

# Techno Systems

## Techno CNC HDS Interface Guide

Click to continue!

On the following screen please click a button or area to learn more about it.

# Contents

This presentation will guide you through Techno HDs machine operation. Please choose a guide or select “controller layout” to learn about HDS operation.

## Advanced Guides

- [Changing Tools](#)
- [Measure Tool Lengths](#)
- [Running a Program](#)
- [Setting an Origin](#)
- [Squaring HDS Machine](#)



Controller Layout



CNC IP.192.168.0.1

**PROGRAM**

IDLE  
MANJOG

**CYCLE START**    **HOLD**    **RESET**

**AUTO**    **Mem S**    **Return To**    **Edit**  
**STEP MODE**    **BLOCK**    **Profile**    **Preview**

**Cut Speed** 100.0 %    0.00

**Spindle Speed** 100.0 %    0

**Spindle**    **AUX**  
 Off     Off  
 On     On

**G-CODE File**

**COORDINATES**

**X** 11.65171  
**Y** 40.49118  
**Z** 7.58016

**TOOL** 1.1  
**ORIGIN NO** 0  
**Z offset** -10.4

**Setup**    **Help**

**X-**    **X+**  
**Y+**    **Y-**  
**Z+**    **Z-**

Handwheel    **Reset Errors**  
 Continuous  
 Step    **MDI**

**Jog Speed** 45.0 %    0.00

**Jog Step** 0.0000

**Home All**  
**SINGLE AXIS HOME**  
**CLEARANCE POSITION**

**Shut Down**

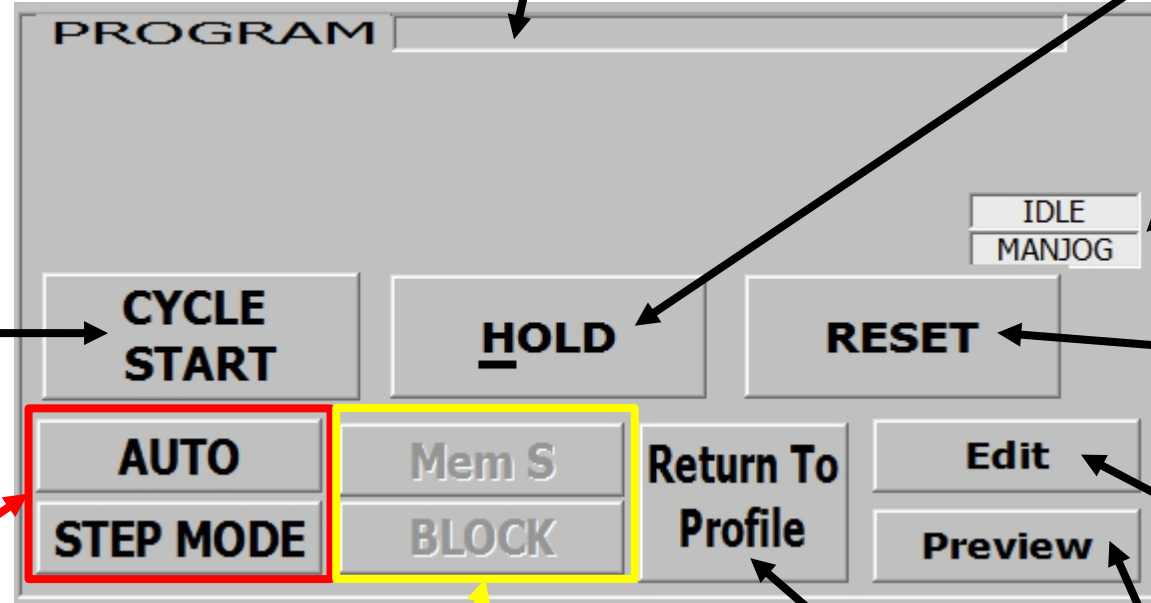
**APPLY ORI 1**  
**GOTO XY0 ORI 1**  
**Save Origin**

**SHROUD DOWN**  
**PINS UP**  
**Vacuum 1 ON**  
**Vacuum 2 ON**

**Screens**

**OFFSETS**  
**I/O**  
**Tool**  
**PRODUCTION**

# Program Execution Area



This shows the name of the currently loaded program. The first few lines of G-code are also shown.

“Hold” is like pause. Click the button to learn more about it.

This shows the current status and mode of the machine. Click for more information

“Reset” is like Stop, Cancel, Clear or Escape.

“Edit” opens up WordPad so you can make changes directly to the G-Code.

“Preview” opens up the Techno G-code Previewer, allowing you to visualize your current program.

“Cycle Start” is like Start, Play or Run. Once a program is loaded, “Cycle Start” will run it.

These choose what mode the selected program will run in. “Auto” mode runs the program in its entirety, start to finish. “Step Mode” allows you to run the selected program line by line. In “Step Mode”, press “Cycle Start” to execute the highlighted line of code.

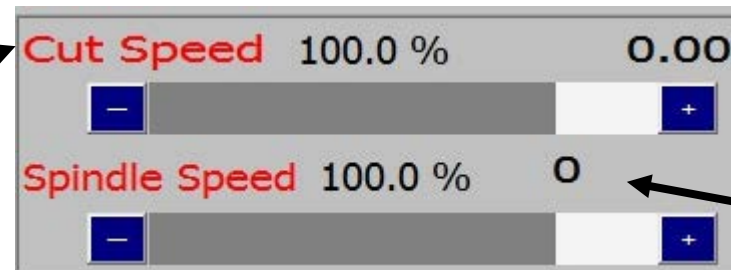
Mem S and Block allow you to skip through the currently loaded program to a different start point. Click the area to learn more about it.

“Return To Profile” allows the user to resume a program after pausing and moving the HDS. Click the area to learn more about it.

# Program Overrides

These overrides will only work when a program is currently running. The override commands can change from 0% to 120% of programmed speeds and feeds.

“Cut Speed” overrides the programmed feed rate of the currently loaded program. To increase the speed, press the “+” button. To decrease the speed, press the “-” button. The current feed rate will be displayed. Units are Inches per Minute.



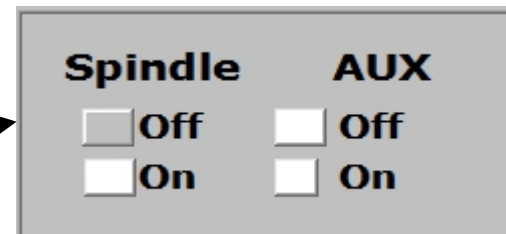
“Spindle Speed” overrides the programmed Spindle speed of the currently loaded program. To increase the speed, press the “+” button. To decrease the speed, press the “-” button. The current spindle speed will be displayed. Units are RPM.

Return to  
Main Screen

# Manual Controls

These controls will manually turn on and off the spindle and any auxiliary components such as coolant systems.

“Spindle” allows the user to turn on the spindle at the minimum RPM of 6000. To turn on the spindle, press the “On” button. To turn off the spindle, press the “Off” button. When the spindle is turned on, you will receive a yellow notification that says “012 Spindle Running in Manual Mode”.



“Aux” allows the user to turn on the auxiliary input, in most cases, a coolant system. To turn on the Aux input, press the “On” button. To turn off the Aux input, press the “Off” button. This input can also be controlled via the M70 and M71 G-code.

Return to  
Main Screen

# Coordinate System

This display shows the current location of the tooltip in terms of Absolute or Relative systems. It also displays the current tool number and associated tool length.

This area shows you the current position of the tool. Units are in Inches. The reference point is indicated by the "Origin No". "0" means Absolute or Machine Coordinates. Any other number indicates Work or Job Coordinates.

COORDINATES	
X	11.65171
Y	40.49118
Z	7.58016
TOOL	1.1
ORIGIN NO	0
Z offset	-10.4

"Origin No" tells you what Origin is currently applied. "0" indicates Absolute or Machine Coordinates. Any other number indicates Work or Job Coordinates.

"Tool" shows you what tool the HDS machine thinks is currently in the spindle. If this is incorrect, please identify the tool correctly. In this example, the HDS thinks tool 1 is in the spindle. The ".1" indicates the tool length is also applied.

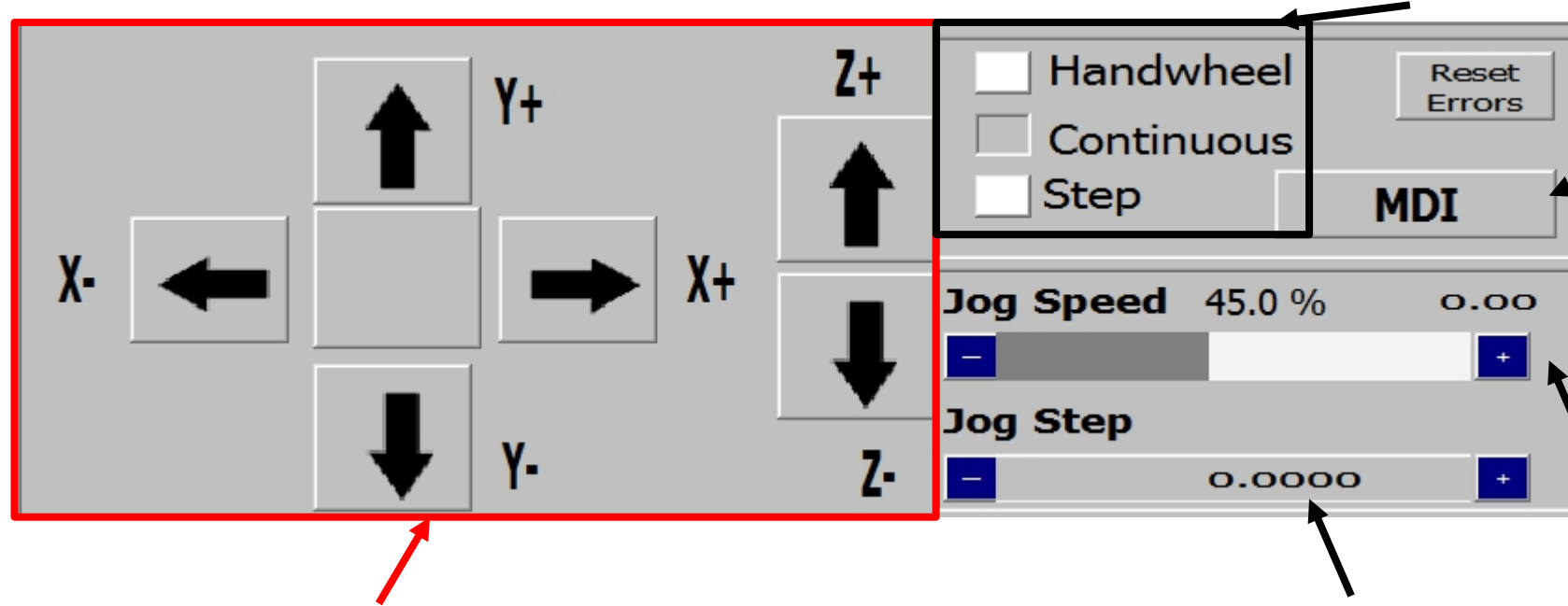
"Z Offset" shows you the length of the current tool. The number is always negative in value. If the number is "0", either the tool length has not been learned or the tool needs to be identified.

Return to  
Main Screen

# Manual Movement Modes

These controls will allow you to manually move the spindle head around the process area.

These buttons allow you to choose which mode you would like to use to manually move the machine. Continuous mode is smooth jogging, where you press and hold the direction you would like to travel in. Step mode is like Continuous mode but moves the machine in set increments. Handwheel mode allows the user to use the handwheel to move the machine. Click on handwheel for more information.



These buttons indicate direction of movement for the HDS machine. When you choose a movement mode, these arrows will move the machine in the indicated direction. The Y-axis is the long axis; the X-axis is the short axis and the Z-axis is the vertical axis.

“Jog Step” indicates the increments that each axis will move when in “Step” mode. To increase the step size, press the “+” button. To decrease the step size, press the “-” button.

“MDI” or manual data input mode. Click to learn more.

“Jog Speed” indicates the speed that each axis will move when in “Continuous” mode. To increase the speed, press the “+” button. To decrease the speed, press the “-” button. The current speed will be displayed.

[Return to Main Screen](#)



# Preset Functions

These controls will allow you to execute some preset functions.

“Home All” allows the user to home the machine to find its limit switches and machine origin. You will need to “Home All” on machine start up. You will receive a yellow notification saying “011 Axes Not Referenced”. To dismiss this error, please choose “Home All”. The Z axis will home first and then the Y and X axes will home together.



“Single Axis Home” is similar to “Home All” except it allows the user to home one axis at a time instead of all 3 concurrently. Press “Single Axis Home” and then press the “Y-”, “X-” or “Z+” button to home the individual axes.

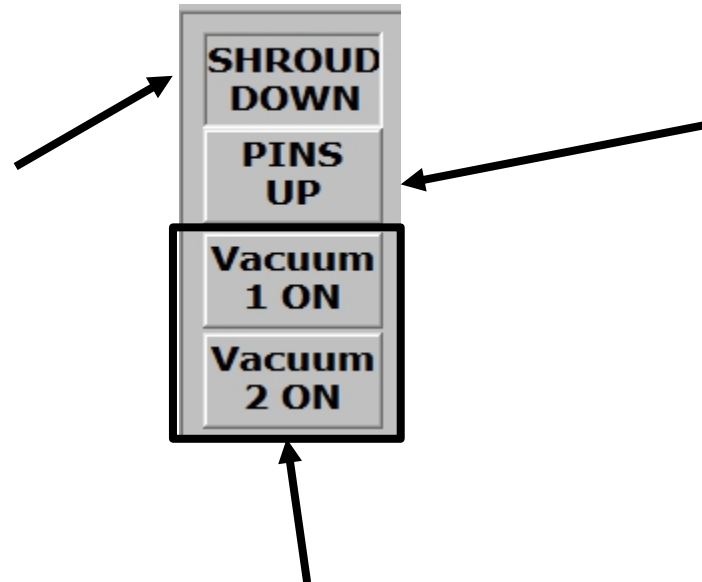
“Clearance Position” moves the machine to a preset park position that is up and out of the way of the process area. The spot can be changed.

Return to  
Main Screen

# Accessory Controls

These controls will allow you to control various accessories such as the dust shroud, pop-up pins and vacuum pumps.

“Shroud Down” / “Shroud Up” allows the user to control the position of the spindle’s dust shroud. Press once to lower, again to raise.



“Pins Down” / “Pins Up” allows the user to control the position of the pop-up alignment pins. Press once to lower, again to raise.

These buttons control the vacuum pumps. Press once to turn on, Press again to turn off.

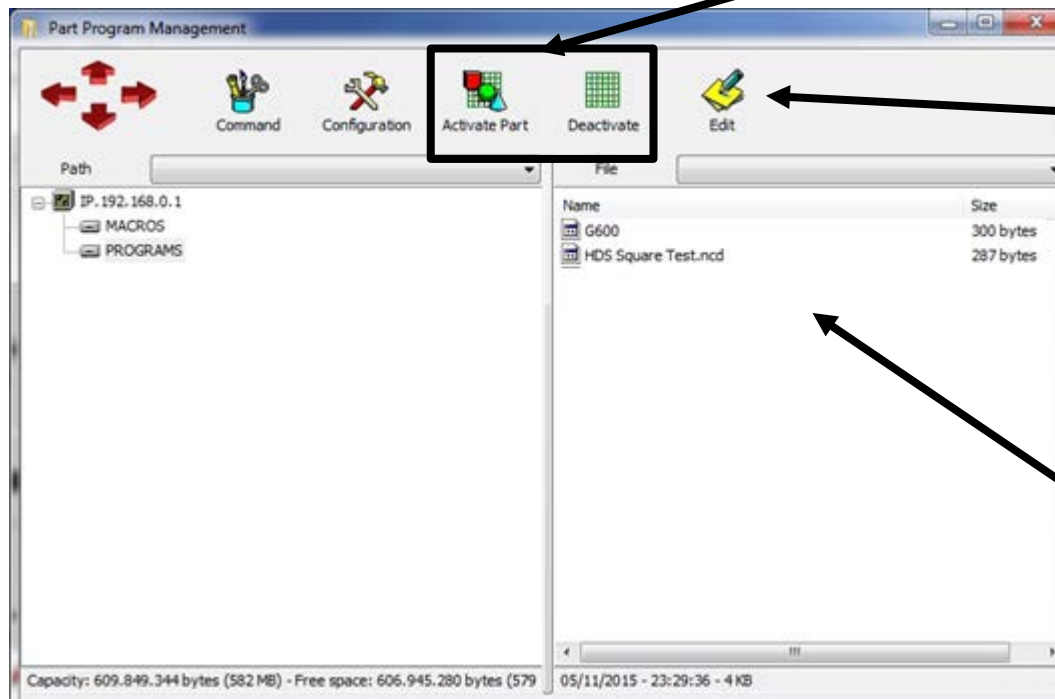
Return to  
Main Screen

# Load G-Code File

**G-CODE File**

These buttons allow the user to bring up the Part Program Management Screen. It allows the operator to load, edit and manage G-code files.

“Activate” and “Deactivate” are used to load and unload programs. Loading prepares the program for execution.



“Edit” opens up WordPad so you can make changes directly to the G-Code.

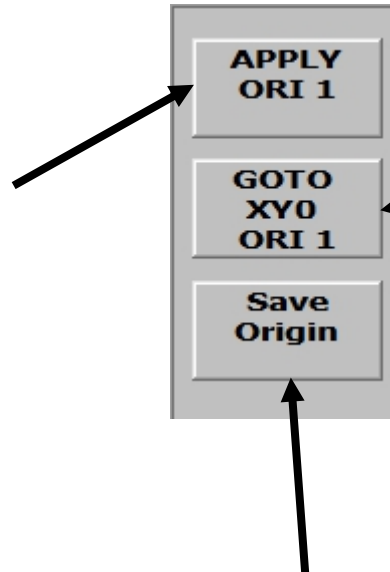
This is the list of programs currently saved to your HDS machine. These files must be loaded in the location “PROGRAMS – Shortcut” located on the computer desktop.

Return to Main Screen

# Quick Origin Commands

These buttons allow the operator to quickly apply and find origin 1, as well as save a new origin.

“Apply Ori 1” allows the operator to change the coordinate system to the work coordinate system with Origin 1 as its zero point. The coordinates with now reflect the position for the tool tip in relation to XYZ=0 for Origin 1.



“Goto XY0 Ori 1” will move the HDS to origin 1. The Z axis will home to the top of travel but the X and Y axes will move to reflect origin 1 X=0 and Y=0.

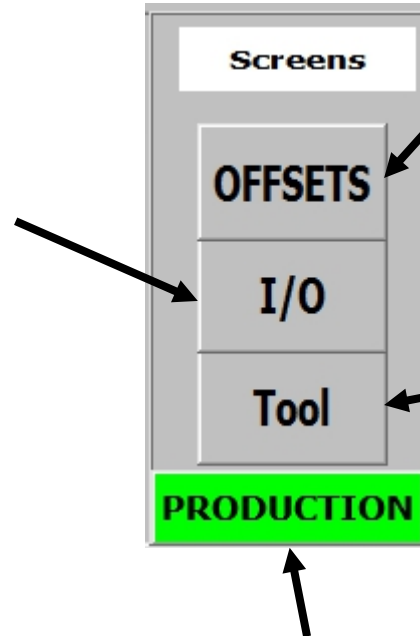
“Save Origin” brings up a new screen that allows the operator to save the current tool tip position as an origin. Please click the button for more information.

Return to  
Main Screen

# Change Screen Buttons

These buttons allow the operator to quickly change between operation screens.

“I/O” bring up the inputs and outputs diagnostic screen. It is useful in trying to identify problems you may have with the HDS.



“Offsets” brings up the offsets screen. This screen allows the operator to apply and go to up to 8 origins. You can also learn tool lengths at this screen. Please click the button for more info.

“Tool” brings up the Tool screen. This screen allows the operator to change tools and learn tool lengths. Please click the button for more info.

“Production” brings up the production screen. It is very basic and allows the operator to merely run a program and not anything else. Please click the button for more information.

Return to  
Main Screen

# Pause and Return to Profile

These buttons allow the operator to pause and then resume a job.

When you are running a job and would like to pause, press the “Hold” button. It stops all movement but keeps the spindle on.

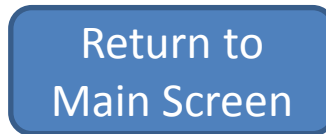
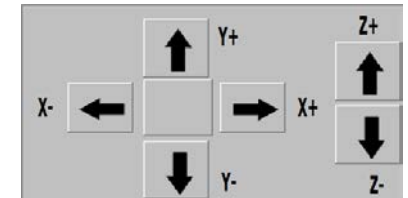
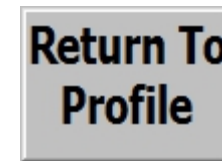
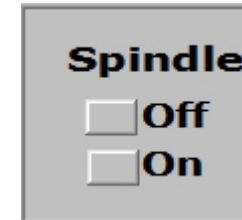
You can shut the spindle off using the manual spindle off button. You can also move the machine out of the way to measure cuts.

To resume, the machine must be on profile. To do so, press the “Return to Profile” button and then jog the machine toward its pause point. It will stop moving when on profile. The red notification will go from “Axes not on profile” to “Axis on Profile”.

**Error: 23/272 - Process: 01  
Axes not on profile**

**Error: 23/272 - Process: 01  
Axis on Profile**

You may now take “Hold” off, choose “Auto” mode and then press “Cycle Start” to resume the job.

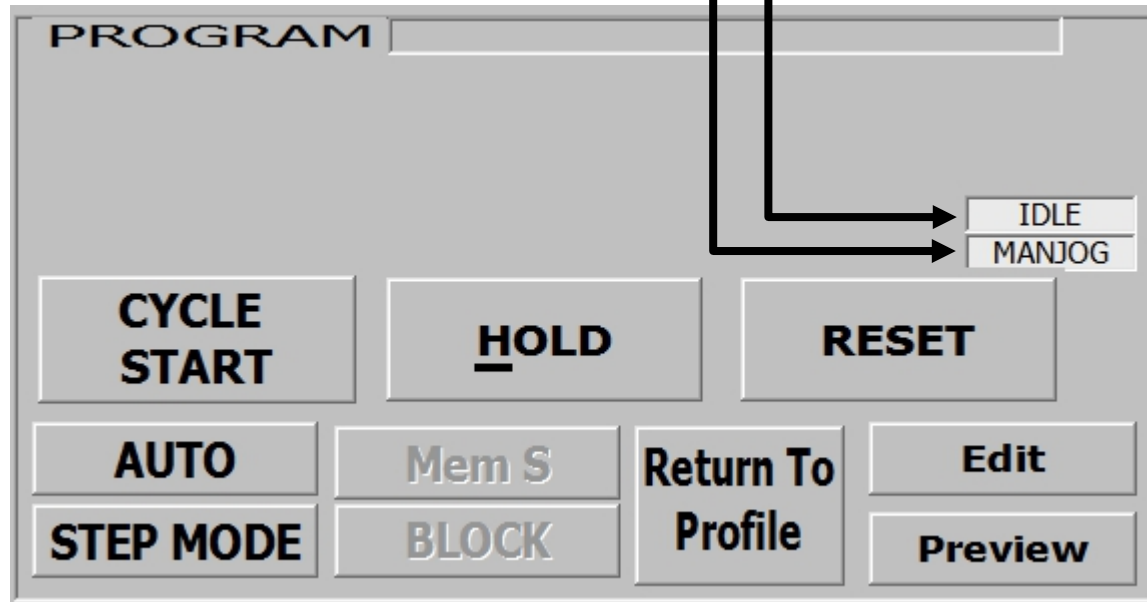


# Status and Mode Indicators

These two displays show the current status and mode the machine is operating in.

The top box shows status.

The bottom box shows mode.



Status	Description
IDLE	The machine is not active or in use.
INCYCLE	The machine is in use. Either running a program or using the hand wheel.
HOLD	The machine is on hold. Program is paused.
<b>HRUN</b>	<b>The machine is on hold. Not on axis.</b>

Status	Description
MANJOG	The machine is in "Continuous" mode.
INCJOG	The machine is in "Step" mode.
HANDWHEEL	The machine is in "Hand Wheel" mode.
PROFILE	"Return to Profile" is selected
AUTO	"Auto" mode selected for G-code execution.
BLK-BLK	"Step" mode selected for G-code execution.
<b>MDI</b>	<b>Manual Data Input selected.</b>

Return to Main Screen

# Mem S and Block

These buttons allow the operator to skip ahead in a program. Mem S searches through the program to a new start point. Block executes a selected block of code.

[Return to  
Main Screen](#)



# Hand Wheel Operation

This knob is used to select which axis is to be controlled. Limited to X, Y, and Z. Use "4" if your machine has a rotary axis.



This knob is used to move the selected axis. Spin the knob clockwise to move in the positive direction. Spin the knob counter-clockwise to move in the negative direction.

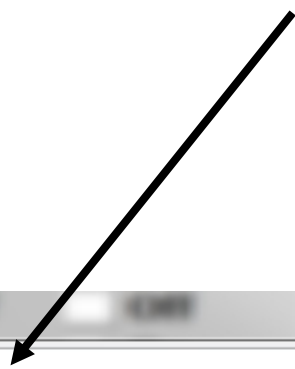
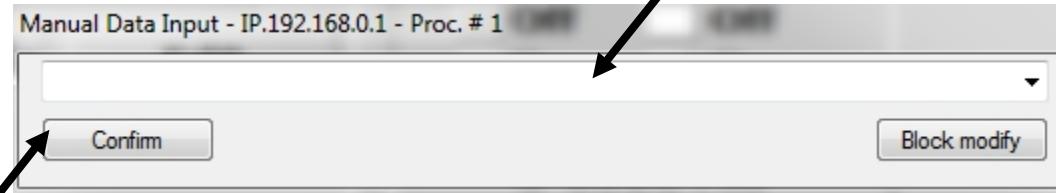
This knob is used to select the magnitude or units of the movement selected. X100 will be large movements and X1 will be very small units (.0001).

Return to  
Main Screen

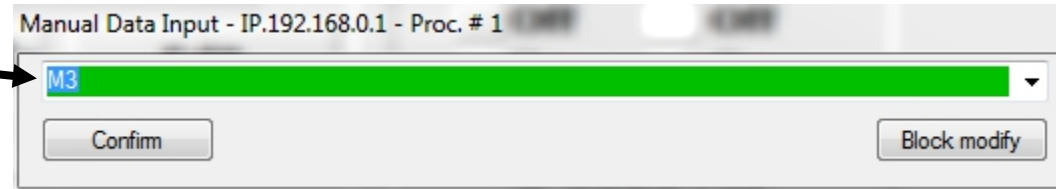
# MDI Mode

MDI mode is Manual Data Input mode. It allows the operator to execute lines of G-code.

Input the G-code you would like to execute here. Please use capital letters. Spacing does not matter.

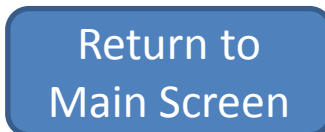


Press "Confirm" to load the line of G-code so that it is ready to run. The G-code will now be highlighted green.



Press "Cycle Start" to run the green highlighted G-code.

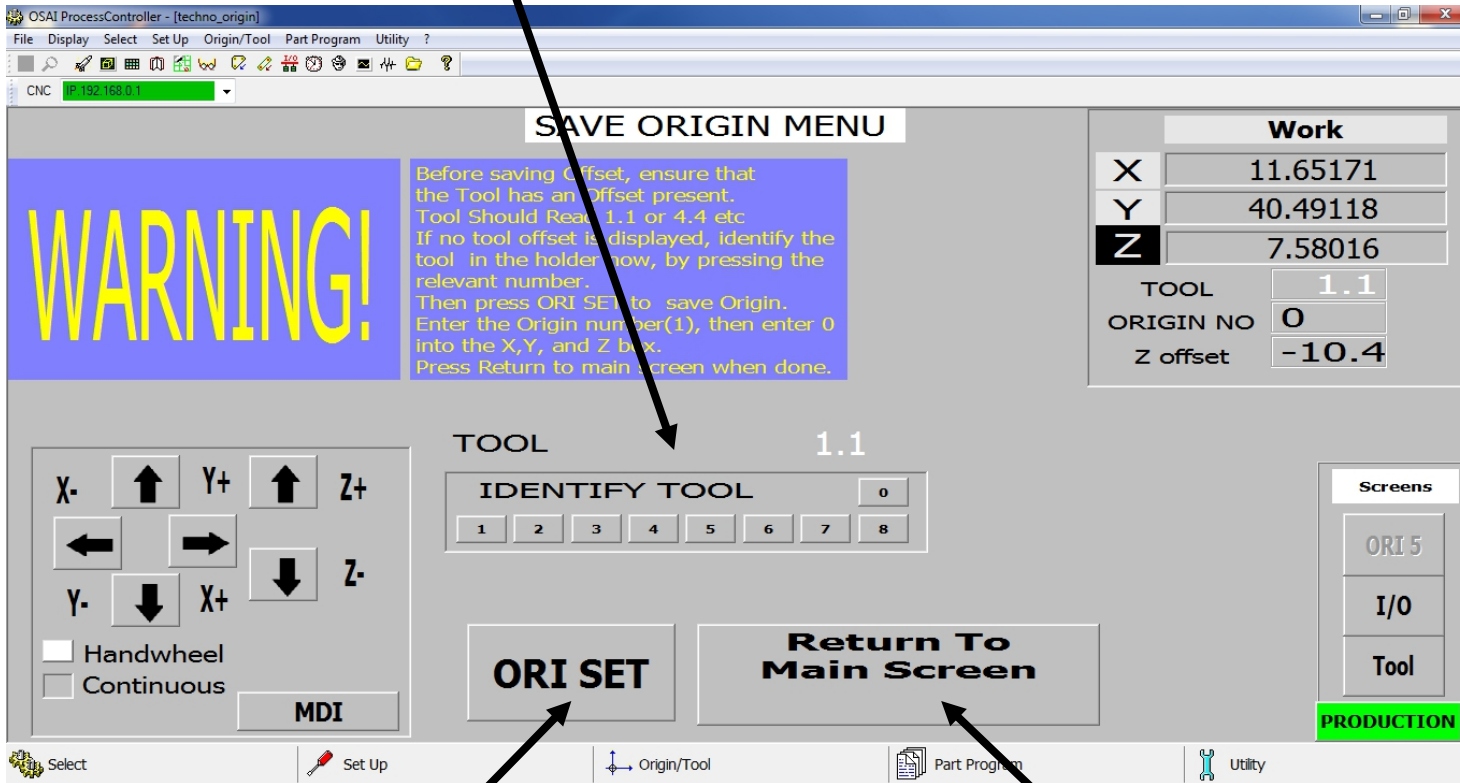
Press "Continuous" to dismiss the MDI mode pop up.



# Save Origin

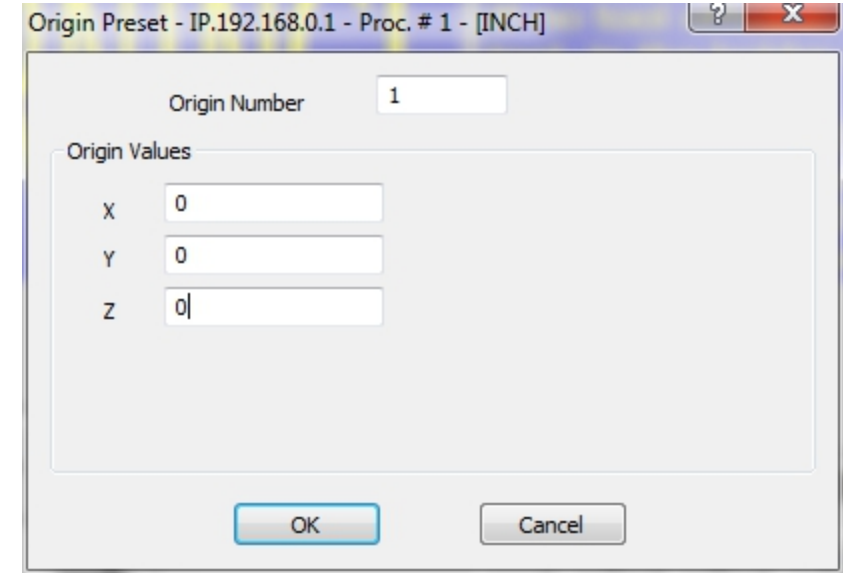
This screen allows the operator to save the current tool tip position as an origin.

1. Identify the current tool in the spindle



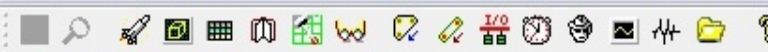
2. Press "ORI SET"

4. Press "Return to Main Screen" to finish



3. In this window, you will choose what origin number you would like to save under and its position. By default, many CAM programs will save under origin 1, so under "Origin Number", please input a "1". For origin values, if you are happy with the current position of the spindle as the start point, please input "0" for X, Y and Z. If for example you put "-.005" for Z, the origin start point will be .005" below the current position. Press OK.

Return to  
Main Screen



CNC IP 192.168.0.1

### Tool Offset Preset

Reference Axis :  
 X  Y  Z

Offset Number :

Tool Tip Position :

## TOOL MENU

**CHANGE TOOL**

1 2 3 4 5 6 7 8

**IDENTIFY TOOL**

1 2 3 4 5 6 7 8

**JOG** 0    45.0 %

**TOOL** 0.00    100.0 %

Version 2.8

Work		ORIGIN NO
X	<input type="text" value="11.65171"/>	X <input type="text" value="0"/>
Y	<input type="text" value="40.49118"/>	Y <input type="text" value="0"/>
Z	<input type="text" value="8.34700"/>	Z <input type="text" value="0"/>
T.OFFSET	<input type="text" value="8.8"/>	
Dia offset	<input type="text" value="0.000"/>	
X offset	<input type="text" value="0.000"/>	
Y offset	<input type="text" value="0.000"/>	
Z offset	<input type="text" value="-11.171"/>	

X- ↑ Y+ ↑ Z+

← →

Y- ↓ X+ ↓ Z-

Handwheel  
 Continuous

**MDI**

**Z-Offset Update/  
Auto Touch Off**

OFST 1 OFST 2 OFST 3

OFST 4 OFST 5 OFST 6

OFST 7 OFST 8

**TOOL REL**

**PINS UP**

**SHROUD DOWN**

**APPLY ORI 1**

**GOTO XY0 ORI 1**

**Save Origin**

**HOLD**

Reset Errors

**RESET**

**Screens**

**OFFSETS**

**I/O**

**MAIN**

**PRODUCTION**

# Manually Setting Tool Length

This display allows the operator to manually set the tool length. This is useful for irregularly shaped or coated tools.

“Reference Axis” refers to which axis you would like to measure the offset for. In most cases you will choose “Z”

“Offset Number” refers to the tool number you would like to manually measure. This tool already be in the spindle.

Tool Offset Preset

Reference Axis :  
X  Y  Z

Offset Number : 0

Tool Tip Position : 0.000

OK Cancel

Tool Tip Position refers to the value you would like to set the tool tip position to. In all cases you would choose “0.000”

To successfully manually measure a tool length, you must lower the tool tip to the top of the black phenolic table top. Then you choose “Z”, input the tool number, choose “0.000” and then press “OK”.

Return to  
Main Screen

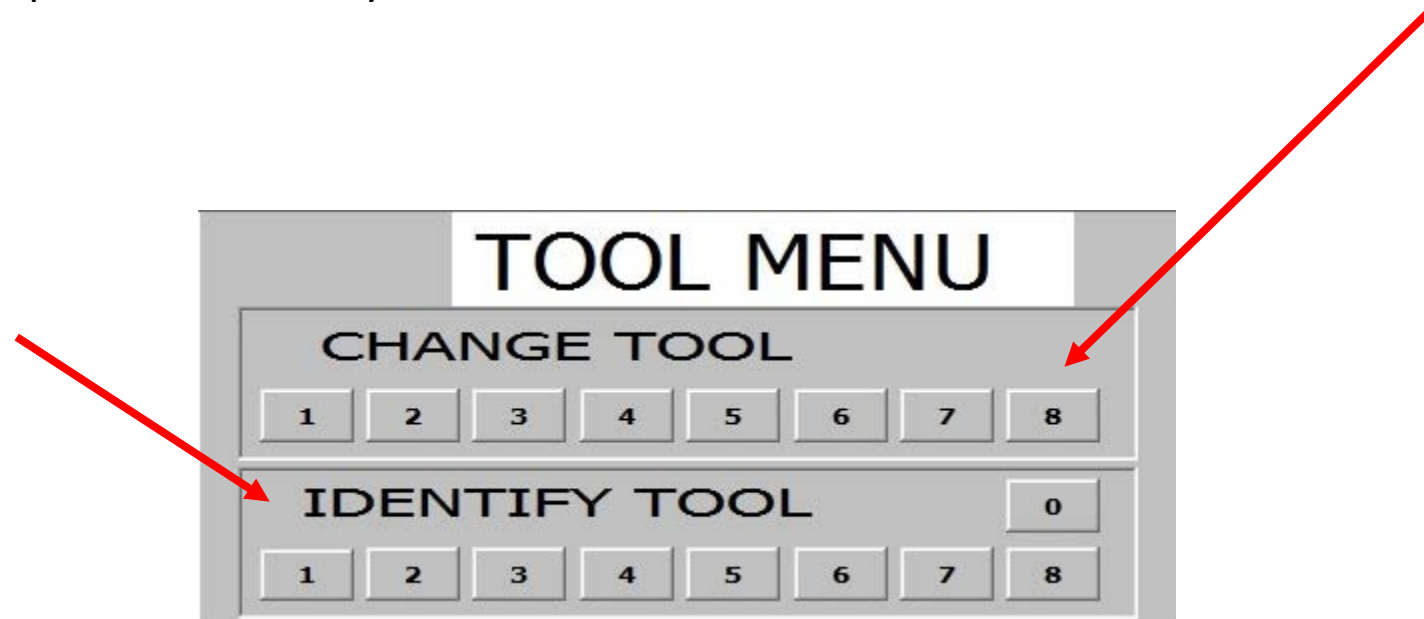


# Tool Menu

The tool menu allows the operator to identify and change tools.

“Identify Tool” allows the operator to indicate to the HDS what tool is currently in the spindle.

“Change Tool” allows the operator to put the current tool away and switch to the selected tool.

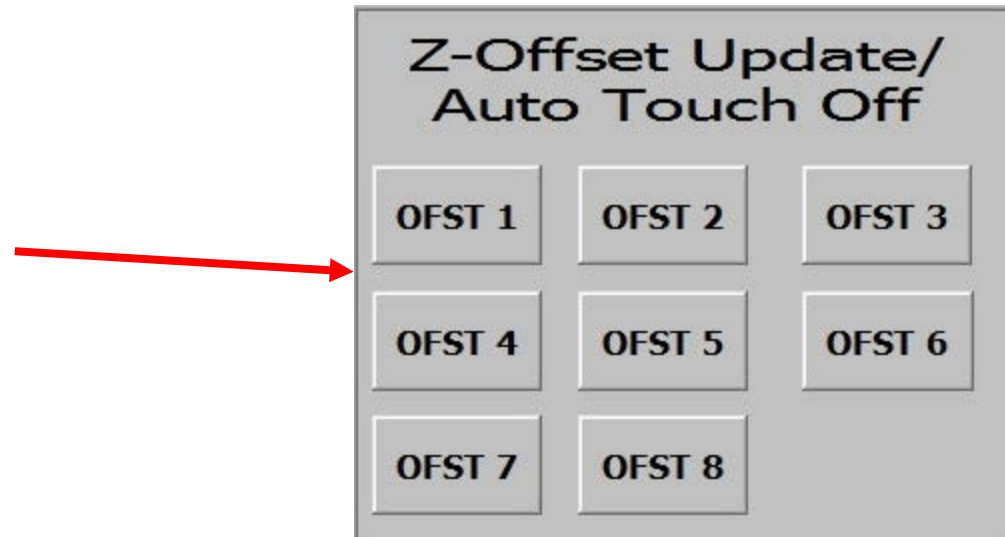


Be sure to always identify the tool in the spindle before changing to another tool. If you do not identify the tool correctly, you may break the tool you currently have in the spindle, break a tool

# Automatic Tool Length Setting

This display allows the operator to automatically learn the length of the current tool in the spindle.

To execute the automatic tool length probing cycle, please choose the correct "OFST" for the tool currently in the spindle.



If you had identified the tool in the spindle as tool number 4, you will then choose "OFST 4". This will cause the machine to measure the length of tool 4 (currently in the spindle). The HDS will move to the touch-off probe and measure twice. The value will be saved as the Z-Offset for tool number 4. Press "Reset" to get rid of the blue notification.

Return to  
Main Screen