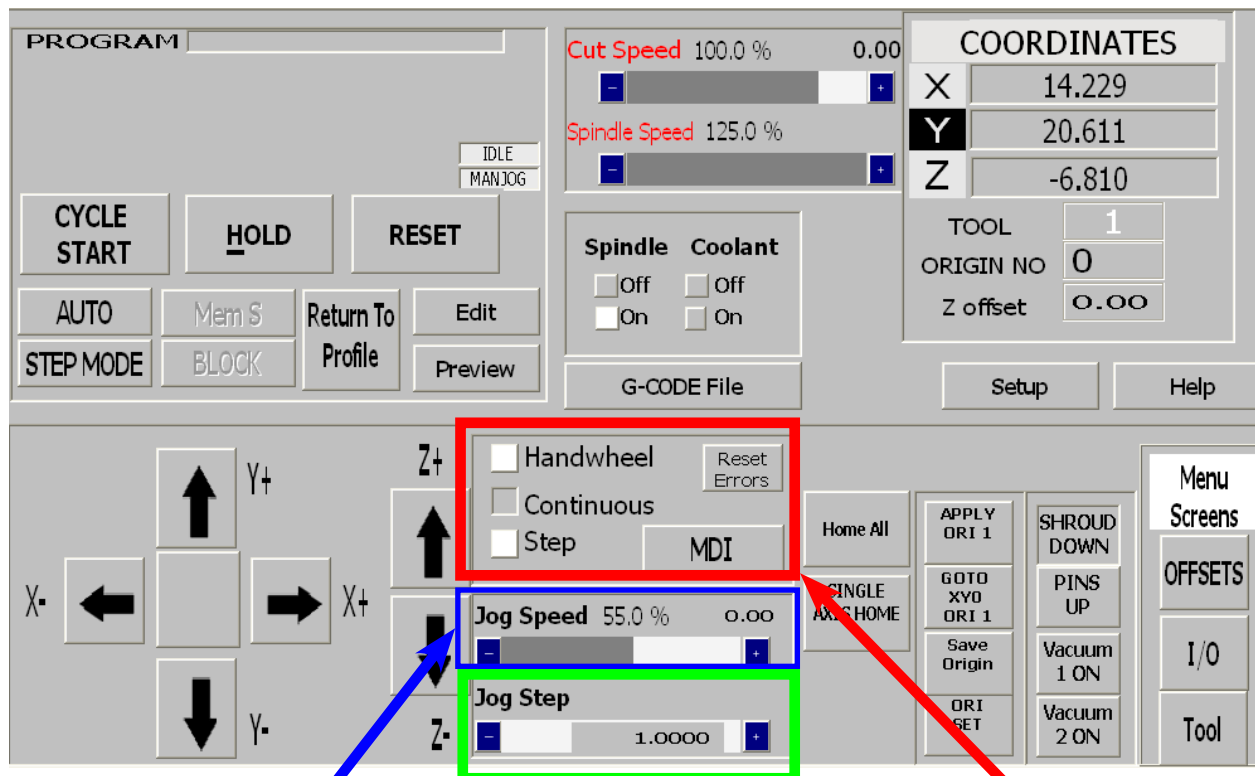


Increase or decrease the jog speed by pressing the blue +or- box. Speed is shown in % of maximum speed. Actually speed is displayed when the machine is running.

Increase or decrease the Jog step by pressing the +or- box. Step size in inches will be shown.

Click on the box beside the mode name to select it.

## Setting the Origin/XYZ zero position.

When setting an origin, make sure the correct tool is in the chuck and this tool has been identified. The T.OFFSET box in the top right hand side of the screen should be 1.1 or 2.2 etc for whatever tool is in the chuck. If the T.OFFSET box only reads one digit the tool must be identified in the Tool screen. See Tools section for instructions in how to do this.

To set the Origin.
-Move the machine to the location on the table you want to set as XYZ. Use the handwheel for precision.
-Once in position switch to Continuous mode.
-Click on Save Origin. The Save Origin screen will open, and a reminder to load the tool number will appear.
-Enter the Origin Number on top and zero in the XYZ boxes.
-Click OK
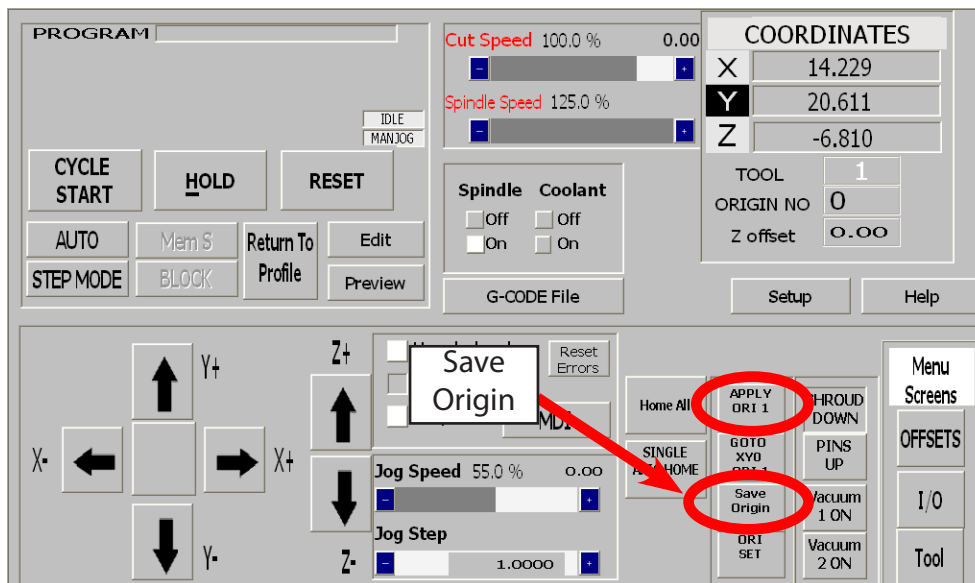


The coordinates displayed will still be the Work Coordinates. That is the coordinate system from the home position.
To display the Origin just saved. Click on Apply ORI 1. The coordinate system will now switch to the origin you saved.
The data (UAO,01) in the g-code file will also switch coordinate systems. So if you hit reset and the coordinate system returns to the Work/Home Coordinates, when the g-code file is started the controller will read the (uao,01) comand and apply Origin 1.

The origin is now set. If you click reset, the coordinate system will switch back to Work system, and the tool offset will be removed.
Identifying the Tool in the chuck and clicking ORI will enter the values again.

**Techno** *Inc.* *SINCE 1986*

## TOOLS

## Learning Tool Lengths.

Tool lengths will record how long the tools in the holder and record the offset value. This will allow multiple tools to be used in a single file.

Tool Locations should be pre-learned from the factory, if the Tool locations are incorrect see Learning Tool Locations document.

To learn tool lengths.
-Click on Tool in the main screen of the Techno interface.

-Place the router bits in the tool holders, ensuring that they are secure.

-Place the tool holder in the chuck.

This can be done manually, by pressing the green button on the side of the spindle, placing the tool holder in the spindle and releasing the button.

Or if the tool is in the tool stand, pressing the corresponding button under the Manual Tool Change bar in the Tool Screen. Warning take note of the T.OFFSET number. If it tries to return a tool to an occupied tool location it can cause damage to the tool stands.

-When a tool is in the spindle place the Touch-Off pad that is mounted to the gantry under the Tool, make sure this is placed on a solid surface that will remain constant for all the tools you learn.

-Click Reset to remove any offsets.

-Click OFST # . Where # is the number of the Tool you have in the chuck.

The machine will go to Top of the Z travel, then slowly move down to the pad. It will touch the brass colored section of the pad, retract, then move down again to confirm the value.

Repeat this procedure for the other tools.

Tool Lengths are now learned.
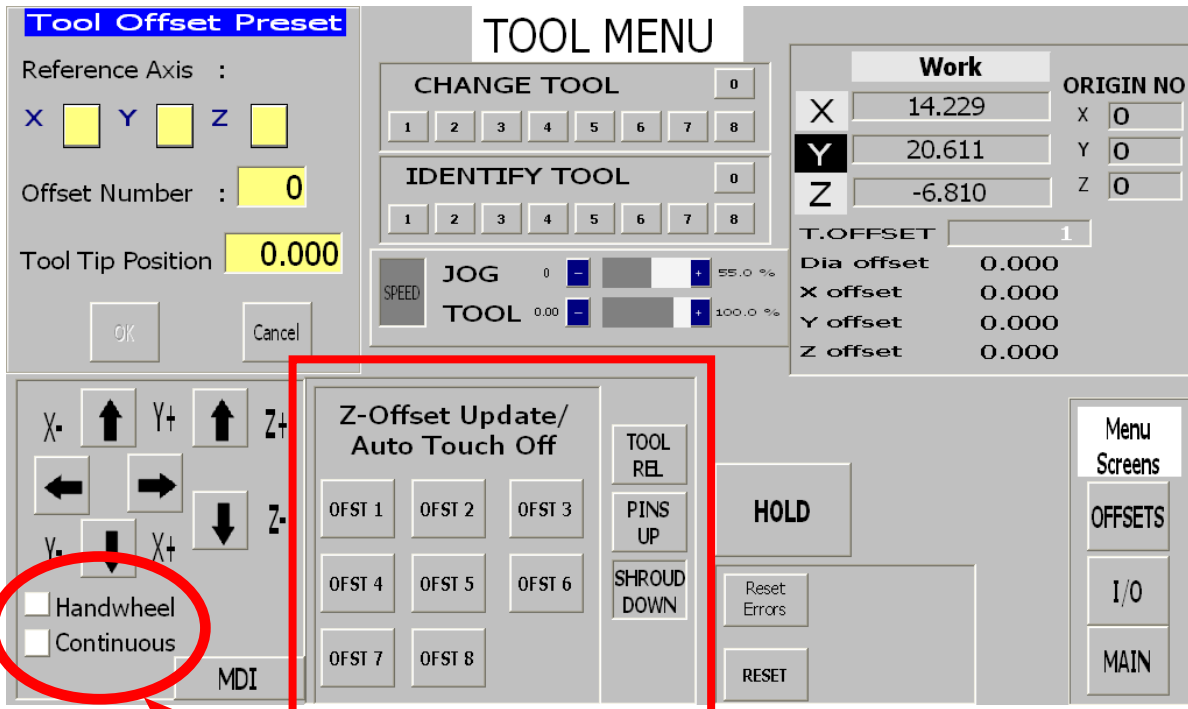
If jogging the machine make sure one of these buttons is clicked, otherwise it will not move.
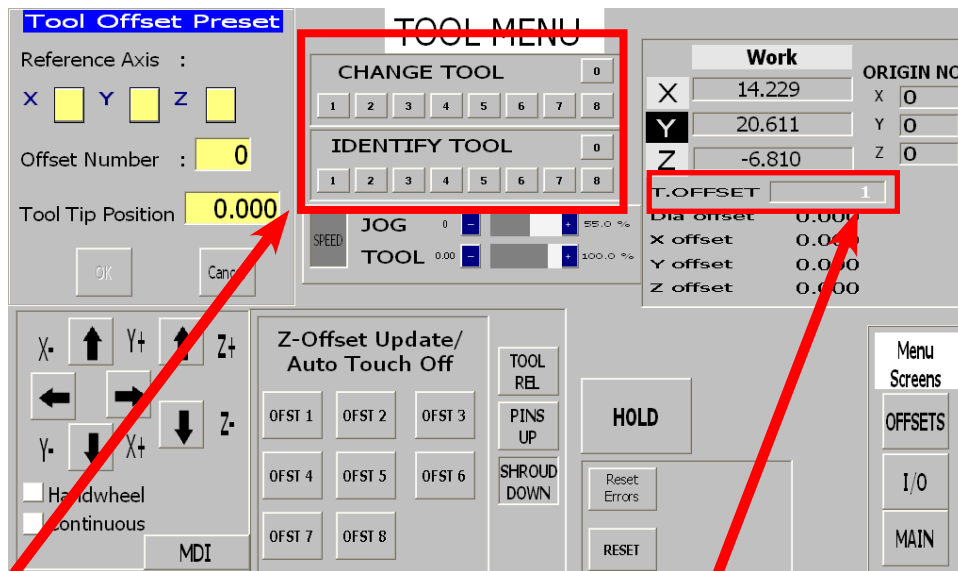


Identify or Change Tools by pressing these buttons. 0 is an empty chuck.

This displays the Tool that is in the chuck. 1 means no offset is applied. 1.1 means offset is applied.

## Loading and Running a G-Code File.

The g-code file must be copied into the programs folder on the PLC directory.

There is a shortcut to this on the main desktop. When operating in virtual XP mode, the usb stick is activated by pressing USB on the top of the screen.

Drag your files from the USB to the desktop icon.

Files must be no more than 8 characters in length. Folders can be copied in to the programs folder but make sure the file G600 is in the same folder as your file.

This data must be in the g-code file:
Data For a Techno-Osai G-code File.

Start of File:

| | |
|---|---|
| G70 | (Programming in Inches) |
| G90 | (Absolute Programing) |
| G40 | (Disables Cutter Diameter compensation) |
| G80 | (Disable Can Cycles) |
| G17 | (Circular interpolation on XY plane) |
| G27 | (Continuous sequence operation with automatic speed reduction on corners) |
| M143 | (Edge Pop Up Pins Down) |
| M49 | (Dust shroud down) |
| G600 | (Loads a set of parameters into controllers amplifiers. This is a txt file that needs to be in the directory of the g-code file. ) |
| (UAO,01) | (Applies Origin 1 to machine coordinates. Ideally user will have an option to select Origin number 1-5) |

Tool Changes:

| | |
|---|---|
| T1 M6 | (Standard Tool change command) |
| M3 S18000 | (Spindle on and RPM value) |
| (DLY,05) | (Wait 5 seconds for spindle to achieve set speed) |
| G27 | (Apply continuous sequence operation with automatic speed speed reduction) |

Circles:

| | |
|---|---|
| G02 | (Clockwise with Arc Center (I-J) Absolute) |
| G03 | (Anti-Clockwise with Arc Center (I-J) Absolute) |
| | R is also accepted, but I-J's must be Absolute. |

End of File:


M05                             (Spindle Off)
M48                             (Dust shroud up)
M30                             (File end)



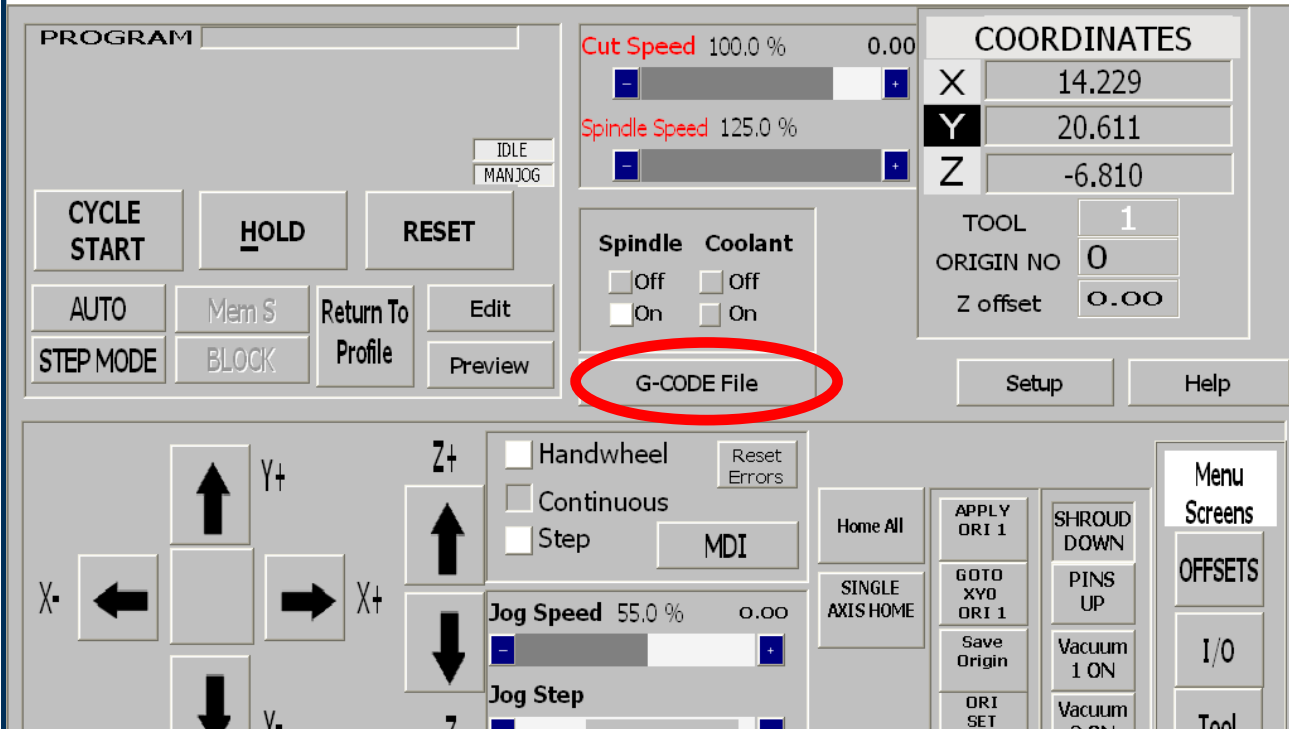So the start of a typical file will look like this.

G70
G90
G40
G80
G17
G27

M143
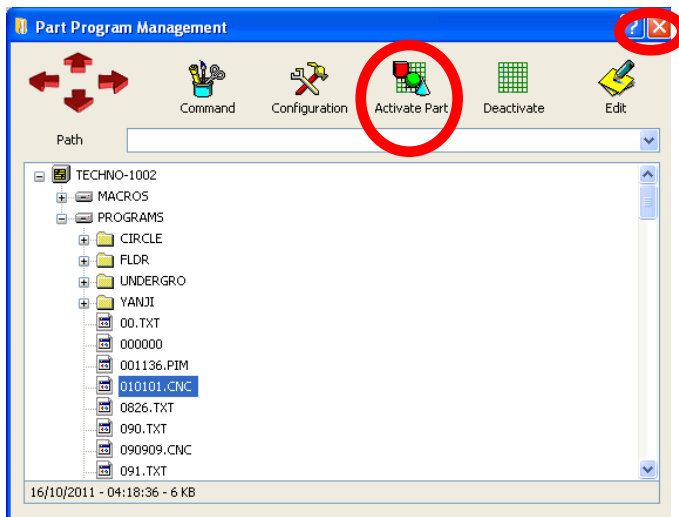M49
G600

(UAO,01)
T1 M6
M3 S18000
(DLY,05)
        .............
            Then the customers G-code.
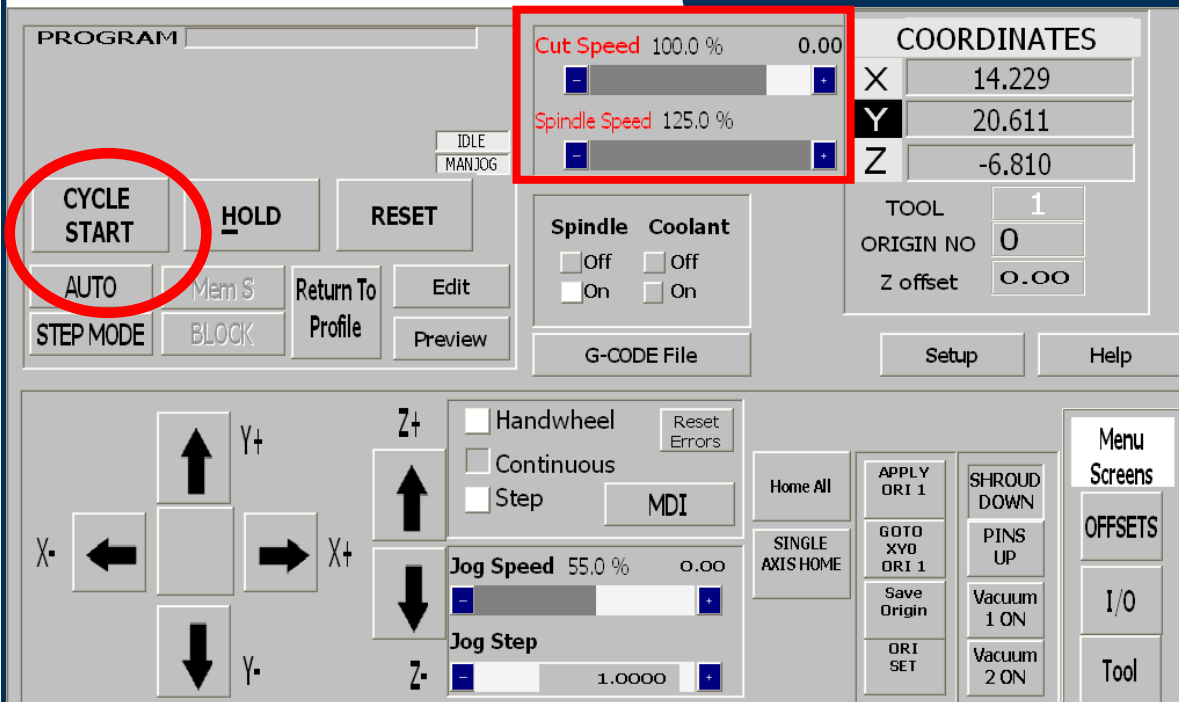                        ...................
M05
M48
M30

Click on G-code File to access the file menu.



Locate the File you want to load.
Left click on the file.
Click Activate to load it to the machine.
Close the window by clicking X.

When Loading a new File click
Deactivate to remove the old file,
before you click activate.

**Techno** *Inc.* *SINCE 1986*



The file name will now appear on the top right of the screen. You are now ready to run the file.

To run the file.
-Ensure that the origin and the Tool offsets are set.
-Check AUTO to run in continuous mode.
-Click Cycle Start to run the file.
-The cut speed can be adjusted by pressing the +or- blue box under cut speed.

The file will now run.
Pressing Hold will pause the machine.
Pressing Cycle Start will continue the file.
DO NOT JOG OFF THE PART WHILE IN HOLD.

Reset will stop the file and reset offsets and the origins.