

Smartshop III CNC Router Manual

SmartShop III 5x10 HSD wo Drill Block

SmartShop III 5x12 HSD w/ ATC - Without Drill Block

SmartShop III 5x12 HSD w Drill Block



LAGUNA TOOLS
2072 Alton Parkway
Irvine, California 92606
Ph: 800.234.1976
www.lagunatools.com

Model Numbers MCNC SS3 5x10 HSD NO Drill Block
MCNC SS3 5x12 HSD NO Drill Block
MCNC SS3 5x12 HSD NO Drill Block

© 2018, Laguna Tools, Inc. LAGUNA® and the LAGUNA Logo® are the registered trademarks of Laguna Tools, Inc. All rights reserved.

Table of Contents

1. GENERAL INFORMATION	5
1.1. SAFETY RULES	5
1.2. LIMITED WARRANTY	6
1.3. NOISE EMISSION	7
1.4. SPECIFICATION SHEET	7
1.5. RECEIVING YOUR MACHINE	7
1.6. MAINTENANCE	8
1.6.1. <i>Lubrication</i>	8
1.6.2. <i>Daily checks</i>	8
1.6.3. <i>Weekly checks</i>	8
1.7. GLOSSARY OF TERMS	9
2. OPERATION BASICS	12
2.1. TURNING ON THE MACHINE	12
2.2. SHUTTING DOWN THE MACHINE	17
2.3. MAIN SCREEN	22
2.3.1. <i>Top Toolbar section</i>	23
2.3.2. <i>CNC Display Section</i>	24
2.3.3. <i>Tool Display Section</i>	25
2.3.4. <i>Function Display Section</i>	26
2.3.5. <i>Modal G-Code/M-Code display section</i>	27
2.3.6. <i>Active G-Code Program Section</i>	28
2.3.7. <i>Coordinate display section</i>	29
2.3.8. <i>Feed and Speed Section</i>	30
2.3.9. <i>Override Buttons</i>	31
2.3.10. <i>Page buttons</i>	32
2.3.11. <i>Function Buttons</i>	33
2.3.12. <i>Pneumatics Control Buttons</i>	35
2.3.13. <i>Operation Mode Selection Button</i>	36
2.3.14. <i>Operation Buttons</i>	37
2.3.15. <i>Bottom Toolbar</i>	38
2.4. AXIS + SPINDLE SCREEN	39
2.4.1. <i>Axis Buttons</i>	40
2.4.2. <i>Feed and Speed Section</i>	41
2.4.3. <i>Speed/Distance Setting</i>	42
2.4.4. <i>Spindle Control</i>	44
2.4.5. <i>Page Buttons</i>	46
2.4.6. <i>Jogging Mode Buttons</i>	47
2.4.7. <i>Operation Buttons</i>	49
2.5. TOOL SCREEN	50
2.5.1. <i>CNC Display Section</i>	51

2.5.2. Tool Display Section	52
2.5.3. Drill Block Buttons	53
2.5.4. Tool Offset Preset	55
2.5.5. Spindle Buttons.....	56
2.5.6. Tool Changer	57
2.5.7. Page Buttons	59
2.5.8. Operation Buttons.....	60
2.6. TOOL LENGTH TABLE	61
2.7. WORK ORIGIN TABLE	63
3. OPERATION PROCEDURES	65
3.1. JOGGING THE MACHINE.....	65
3.1.1. Continuous Jogging Procedure	65
3.1.2. Step Jogging Procedure	68
3.1.3. Manual Pulse Generator (MPG)	71
3.2. SETTING TOOL LENGTH	74
3.3. SETTING WORK ORIGIN	77
3.4. RUNNING A G-CODE FILE	80
4. ADVANCED FUNCTIONS.....	92
4.1. MPG RUN	92
4.2. BLOCK BY BLOCK MODE.....	96
4.3. MDI	98
4.4. AUTOMATIC TOOL CHANGE	101
4.5. AUTOMATIC TOOL TOUCHOFF.....	104
4.6. START FROM BLOCK	108
4.7. MEMORY SEARCH FUNCTION	111
5. TROUBLESHOOTING	114
5.1. FAQ	114
5.2. SYSTEM RESTORE.....	115
5.3. SYSTEM BACKUP.....	125
6. APPENDIX.....	130
6.1. MDI COMMAND LIST	130
6.2. G-CODE LIST	131
6.3. M-CODE LIST	133
6.4. CONTROLLER SPECIFIC FUNCTIONS.....	134
6.5. SAMPLE G-CODE	134

General Information

Safety Rules

As with all machinery, there are certain hazards involved with the operation and use of your machine. Using it with caution will considerably lessen the possibility of personal injury. However, if normal safety precautions are overlooked or ignored, personal injury to the operator may result. If you have any questions relating to the installation and operation, do not use the equipment until you have contacted your supplying distributor.

Read the following carefully and fully before operating the machine.

1. Keep the working area clean and ensure adequate lighting is available.
2. Do not wear loose clothing, gloves, bracelets, necklaces or ornaments.
3. Do wear face, eye, respiratory and body protection devices as indicated for the operation or environment.
4. Ensure that the power is disconnected from the machine before tools are serviced or any attachment is to be fitted or removed.
5. Never leave the machine with the power on.
6. Do not use dull, gummy or cracked cutting tools.
7. Ensure that the keys and adjusting wrenches have been removed and all the nuts and bolts are secured.

Limited Warranty

New machines and accessories sold by Laguna Tools carry a one-year warranty effective from the date of shipping. Machines sold through dealers must be registered with Laguna Tools within 30 days of purchase to be covered by this warranty. Laguna Tools guarantees all new machines and accessories sold to be free of manufacturers' defective workmanship, parts and materials. We will repair or replace, without charge, any parts determined by Laguna Tools, Inc. to be a manufacturer's defect. We require that the defective item/part be returned to Laguna Tools with the complaint. Any machines returned to Laguna Tools must be returned with packaging in the same manner in which it was received. If a part or blade is being returned it must have adequate packaging to ensure no damage is received during shipping. In the event the item/part is determined to be damaged due to lack of maintenance, cleaning or misuse/abuse, the customer will be responsible for the cost to replace the item/part, plus all related shipping charges. This limited warranty does not apply to natural disasters, acts of terrorism, normal wear and tear, product failure due to lack of maintenance or cleaning, damage caused by accident, neglect, lack of or inadequate dust collection, misuse/abuse or damage caused where repair or alterations have been made or attempted by others.

Laguna Tools, Inc. is not responsible for additional tools or modifications sold or performed (other than from/by Laguna Tools, Inc.) on any Laguna Tools, Inc. machine. Warranty maybe voided upon the addition of such described tools and/or modifications, determined on a case-by-case basis.

Software purchased through Laguna Tools Inc. is not covered under this warranty and all technical support must be managed through the software provider. Software is non-refundable.

Normal user alignment, adjustment, tuning and machine settings are not covered by this warranty. It is the responsibility of the user to understand basic machinery operation, settings and procedures and to properly maintain the equipment in accordance with the standards provided by the manufacturer.

Parts, under warranty, are shipped at Laguna Tools, Inc.'s cost either by common carrier, FEDEX ground service or a similar method. Technical support to install replacement parts is primarily provided by phone, fax, e-mail or Laguna Tools Customer Support Website. The labor required to install replacement parts is the responsibility of the user.

Laguna Tools is not responsible for damage or loss caused by a freight company or other circumstances not in our control. All claims for loss or damaged goods must be notified to Laguna Tools within twenty-four hours of delivery. Please contact our Customer Service Department for more information.

Only **new** machines sold to the original owner are covered by this warranty. For warranty repair information, **call 1-800-332-4094**.



Noise emission

Given that there exist a relationship between noise level and exposure times, it is not precise enough to determine the need for supplementary precautions. The factors affecting the true level of exposure to operators are clearly the amount of time exposed; the characteristics of working environment; other sources of dust and noise, etc. For example, adjacent machines may impact the level of ambient noise. It is also possible that exposure level limits will vary from country to country.

Specification sheet

Note: The specification will depend on the machine that is ordered. Most machines are made to order, and therefore the specification will vary depending on the machine you require.

Receiving your machine

Note: It is probable that your machine will be delivered by a third party. Before you unpack your machine, you will first need to inspect the packing, invoice and shipping documents supplied by the driver.

Ensure that there is no visible damage to the packing or the machine. You must do this prior to the driver leaving. All damage must be noted on the delivery documents and signed by you and the delivery driver. You must then contact the seller, Laguna Tools, within 24 hours. It is advisable to photograph any shipping damage to support an insurance claim.

Note: Laguna Tools, Inc. endeavors to test each machine prior to shipping, and you may find sawdust in or on your machine.

Maintenance

Lubrication

You must regularly (minimum every 12 hours) lubricate the rails. Use a thin layer of 30wt oil lubricant. Spray daily and wipe off the excess.

Daily checks

1. Clean the machine and lubricate unpainted surfaces with a 30wt oil lubricant. Wipe off any excess and buff with a dry polishing cloth. This will reduce the likelihood of rust forming.
2. Check cutter teeth for chips and dullness.
3. Generally inspect the machine for damage and loose or worn parts.
4. Collets and spindle collet holes must be cleaned regularly. Ensure that the slots in the collets are free of sawdust, as sawdust builds up and will stop the collet compressing. If the collet or spindle holes are not clean, the router bit may not run true, and this will affect the performance of your machine.



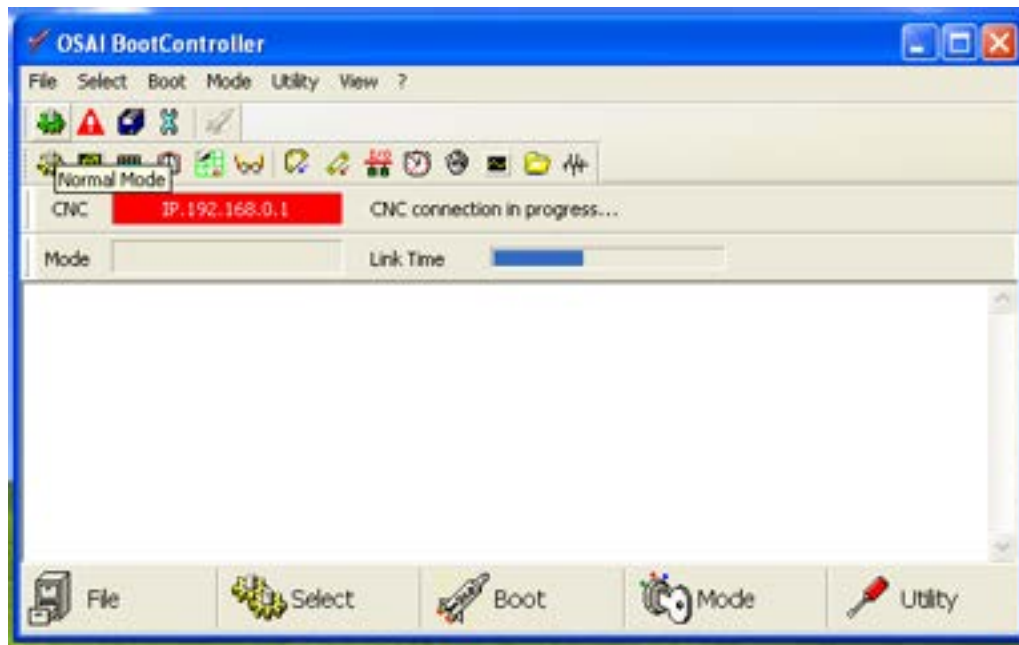
Collet fitted to spindle nut

Weekly checks

1. Clean the cutters.
2. Check cutter teeth for chips and dullness.
3. Generally inspect the machine for damage and loose or worn parts.
4. Check the dust extraction for blockages, as any large bits could cause blockages.
5. Replace the water every week.
6. Rotate the water pipes every week. (Inlet to out let and reverse for the other pipe. This will clean out any dirt or other debris that is accumulating inside the cooling system.)
7. Check that all the electrical connectors are fitted correctly and are not loose.
8. Check that all the motor couplers are connected and that the screws are tight.

Glossary of terms

Boot Controller – The window used to boot the machine.



Process Control Window – The window that contains all the control screens for the machine. It includes the Main screen, Axis + Spindle Screen and Tool Screen.



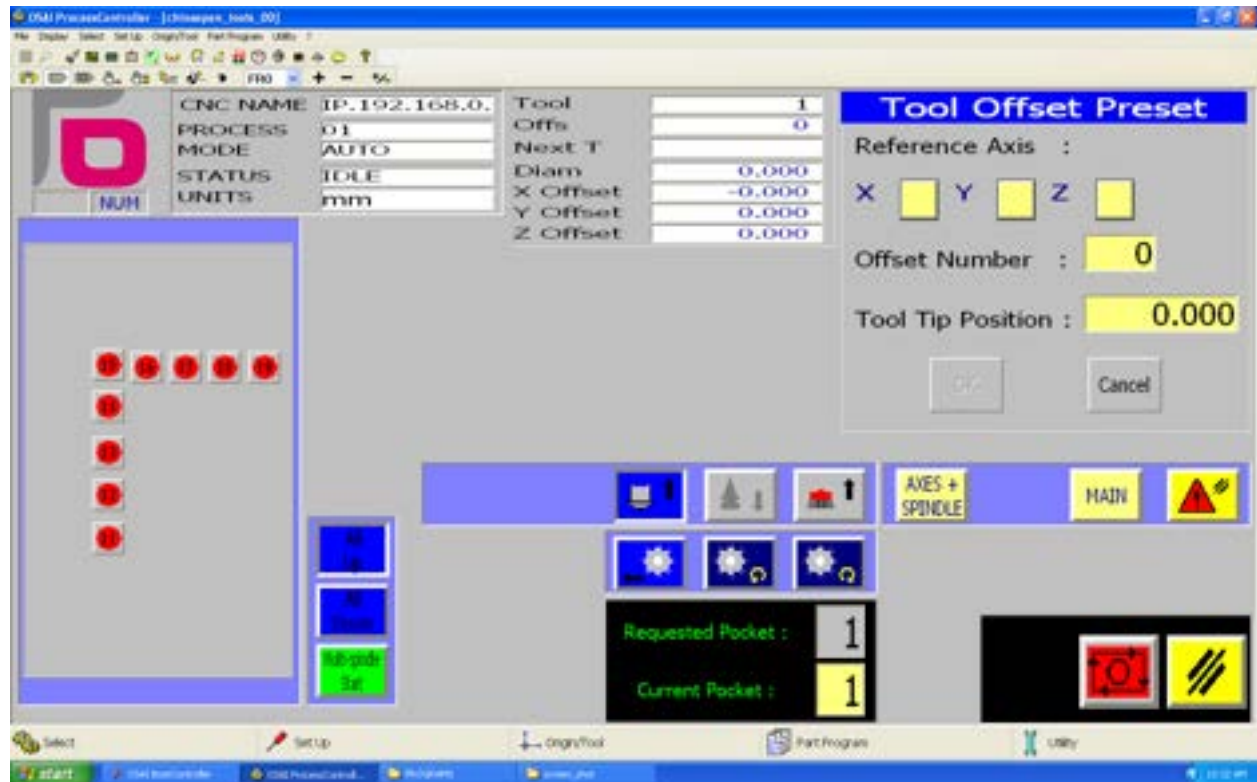
Main Screen – the screen that shows the active button, MDI button, and the active G-Code.



Axis + Spindle Screen – The screen that allows the user to move the spindle manually.



Tool Screen –The screen that displays tool information.



Manual Pulse Generator (MPG) – The axis



Operation Basics

Turning On the Machine

1. Make sure there is no object or people near the machine that can be damaged by machine movement.
2. Rotate the main contact switch clock-wise until it clicks in place.



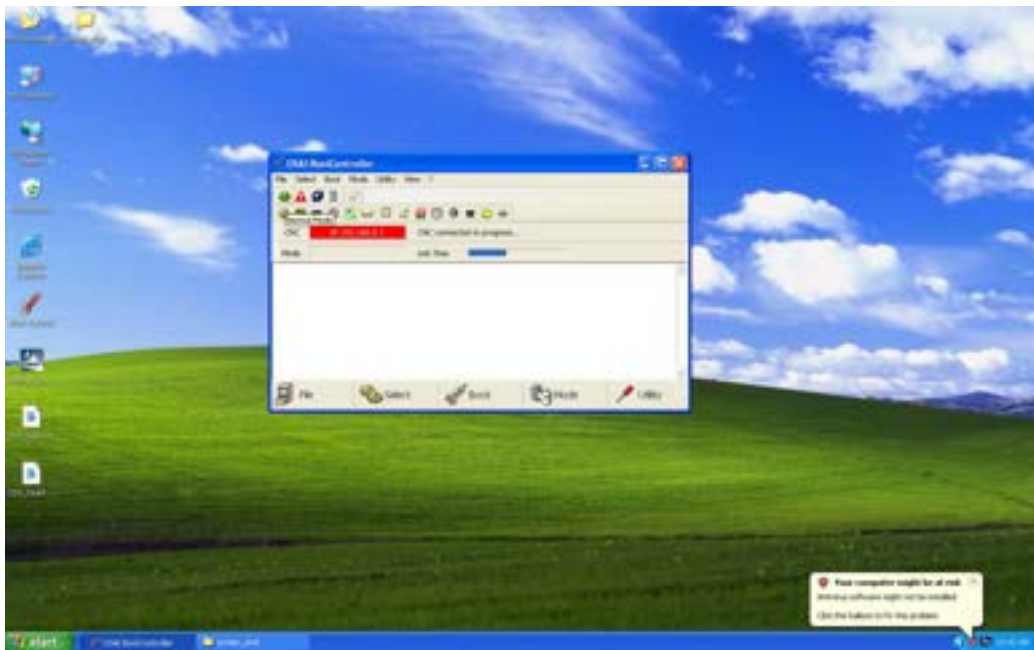
3. Press the PC power button.



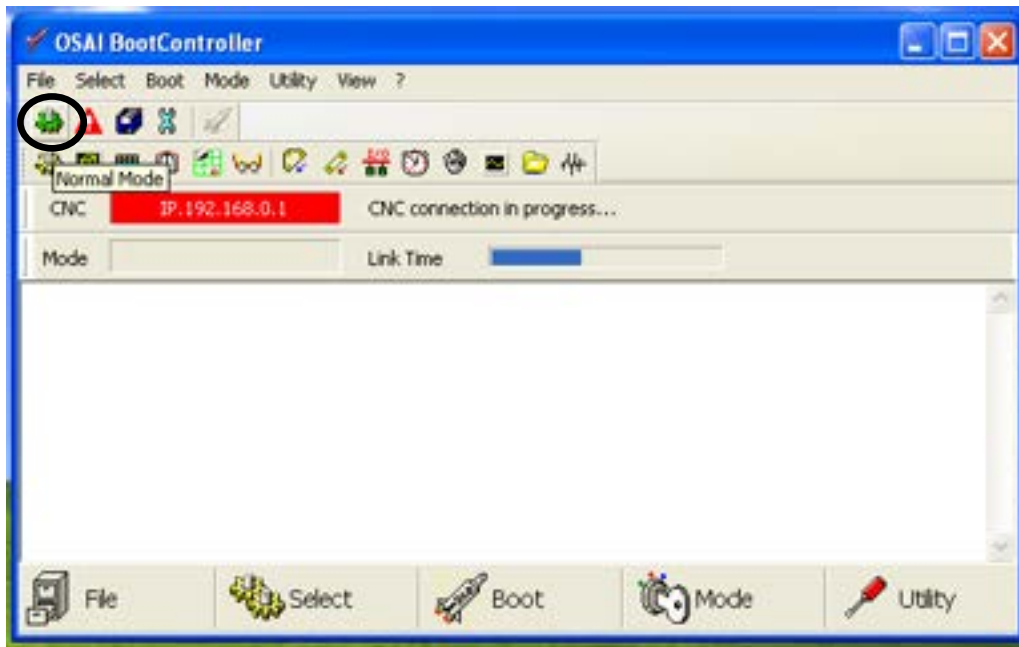
4. Press the controller power button. There might be an additional controller on button that needs to be pressed, depend on your machine setup.



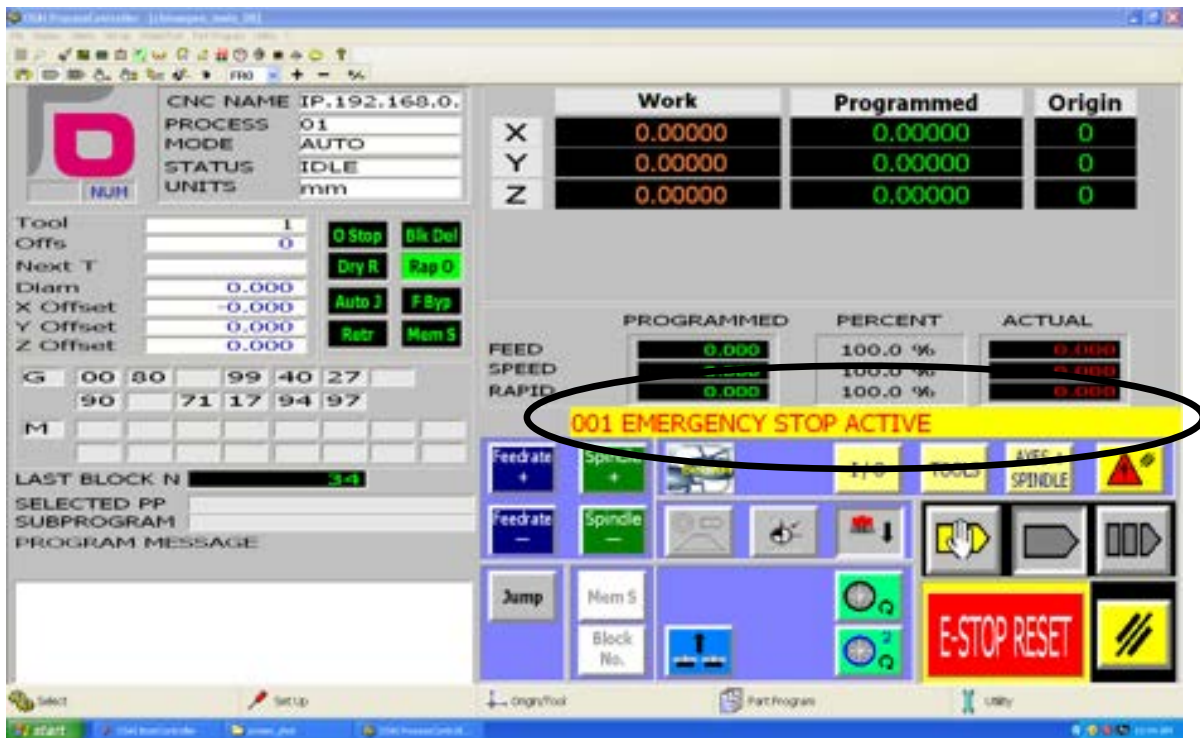
5. The Boot controller will load as in picture below.



- Press the Normal Mode button.



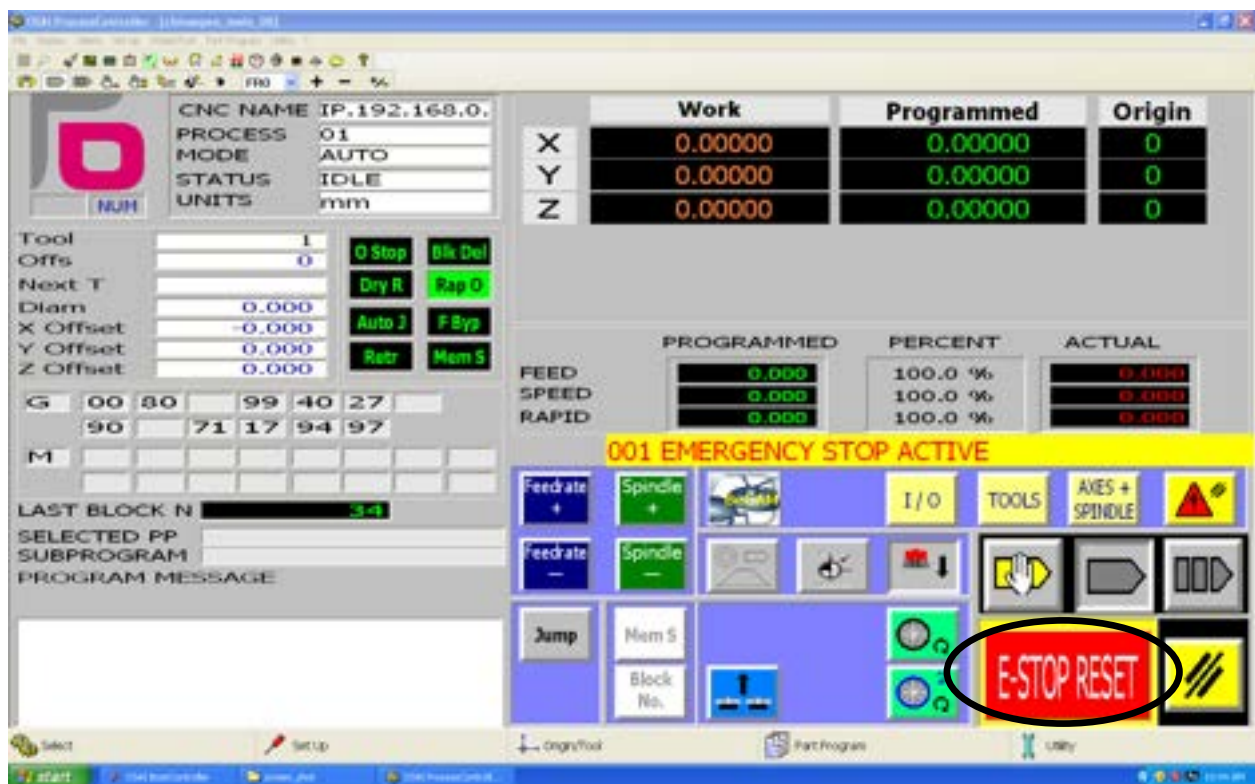
- Wait for the machine to start up. Once start up is completed, the Process Control window will display the Main screen as in picture below. Notice that there is a message box that displays "001 EMERGENCY STOP ACTIVE."




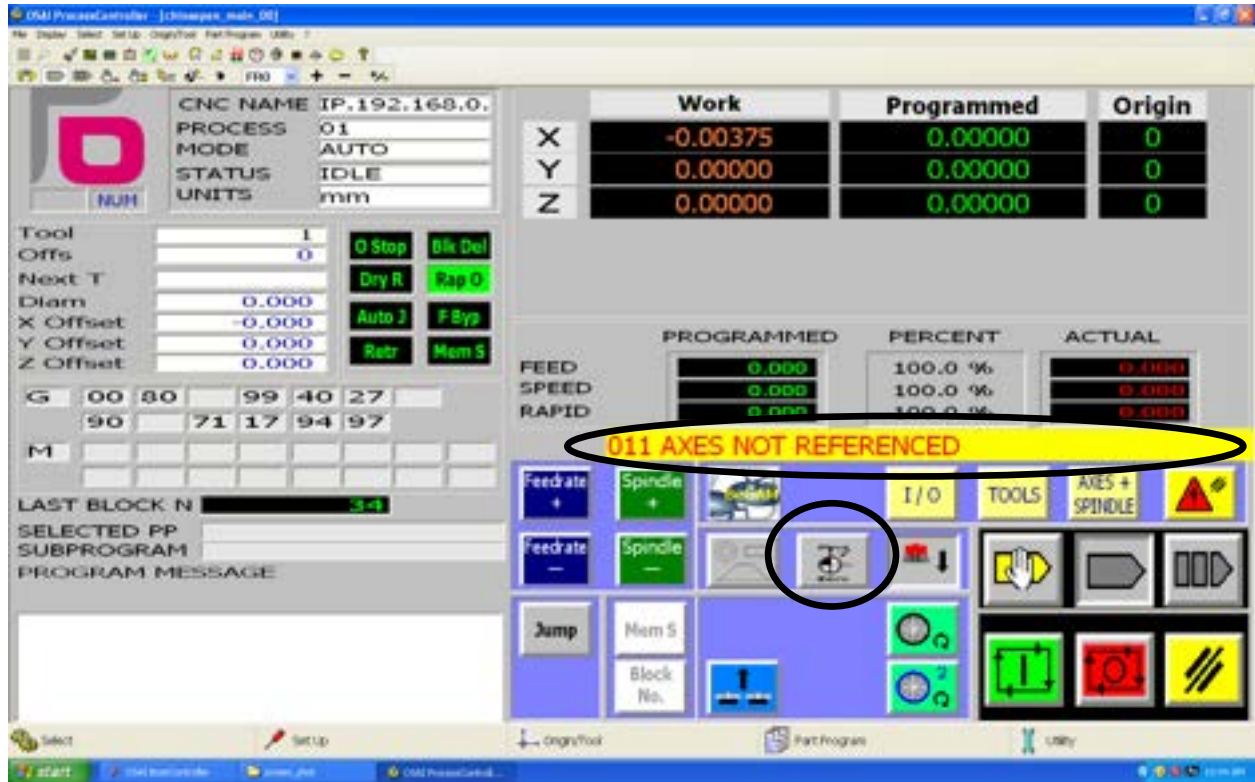
8. Make sure the E-stop is released by rotating it clockwise.




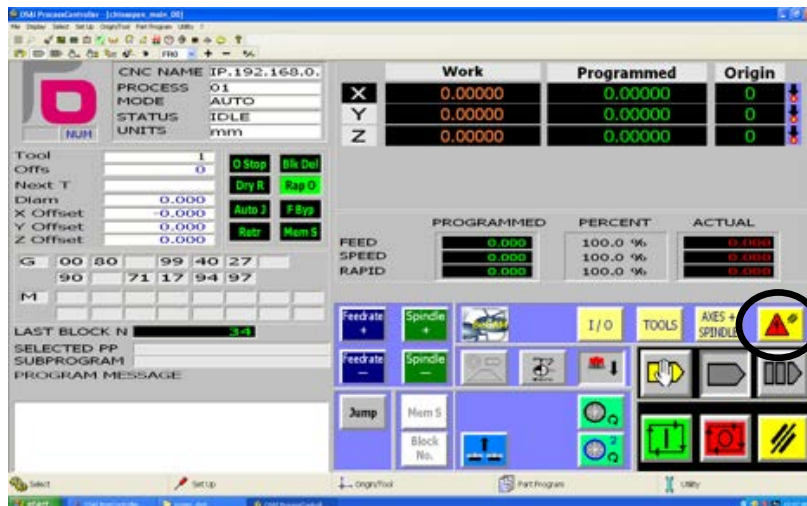
9. Click on the E-STOP RESET button. This will clear the emergency stop mode.



10. Notice that the message box displays “011 AXIS NOT REFERENCED”, it is because the machine needs to be homed at start up. Click the home button , machine will start moving to its homing position.



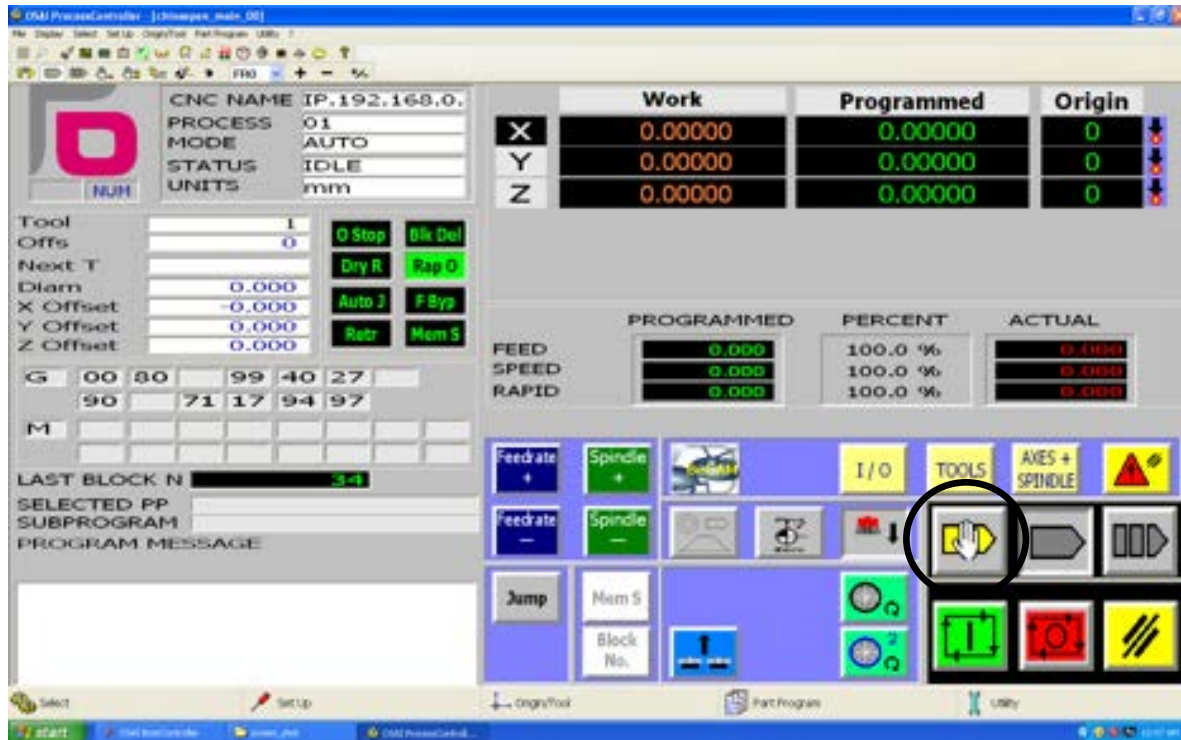
11. Wait until the machine has finished homing. The machine should now have no message in the message box. The machine is now ready for operation. If message box shows the message “LOW AIR PRESSURE”, make sure there is adequate compressed air supplied to the machine then press the Reset Errors button  to remove the message.



Shutting Down the Machine

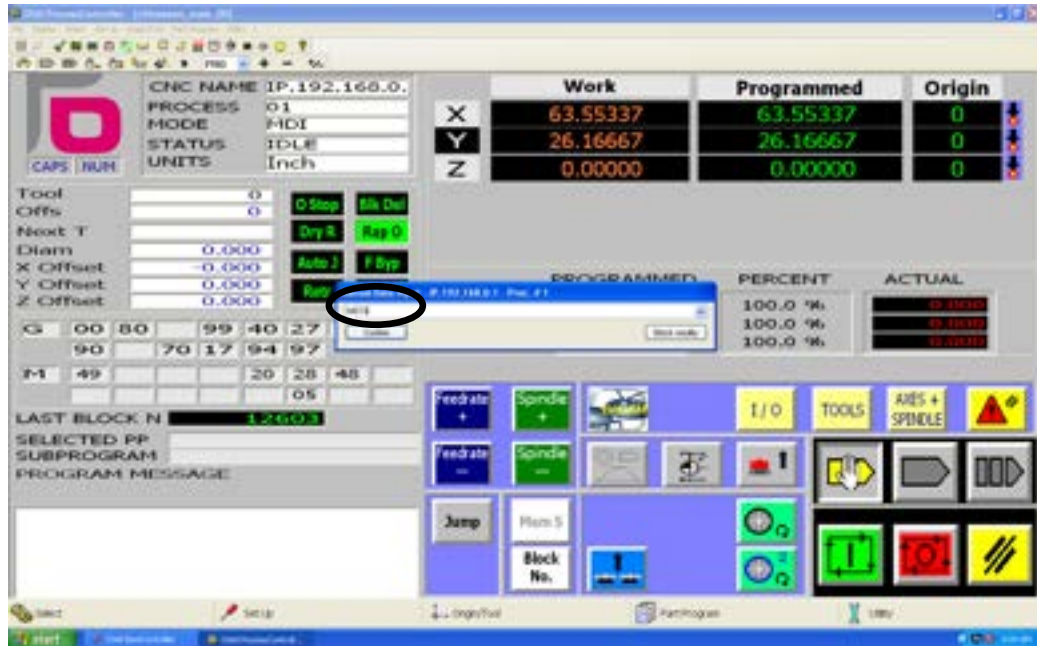
1.) Make sure nobody can be harmed by machine movement.

2.) Click on the MDI button to enable MDI input.

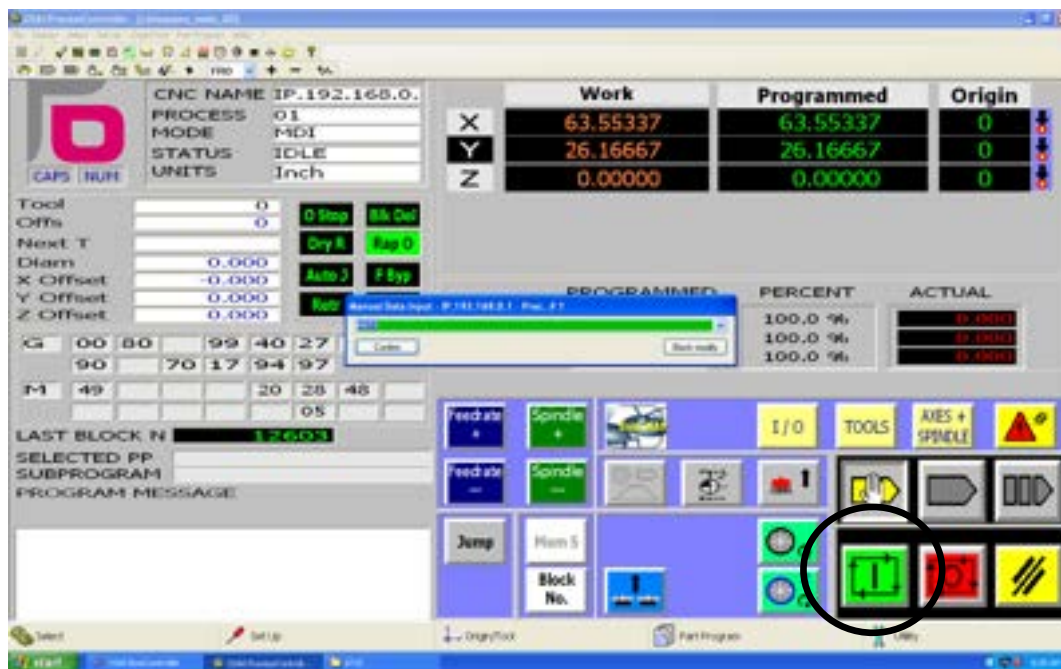



3.) Enter "M6T0" in the dialogue box then press Confirm.

Confirm




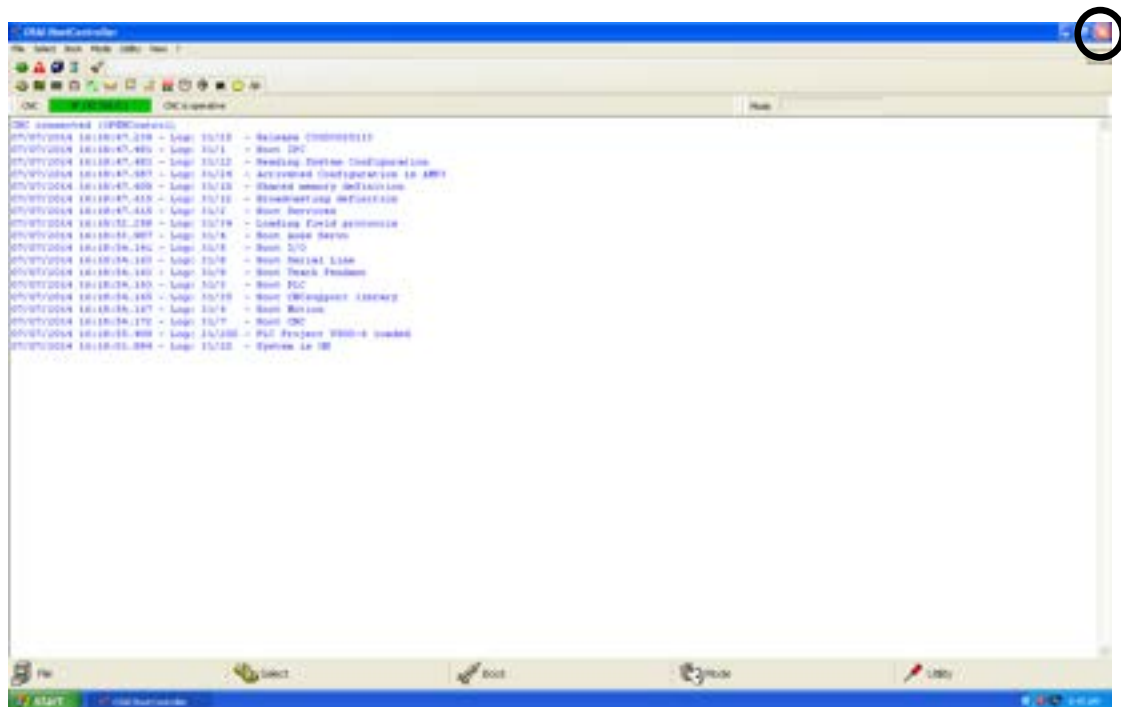
4.) Press the RUN button, the machine will put away the tool in spindle. Putting away the tool in spindle at the end of the day prevents the spindle from rusting.



- 5.) Close the process controller window by clicking the “X”  button on the upper right corner of the window.



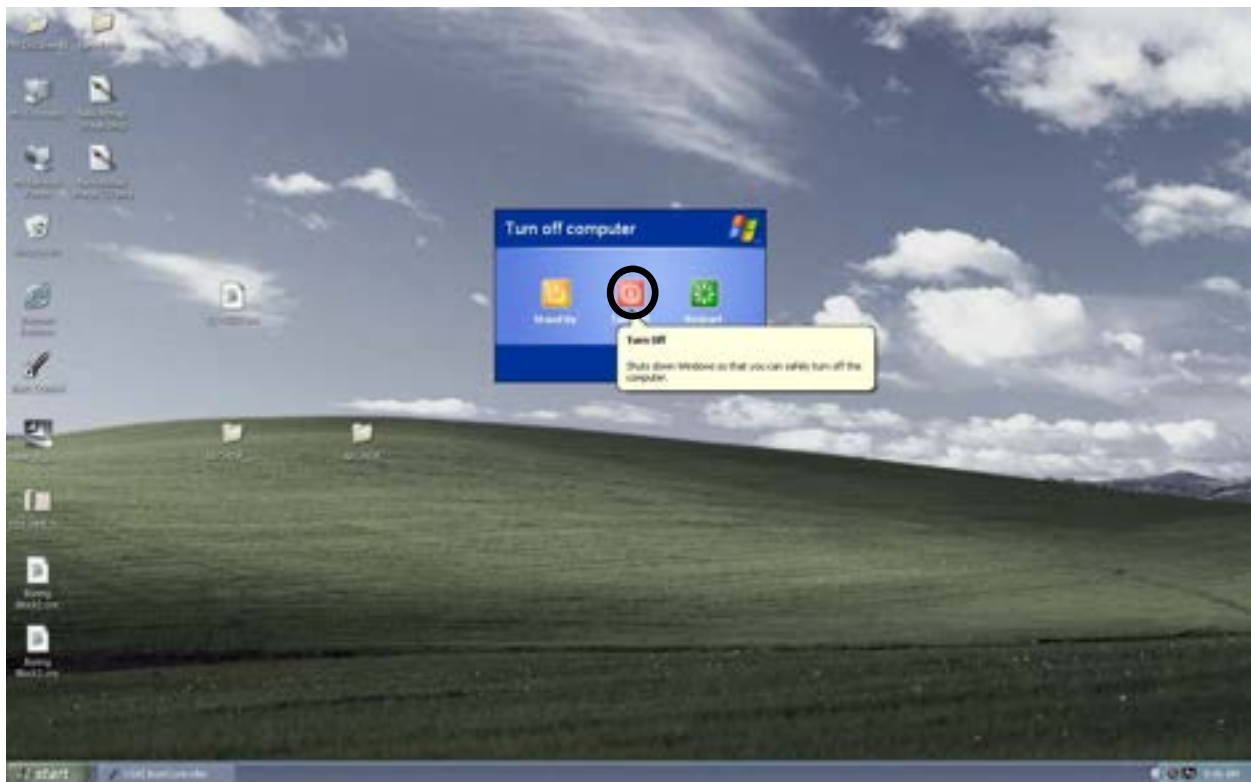
- 6.) Close the boot controller window by clicking the “X”  button on the upper right corner of the window.



7.) Select start -> shutdown



8.) Click on the shutdown button, and then wait for the computer to shut down.



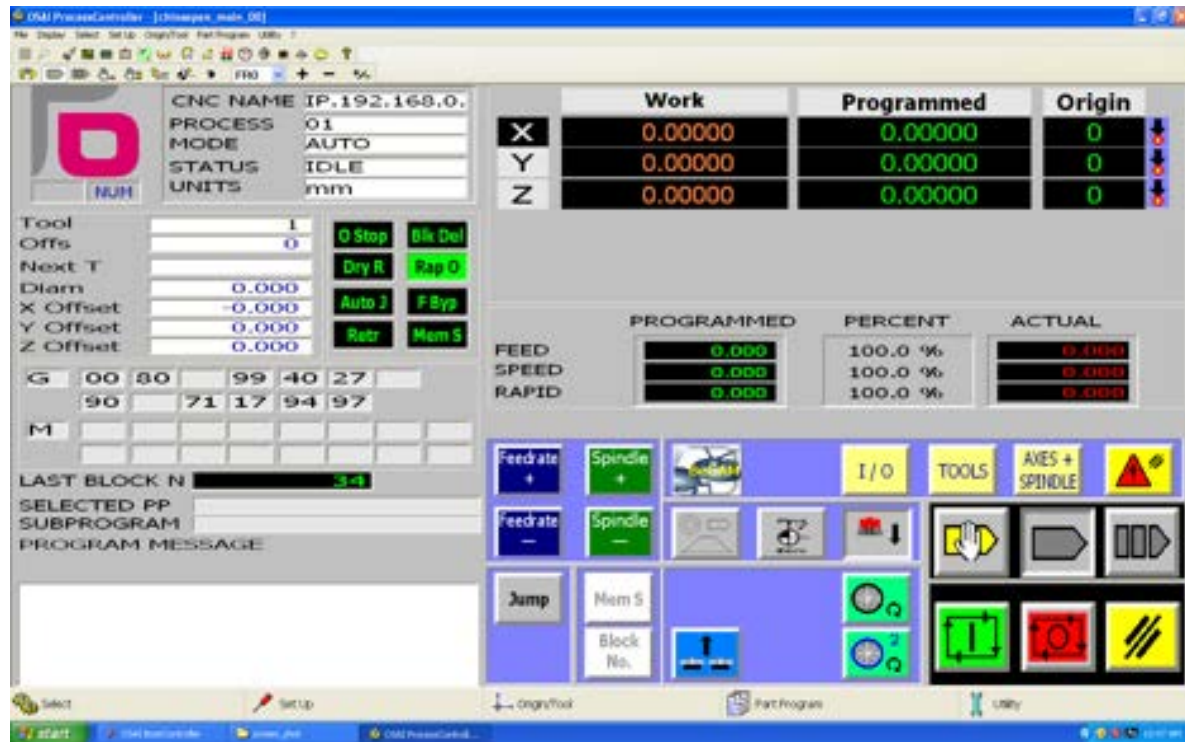
9.) Press the red button to shutdown the controller



10.) When the computer has been shut down, turn off the main contactor by rotating it counter-clockwise.



Main Screen



The main page will load up at machine start up. The sections are explained below.

Top Toolbar section

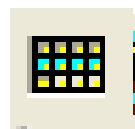
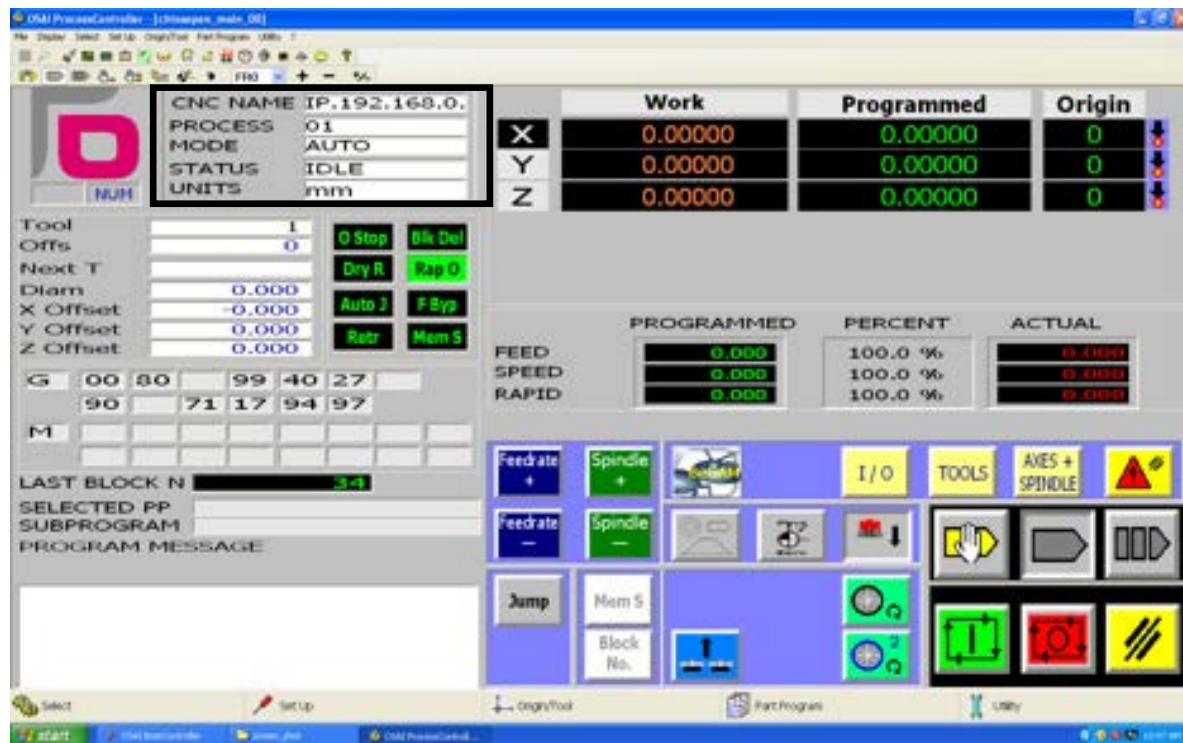


Table editor button – opens the table editor window.

The rest of the buttons are reserved for use by Laguna Tools technicians.

CNC Display Section



CNC NAME	NC0001
PROCESS	01
MODE	AUTO
STATUS	IDLE
UNITS	mm

Here are the description of each fields of display:

CNC NAME – The name of the CNC machine being controlled.

PROCESS – This field is reserved for use by Laguna Tools technicians.

MODE – Whether the machine is in AUTO mode or MANUAL mode. AUTO mode allows the machine to run G-code programs and MANUAL mode allows the user to move the spindle manually.

STATUS – displays the status of the machine

UNITS – Whether the coordinate display is in inches or milimeter. Note that while running a G-Code program, the unit is set with G70 and G71, thus the units displayed here does not override the unit set in the G-Code.

Tool Display Section



Tool	7
Next T	
Diam	0.000
X Offset	0.000
Y Offset	0.000
Z Offset	0.000

This section displays information regarding the tools. The machine can hold 8 tools in the tool changer.

The tools are numbered from 1 through 8. Each tool has its own diameter and X, Y, Z offset data.

Here are the descriptions of each fields of display:

Tool – Tool number of current tool in spindle

Next T – Tool number of the next tool to be picked up according to the active G-Code.

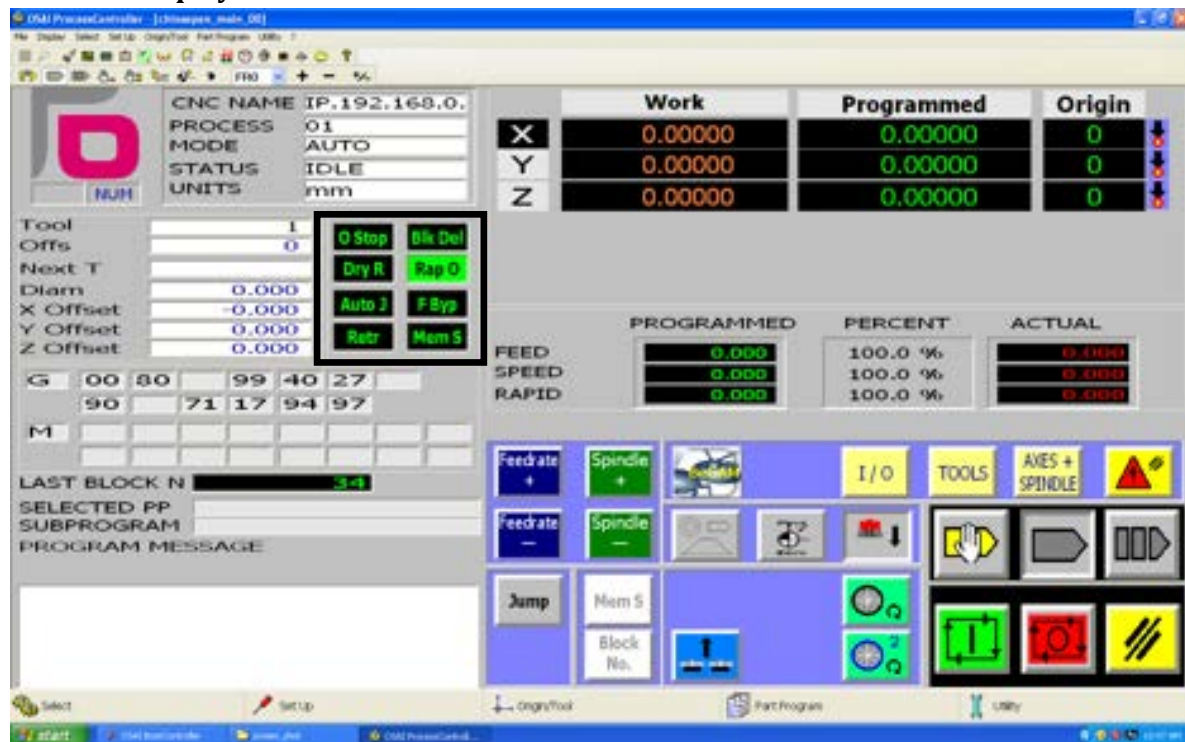
Diam – Diameter of the curent tool.

X Offset – the X offset of the current tool, this is factory set and is only used by Laguna Tools technician.

Y Offset – the Y offset of the current tool, this is factory set and is only used by Laguna Tools technician.

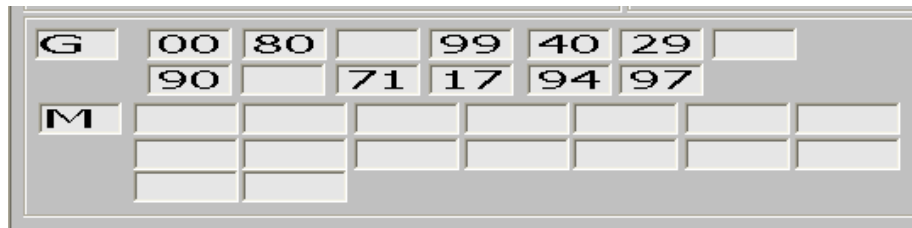
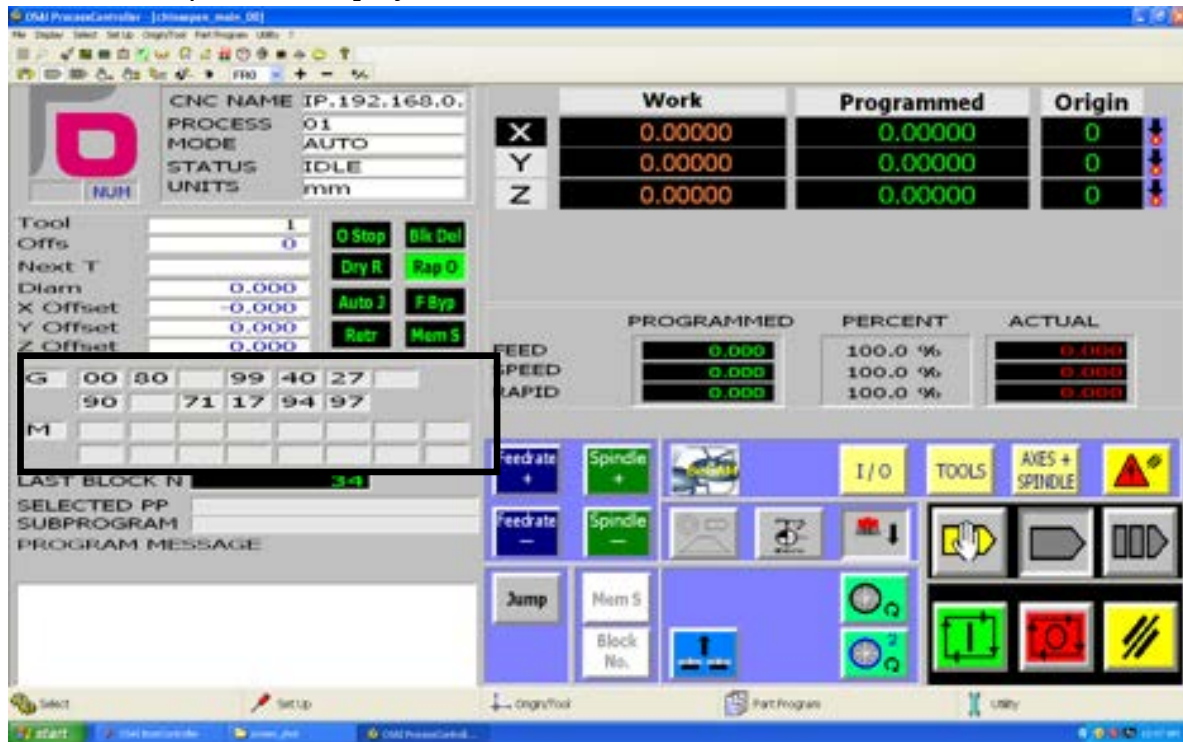
Z Offset – the Z offset of the current tool, this is the tool length compensation for the current tool.

Function Display Section



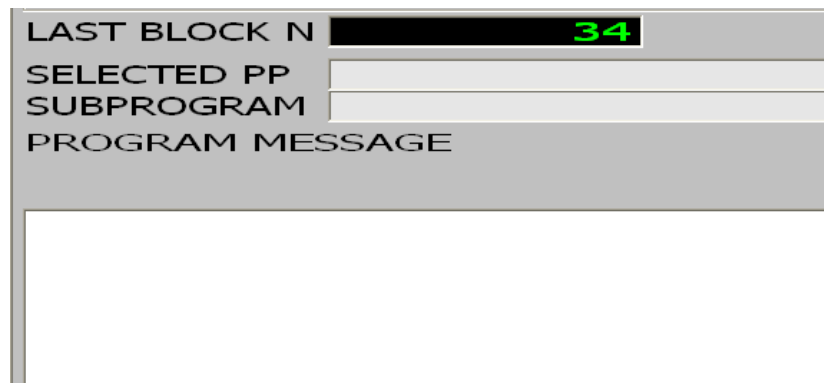
This section displays whether each machine functions are on or off. They are reserved for use by Laguna Tools technicians.

Modal G-Code/M-Code display section



This section displays the modal G-Code and M-Code currently active. Please refer to “Appendix, G-Code List”, and “Appendix, M-Code List” for detail.

Active G-Code Program Section



This section displays the active G-Code.

Below is a description of each display:

Last Block N – Number of the last block ran.

Selected PP – Reserved for use by Laguna Tools technicians.

Subprogram – Reserved for use by Laguna Tools technicians.

Program Message – First few lines of the active G-Code.

Coordinate display section



	Work	Programmed	Origin
X	0.00000	0.00000	0
Y	0.00000	0.00000	0
Z	0.00000	0.00000	0

This section displays the programmed position and actual position of the spindle.

When the origin display column displays “0”, the displayed coordinate is in Machine Coordinate

When the origin display column displays “1”, the displayed coordinate is in Work Coordinate 1, which can be activated by the command “(UAO, 1)”.

There are 9 different Work Coordinates which the user can configure, from “(UAO, 1)”, to “(UAO, 9)”. Please see the section “Setting Work Origin” for detail.

Feed and Speed Section



	PROGRAMMED	PERCENT	ACTUAL
FEED	0.000	100.0 %	0.000
SPEED	0.000	100.0 %	0.000
RAPID	0.000	100.0 %	0.000

This section shows the programmed speed, override percentage, and actual speed of the spindle.

The programmed speed is multiplied by the override percentage to result in the actual speed. The machine moves at the actual speed.

The override percentage can be changed using the override buttons which will be explained in more detail in the next section.

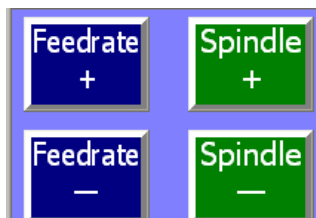
Here is a description of each display:

FEED– The feed rate, the speed of the machine when making a G01 move.

SPEED – The spindle speed, the rotational speed of the spindle in RPM.

RAPID- The rapid speed, the speed of the machine when making a G00 move.

Override Buttons

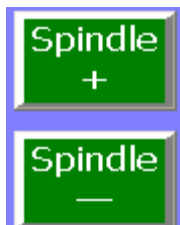


The override buttons allows the user to adjust the feed rate and speed override.

Below are the descriptions:



- Feed rate button, used to adjust the override percentage of the feed rate.

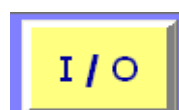


- Spindle button, used to adjust the override percentage of the spindle speed.

Page buttons



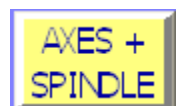
These buttons allows the user to move to different pages of the Process Control Window, and clear messages.



I/O Button – go to I/O page. It lists different input output ports.



Tools Button – go to the tool page.

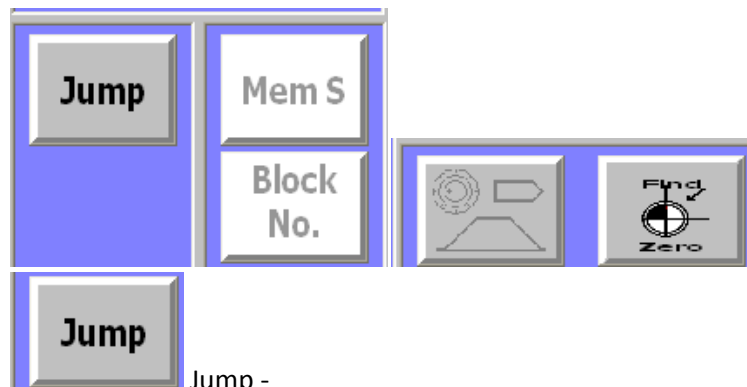


Axis+Spindle Button – goes to the axis page which allows manual jog.



Clear Message Button – used to clear messages in the yellow message box. An message can only be cleared if the cause of the message had be alleviated.

Function Buttons



Jump -



Memory search button – used to return to the last line of G code executed after unexpected system shutdown. For example, a power outage.



Block number button – used for run from a specific line number.

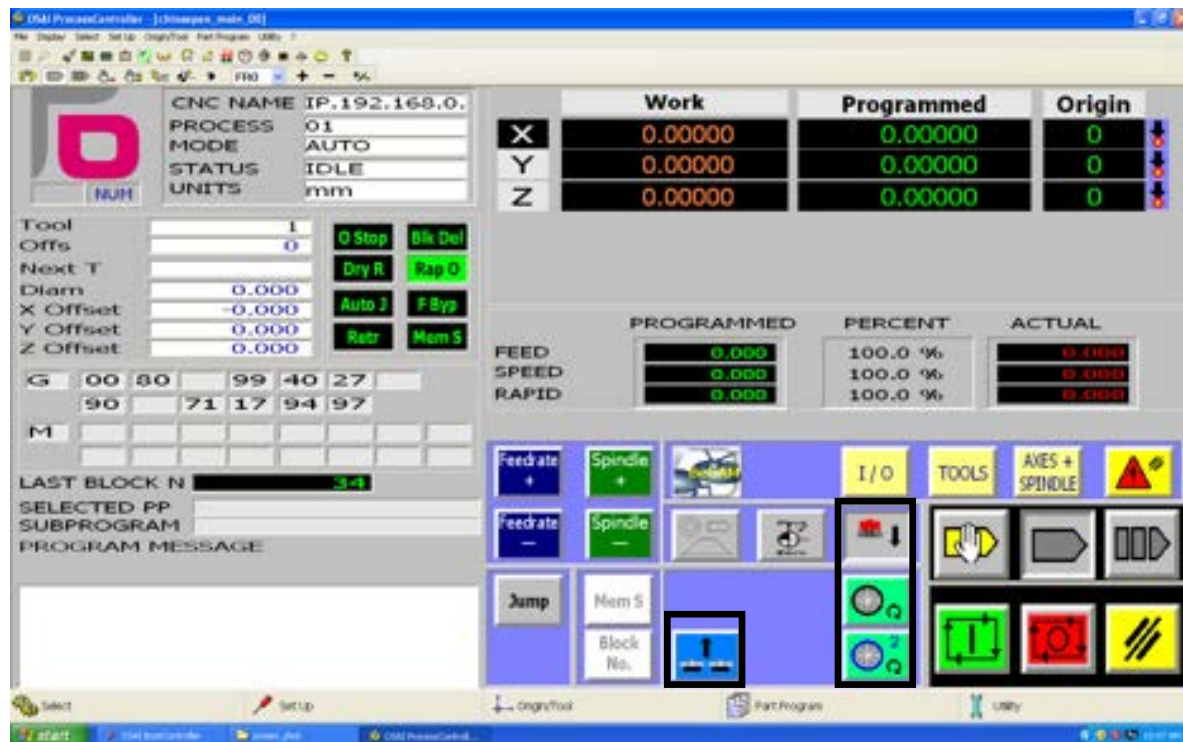


MPG Run Mode – When running G-Code with this button the feed rate will depend on the speed in which the user rotates the MPG main wheel. It is used to test out a G-Code slowly while allowing the user to pause.



Find Zero Button – AKA, the home button. When pressed, machine will attempt to home the machine.

Pneumatics Control Buttons



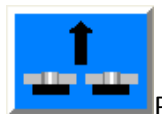
Dust hood control – lifts and drops the dust hood.



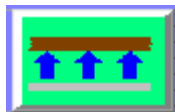
Vacuum button – turns the first vacuum pump on and off.



Vacuum 2 button – turns the second vacuum pump on and off.

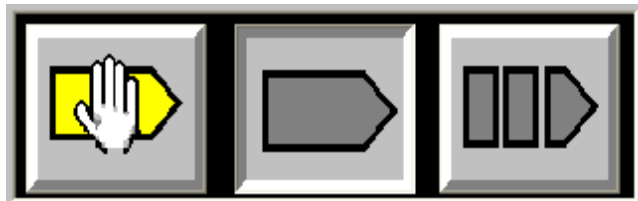


Pop-Up Pin Button – raise and lowers the pop-up pins used to locate the work piece.



Vacuum table on (Optional) - There might be a Vacuum table on button if your machine is fitted with a butterfly valve for the vacuum table. This function allows the vacuum table to be turned off without turning off the vacuum pump. Please depress this button to activated when you want the Vacuum table to be on.

Operation Mode Selection Button



MDI – used to input command manually.

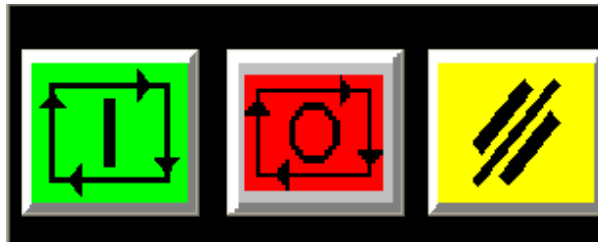


Auto mode button – auto mode allows continuous G code as well as MPG Run mode when the MPG button is pressed.



Block by Block Mode – runs G-code one line at a time.

Operation Buttons



Run Button – used to start running a G code



Pause Button – used to pause a G code while it is running. To continue running, press the pause button again to unpress it, then press the Run button again.



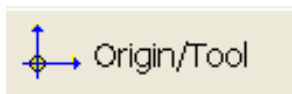
Reset Button – used to stop a G code while it is running and reset.

Bottom Toolbar

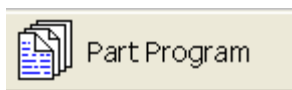


Only two buttons on this tool bar is used, the other buttons are reserved for used by Laguna Tools technicians.

Below are the descriptions for the two buttons:

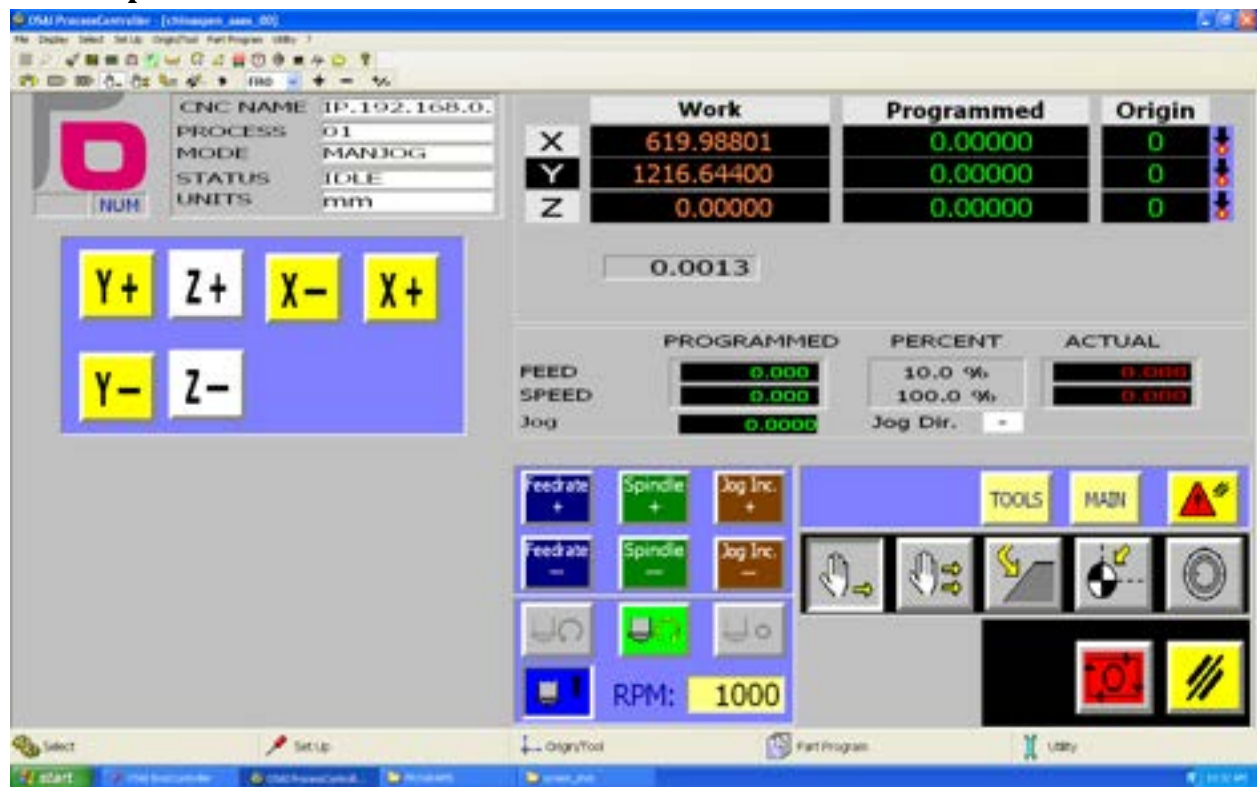



Origin/Tool button – Used to set work origin. Please see the section “Setting Work Origin” for detail.



Part Program – Used to select a G-code program. Selected program will be shown in the Active file section.

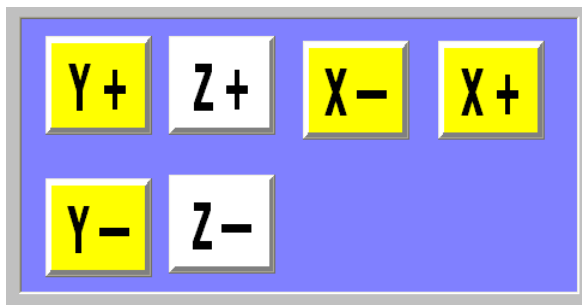
Axis + Spindle Screen



The Axis + Spindle Screen can be accessed by pressing the Axis + Spindle button  from the main screen and the tool screen.

Here are the descriptions on the different parts of this screen.

Axis Buttons



Axis buttons, used to move the axis in continuous jog mode or step jog mode. Please read “Continuous Jogging”, and “Step Jogging” sections for detail. There might be additional buttons if your machine has additional axis.

Feed and Speed Section



	PROGRAMMED	PERCENT	ACTUAL
FEED	0.000	10.0 %	0.000
SPEED	0.000	100.0 %	0.000
Jog	0.0000	Jog Dir. -	0.000

This section shows the programmed speed, override percentage, and actual speed of the spindle.

The programmed speed is multiplied by the override percentage to result in the actual speed. The machine moves at the actual speed.

The override percentage can be changed using the override buttons which will be explained in more detail in the next section.

Here is a description of each display:

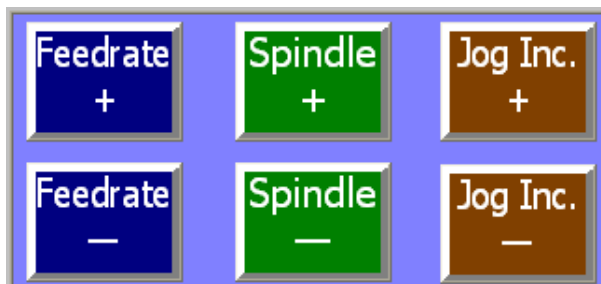
FEED– The feed rate, the speed of the machine when making a G01 move.

SPEED – The spindle speed, the rotational speed of the spindle in RPM.

Jog – The step jog increment, the distance an axis will move when the axis is clicked in step jogging mode.

Jog Dir – The direction in which the machine is currently moving.

Speed/Distance Setting

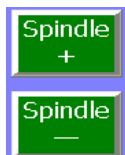


The override buttons allows the user to adjust the feed rate and speed override.

Below are the descriptions:



- Feed rate button, used to adjust the override percentage of the feed rate.

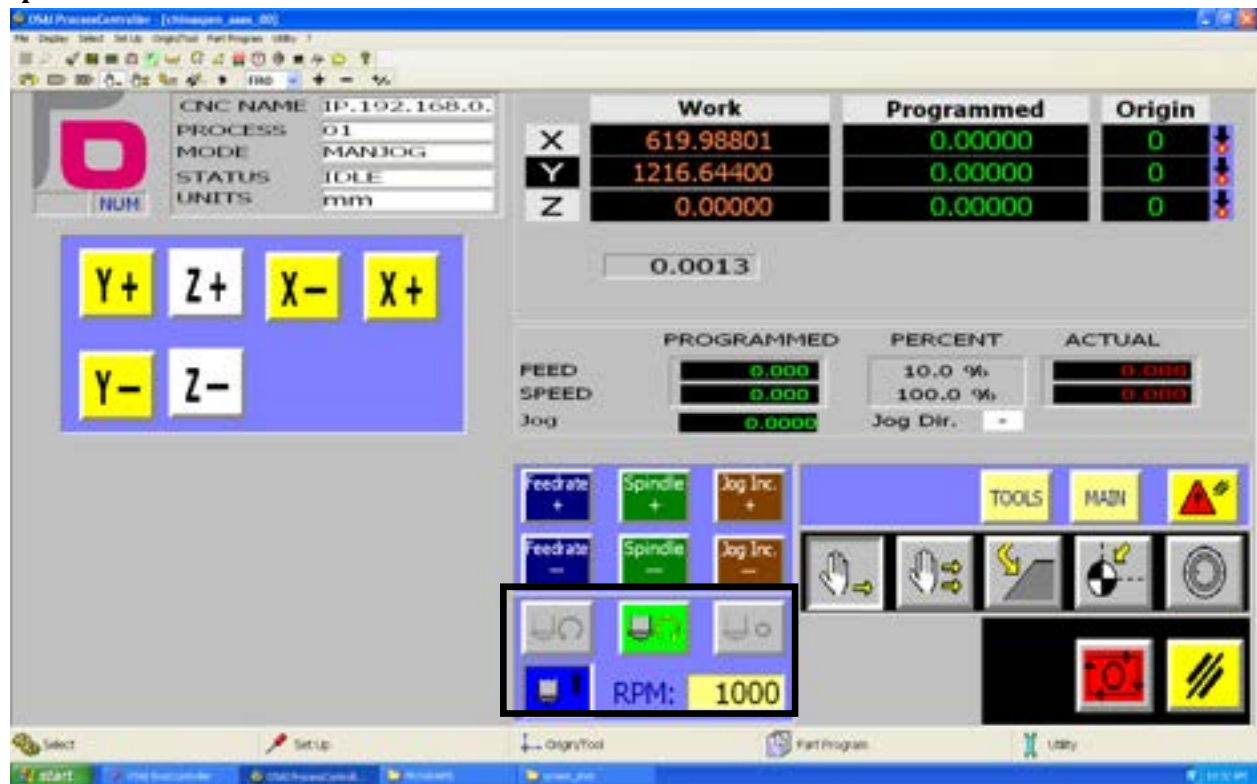


- Spindle button, used to adjust the override percentage of the spindle speed.



Jog increment control – controls the step jog increment, that is the distance an axis will travel in step jog mode.

Spindle Control



These buttons are used to control the spindle, below is the description:



Spindle Reverse Run – Not in use.



Spindle Forward Run – Turns on the spindle to preset RPM.



Spindle Stop – Stops the spindle when it is on.

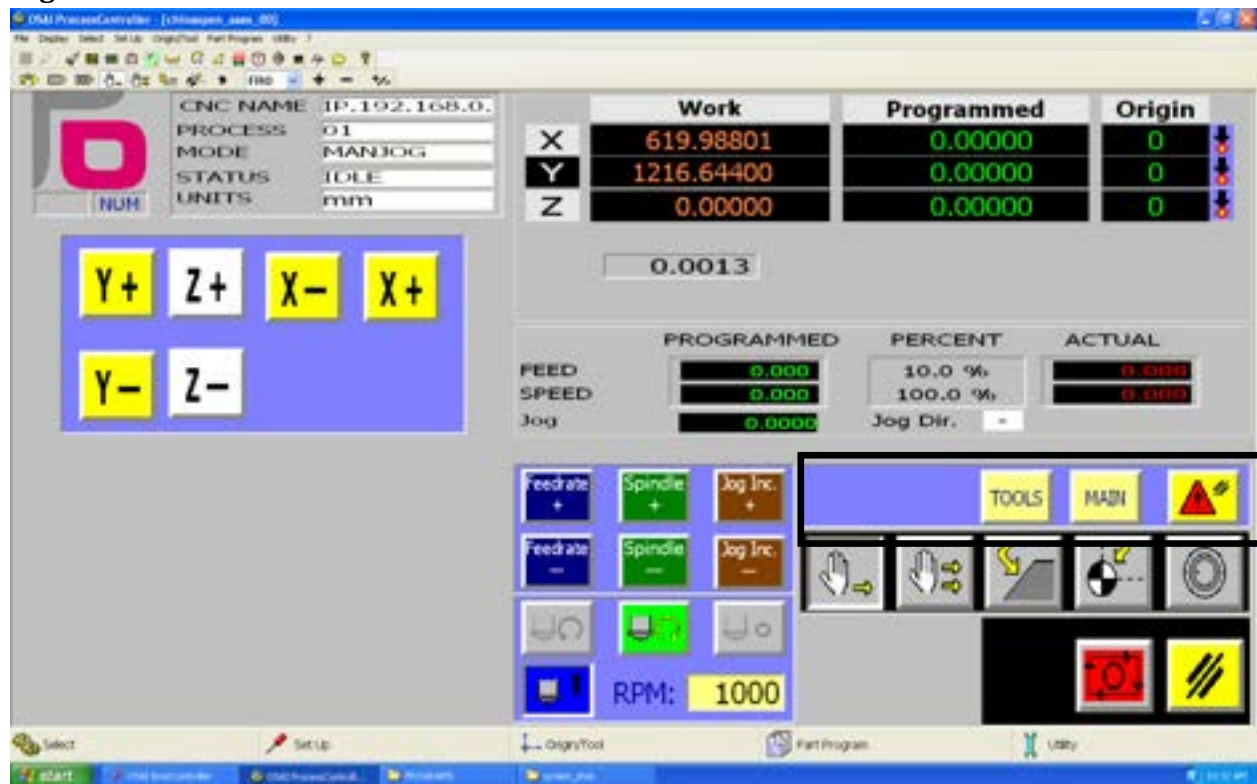


Spindle Up – Raises and drops the spindle. The spindle needs to be dropped when machining using the spindle and it needs to be raised when using the drill block.



RPM Setting – Revolution per minute; this is used to set the spindle speed when turning on the spindle manually.

Page Buttons



These buttons allows the user to move to different pages of the Process Control Window, and clear messages.



Tools Button – go to the tool page.



Main Button – go to the Main page.



Clear Message Button – used to clear messages in the yellow message box. An message can only be cleared if the cause of the message had be alleviated.

Jogging Mode Buttons



Continuous jog mode button – used to put the machine into continuous jog mode.



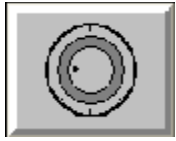
Step jog mode button – used to put the machine into step jog mode.



Button not in use - Reserved for use by Laguna Tools technicians.

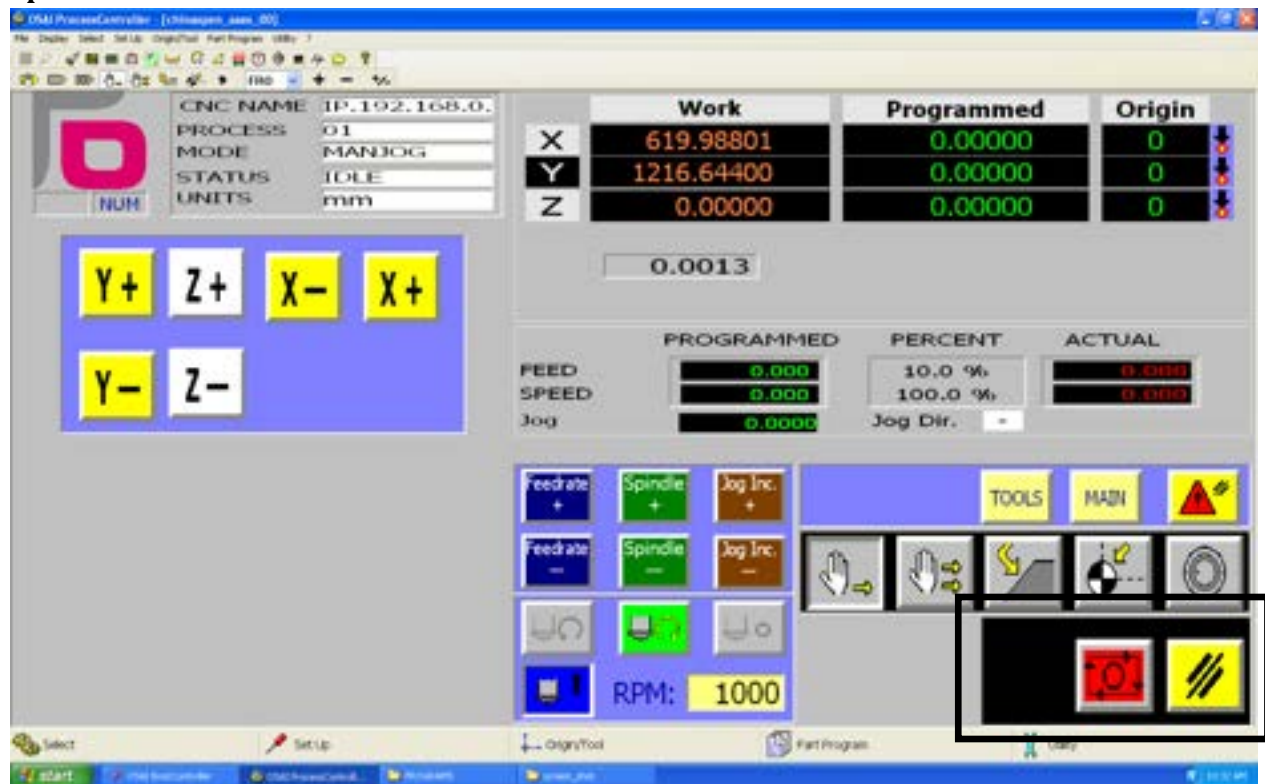


Single axis home button – Used to put the machine into single axis home mode. When in this mode, press the desired axis button to home it.



MPG – Manual Pulse Generator; used to put the machine into hand wheel control mode.

Operation Buttons

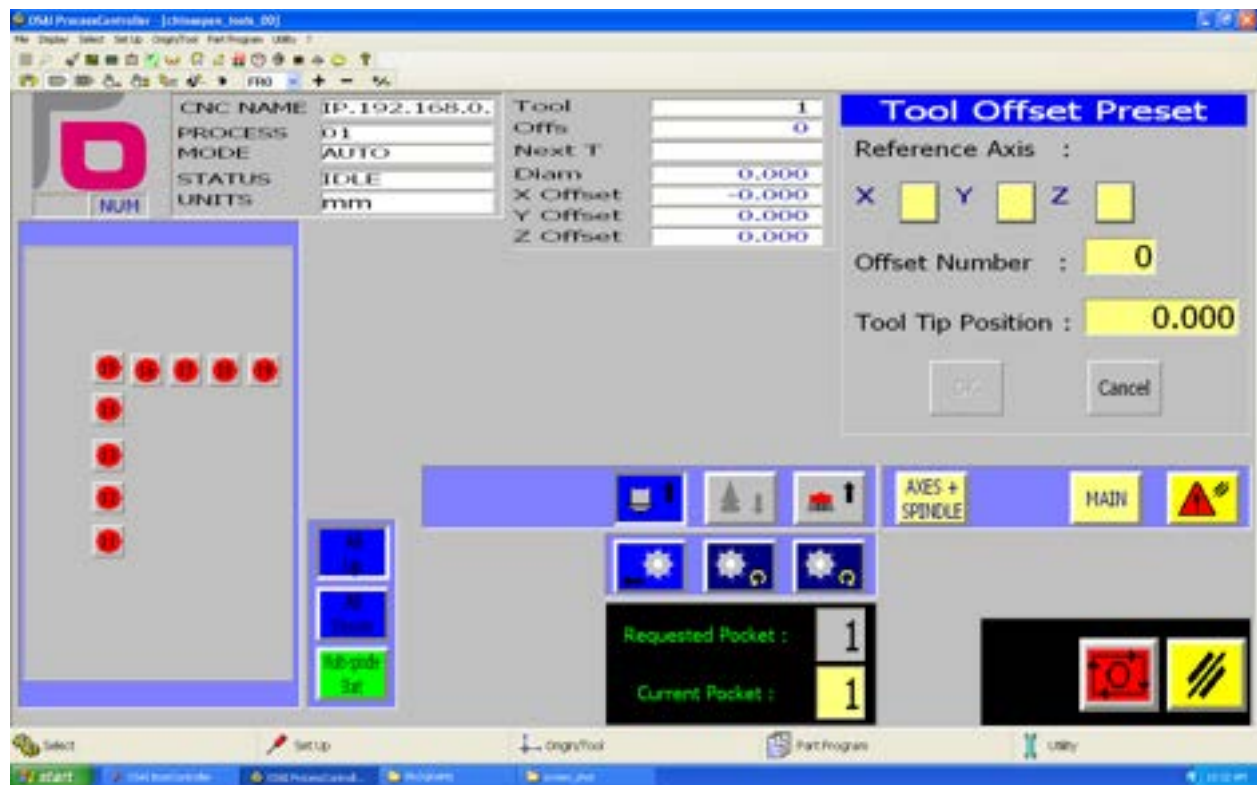


Pause Button – used to pause a G code while it is running. To reset, click the reset button.

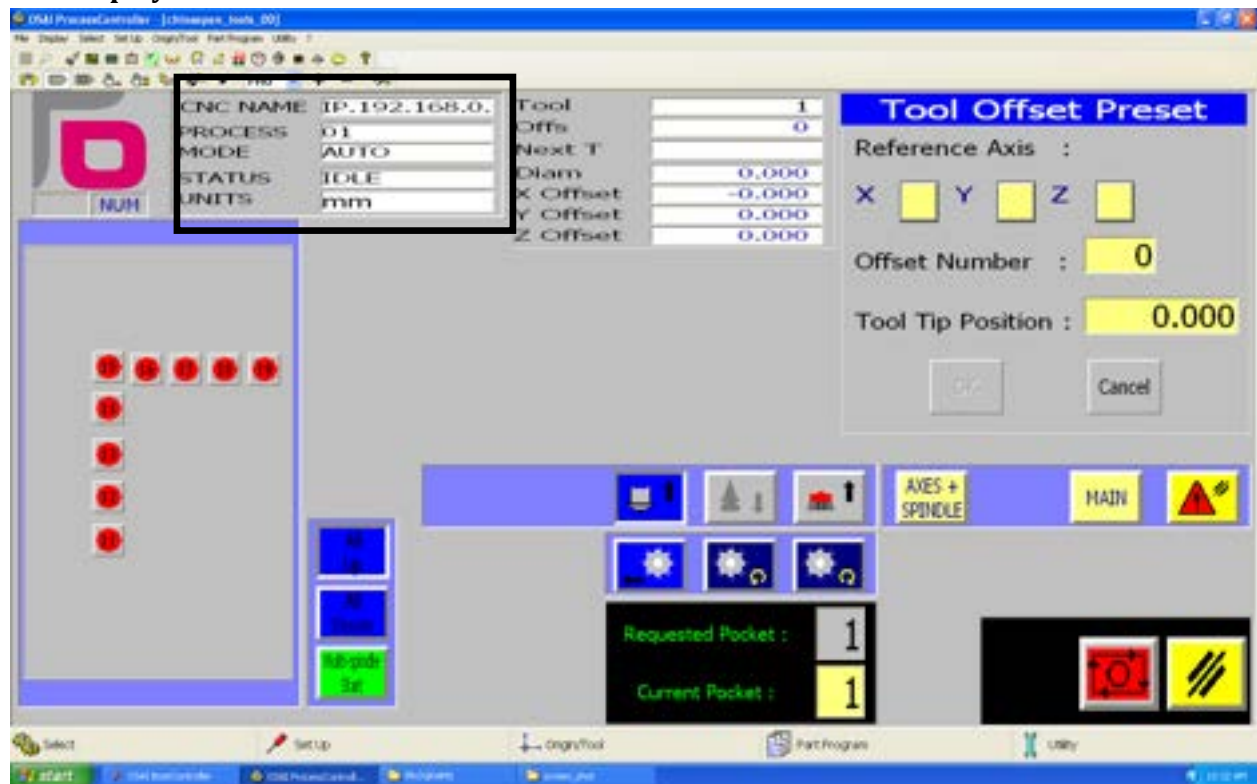


Reset Button – used to stop a G code while the system is moving.

Tool Screen



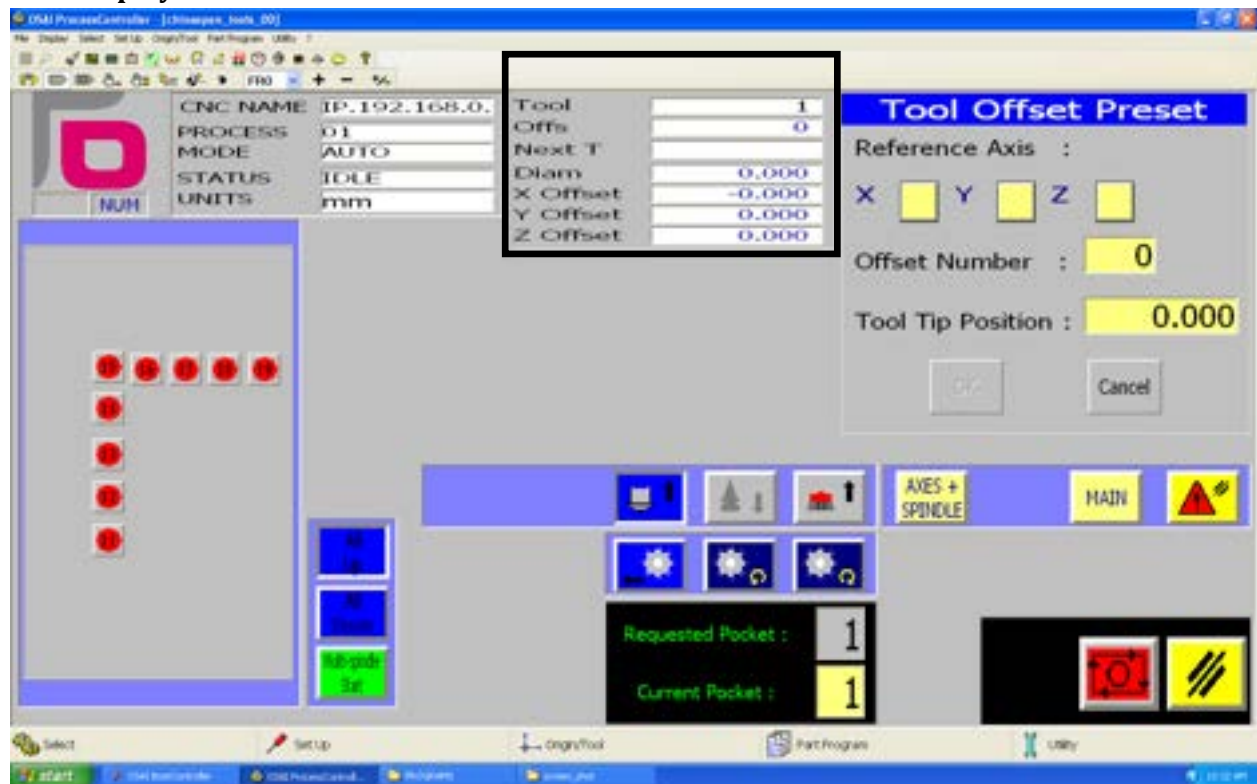
CNC Display Section



CNC NAME	IP.192.168.0.
PROCESS	01
MODE	AUTO
STATUS	IDLE
UNITS	mm

Please see "Error! Reference source not found. CNC Display Section" for reference.

Tool Display Section



Tool	1
Offs	0
Next T	
Diam	0.000
X Offset	-0.000
Y Offset	0.000
Z Offset	0.000

This section displays information regarding the tools. The machine can hold 8 tools in the tool changer.

The tools are numbered from 1 through 8. Each tool has its own diameter and X, Y, Z offset data.

Here are the descriptions of each fields of display:

Tool – Tool number of current tool in spindle

Offs – The tool number whos offset is used.

Next T – Tool number of the next tool to be picked up according to the active G-Code. This only functions when the G-Code is running.

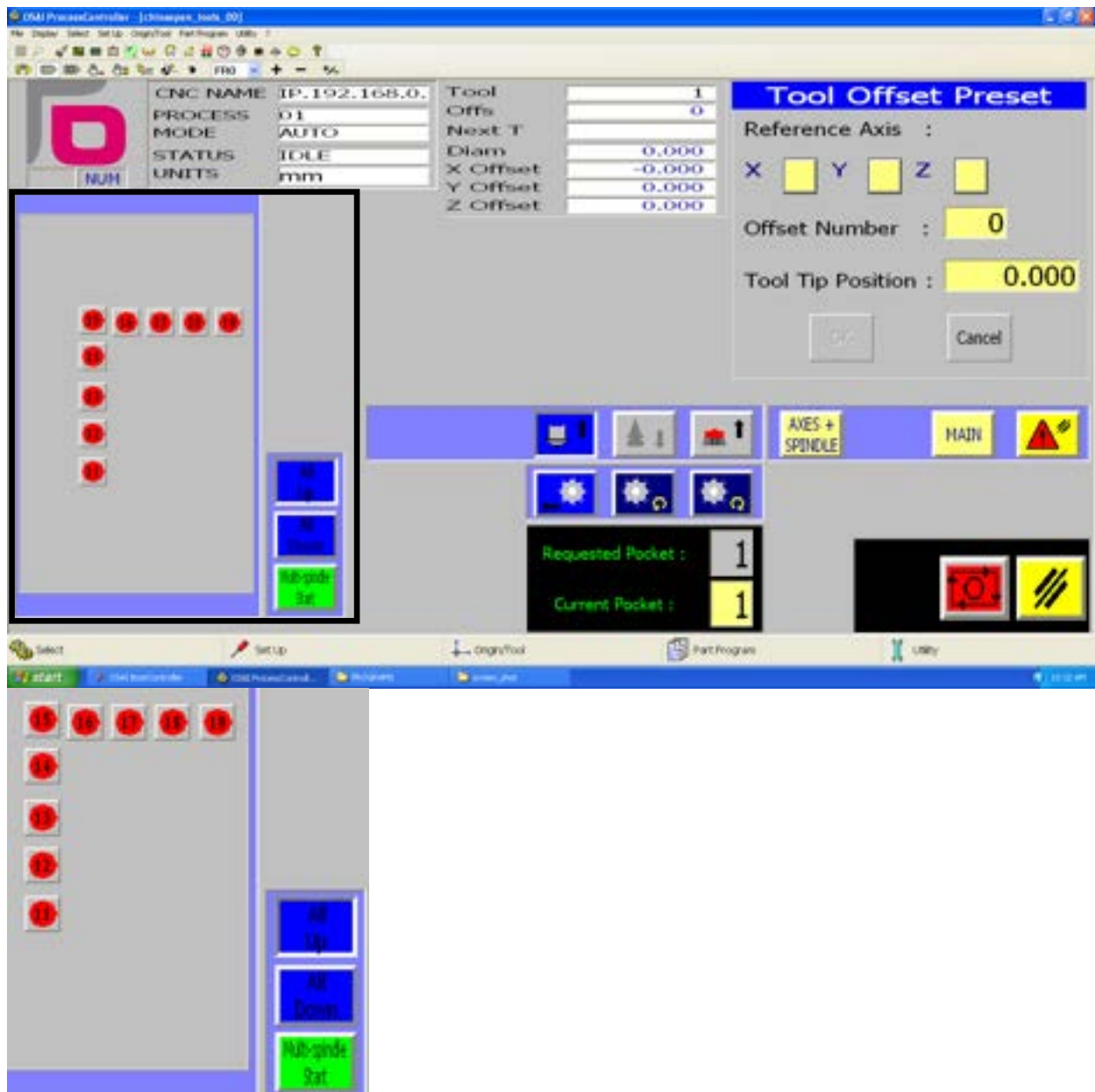
Diam – Diameter of the curent tool.

X Offset – the X offset of the current tool, this is factory set and is only used by Laguna Tools technician.

Y Offset – the Y offset of the current tool, this is factory set and is only used by Laguna Tools technician.

Z Offset – the Z offset of the current tool, this is the tool length compensation for the current tool.

Drill Block Buttons



Note that some machine might not have the drill block, thus will not have these buttons.



11 through 19 - Single Drill Actuation Button – raise and drops a single drill bit on the drill block. Click once to drop a drill bit, click again to raise it.



All Up Button – Raises all of the drill bits on the drill block.

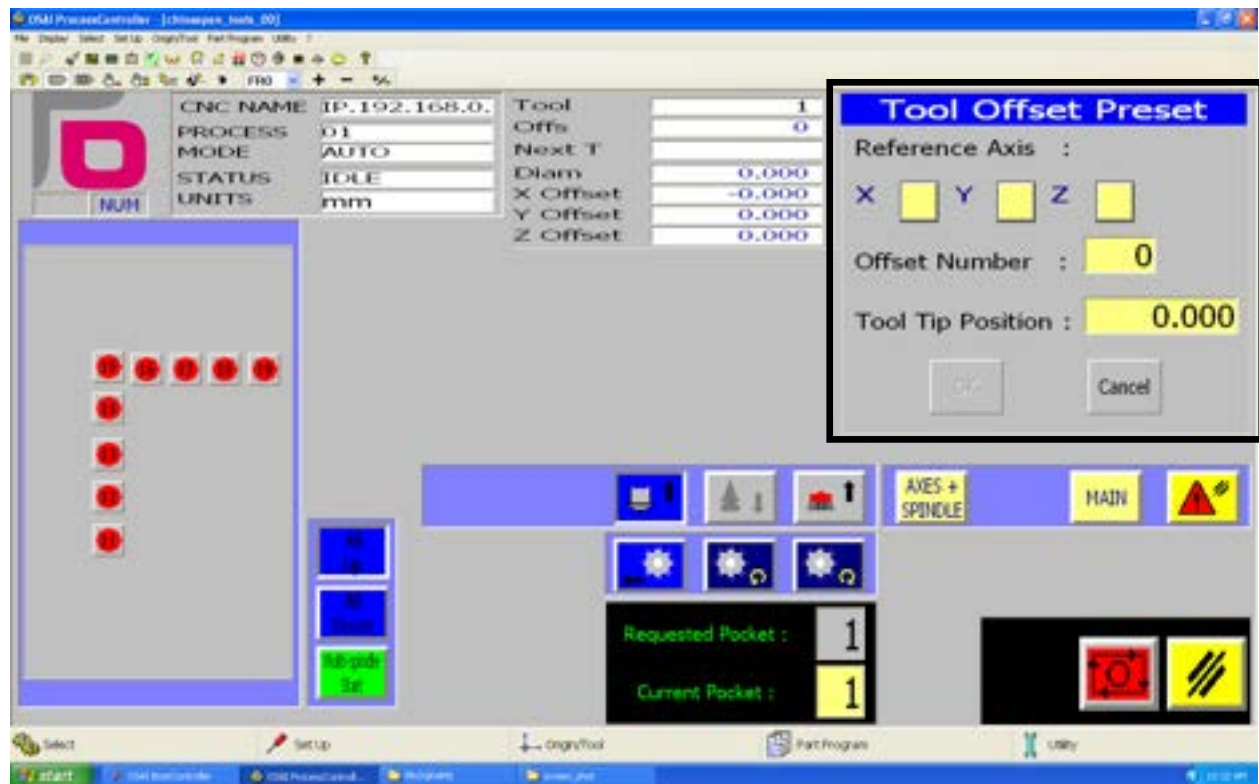


All Down Button – Drops all of the drill bits on the drill block.



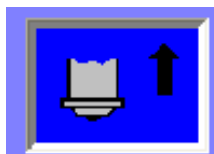
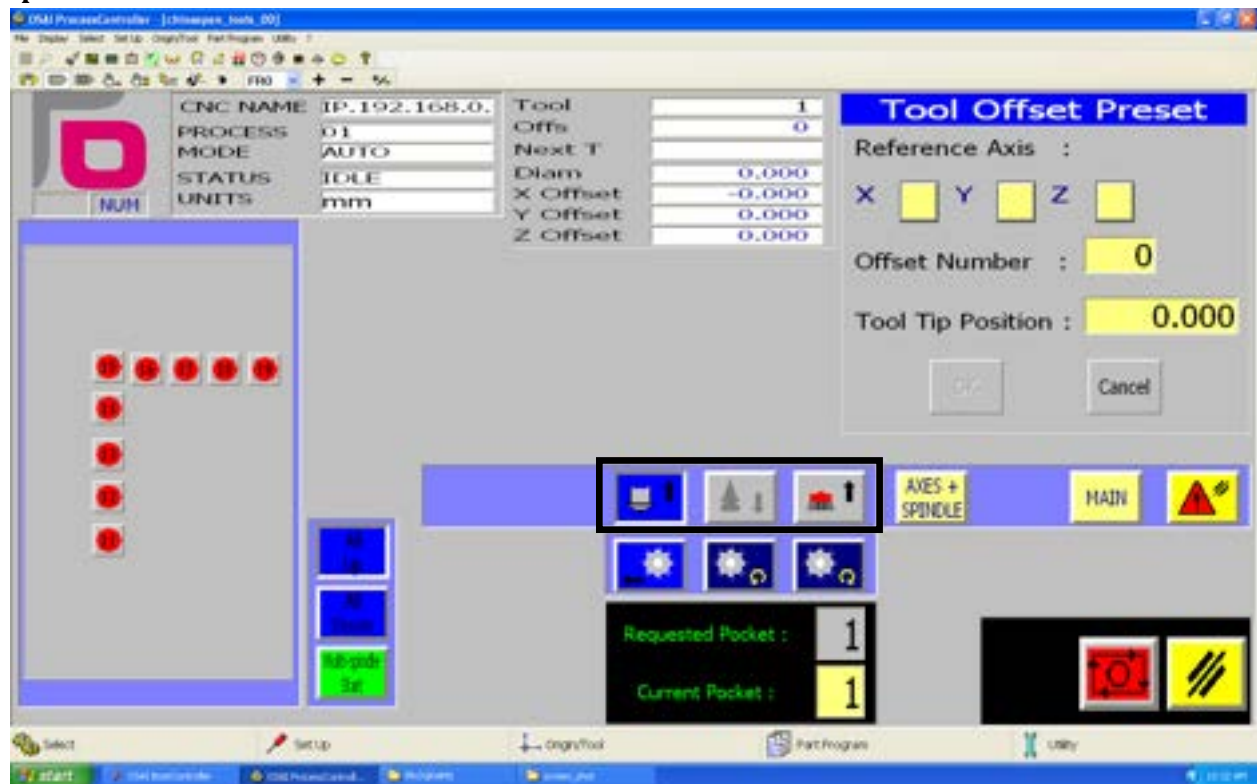
Multi-spindle Start – Turns on the drill block rotation. Press this button again to turn the drill block off.

Tool Offset Preset



This section is used to set tool offset and tool length. Please read the section "Setting Tool Length" for more detail.

Spindle Buttons



Spindle Up – Raises and drops the spindle. The spindle needs to be dropped when machining using the spindle and it needs to be raised when using the drill block.

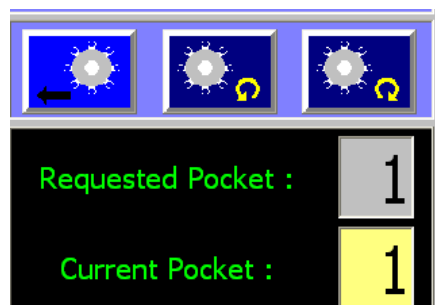
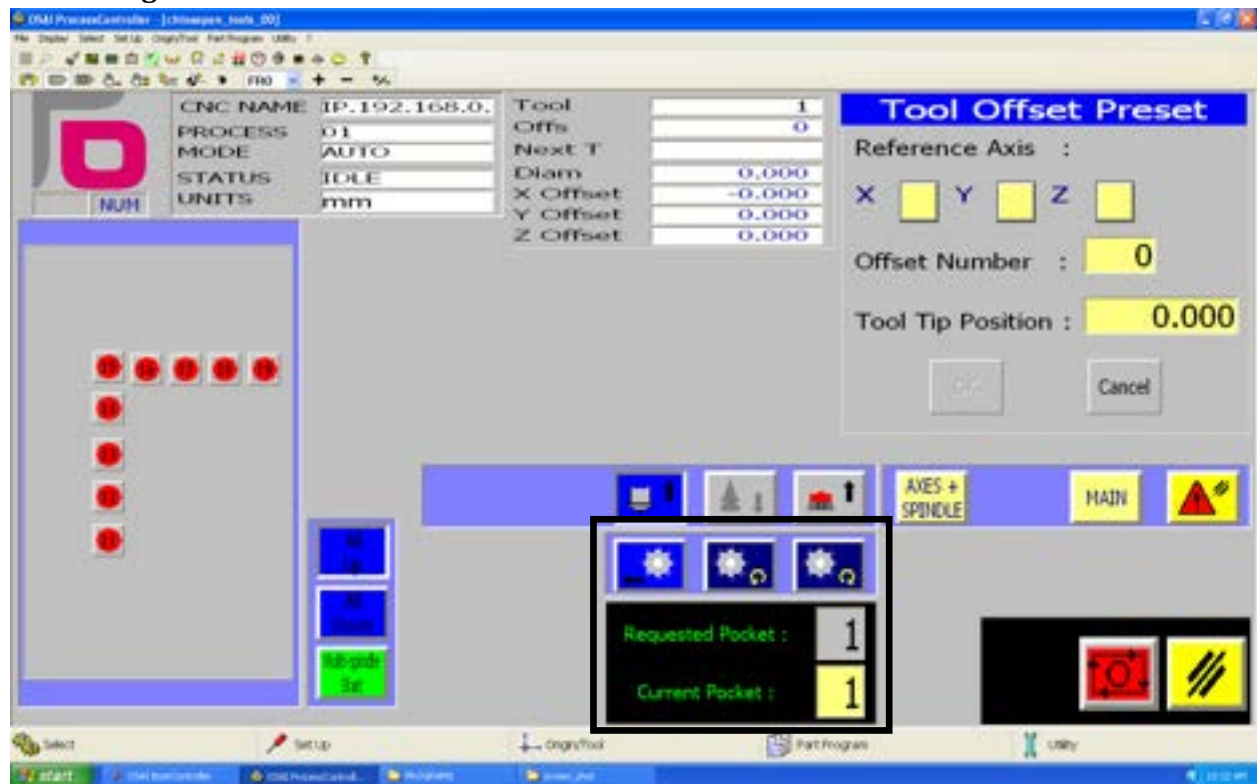


Tool Release – When clicked, the spindle will release the tool it is holding.



Dust hood control – lifts and drops the dust hood.

Tool Changer



Tool Changer Slide – Slides the tool changer.



Tool Changer Rotation (counter-clockwise) – rotate the tool changer counter-clockwise by one increment.



Tool Changer Rotation (clockwise) – rotate the tool changer clockwise by one increment.

Requested Pocket :

1

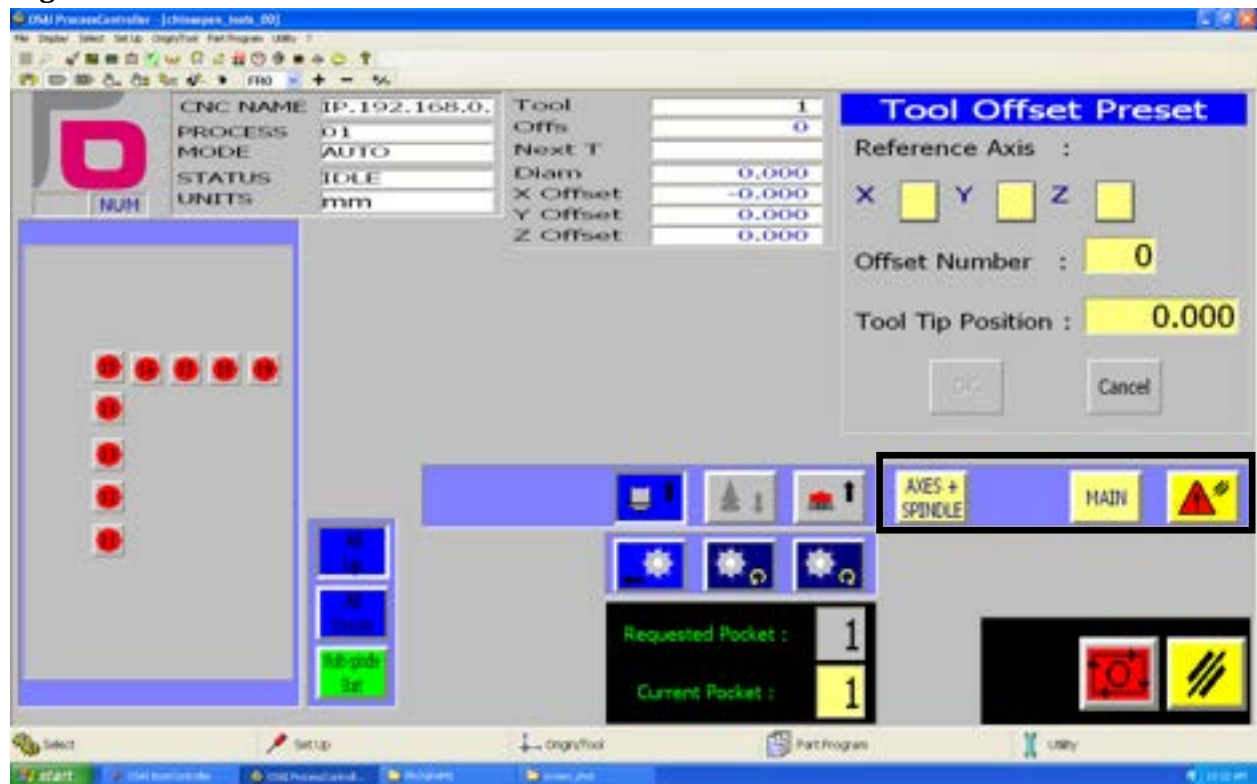
Requested Pocket – pocket requested by the G-Code.

Current Pocket :

1

Current Pocket – The current active tool changer pocket.

Page Buttons



Axis + Spindle Button – go to the axis + spindle page.

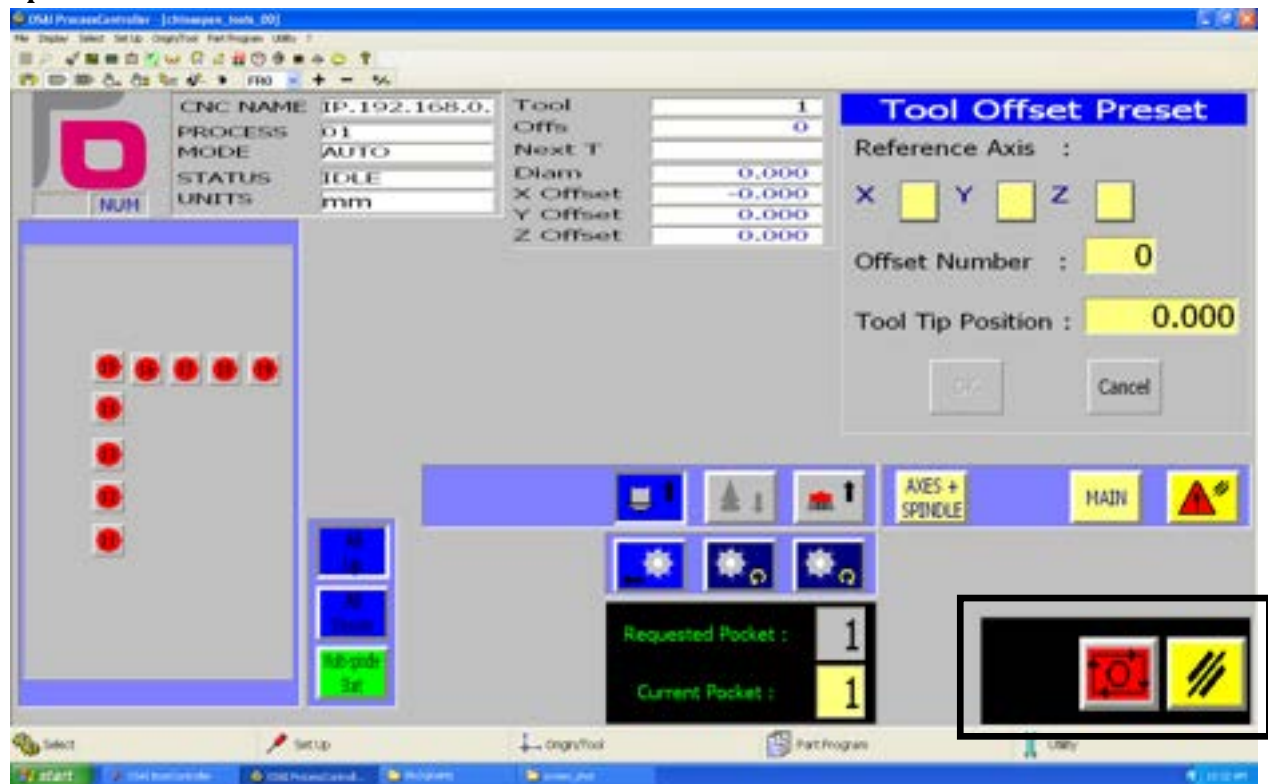


Main Button – go to the Main page.



Clear Message Button – used to clear messages in the yellow message box. A message can only be cleared if the cause of the message had been alleviated.

Operation Buttons



Pause Button – used to pause a G code while it is running. To reset, click the reset button.

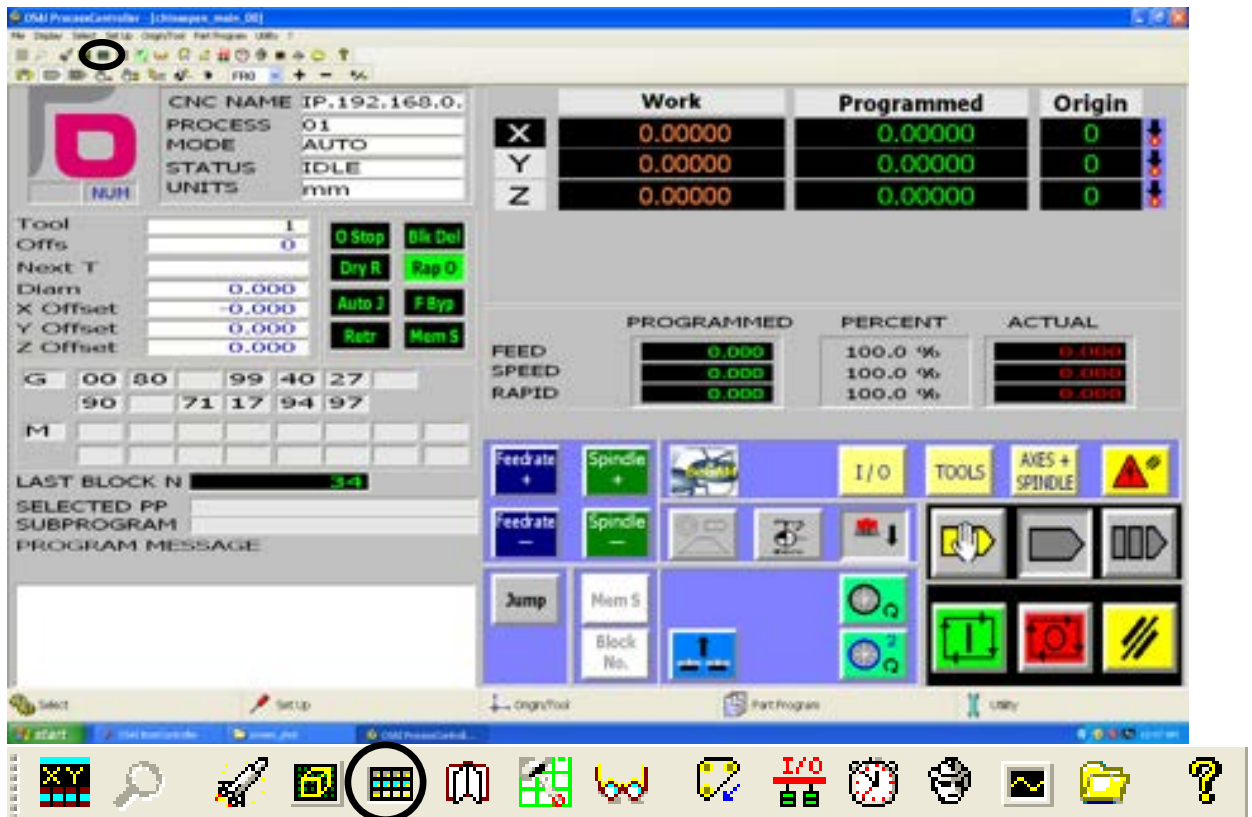


Reset Button – used to stop a G code while the system is moving.

Tool Length Table

The tool length table displays the tool length and offset for a particular tool. Here are the instructions to open the Tool Length Table.

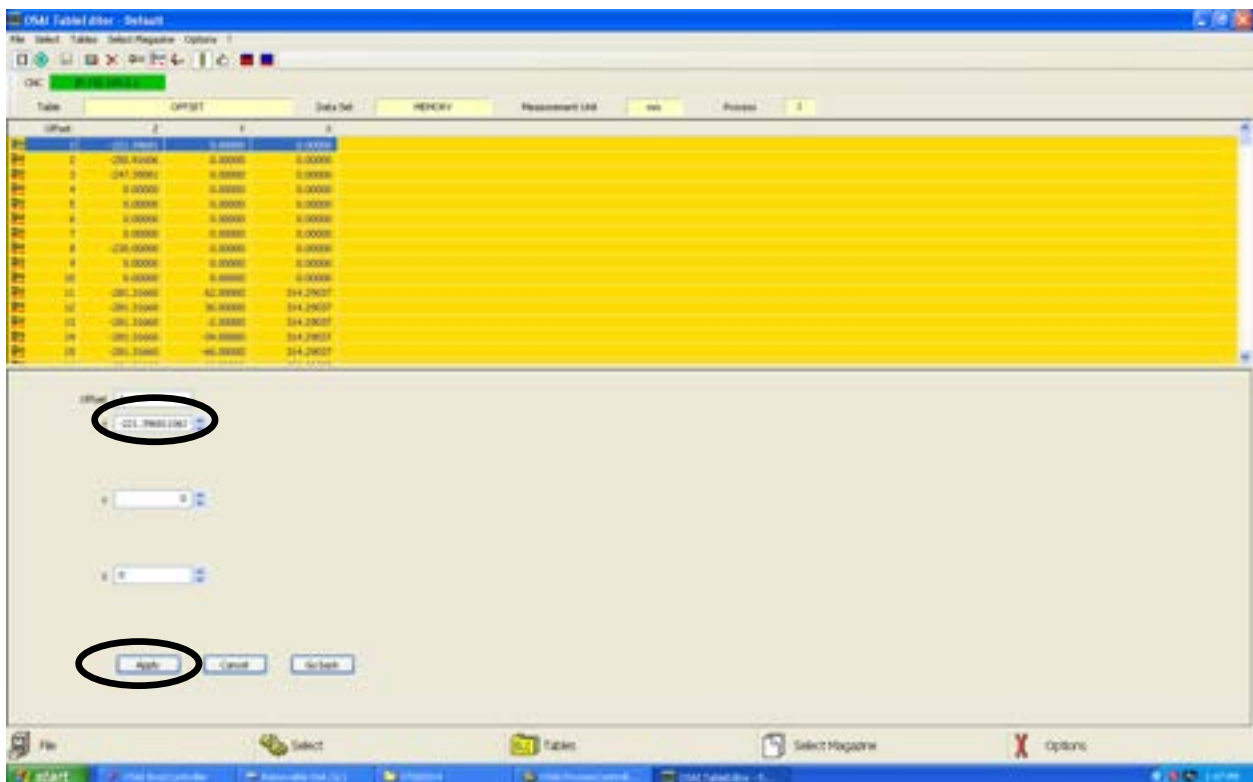
1. From the Main screen, click on the Table Editor button



- Click on the Tool Length Table button.



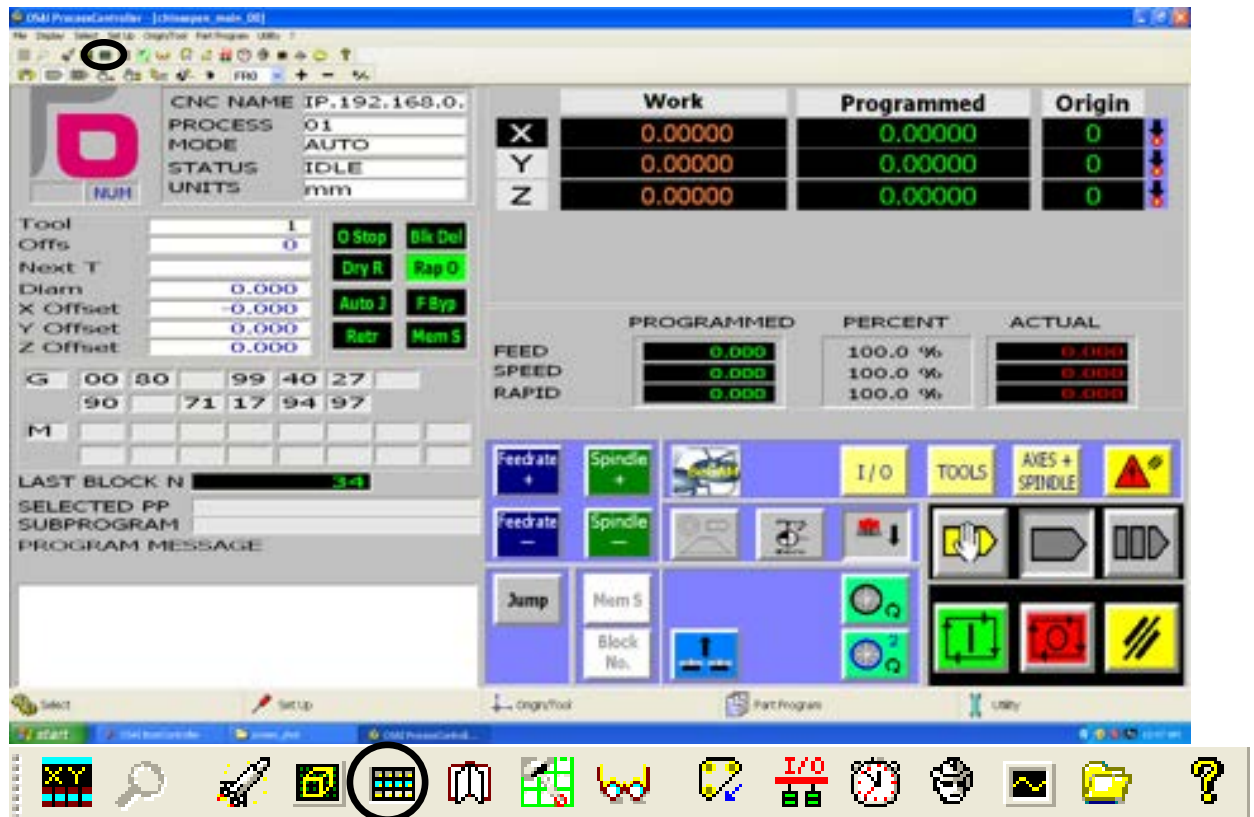
- Click on the desired tool and enter number in the desired field, then press apply to apply the change.



Work Origin Table

The work origin table displays the X,Y, Z offset for a particular tool. Here are the instructions to open the Work Origin Table.

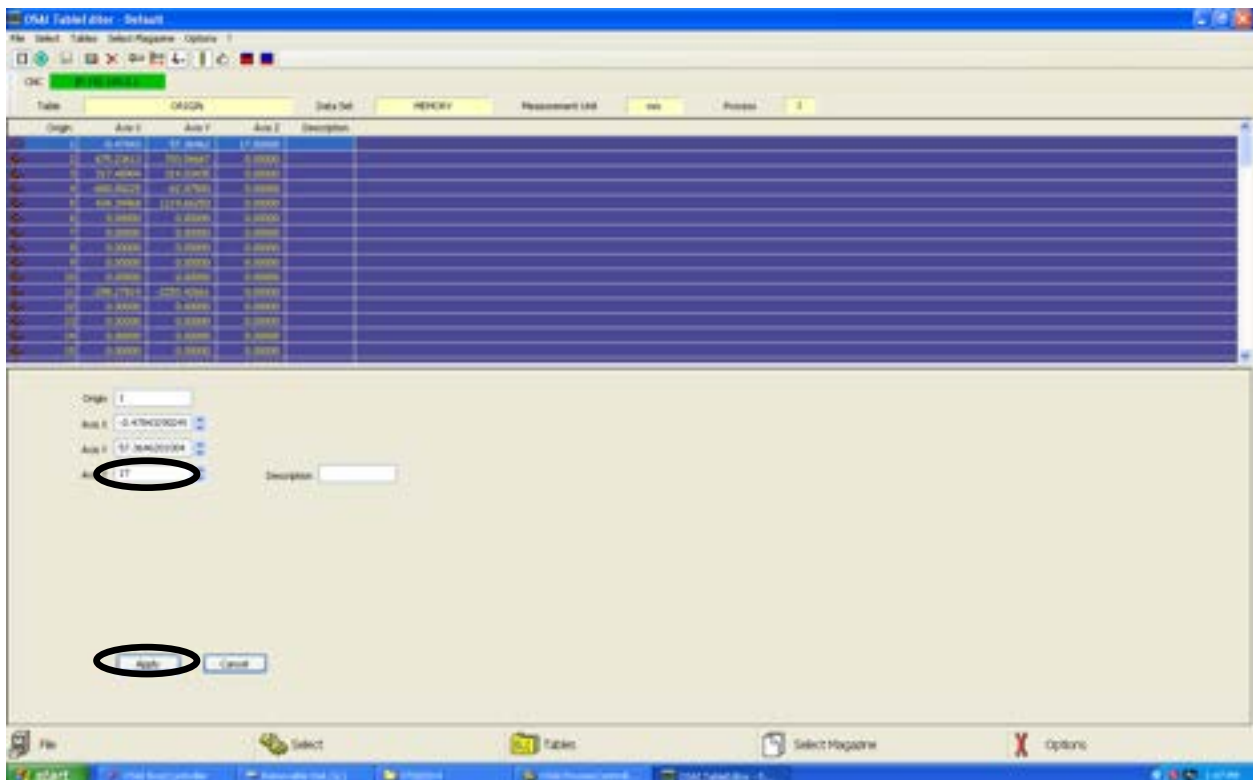
1. From the Main screen, click on the Table Editor button



- Click on the Work Origin Table button.



- Click on the desired work origin and enter number in the desired field, then press apply to apply the change.



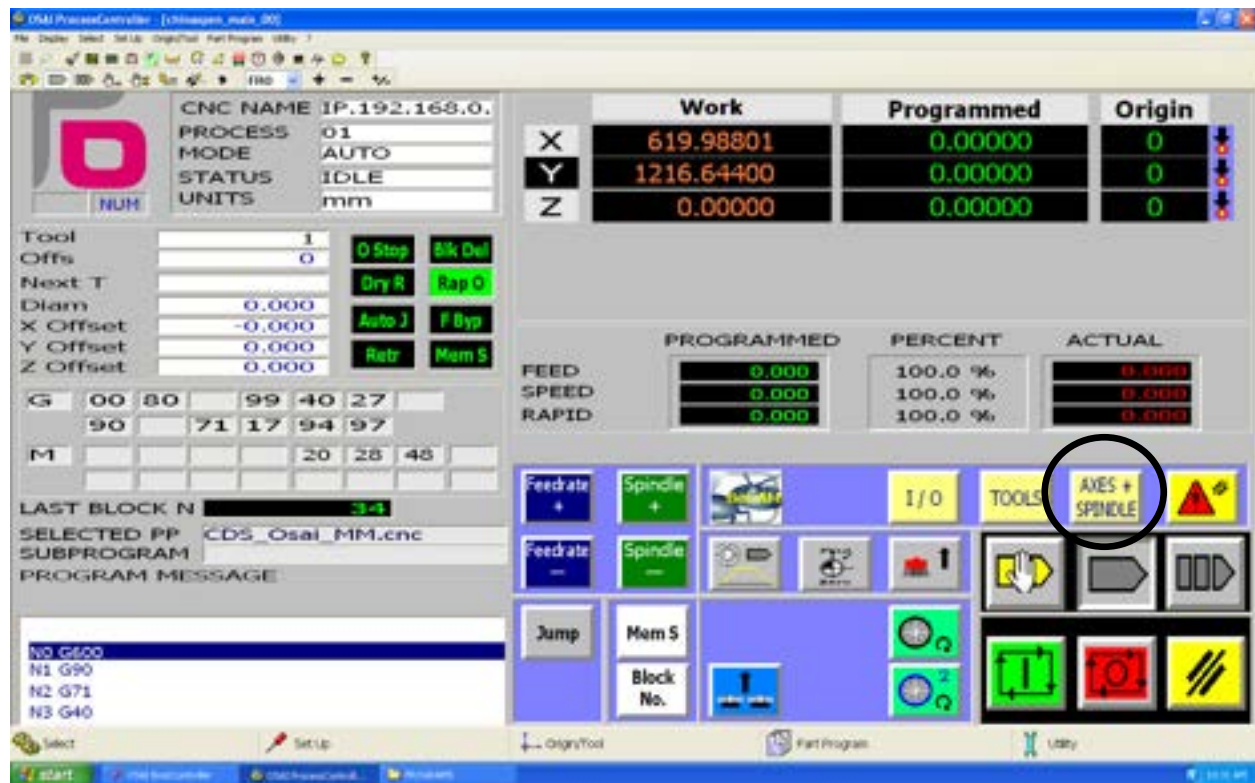
Operation Procedures

Jogging the machine

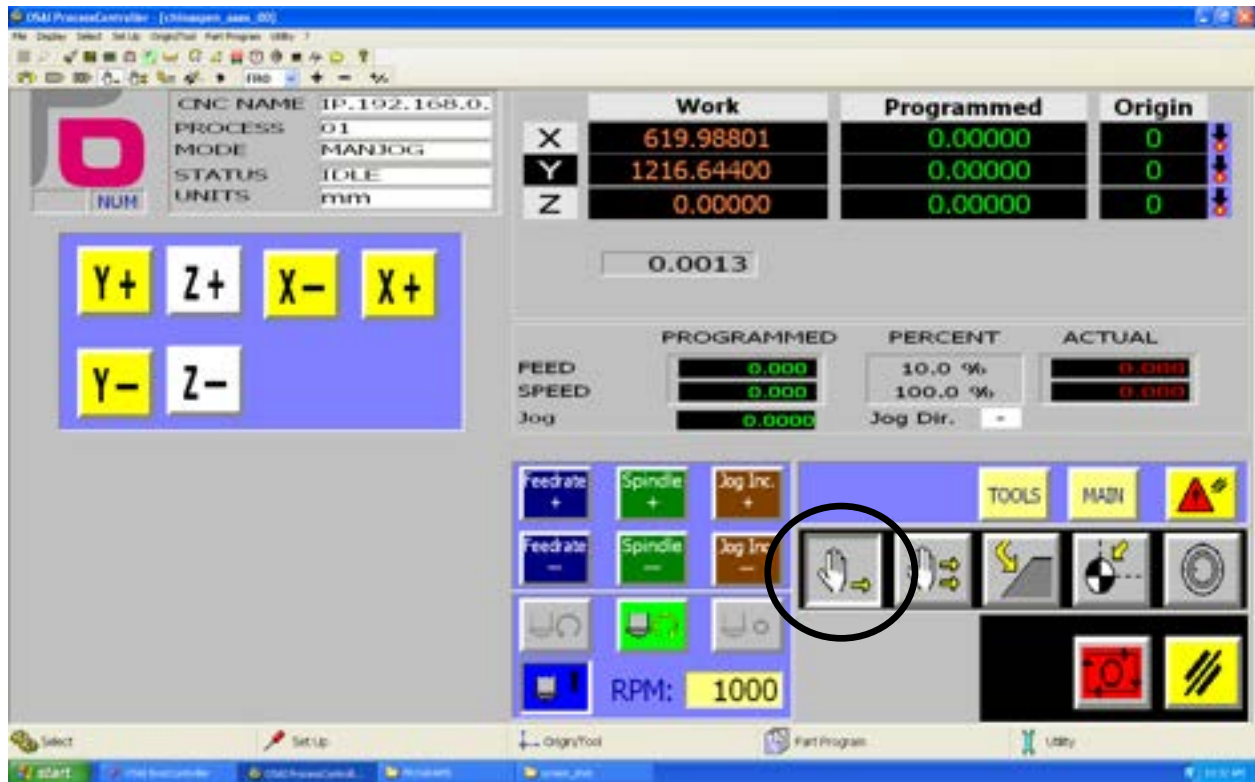
Jogging the machine refers to manually telling the machine to move its X, Y, or Z axis. There are three ways to jog the machine: Continuous Jogging, Step Jogging, and MPG jogging. Continuous jogging is used to move the machine over a long range, since it allows jogging at a high speed. Step jogging is used to move the axis for a user specified distance. Lastly MPG jogging is used when precise movement is needed; for example, when aligning the spindle to a work origin. The procedures for these three ways of jogging are described in the following sections.

Continuous Jogging Procedure

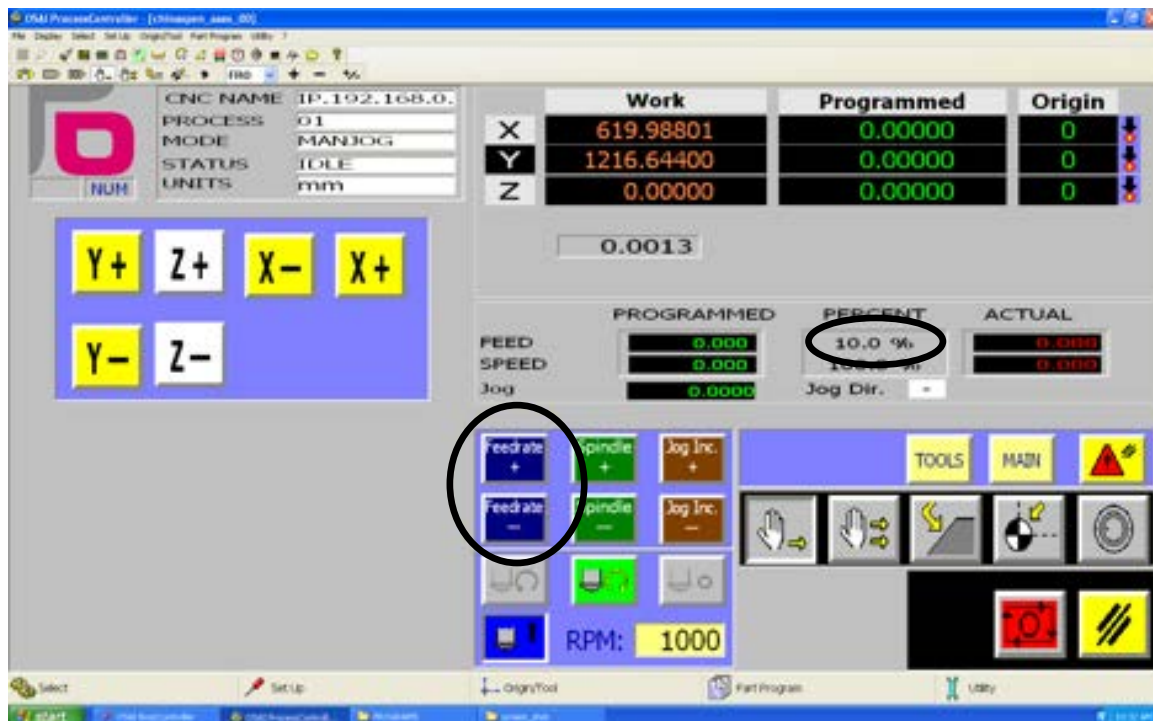
1. Make sure nobody can be harmed by any machine movements and that there is nothing that obstructs machine movements.
2. From the Main Screen, click the Axis + Spindle Button to go to the Axis + Spindle page.



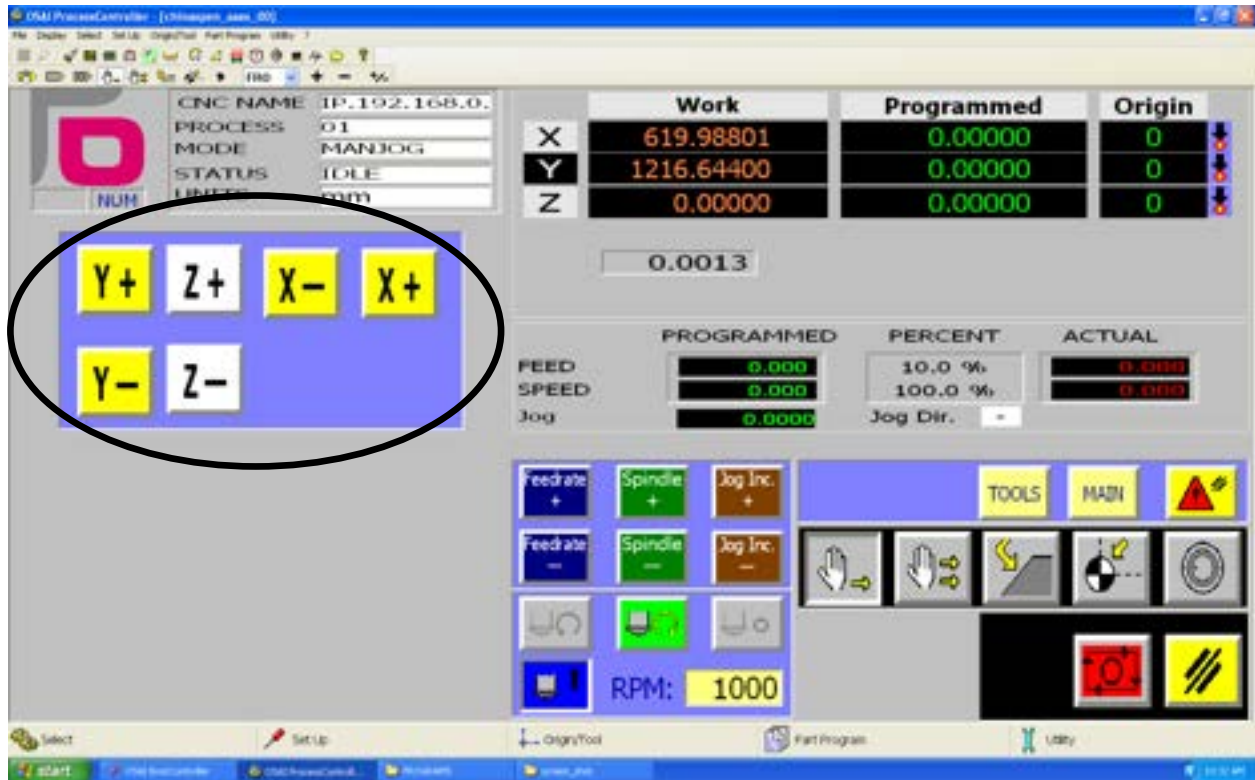
- From the Axis + Spindle Screen, press the Continuous Jog Button.



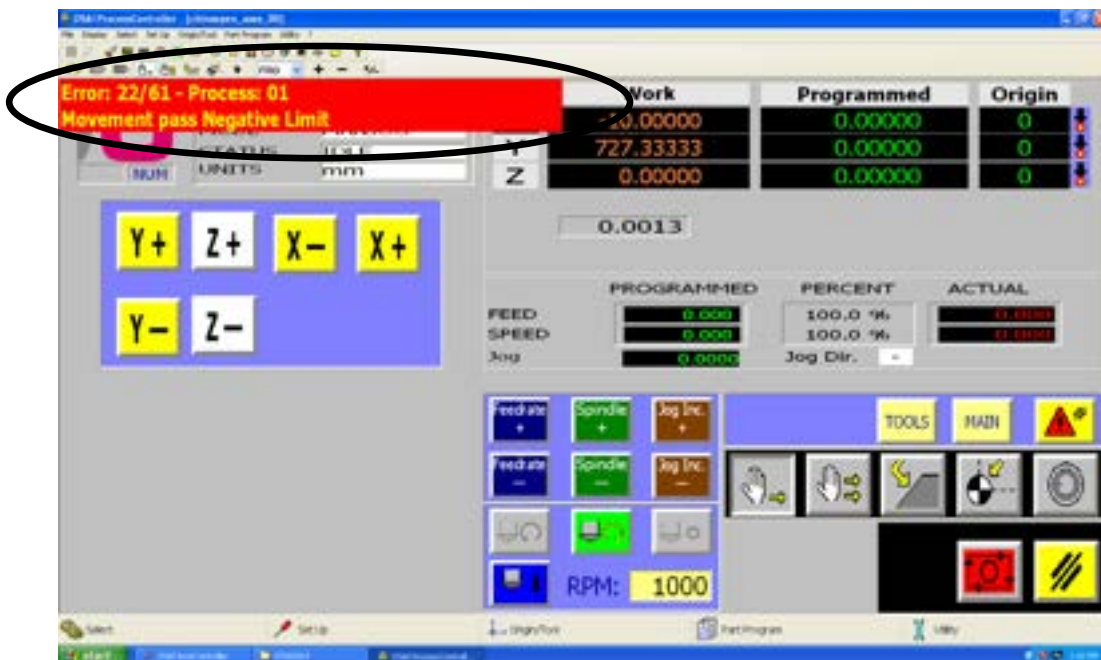
- Use the feed rate buttons to adjust the feed rate percentage. The higher the percentage the faster the machine will move.



- Press and hold the axis buttons to move the machine. On a touch screen, there is a safety delay so that the user needs to hold the button for about one second before the machine starts moving.



- If an axis limit message shows up, jog the machine away from its positive or negative limit.



Step Jogging Procedure

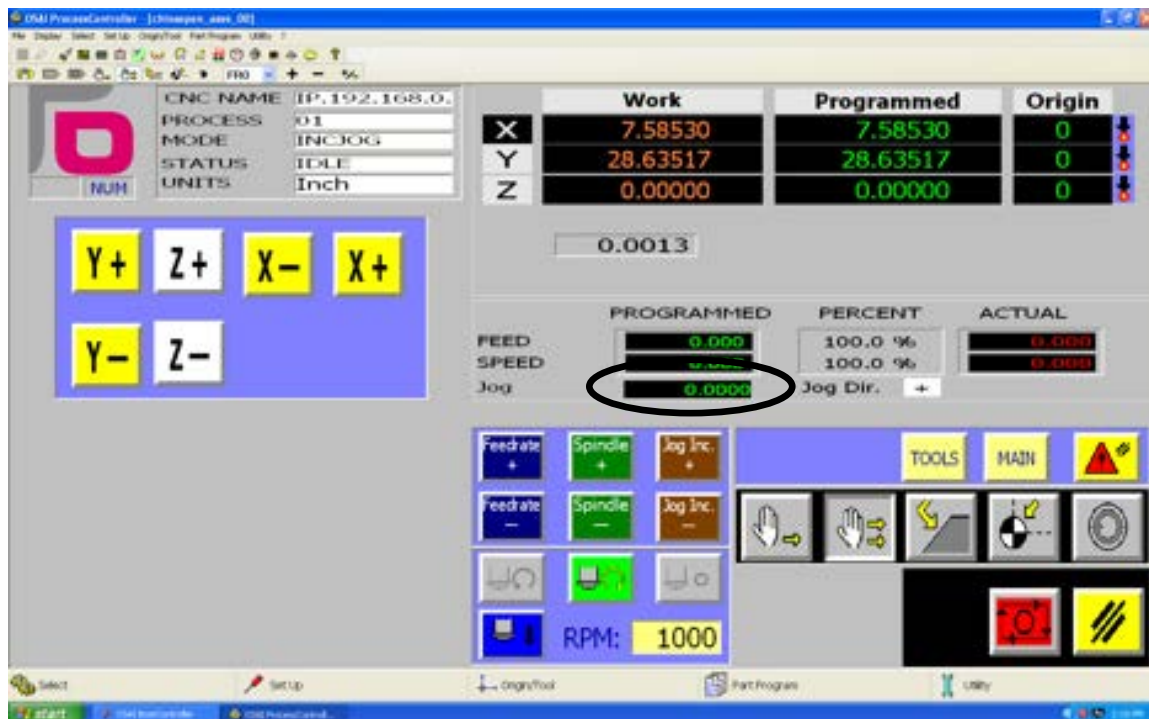
1. Make sure nobody can be harmed by any machine movements and that there is nothing that obstructs machine movements.
2. From the Main Screen, click the Axis + Spindle Button to go to the Axis + Spindle page.



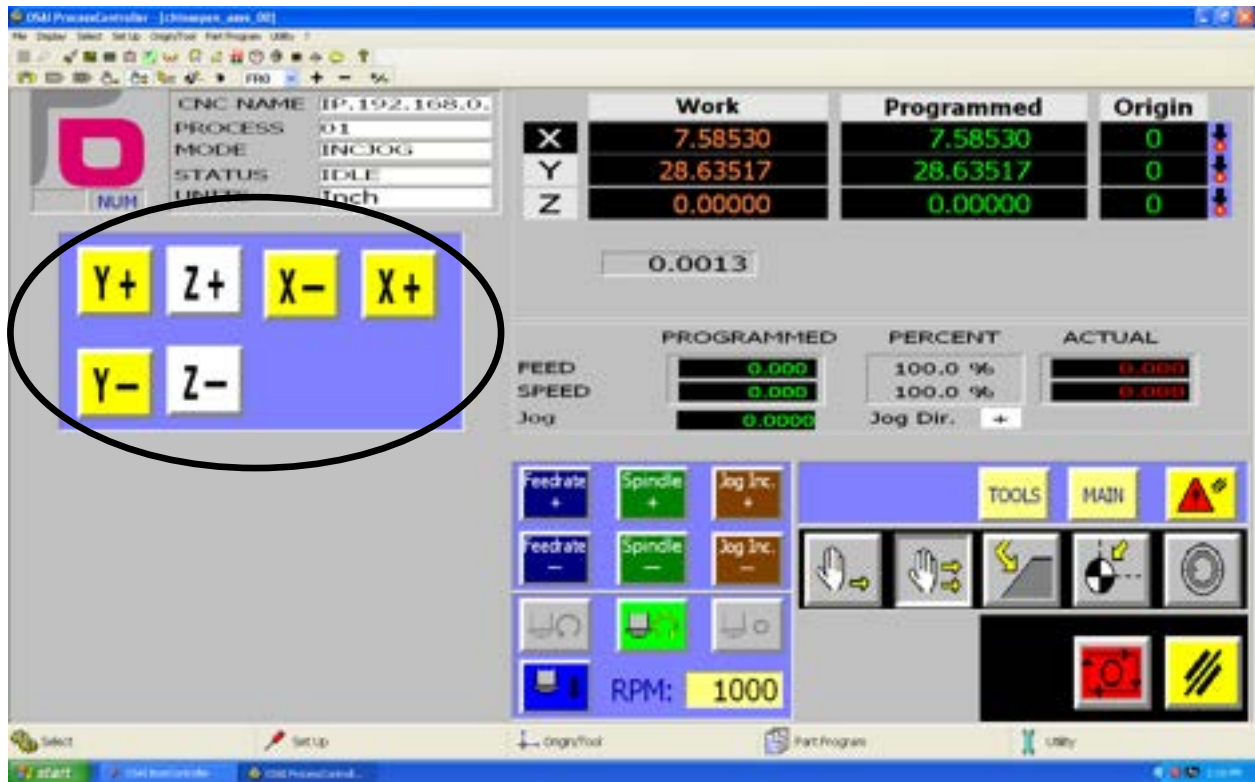
3. From the Axis + Spindle Screen, press the Step Jog Button.



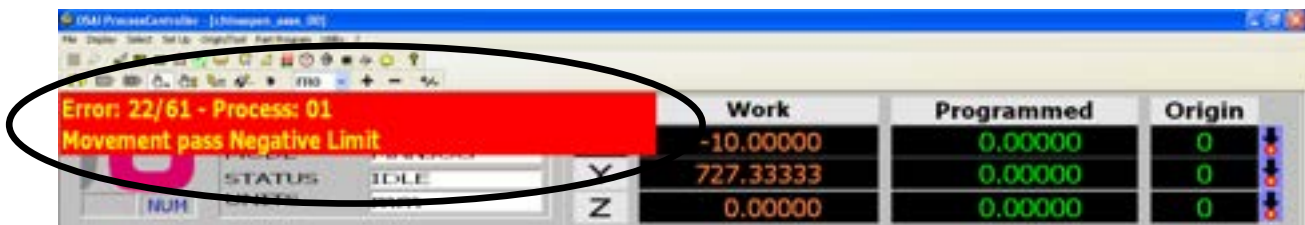
4. Use the Jog Increment buttons to adjust the Jog Increment. The Jog Increment is the distance an axis will move when it is clicked.



5. Single click the axis buttons to move the machine.

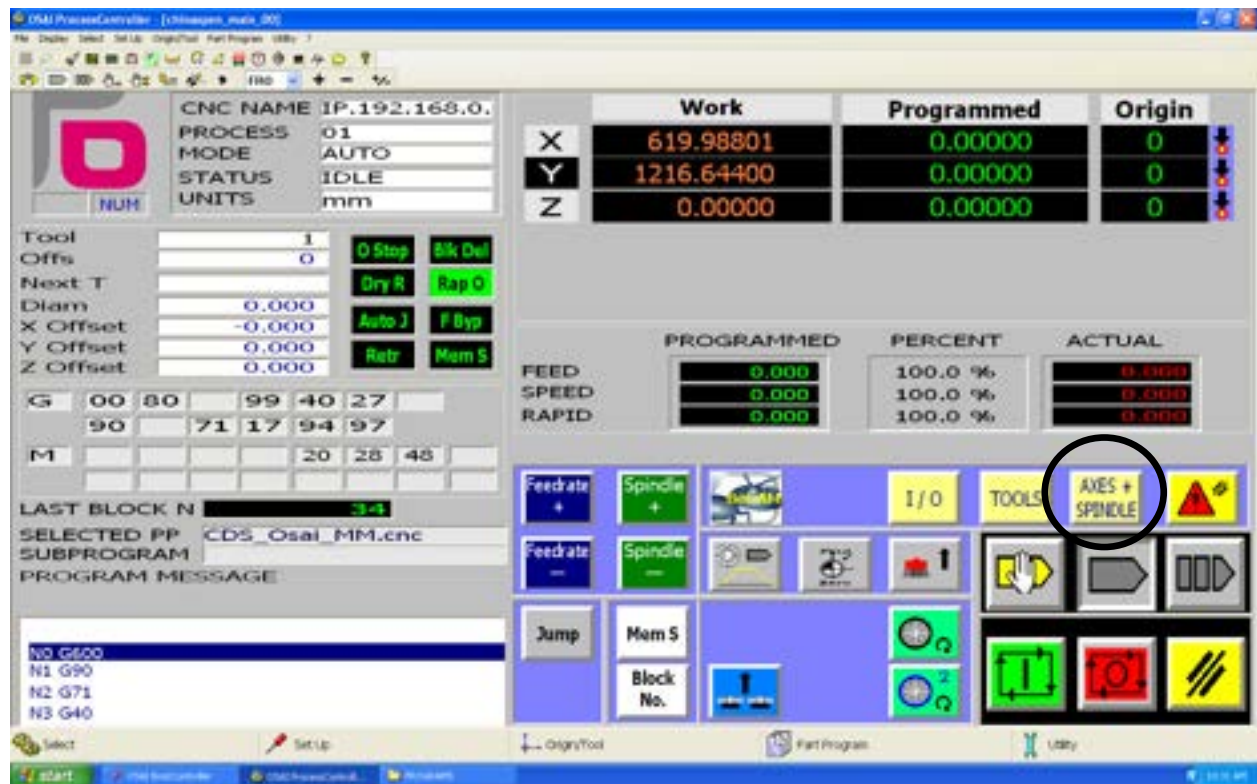


6. If an axis limit message shows up, jog the machine away from its positive or negative limit.



Manual Pulse Generator (MPG)

1. Make sure nobody can be harmed by any machine movements and that there is nothing that obstructs machine movements.
2. From the Main Screen, click the Axis + Spindle Button to go to the Axis + Spindle page.



3. From the Axis + Spindle Screen, press the MPG Jog Button.



4. Use the middle knob on the MPG to select the axis to move.



5. Use the bottom knob to select speed, with 100X being the fastest.



6. Rotate the top knob clockwise to move the axis in the positive direction and counter-clockwise for negative direction.



7. After the manual jog is completed, turn the middle knob to the “Off” position to prevent machine movement by accident.
8. If an axis limit message shows up, jog the machine away from its positive or negative limit.



Setting Tool Length

Please refer to the 5-Axis manual for Setting Work tool length on a 5-Axis machine.

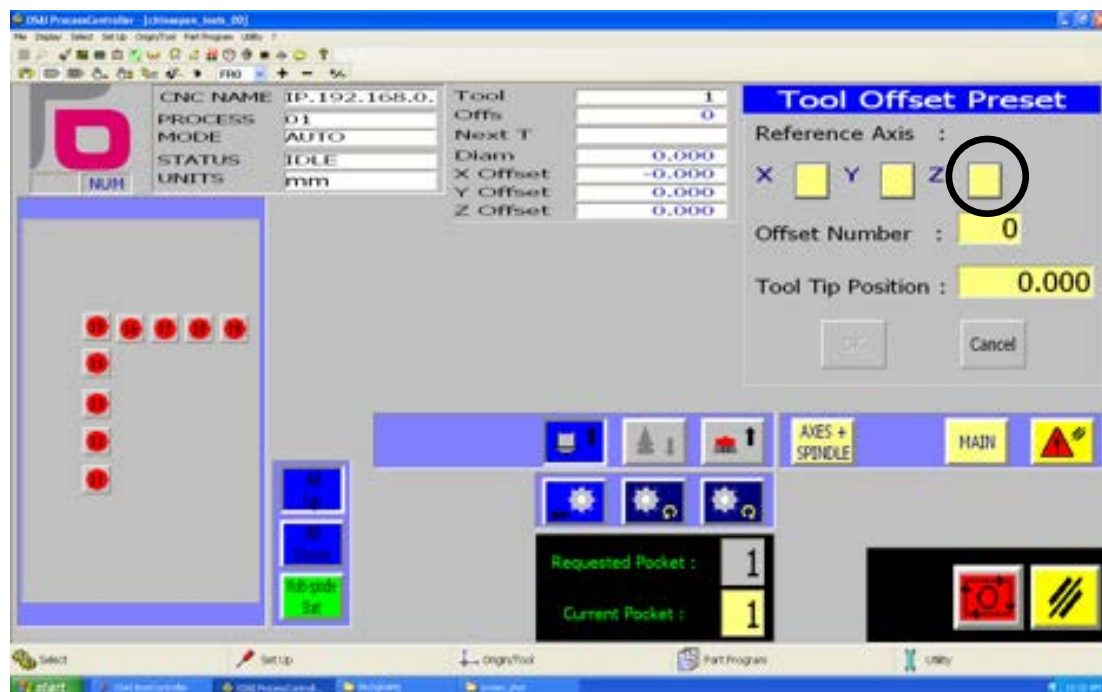
1. Make sure nobody can be harmed by any machine movements and that there is nothing that obstructs machine movements.
2. Make sure it is safe to drop the spindle or drill, whichever applies. That is, make sure the Z axis is high enough that dropping the spindle or the drill will not cause it to hit anything.
3. Drop the spindle or drill for setting tool length.
4. Jog the machine to a desired Z height. Usually on top of the material. The X, Y coordinate does not matter.



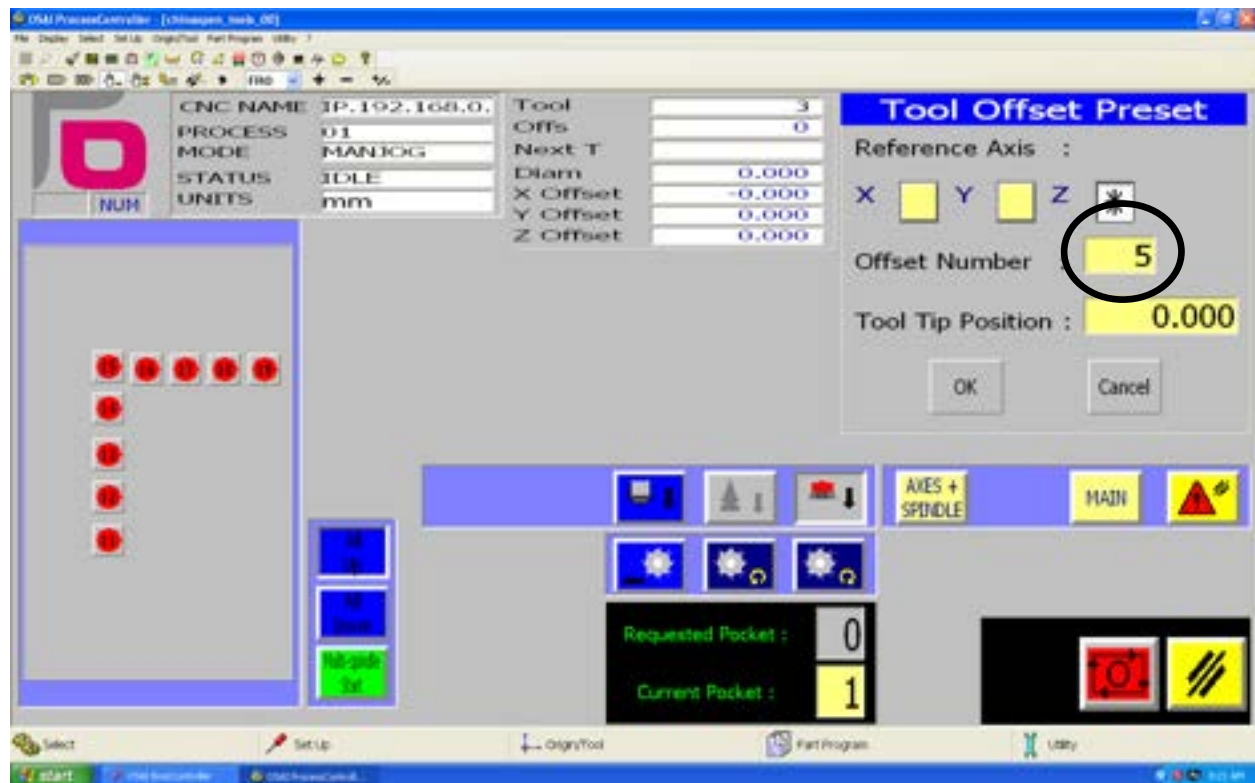
5. From the Main page click Tools button to go to the tool page.



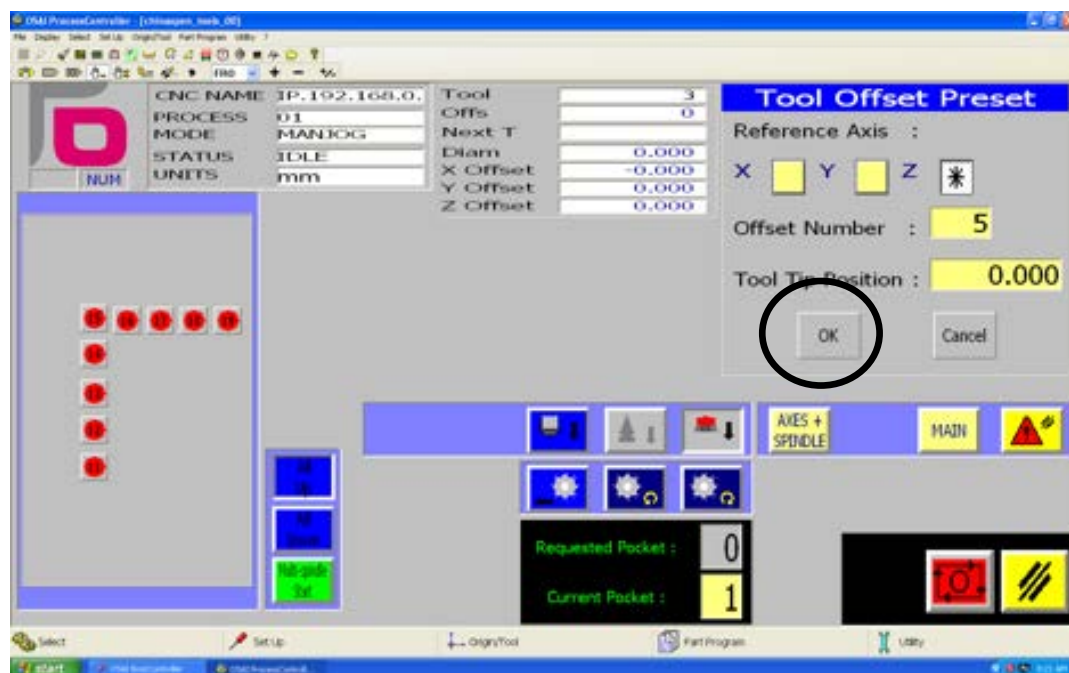
6. Select the Z axis at Tool Offset Preset field.



7. Enter the tool number for Offset Number field then press “Enter” key on keyboard. The tools in main spindle are tools 1 through 8, depend on its location in the tool holder. The drill bits in the drilling block has tool numbers from 11 to 19.



8. Click “OK”. The specified tool’s length is now set.

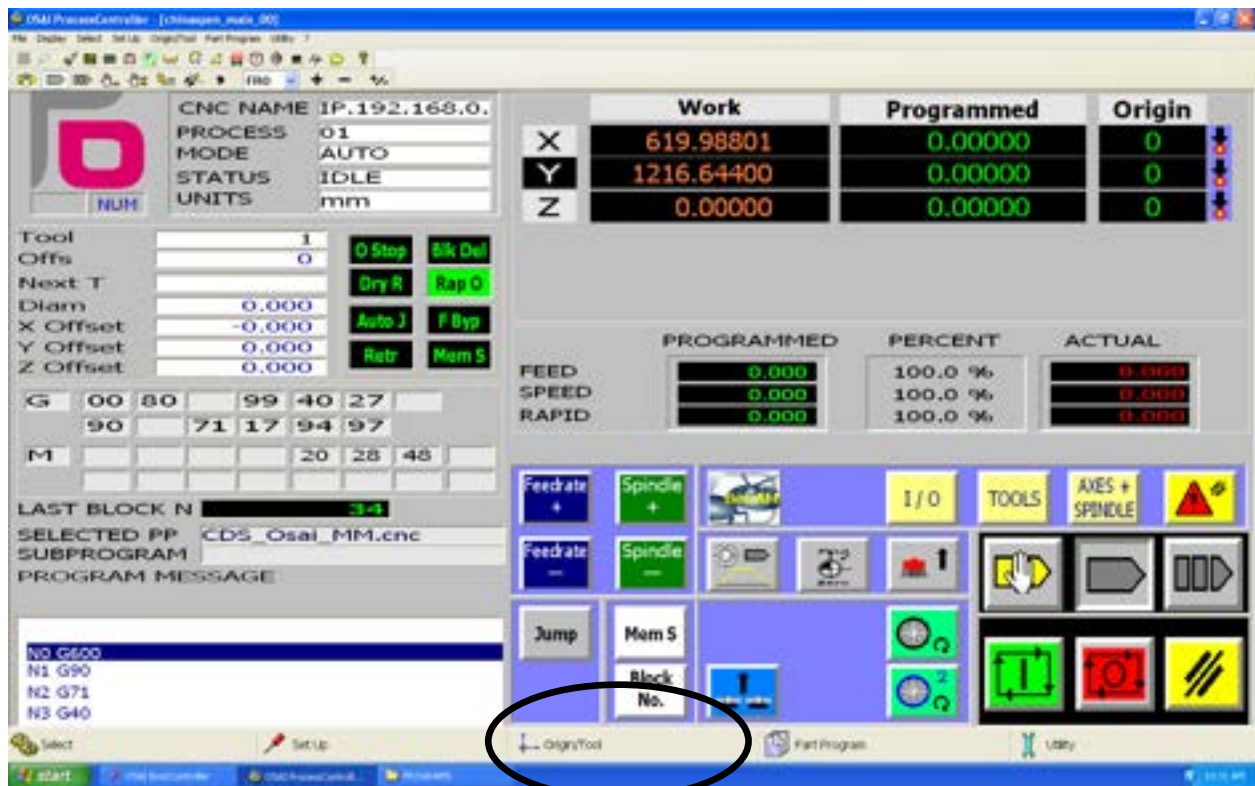


Setting Work Origin

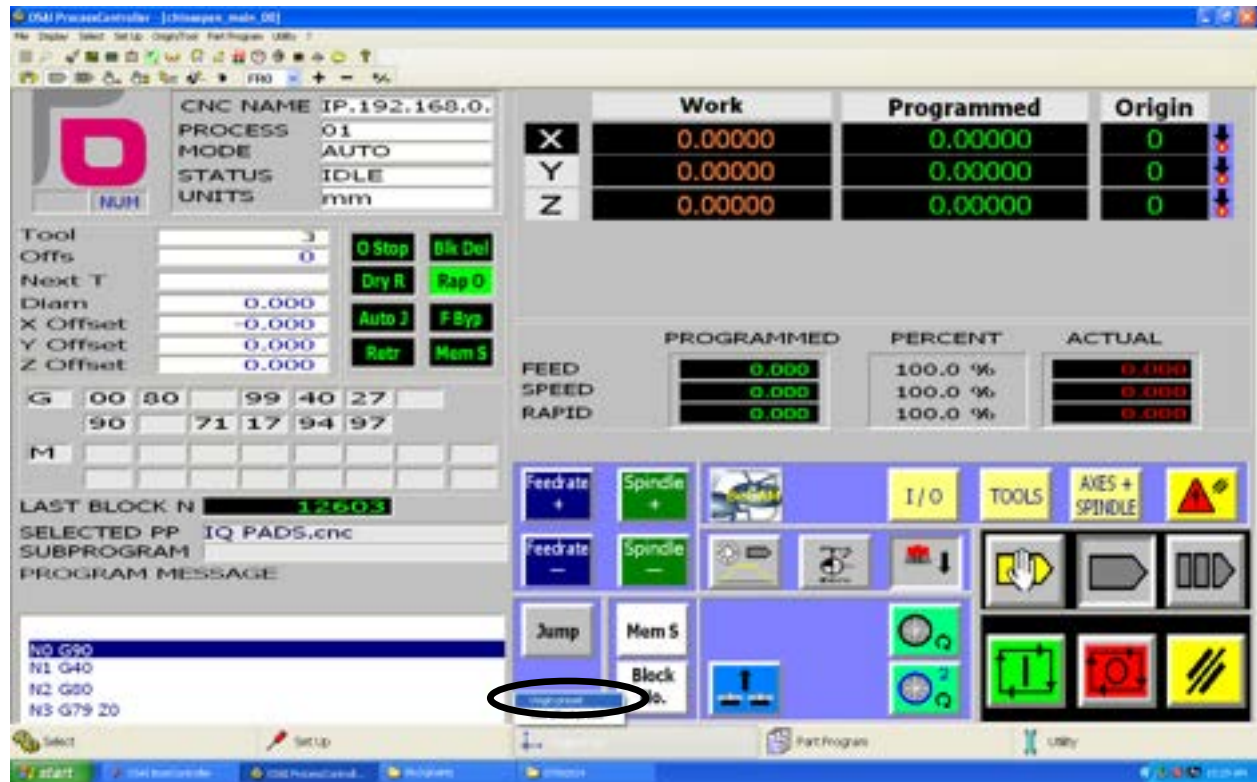
Please refer to the 5-Axis manual for Setting Work Origin on a 5-Axis machine.

1. Make sure nobody can be harmed by any machine movements and that there is nothing that obstructs machine movements.
2. Manually jog the machine to a X,Y coordinate you wish to set as the Work Origin. The Z coordinate does not matter.

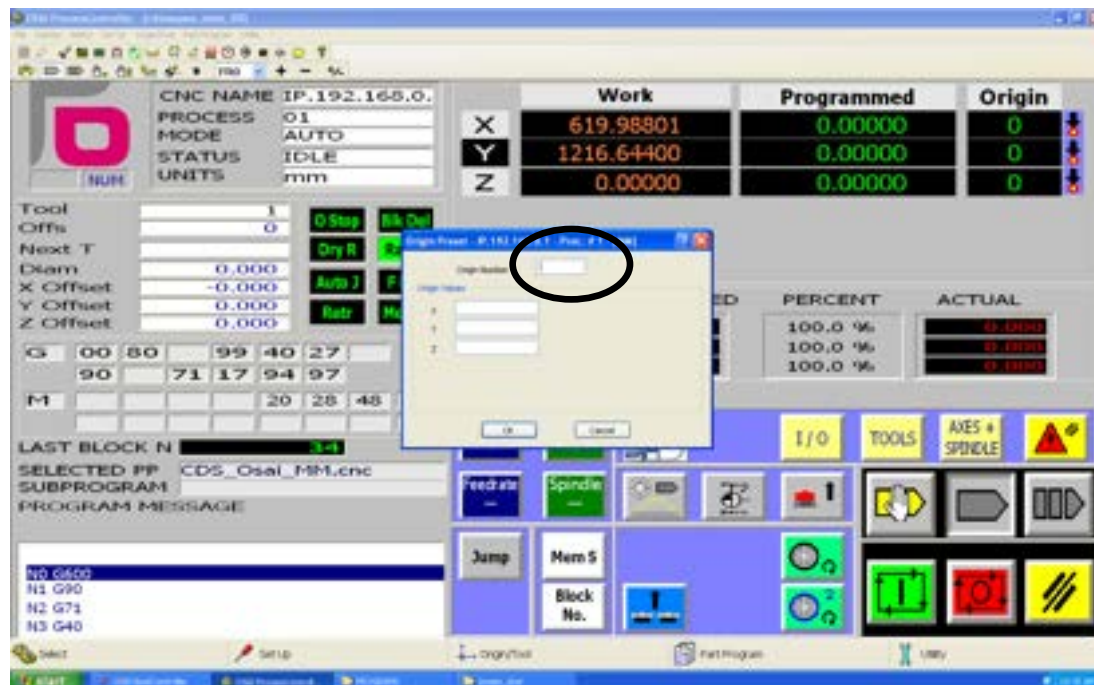
3. From the Main page click on Origin/Tool button



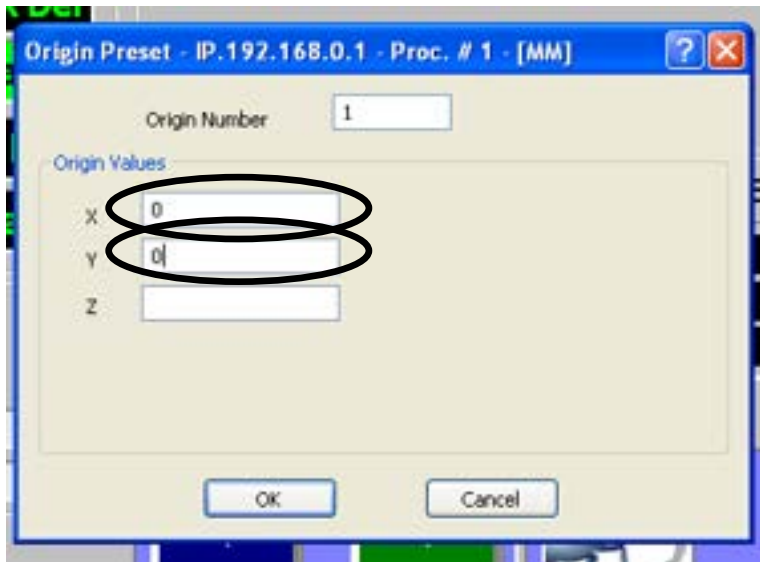
- Click on the Origin Preset option.



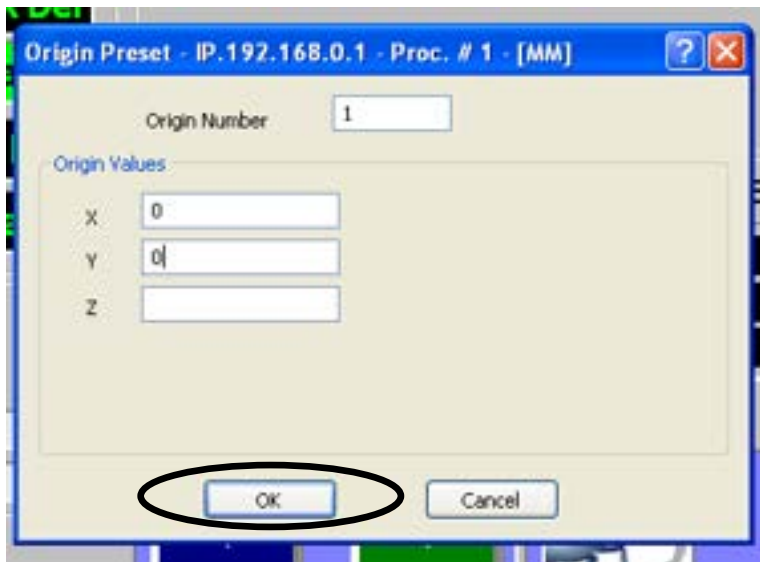
- Enter the desired offset number. There are nine off sets, from 1 to 9. The offset is activated in G-Code with the syntax "(UAO, 1)" where "1" can be any number from 1 to 9.



6. Enter "0" for both X and Y, and leave Z blank. If the machine has a C axis, please leave that blank as well.



7. Click OK. The Work Origin is now set.

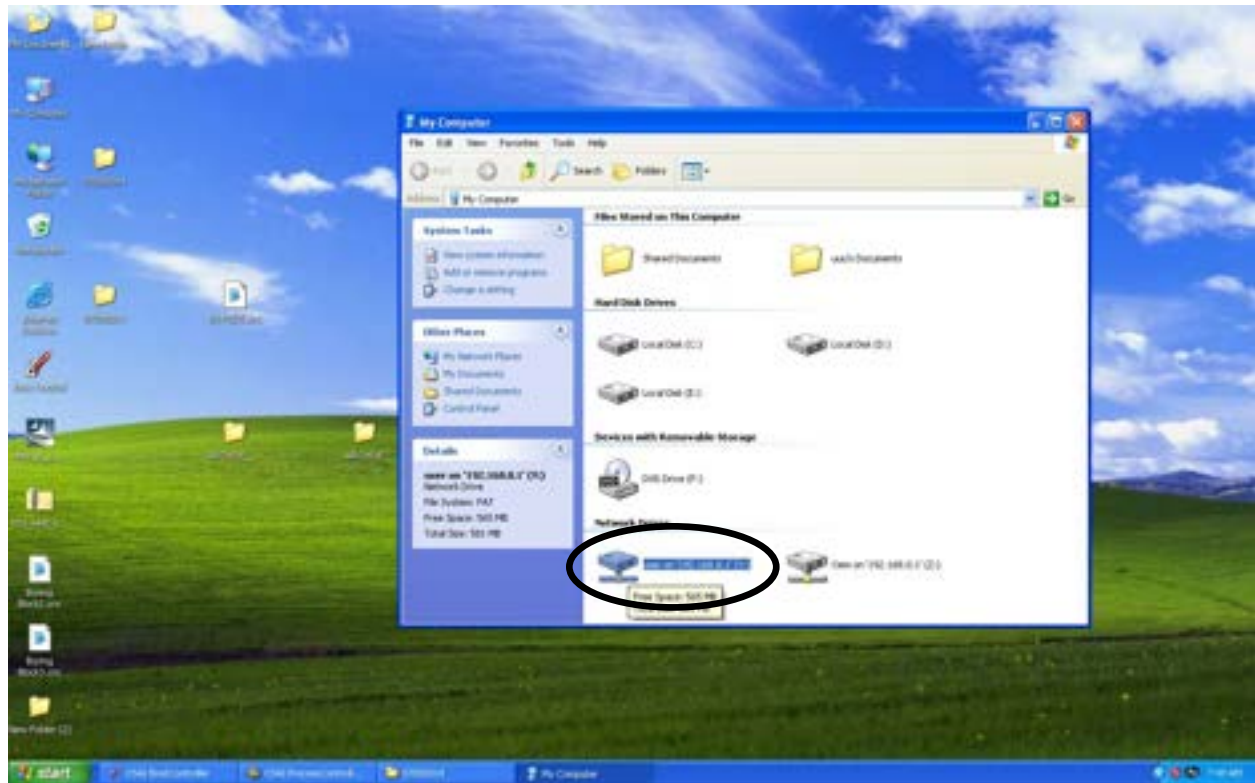


Running a G-Code File

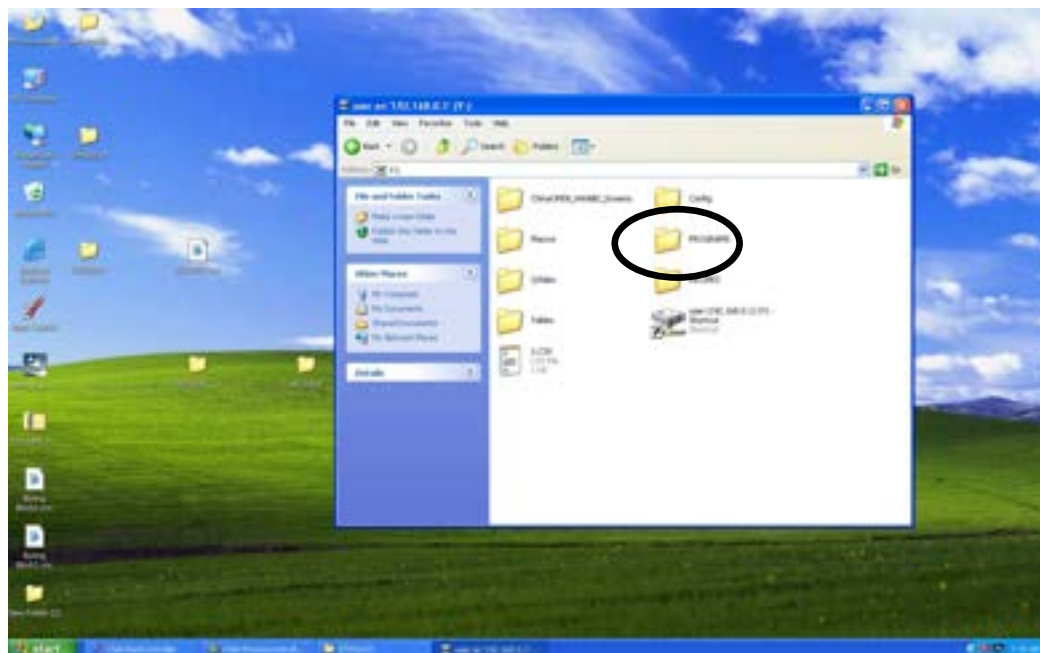
1. Make sure nobody can be harmed by any machine movements and that there is nothing that obstructs machine movements.
2. Make sure the Tool Length and Work Origin is setup correctly.
3. Transfer the G-Code program to the machine. This can be done via a USB Disk.
4. From the Windows home screen, double click the Computer icon.



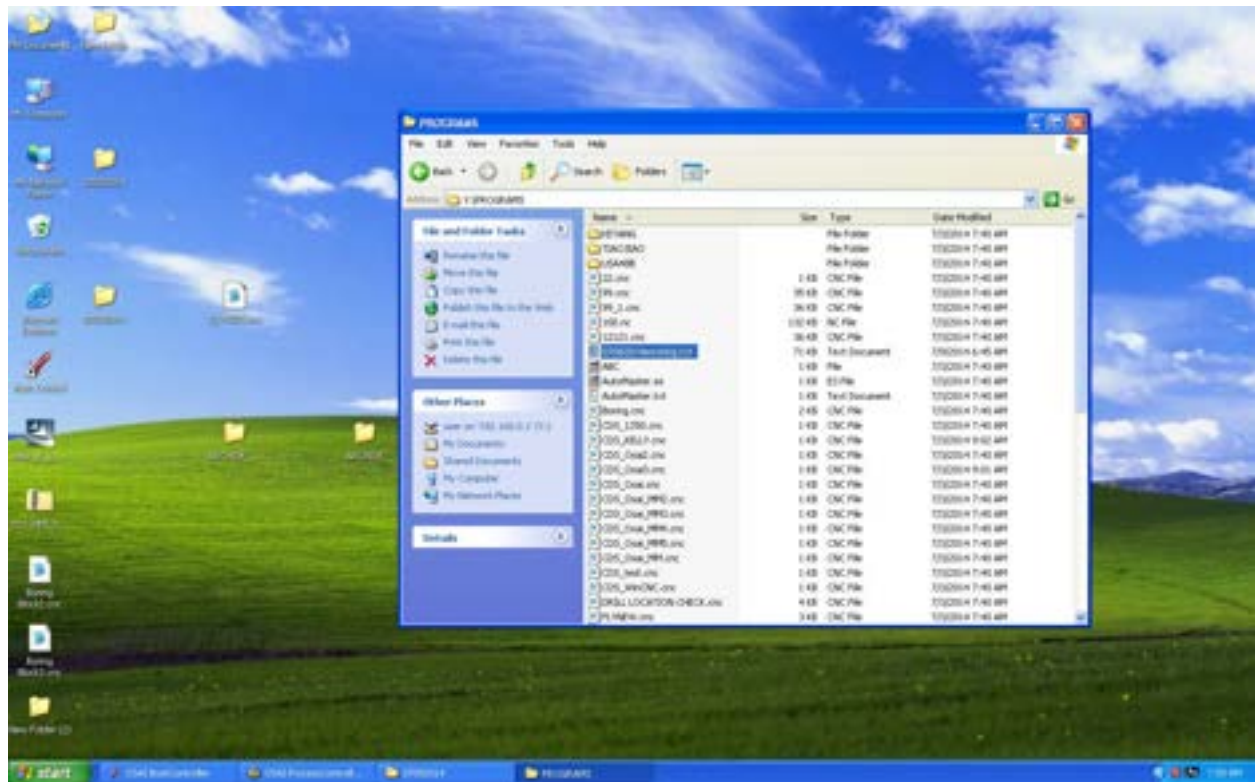
5. In My Computer window, double click on the user on '192.168.0.1' (Y:) Disk



6. Double click on the PROGRAMS folder.



7. Copy the G-Code file into this folder: Y:\PROGRAMS



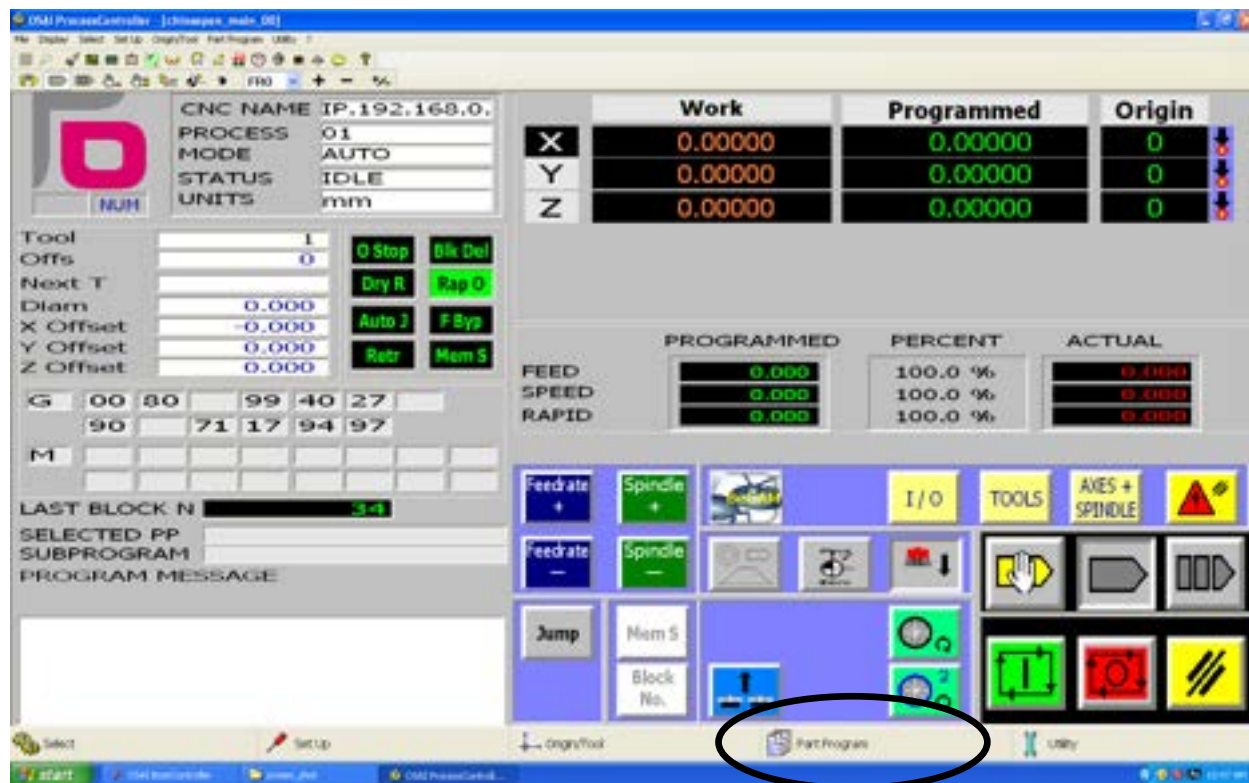
8. Make sure the machine is in Auto Mode by clicking on the Auto Mode button.



9. Click the Part Program button

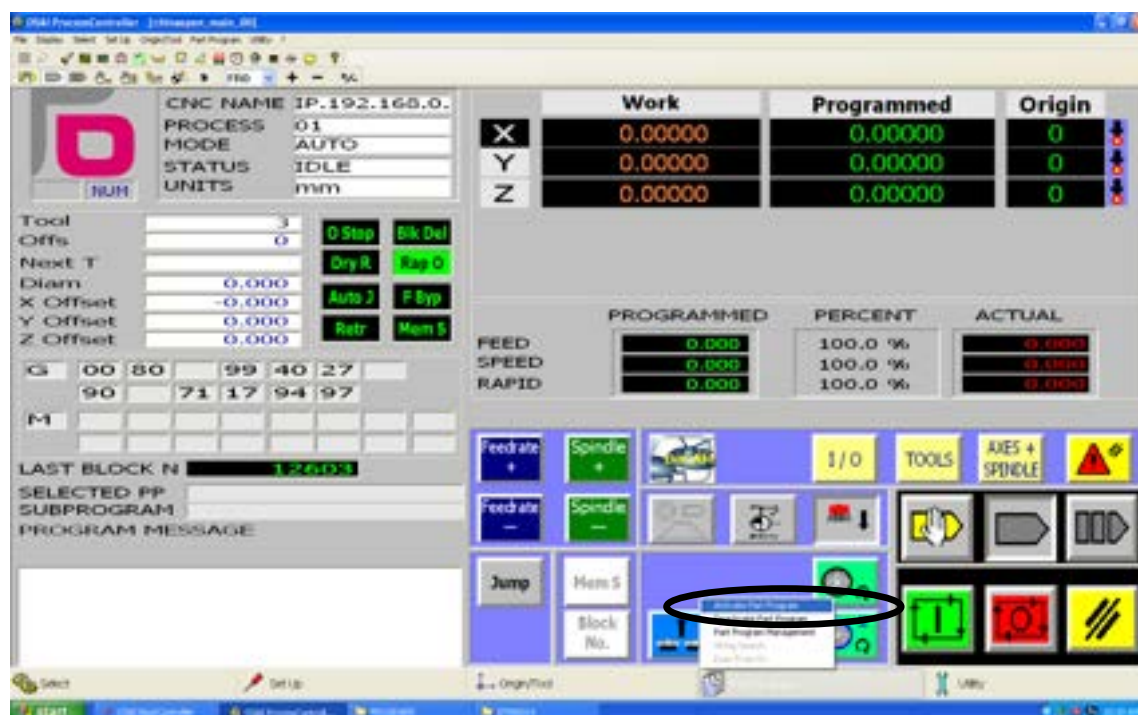


Part Program

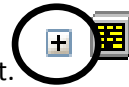


10. Select Activate Part Program

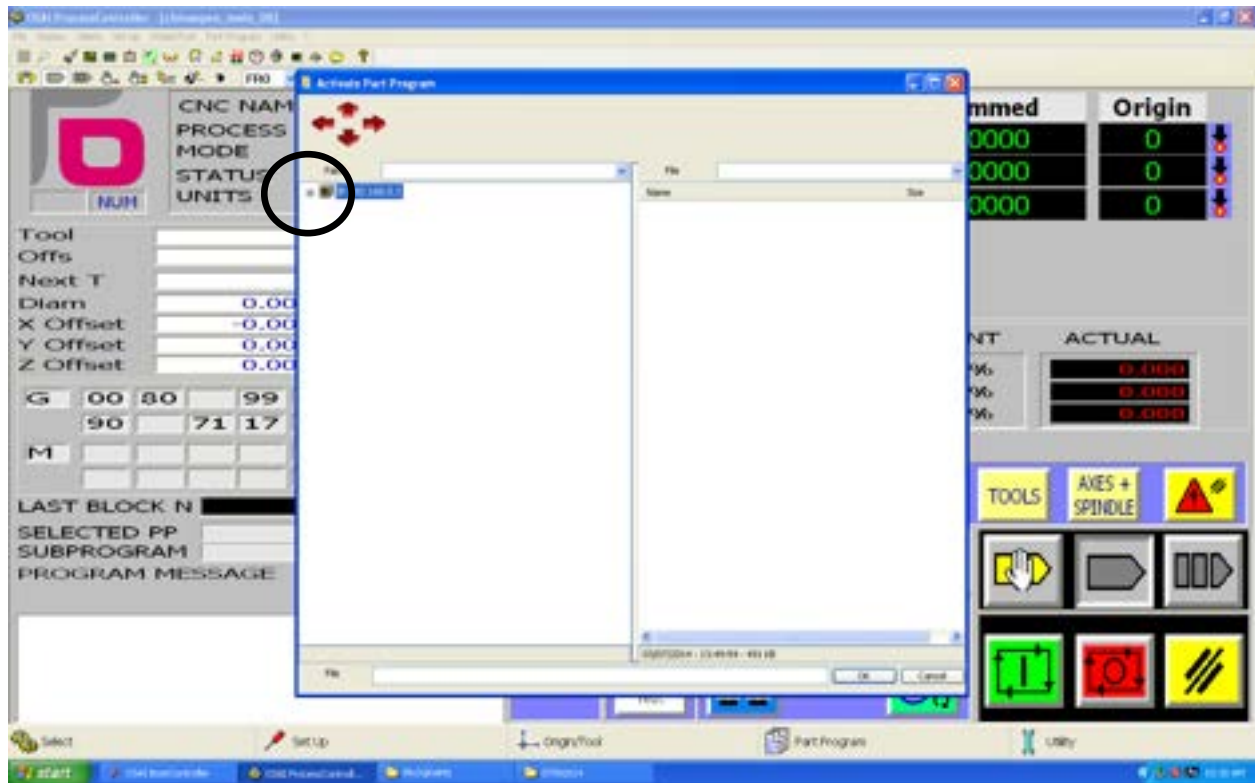
Activate Part Program

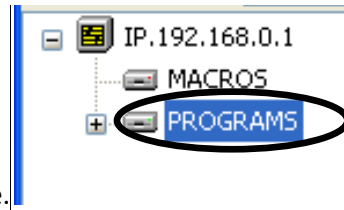


11. If the directory isn't expanded, click on the + sign next to expand it.

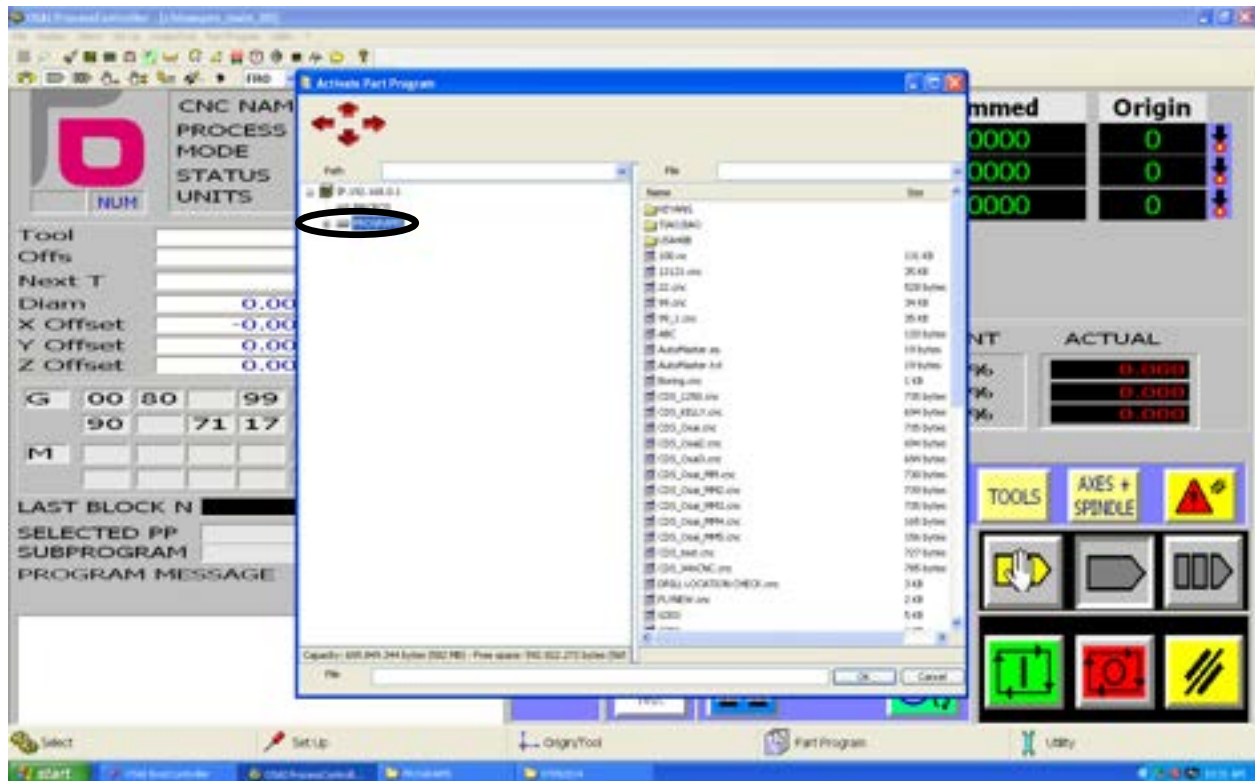


IP:192.168.0.1





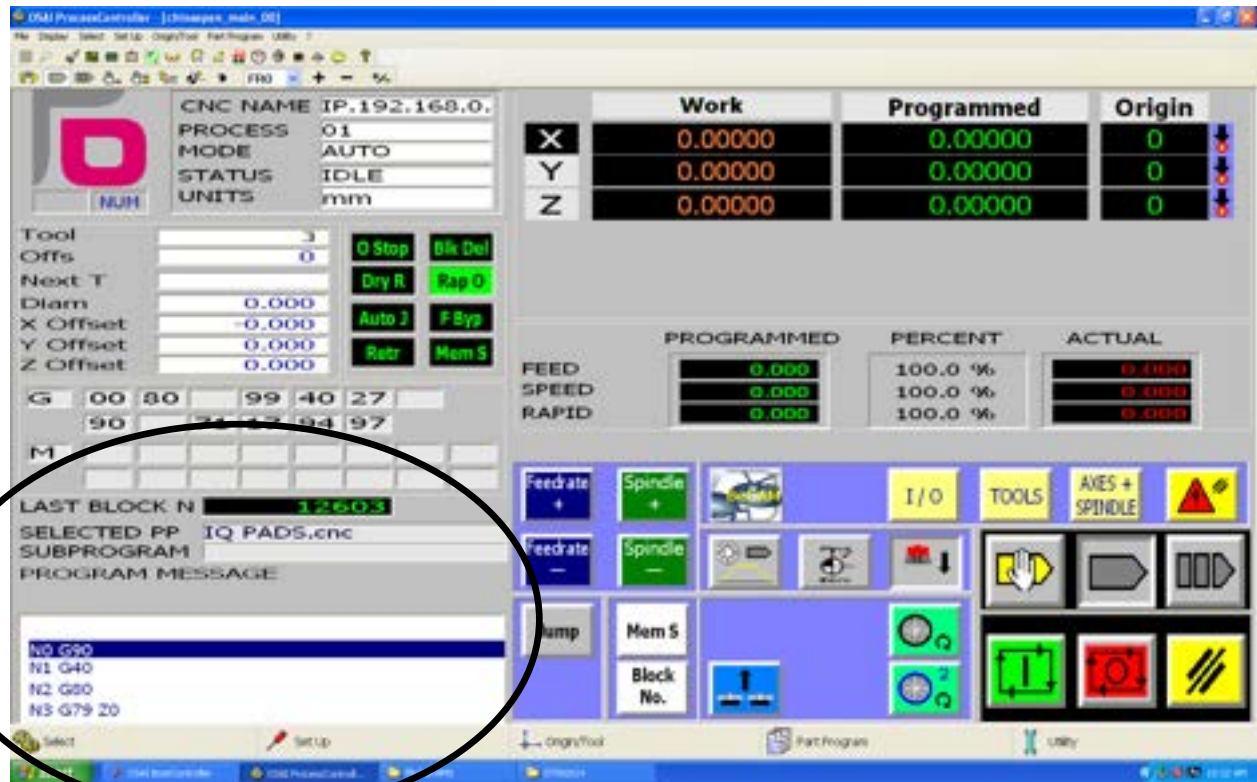
12. Select PROGRAMS drive.



[illegible]

The screenshot shows a Windows XP desktop with the 'Activate Part Program' dialog box open. The dialog box has a 'File' field containing a list of files, with 'C:\Program Files\Festo\...' highlighted. A red circle is drawn around the 'File' field. The background shows the CNC control interface with various buttons and displays.

15. Upon activation, the G-Code program will be shown in the Active G-Code Program section.

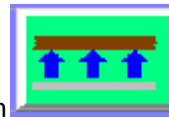


16. If necessary, turn on the vacuum pumps by clicking on the vacuum pump buttons.

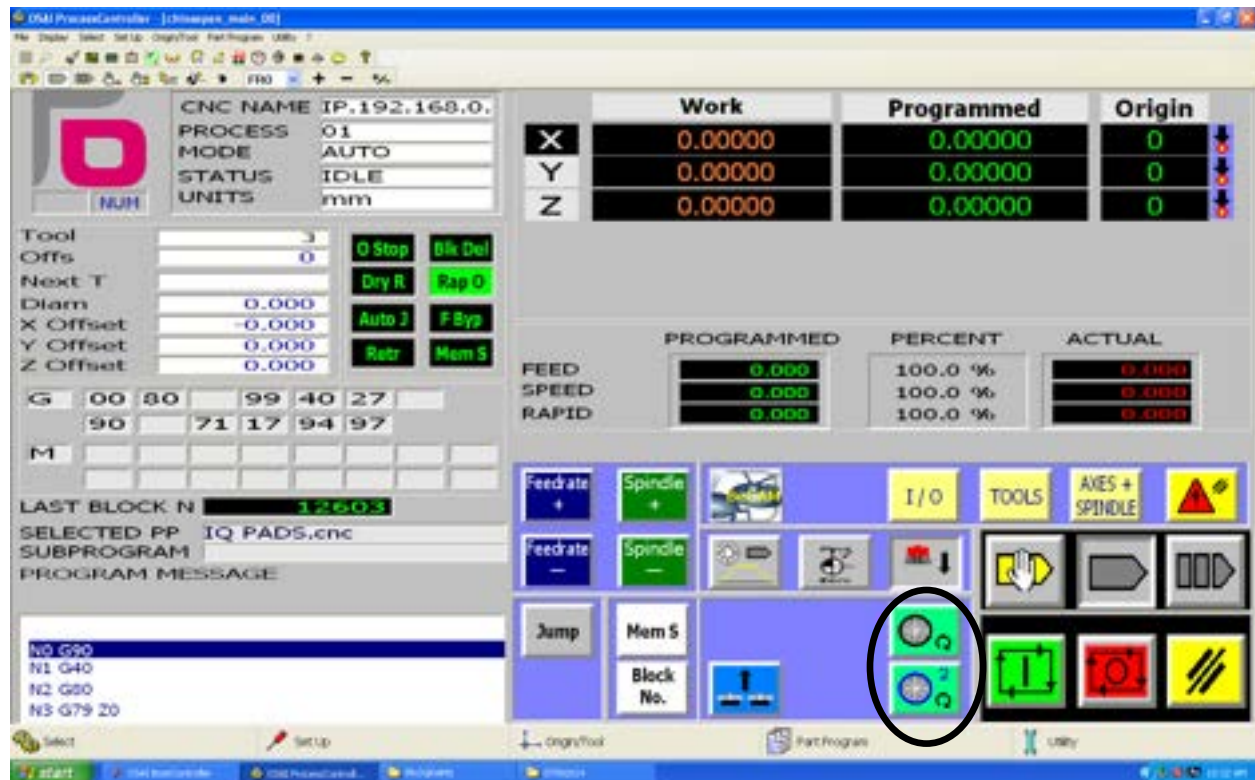


. Also

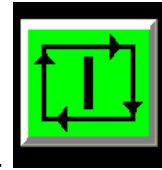
turn on the vacuum table by pressing the Vacuum table on button



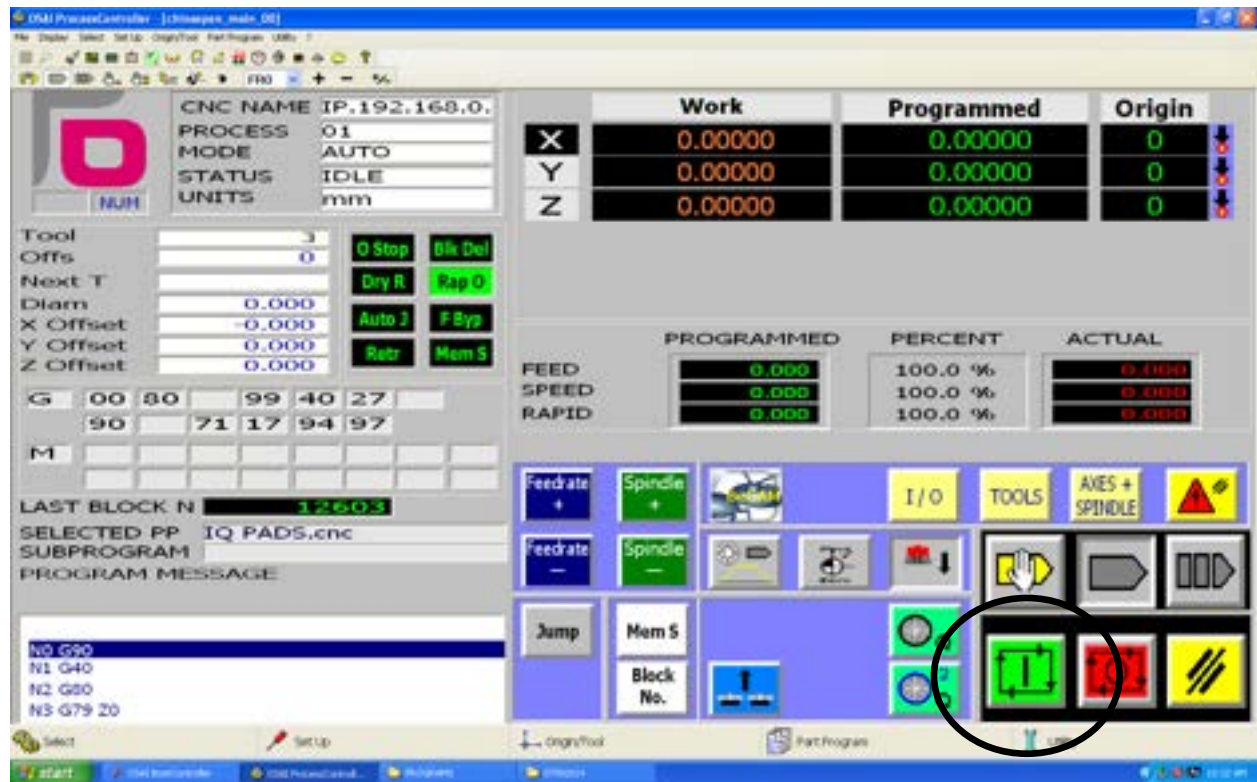
if necessary.

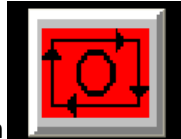


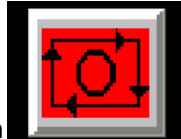
17. Make sure the work piece is secured and that machine movements will not damage any person or object.



18. Click on the run button. The machine will start executing the G-Code file.





19. To pause the machine movement, click the Pause Button . Note that the spindle might not stop when pausing.



20. To continue, press the Pause Button again then press Run button.



21. To Stop the program completely, press the Reset button.



Advanced Functions

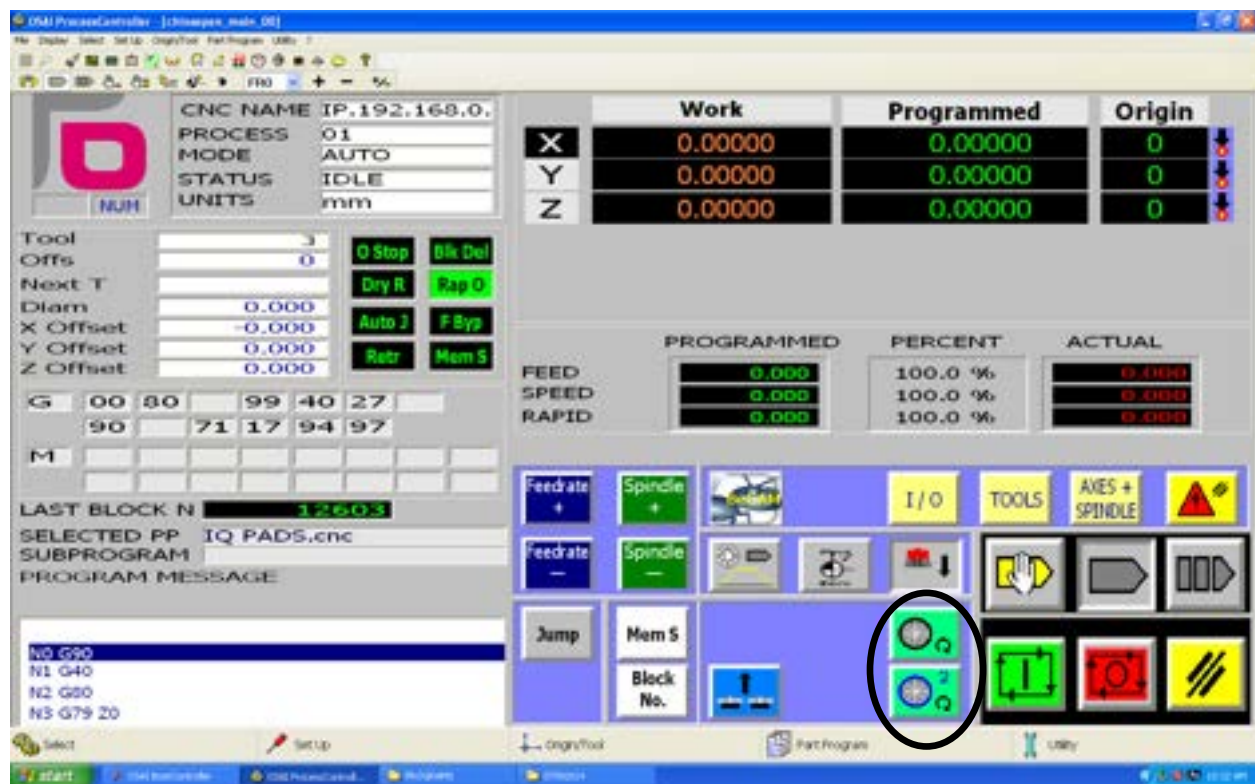
MPG Run

When running G-Code in MPG Run mode the feed rate will depend on the speed in which the user rotates the MPG knob. It is used to test out a G-Code slowly while allowing the user to pause. Following is the MPG Run mode procedure.

1. Activate a G-Code file by following the instructions in the “Running a G-Code File” section.



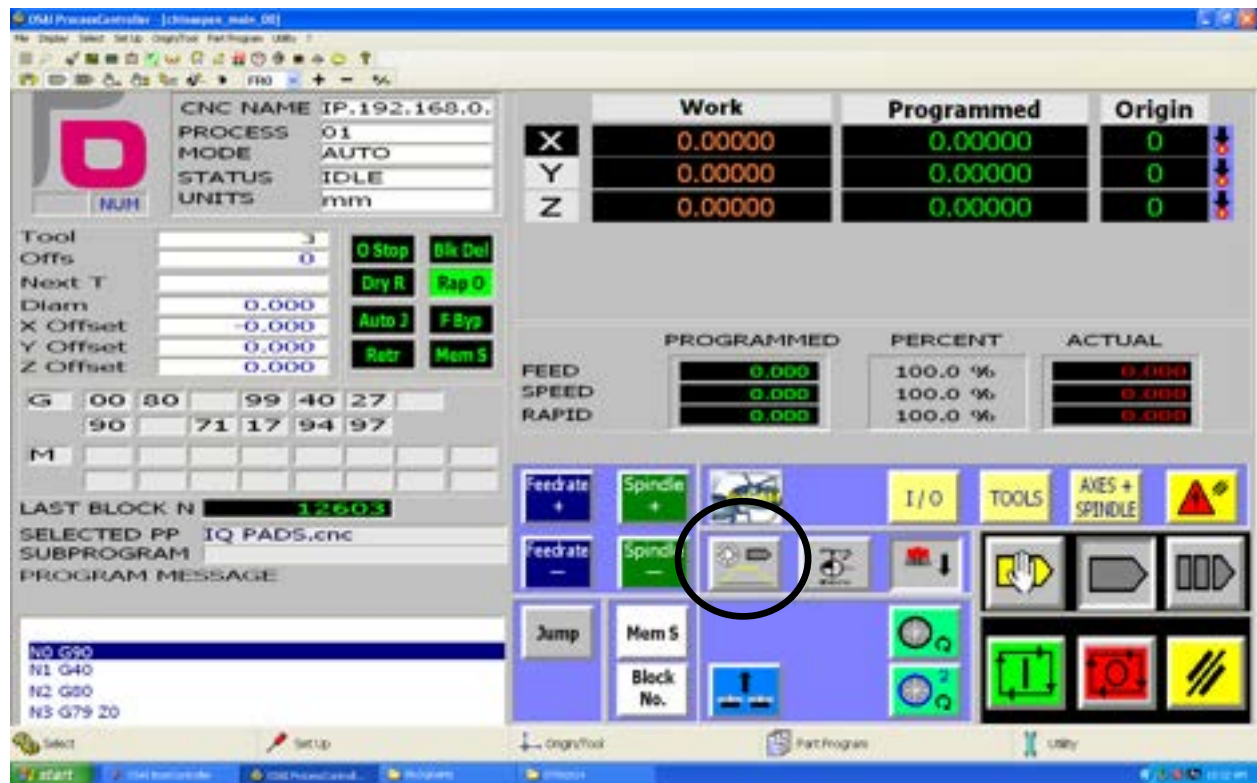
2. If necessary, turn on the vacuum pumps by clicking on the vacuum pump buttons.



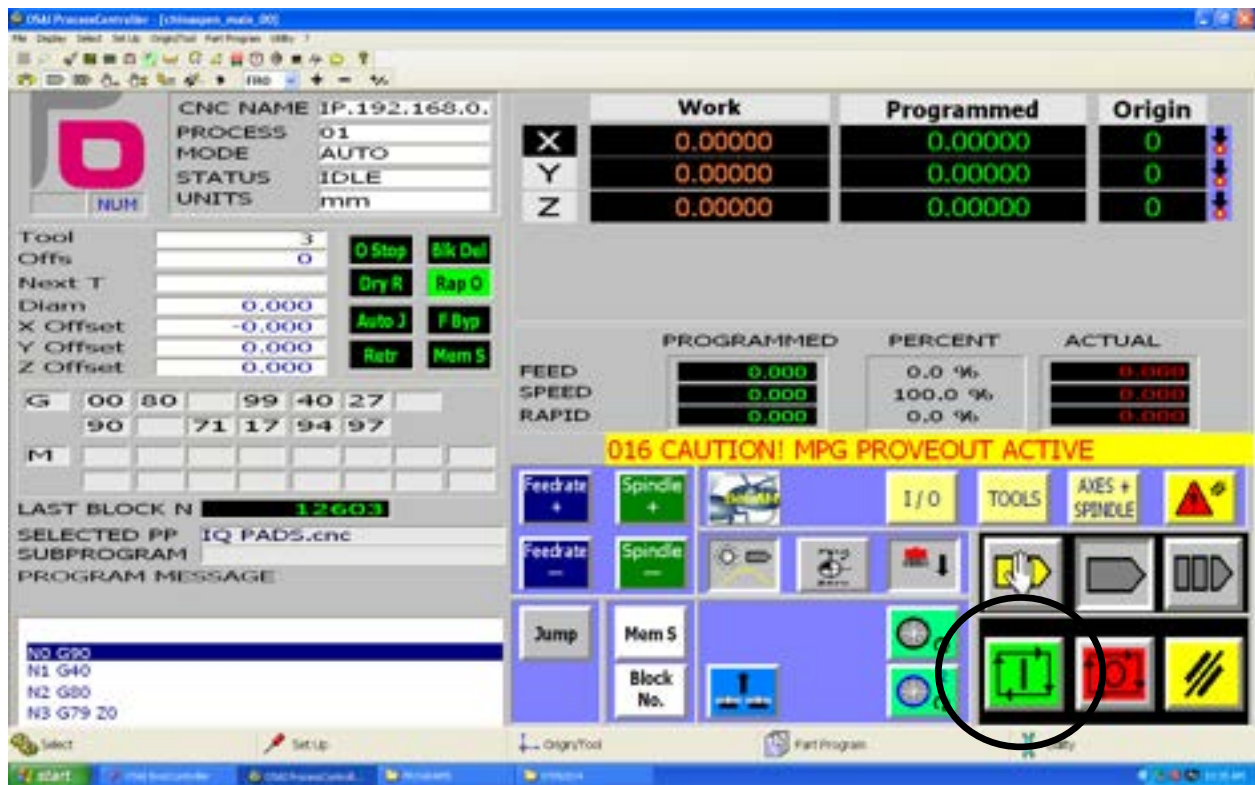
3. Make sure the work piece is secured and that machine movements will not damage any person or object.



4. Click on the MPG Run button to activate it.



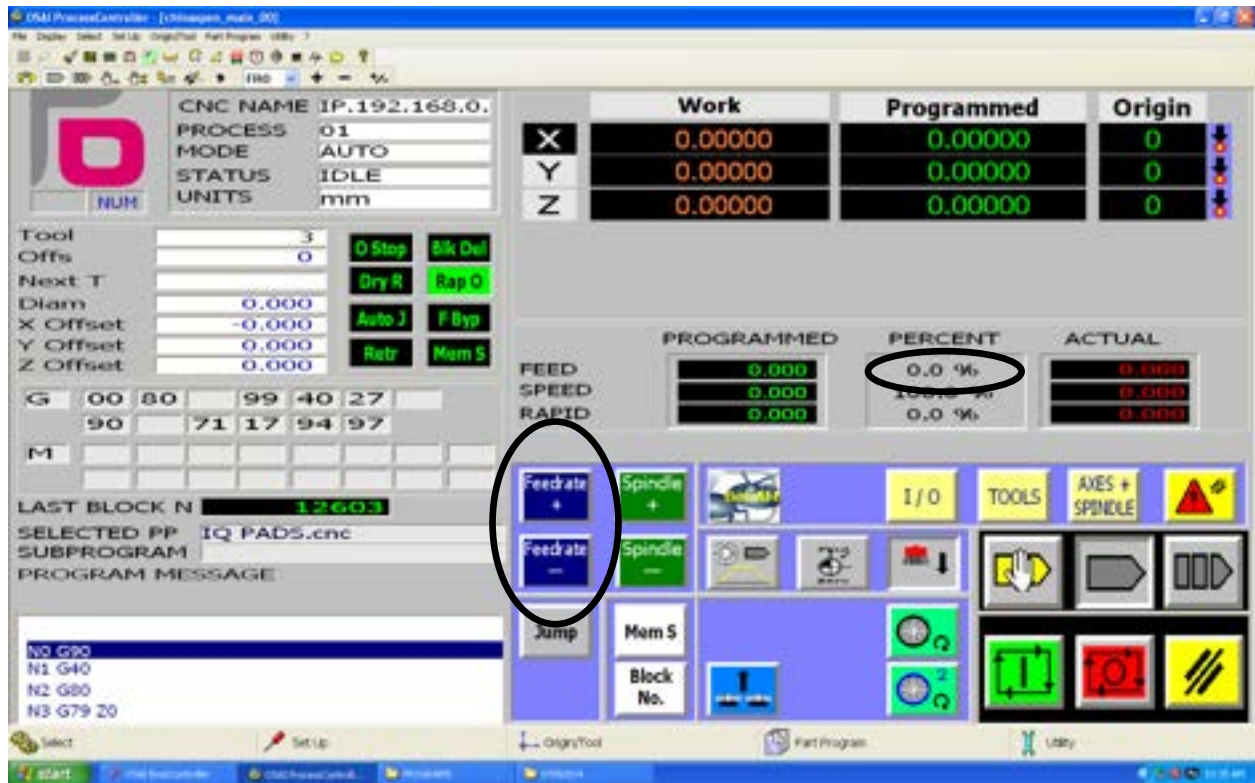
5. Press the Run Button to run the G-Code file.



6. Turn the MPG's top knob to run the program. The faster it is turned, the faster the program will run.



7. To exit the MPG Run mode, press the MPG button again then increase the Feedrate Percent by pressing the Feedrate button.



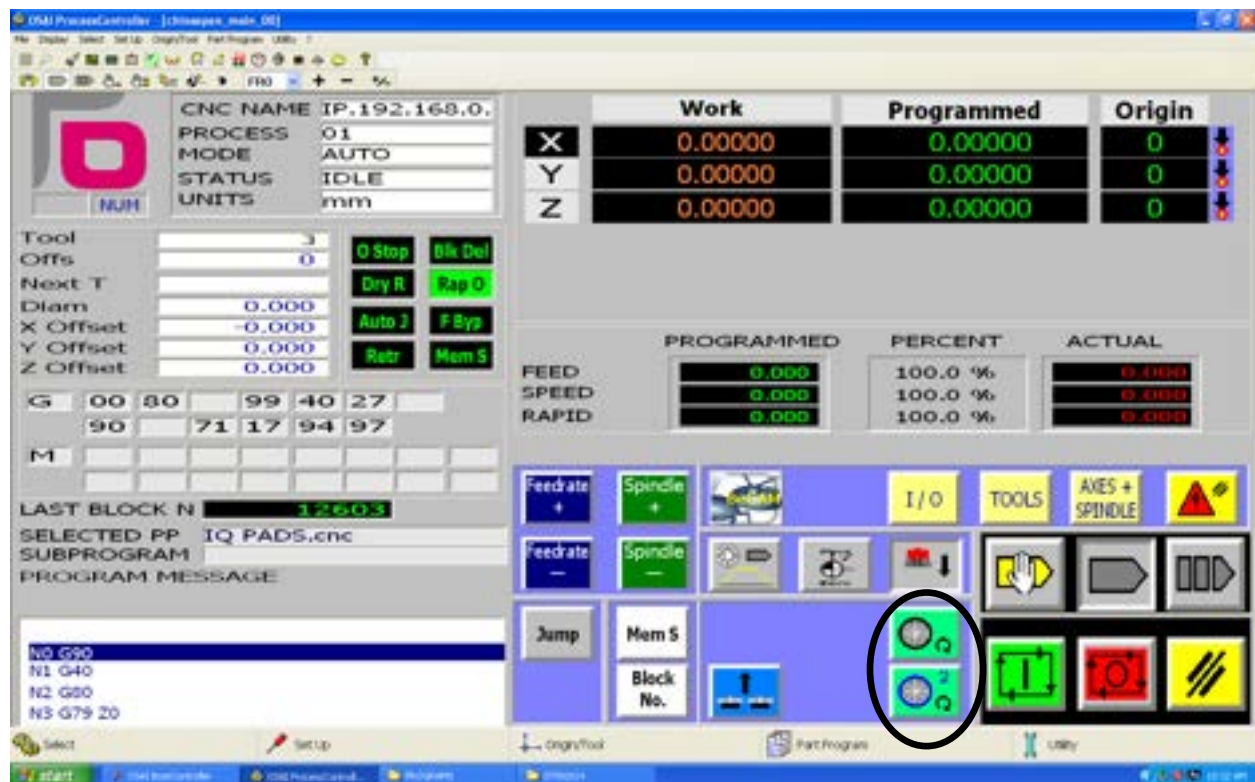
Block by Block Mode

The Block by Block mode allows user to run a G-Code file line by line, with one line executed per click of the Run button. The following section is the procedure.

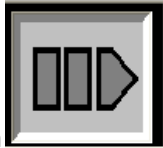
1. Activate a G-Code file by following the instructions in the “Running a G-Code File” section.



2. If necessary, turn on the vacuum pumps by clicking on the vacuum pump buttons.



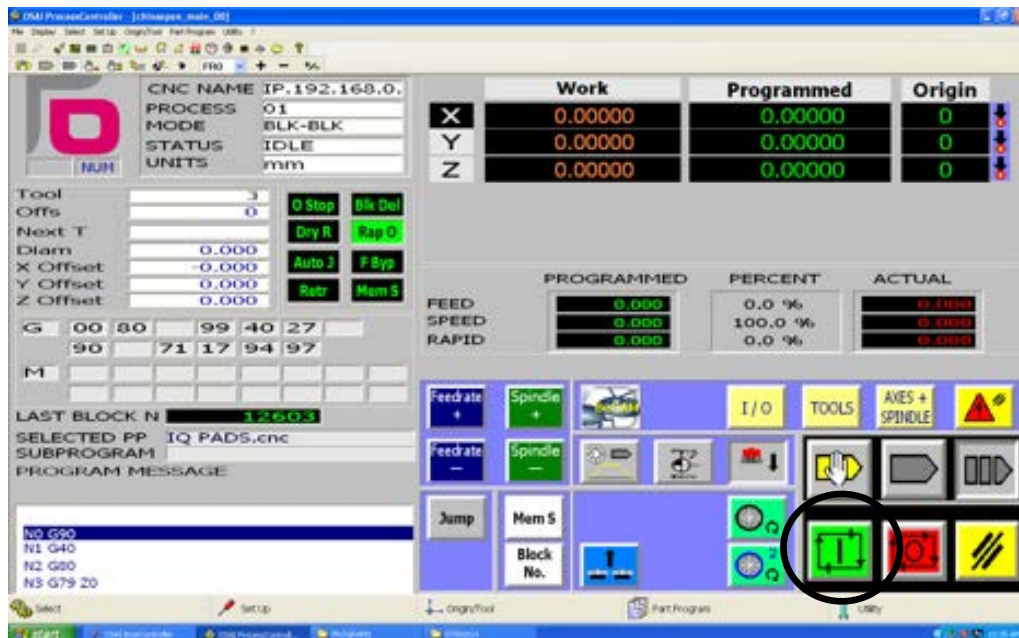
3. Make sure the work piece is secured and that machine movements will not damage any person or object.



4. Click the Block by Block button . This will put the machine in Block by Block mode.



5. Click run to run a single line of the code , click Run again to run another line.

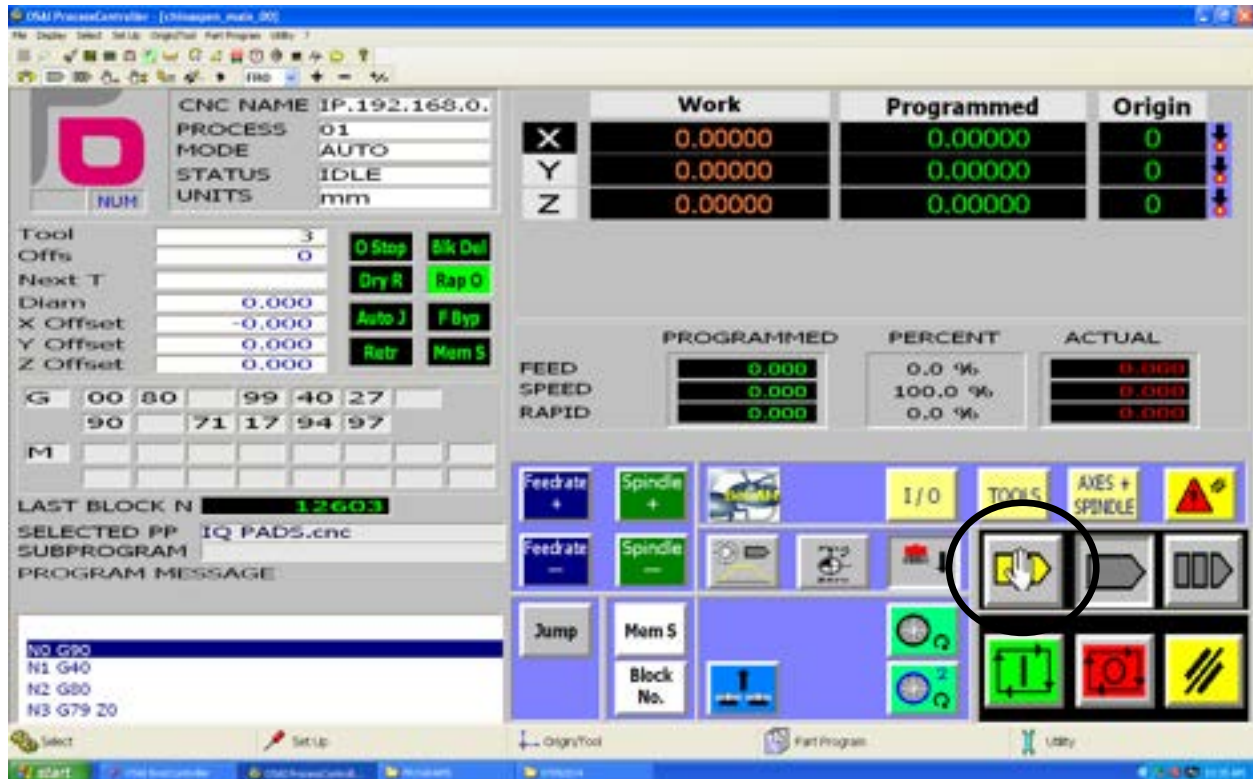


MDI

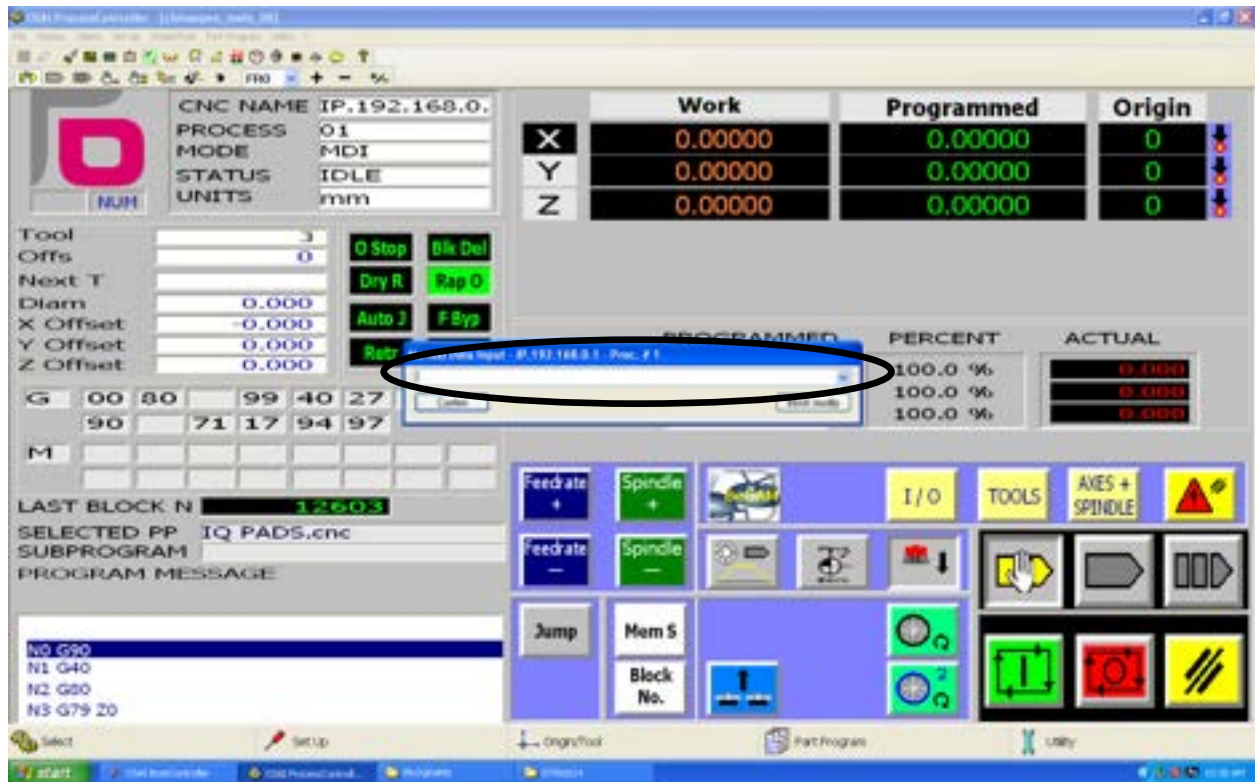
The MDI command allows the user to input G-Code and M-Code manually. Please refer to appendix section “MDI Commands” for a list of commands available. The following is the procedure for using MDI commands.



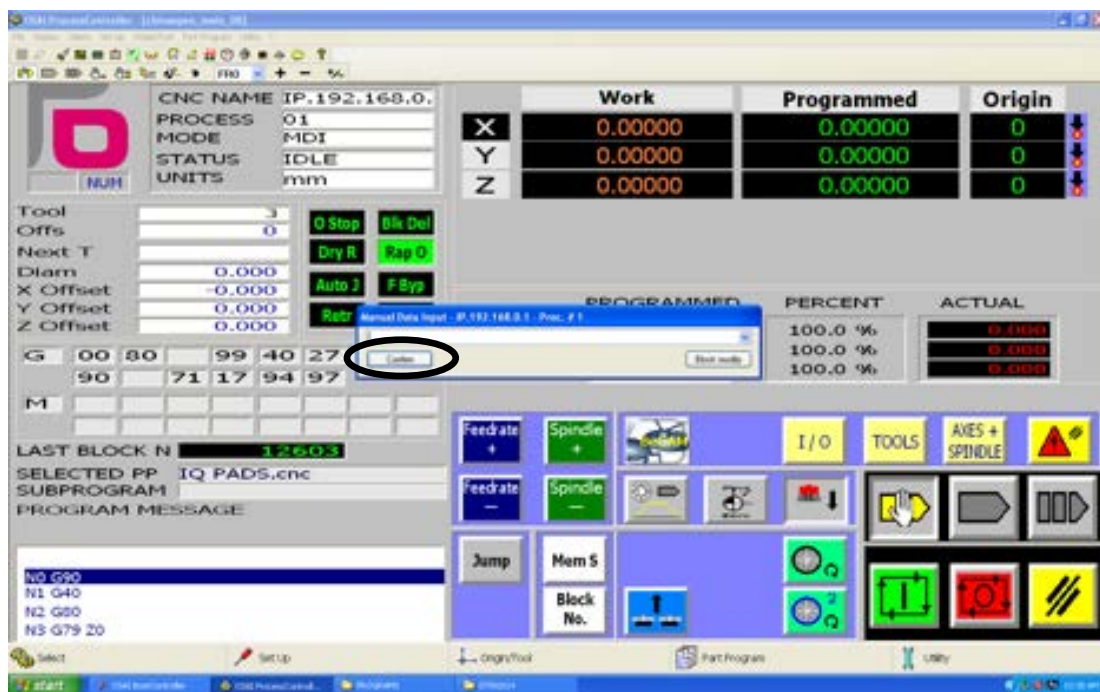
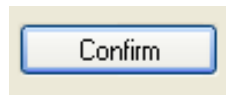
1. From the Main page, click the MDI button.



2. Manually input the command in the command box.



3. Click confirm.



4. Click the run button. The MDI entry is completed.



5. Click Auto mode button  to exit the MDI mode.

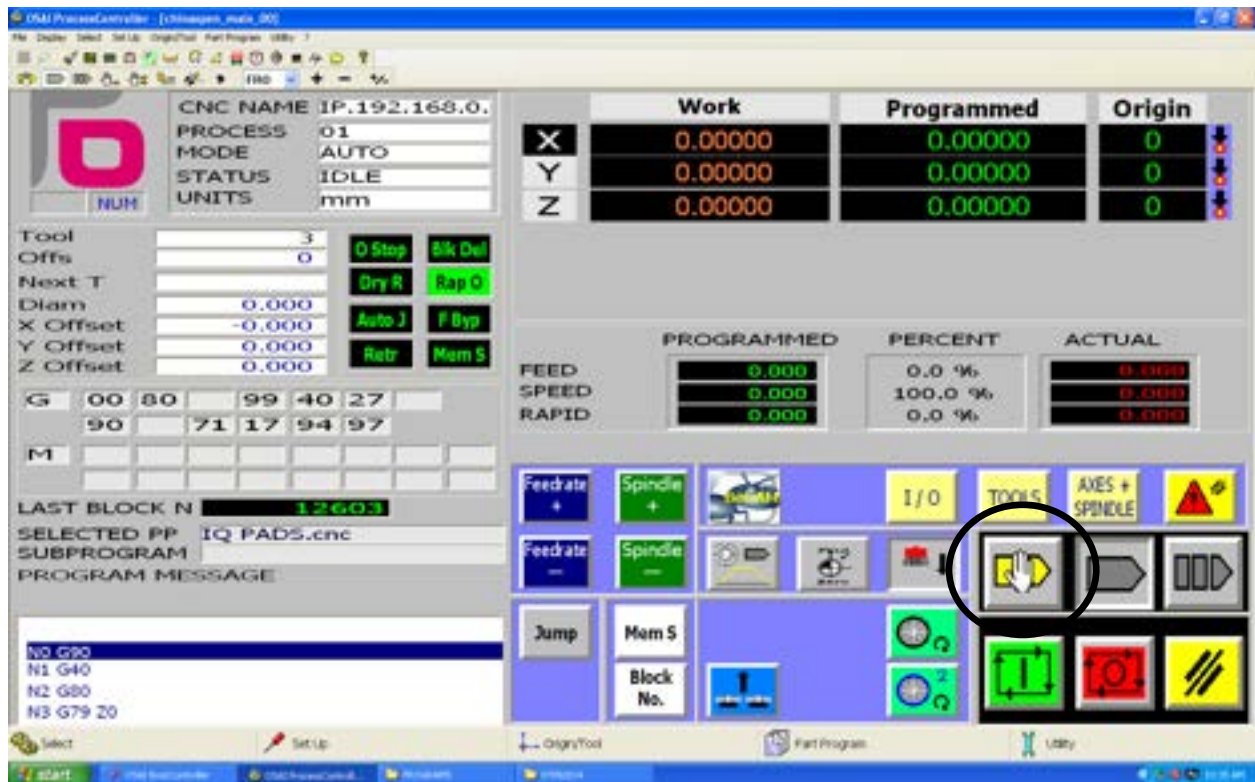


Automatic Tool Change

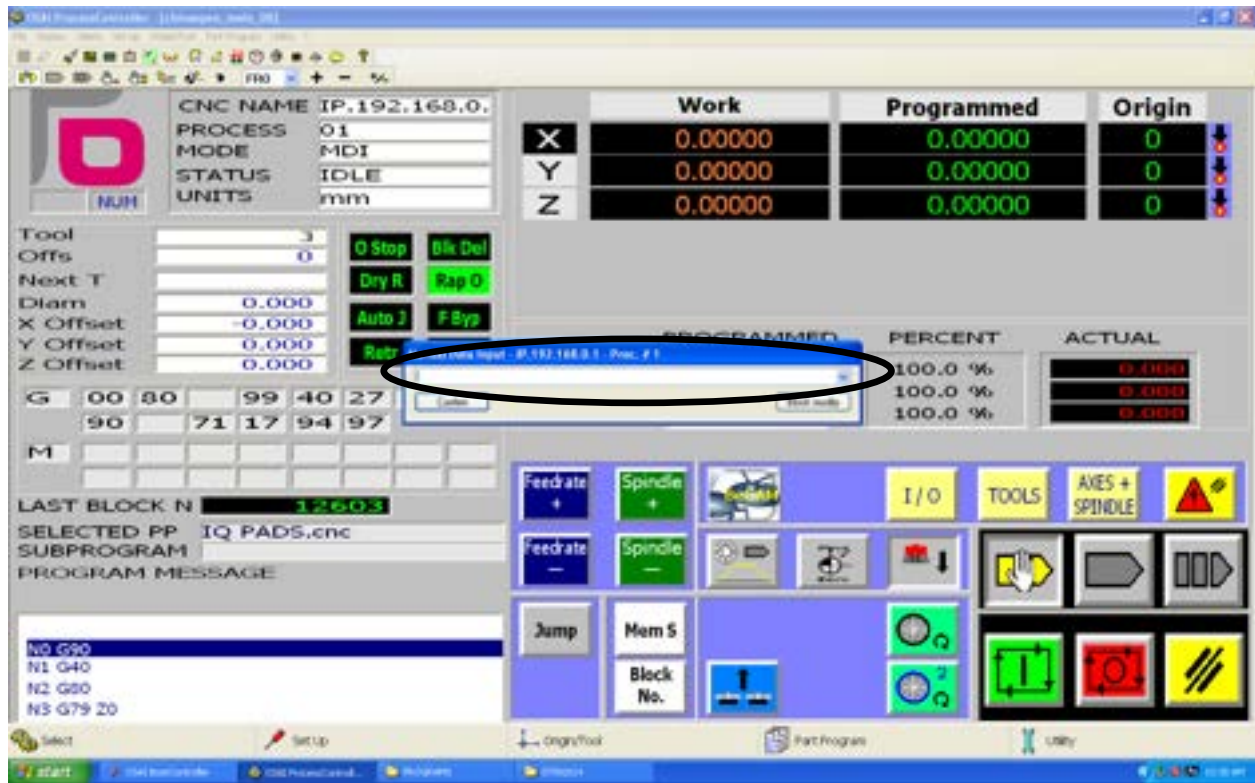
1. Make sure that it is safe for the machine to move.



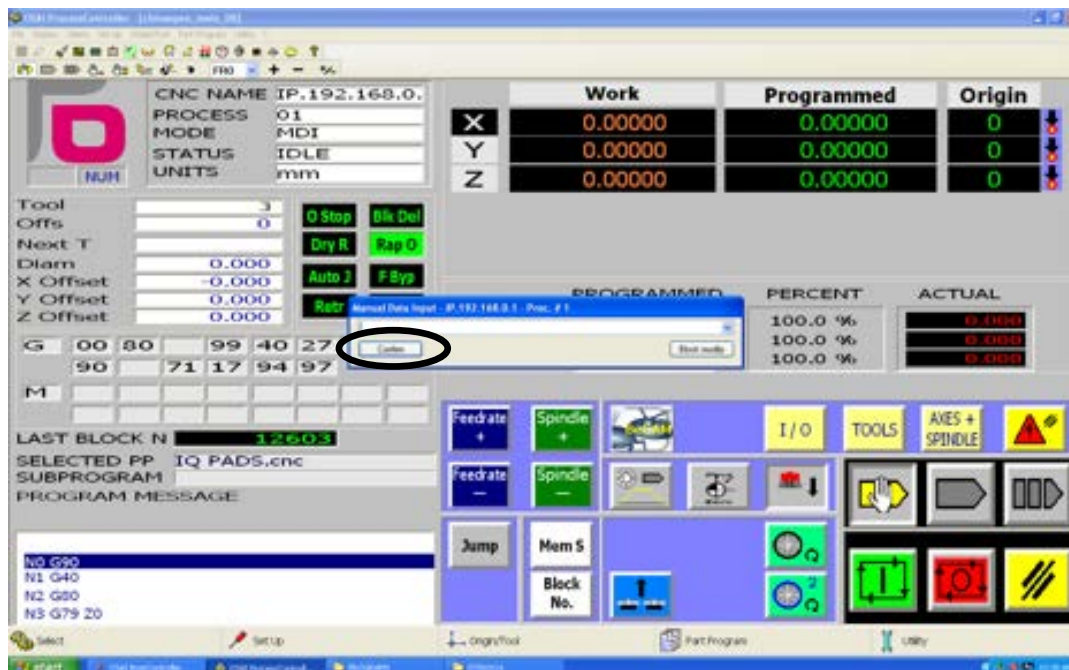
2. From the Main page, click the MDI button.



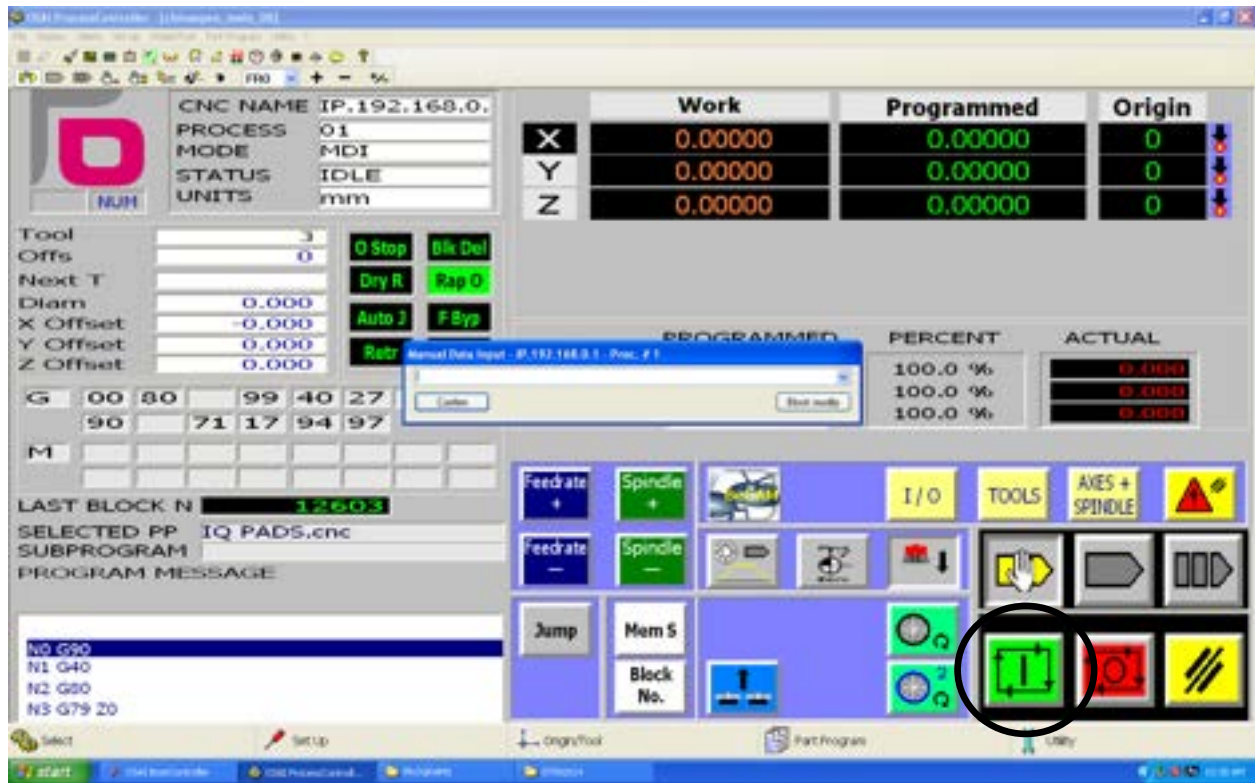
- Manually input the command “M6T3” in the command box. Where T3 refers to tool number 3.
To change to tool number 8, for example, enter “M6T8”



- Click confirm.



- Click the run button. The MDI entry is completed and the machine will change to the tool specified.



- Click Auto mode button  to exit the MDI mode.



Automatic Tool Touchoff

Automatic tool touch off utilizes a tool touch off switch to automatically detect the length of the tool in spindle. The following section includes the procedure.

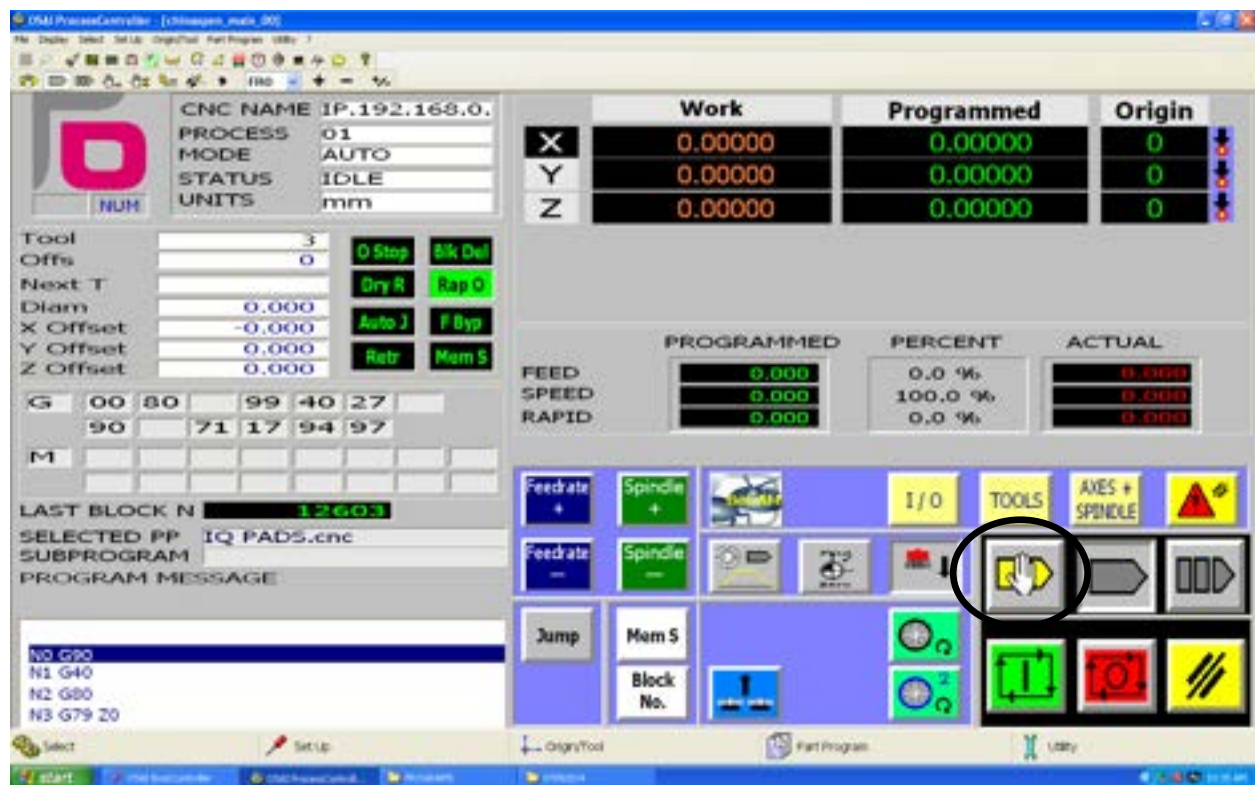
1. Make sure nobody can be harmed by any machine movements and that there is nothing that obstructs machine movements.
2. Follow the Automatic Tool Change procedure to change to the desired tool to touch off.



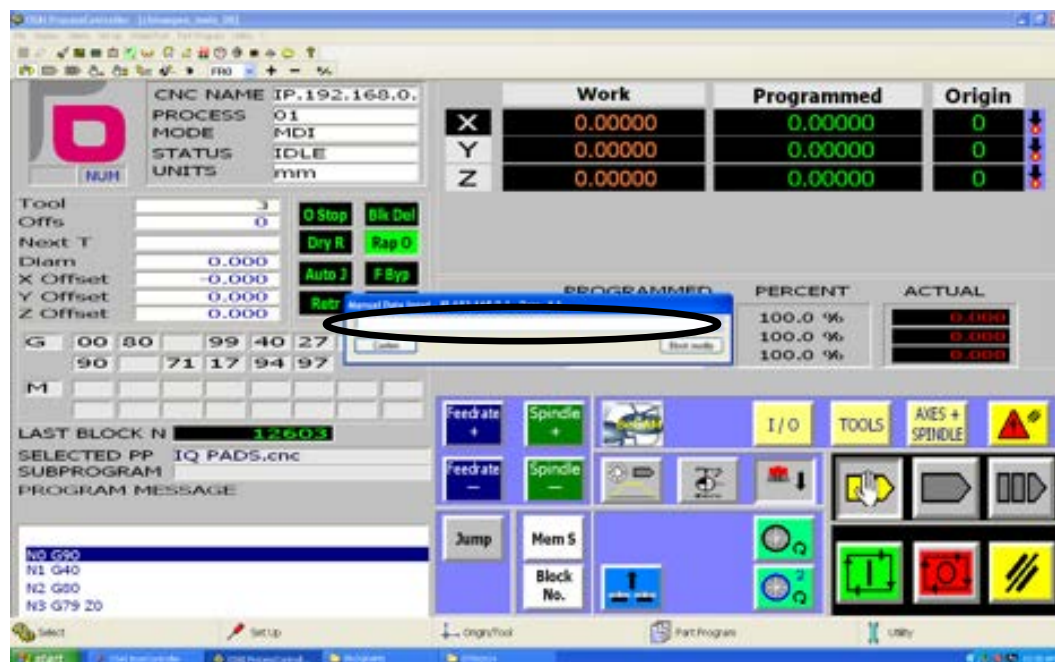
3. From the Main page, click the Reset button.



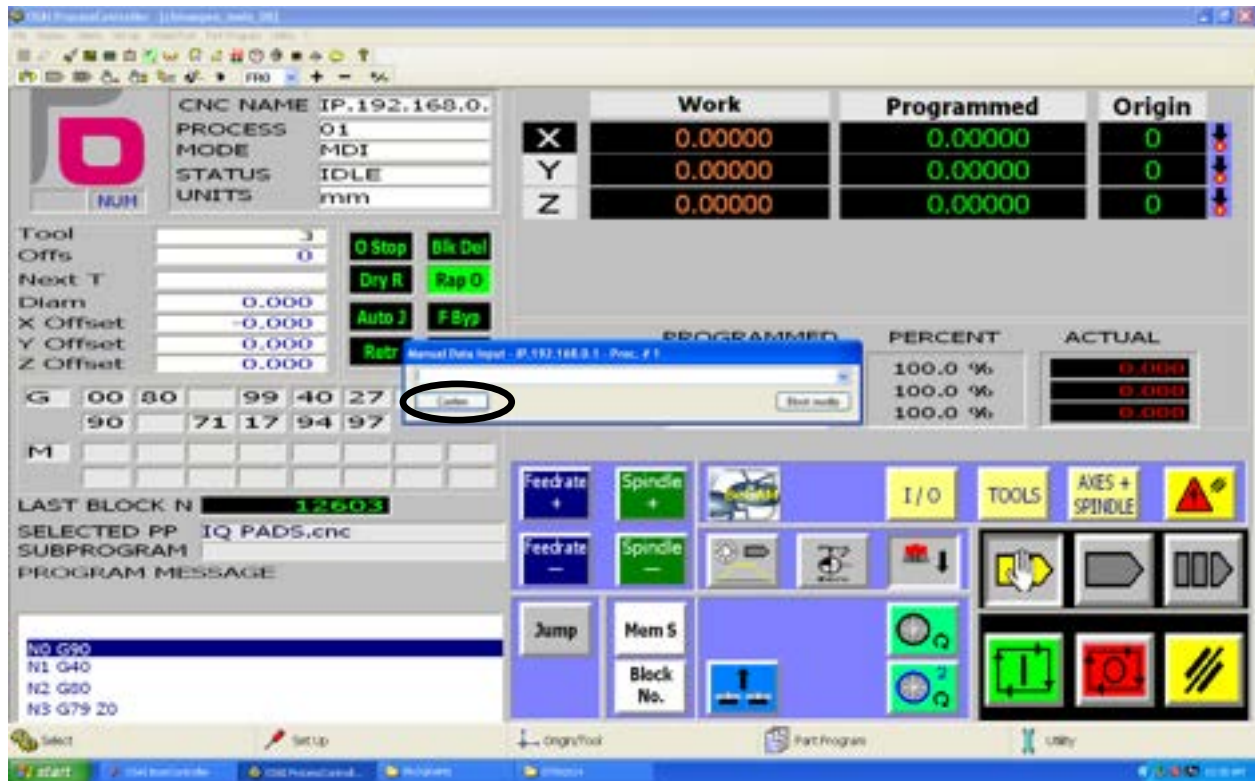
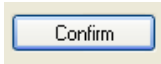
4. From the Main page, click the MDI button .



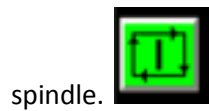
5. Manually input the command “M401T3” in the command box. Where T3 refers to tool number 3 currently in spindle. If tool 8 is in spindle, for example, enter “M401T8.”



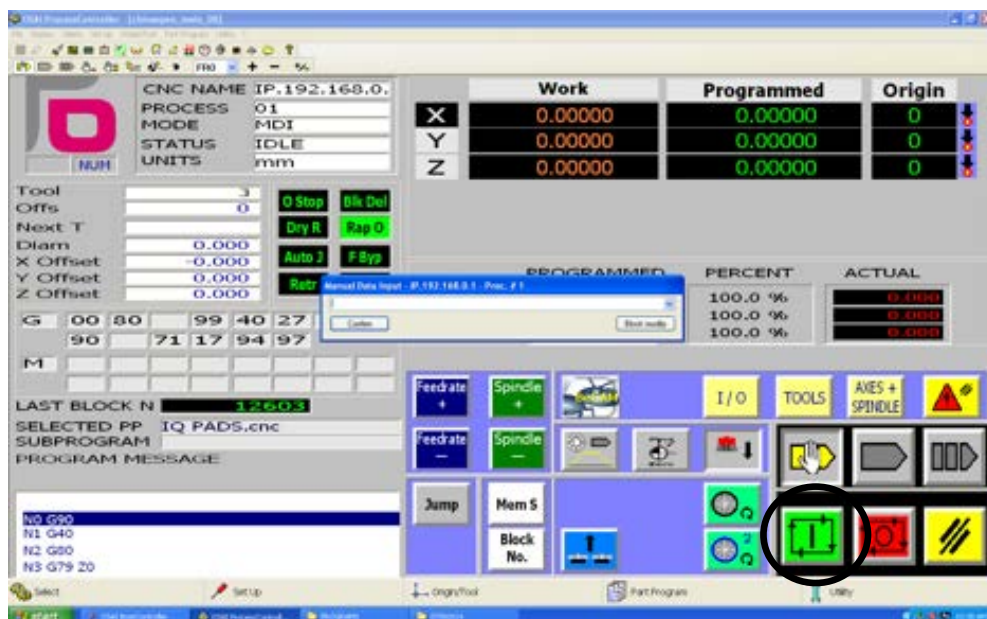
6. Click confirm.



7. Click the run button. The MDI entry is completed and the machine will touch off the tool in



spindle.





8. Click Auto mode button  to exit the MDI mode.



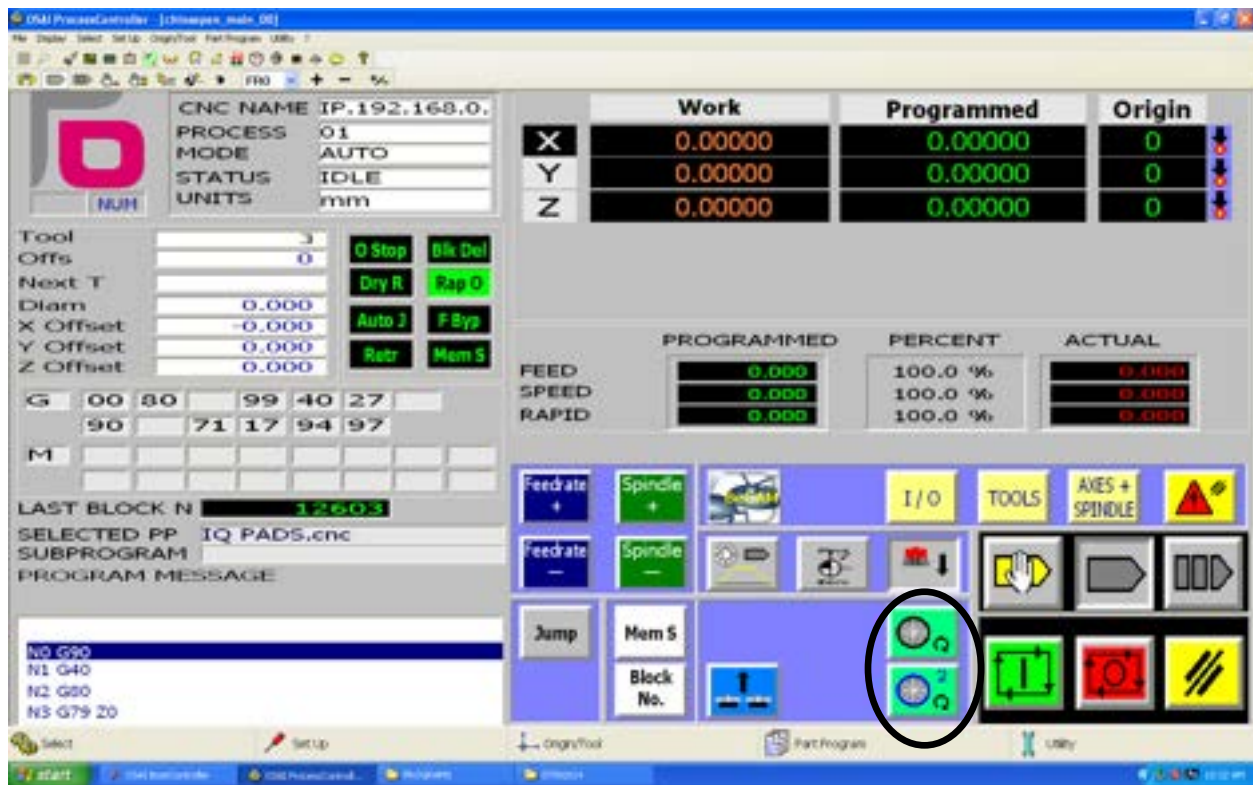
Start from Block

Start from block function allows the user to start running a G-Code file at a specific line. The following is the procedure to use it. Note that the modal G-Code are usually at the start of the program and might be skipped when doing Start from Block. Also, the Reset button resets the modal G-Code to default.

1. Activate a G-Code file by following the instructions in the "Running a G-Code File" section.



2. If necessary, turn on the vacuum pumps by clicking on the vacuum pump buttons.

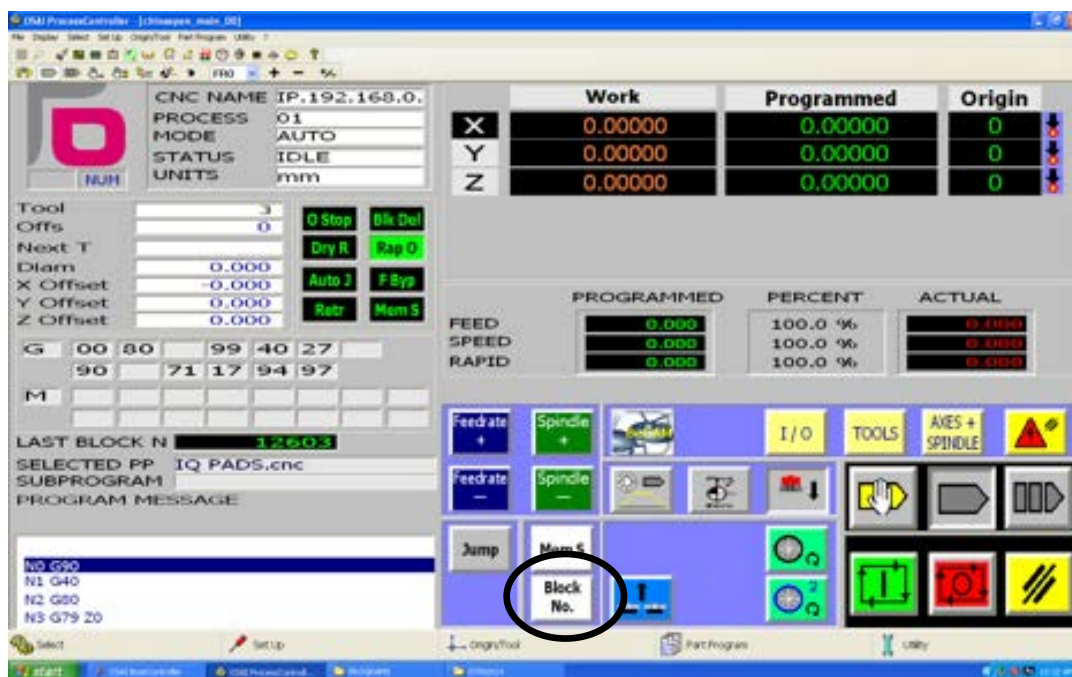
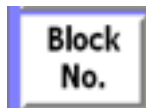


3. Make sure the work piece is secured and that machine movements will not damage any person or object.

4. Make sure the machine is in Auto Mode by clicking on the Auto Mode button.



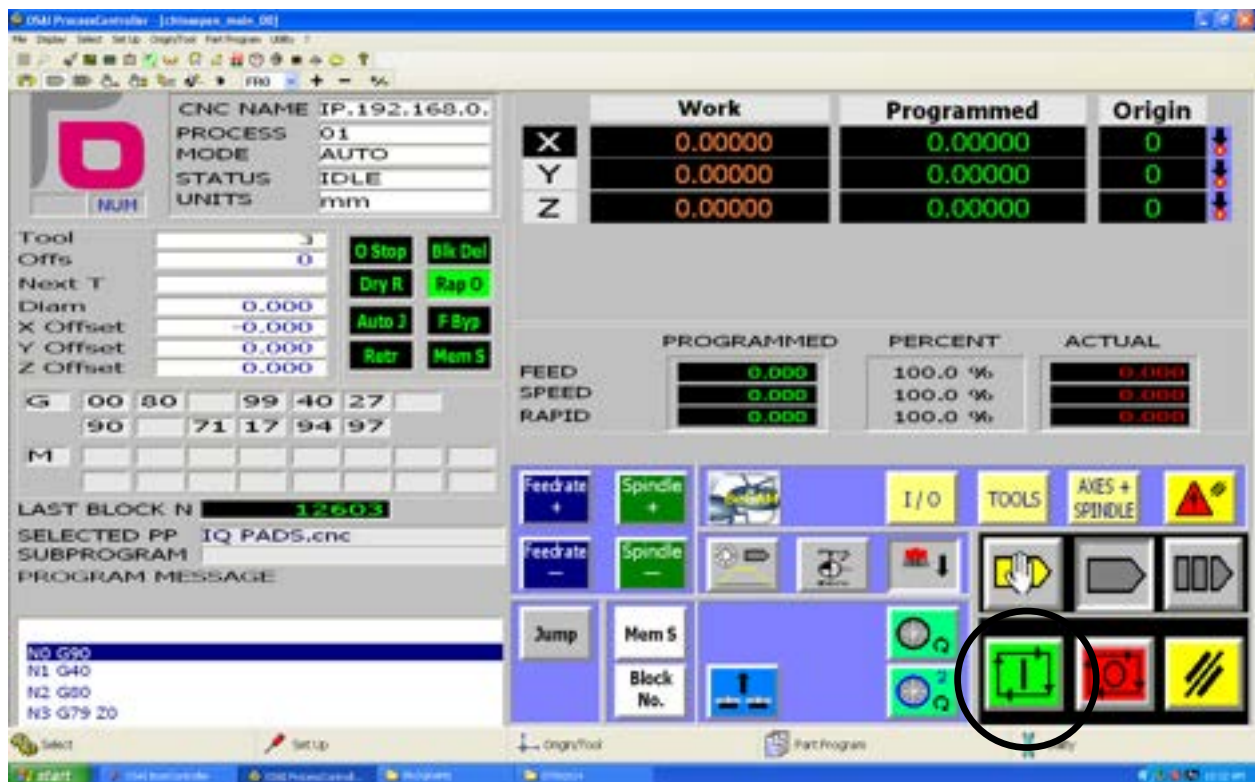
5. Click on the Block No. button.



6. Enter the desired block, label, or line number to start from. Then click “OK”.




7. Click the Run button to start the program.



Memory Search Function

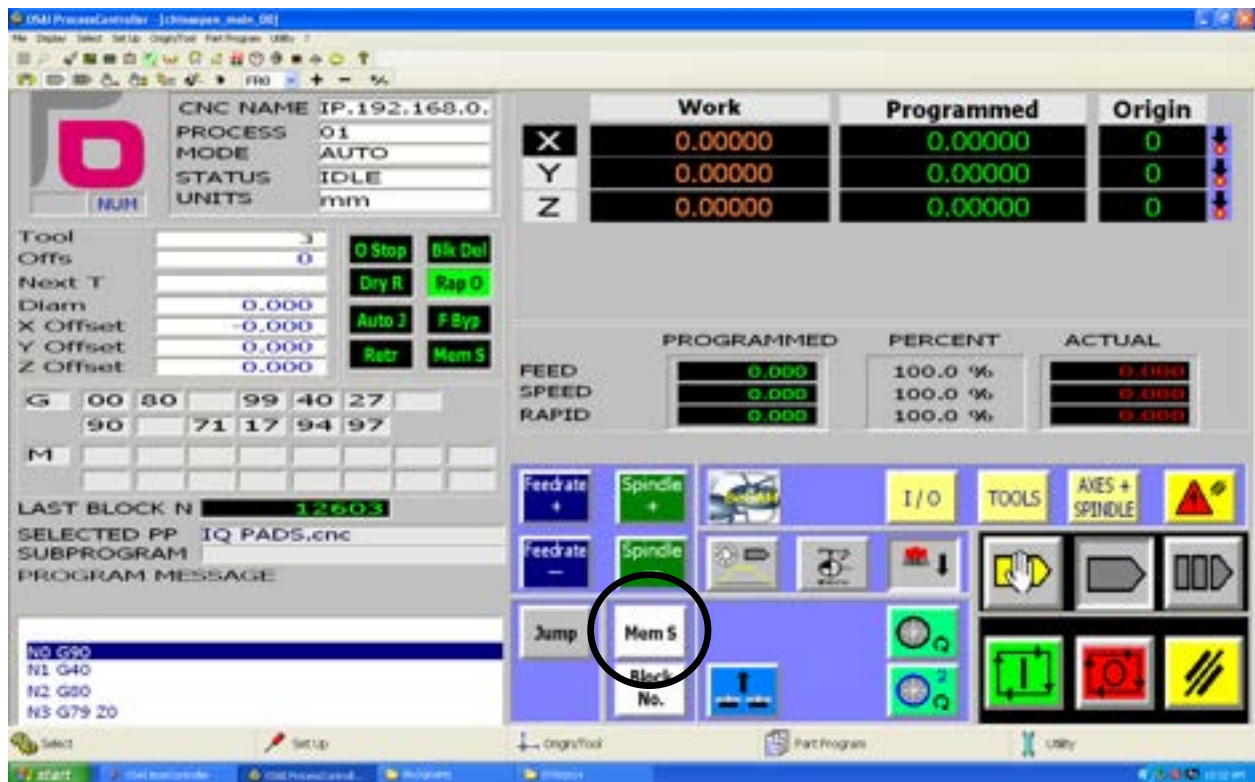
The Memory Search function is used to find the last executed line in a file. It can be useful in the event of a power outage.

1. Make sure the machine is in Auto mode by pressing the Auto mode button .

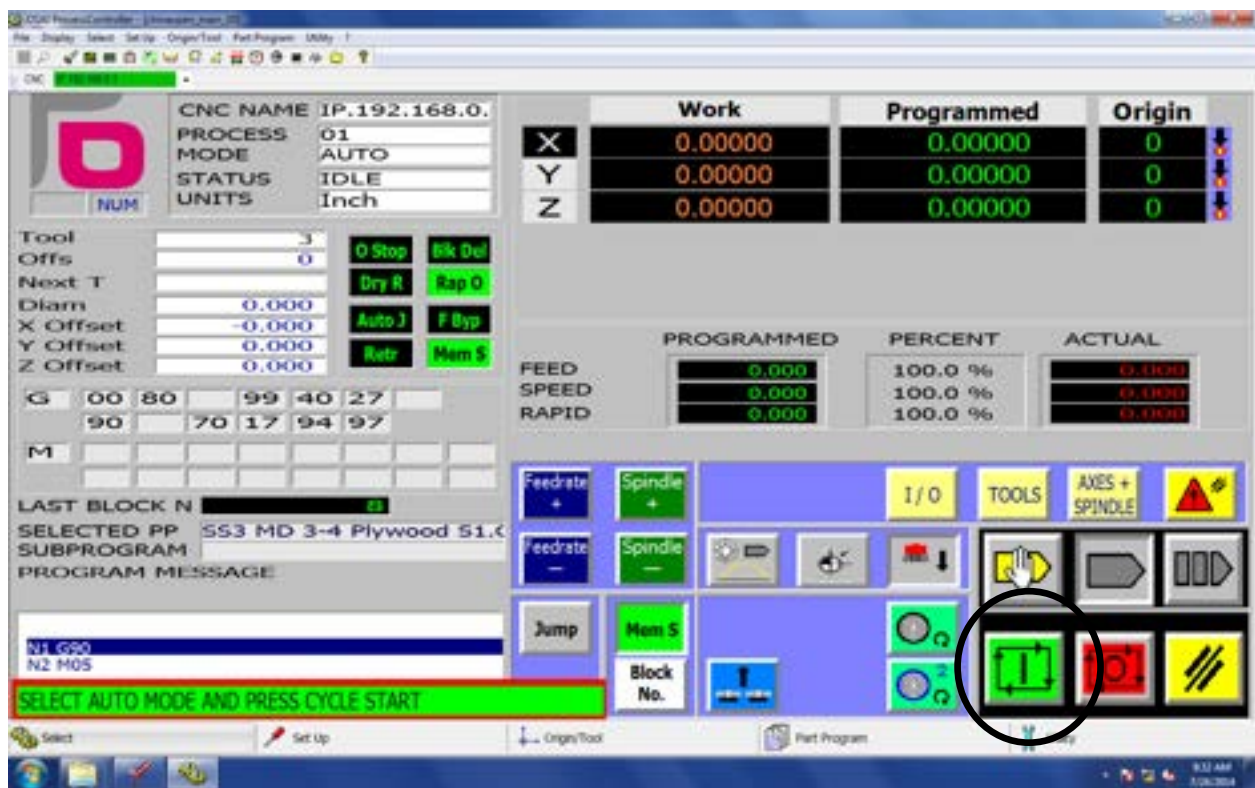


2. Click on the Memory Search button to depress it.

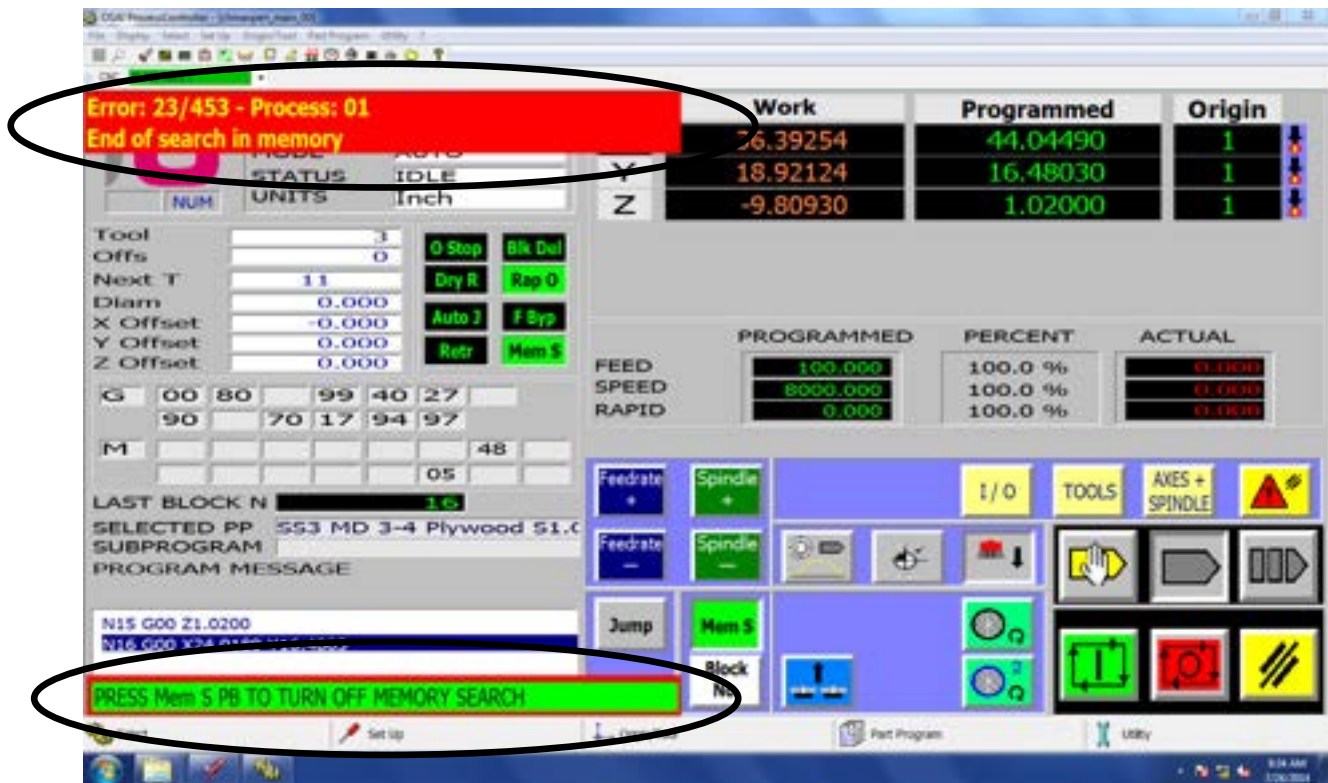




3. Click on the run button. The machine will start the memory search.



- Wait for the process to complete. Once the memory search concluded, the machine will display “End of search in memory” on upper left corner. The last ran G-code will be highlighted in the Active G-Code area.



- Click the reset button to clear the message and click the Memory Search button to disable Memory Search mode.



Troubleshooting

FAQ

The menu bar is missing in the Process Controller. – Press Ctrl+M to toggle it on and off.



File Display Select Set Up Origin/Tool Part Program Utility ?

MPG run mode is grayed out –make sure the machine is in Auto mode not MPG mode on the Axis screen, make sure there is an active program on the Main screen.

The machine freezes during tool change – Raise the dust hood and check if dust hood up sensor lights up. If it does not light up, the sensor is faulty.

The process controller window does not response to clicking. You might be in the process of entering a number into a field. Enter 0 on the keyboard then press Enter on the keyboard.

088 Spindle Stop Error – the spindle might still be running. Please make sure the spindle has stopped then check that the relay functions correctly.

WRONG fixed cicle motion lengths - make sure the z or w axis can reach the R value for a canned drilling cycle G80

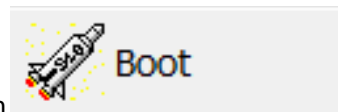
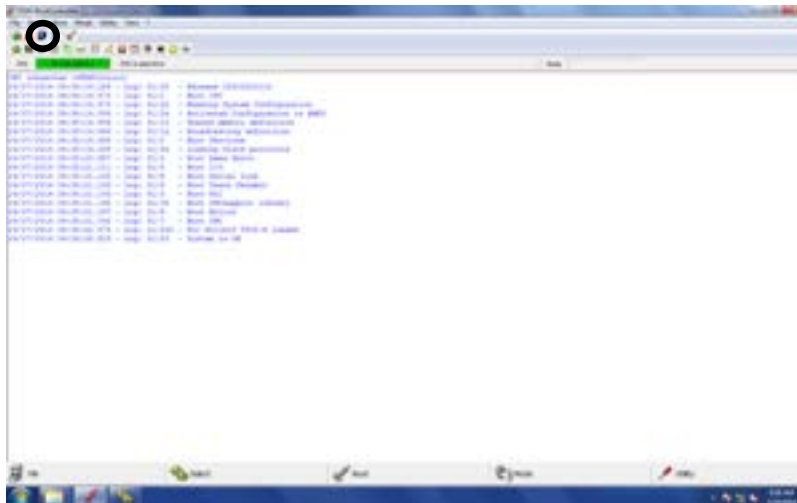
System recovery sets the machine settings according to a specific recovery file. It can be used to change machine to its previous state when the machine setting is accidentally changed.

-

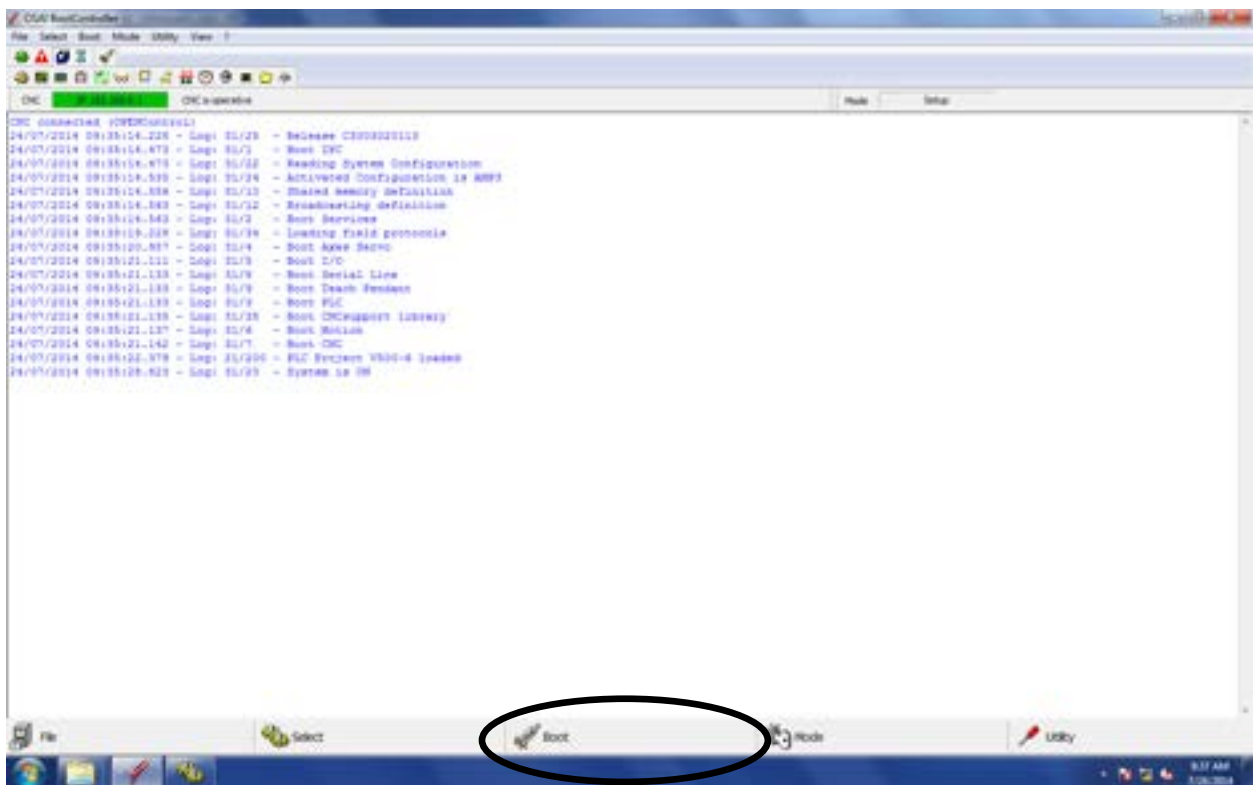
-
- The screenshot displays the Fusion 360 CAM software interface, specifically the CNC control panel. The interface is divided into several sections:
- CNC Information:** CNC NAME: IP.192.168.0., PROCESS: 01, MODE: AUTO, STATUS: IDLE, UNITS: Inch.
 - Work, Programmed, and Origin Coordinates:**

	Work	Programmed	Origin
X	0.00000	0.00000	0
Y	0.00000	0.00000	0
Z	0.00000	0.00000	0
 - Tool and Offsets:** Tool: 3, Offsets: 0.000, 0.000, 0.000, 0.000.
 - G-Code and M-Code:** G 00 80 99 40 27, 90 70 17 94 97, M.
 - FEED SPEED RAPID:** 100.0 %, 100.0 %, 100.0 %.
 - Buttons:** O Stop, Bk Del, Dry R, Rap O, Auto J, F Byp, Retr, Mem S, Feedrate +, Spindle +, I/O, TOOLS, AXES + SPINDLE, Feedrate -, Spindle -, Jump, Mem S, Block No., and various axis movement buttons.
 - Instruction:** A red box at the bottom contains the text: "SELECT AUTO MODE AND PRESS CYCLE START".
 - Status Bar:** Shows "Set Up" and "Origin Tool" buttons.

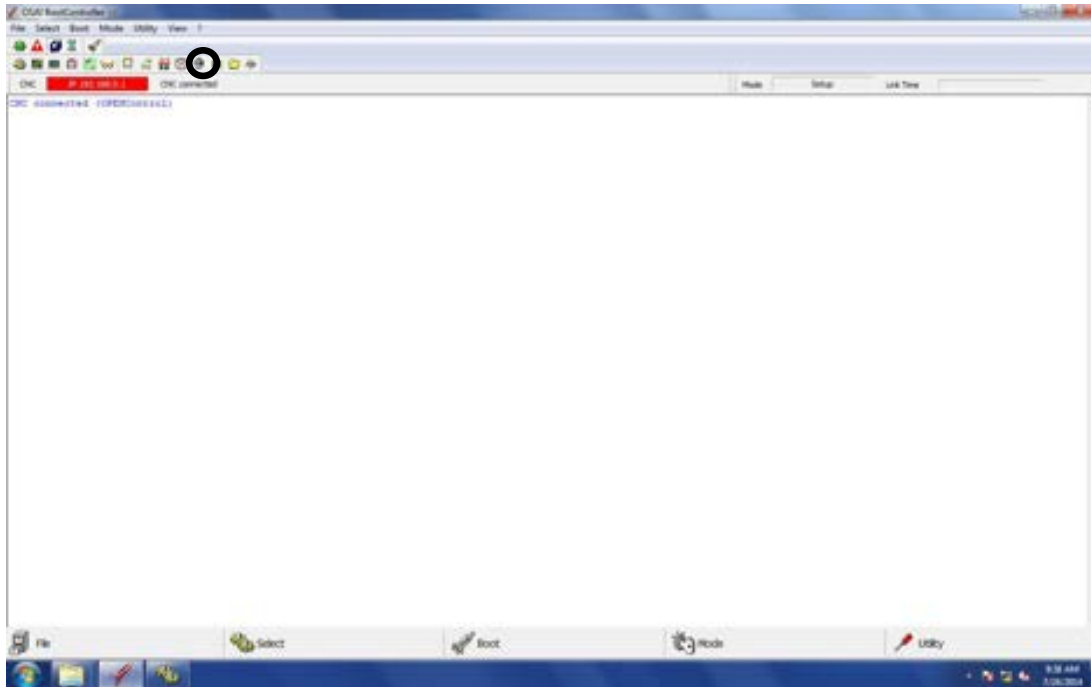
- 



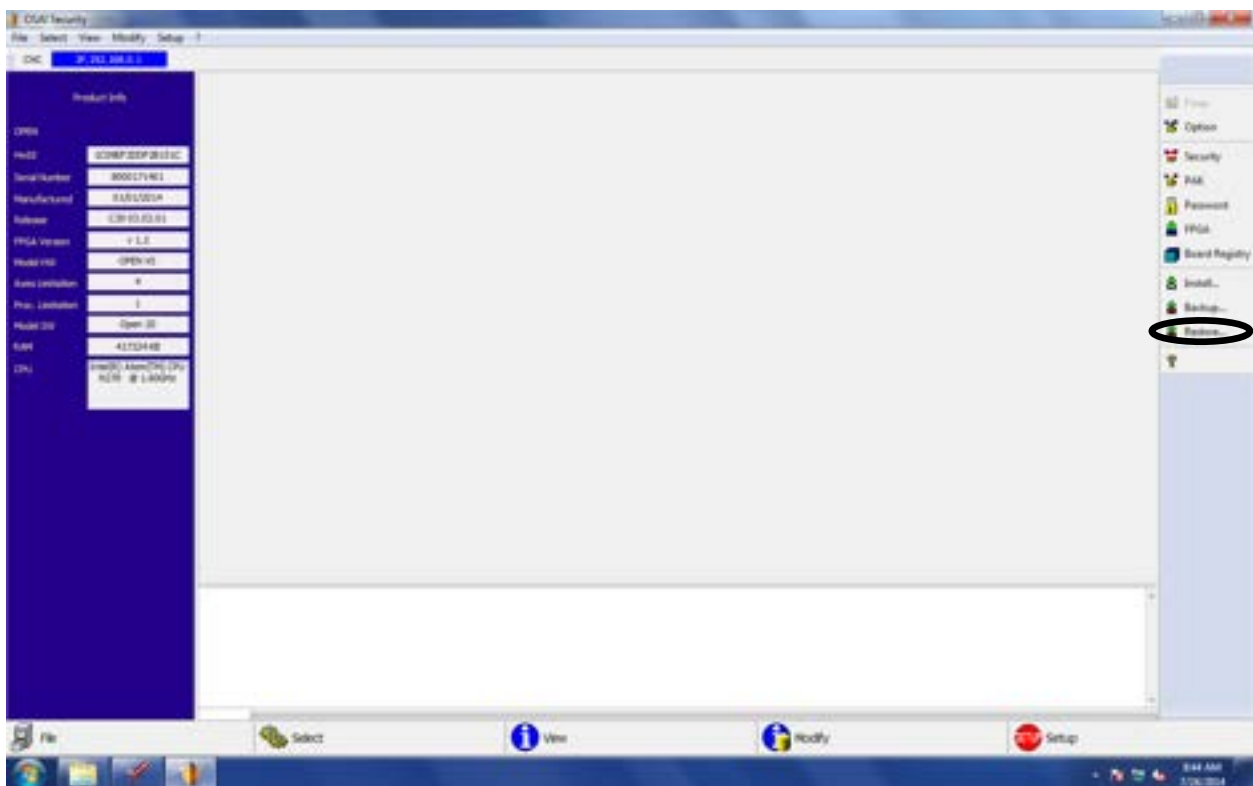
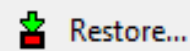
4. Click the boot button then click "Shutdown."



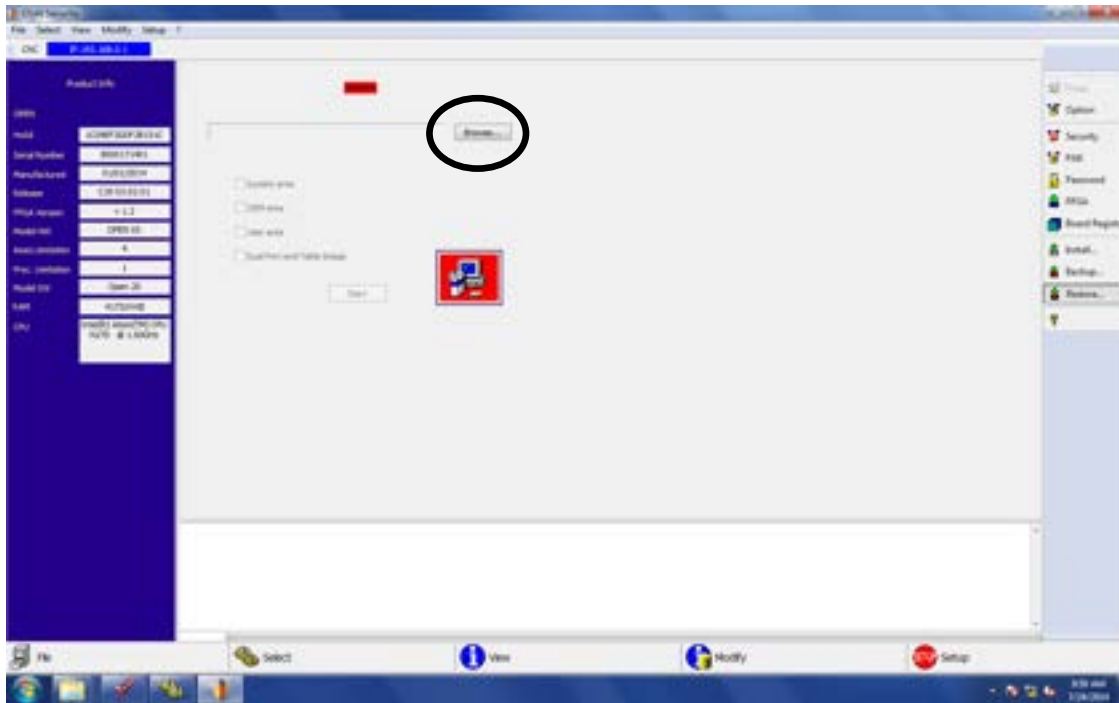
5. Click OK in the pop-up window. This will restart the controller.



8. From the security window, click on the Restore button.



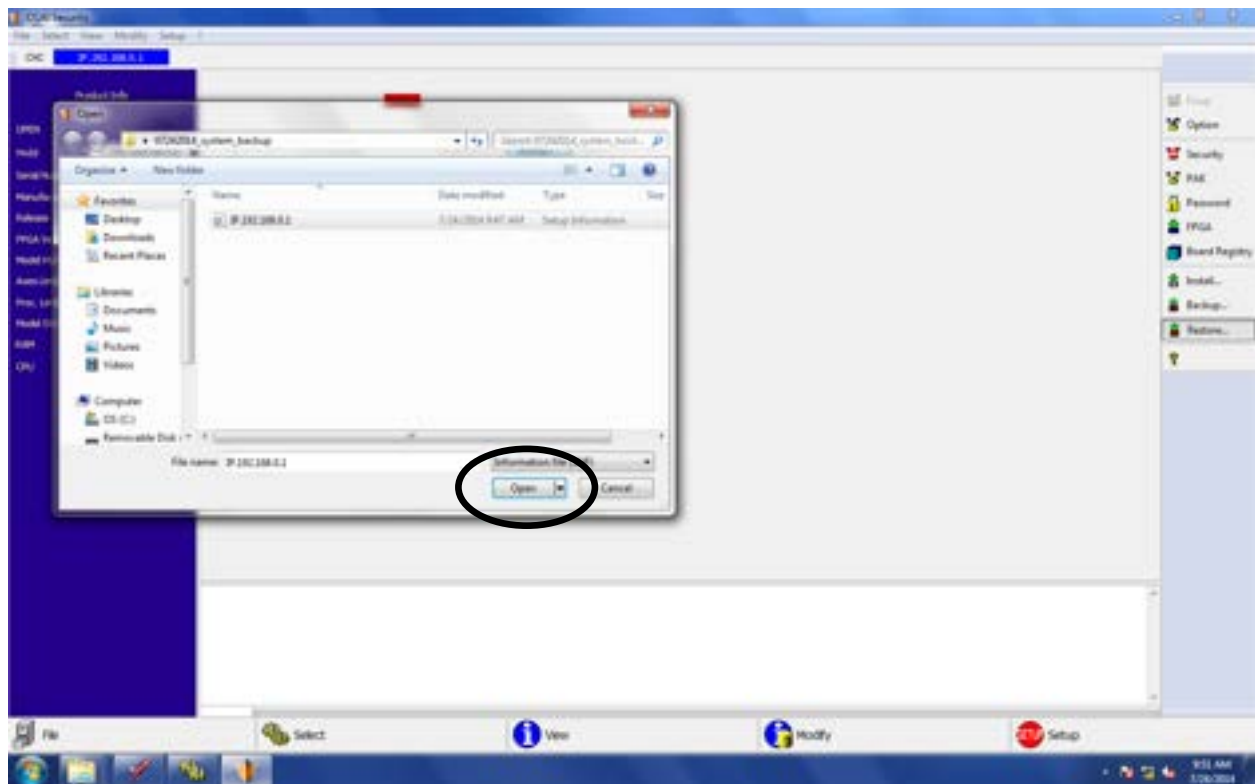
9. Click on the browse button.



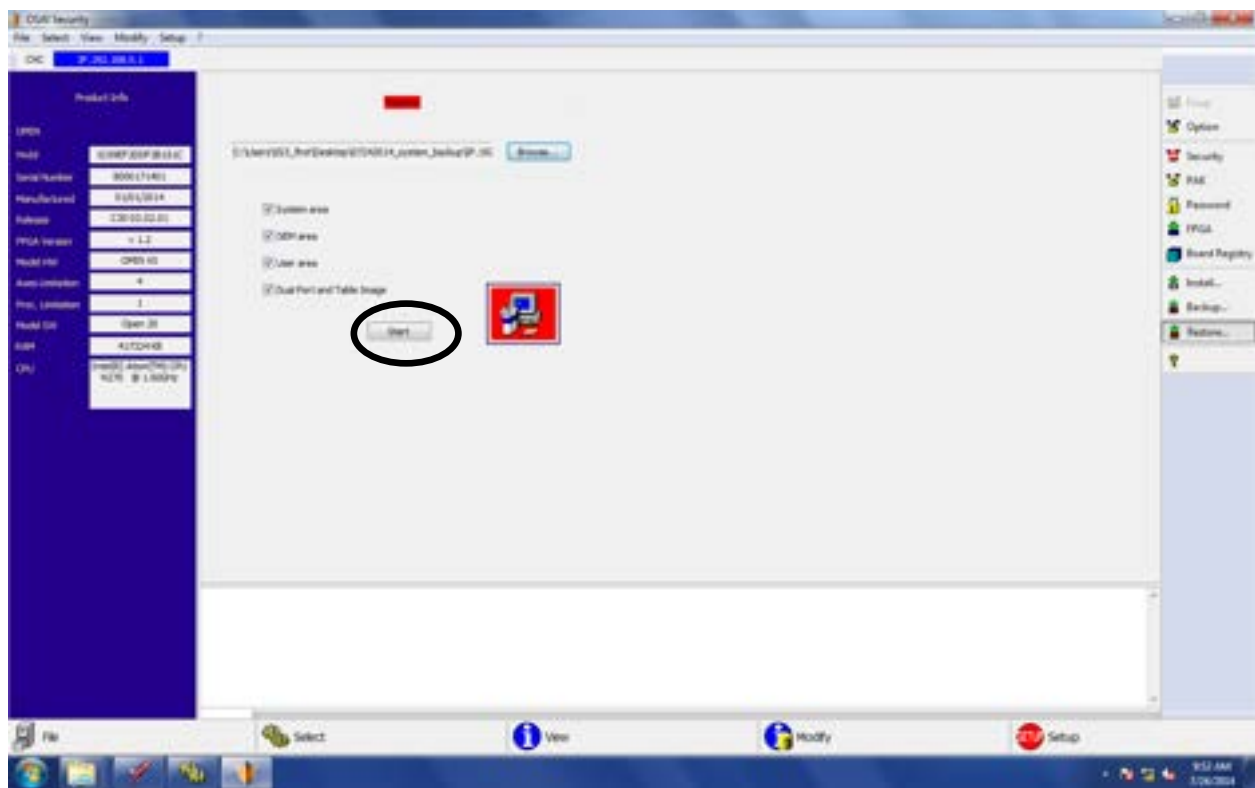
10. Navigate to the desired system backup file, it should be named "IP.192.168.0.1".



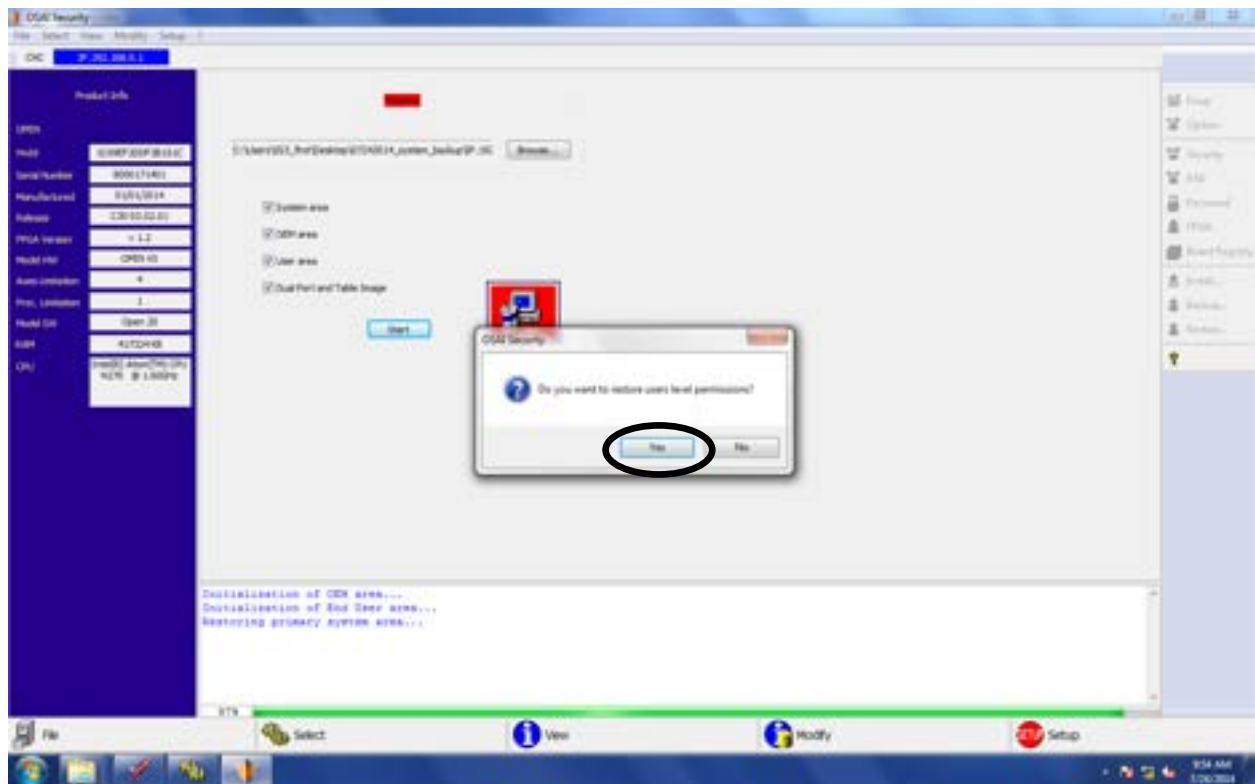
11. Click "Open"



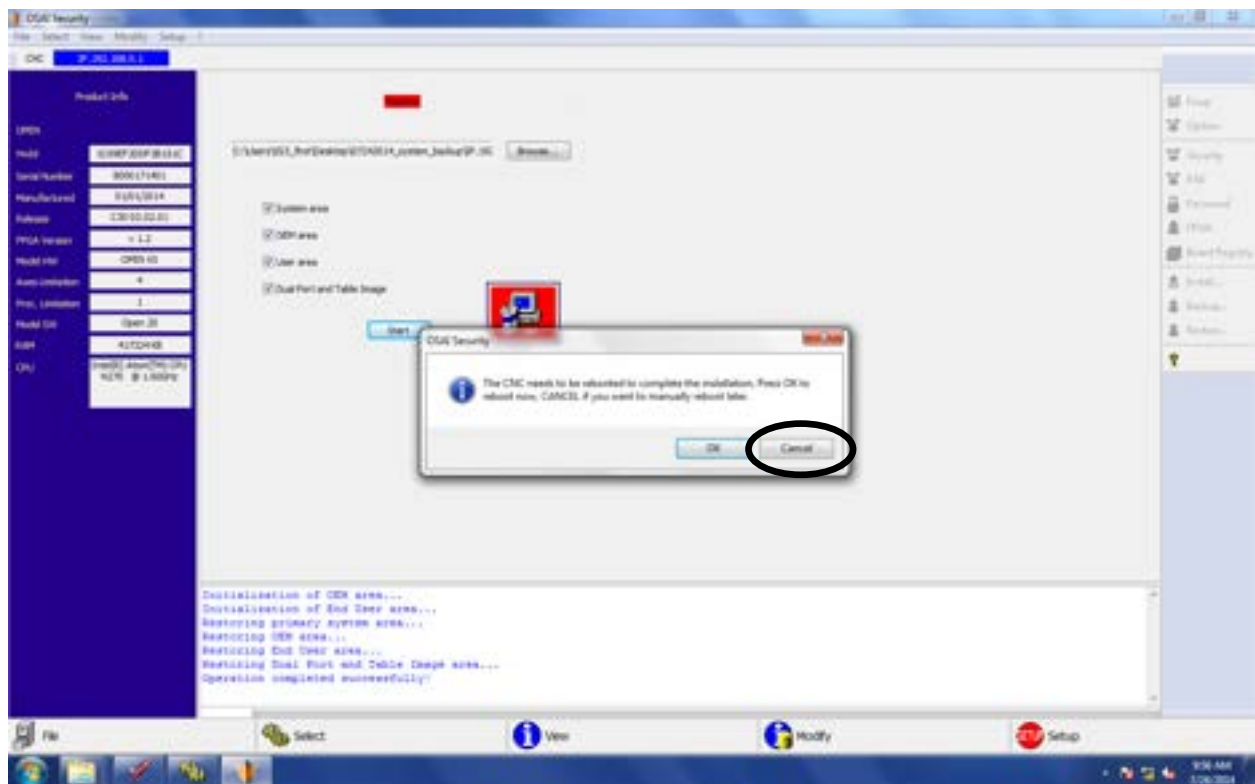
12. Click “Start,” the system will begin the system restore.



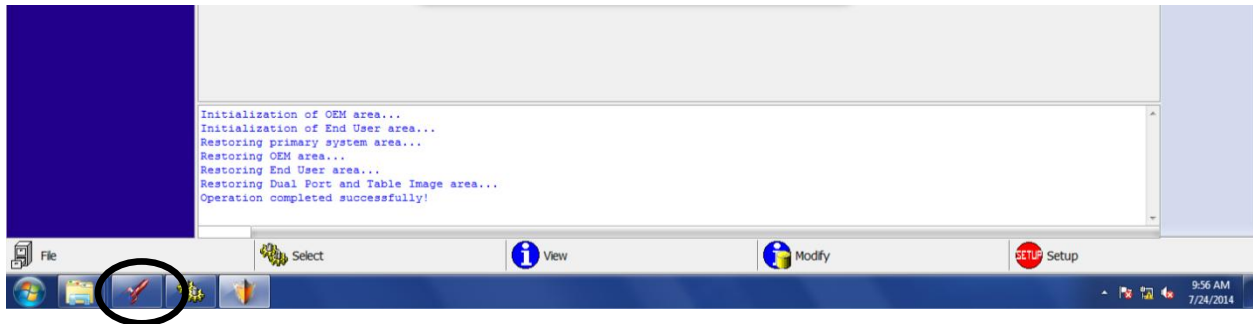
13. Click “Yes” when prompted with the message “Do you want to restore users level permissions?”



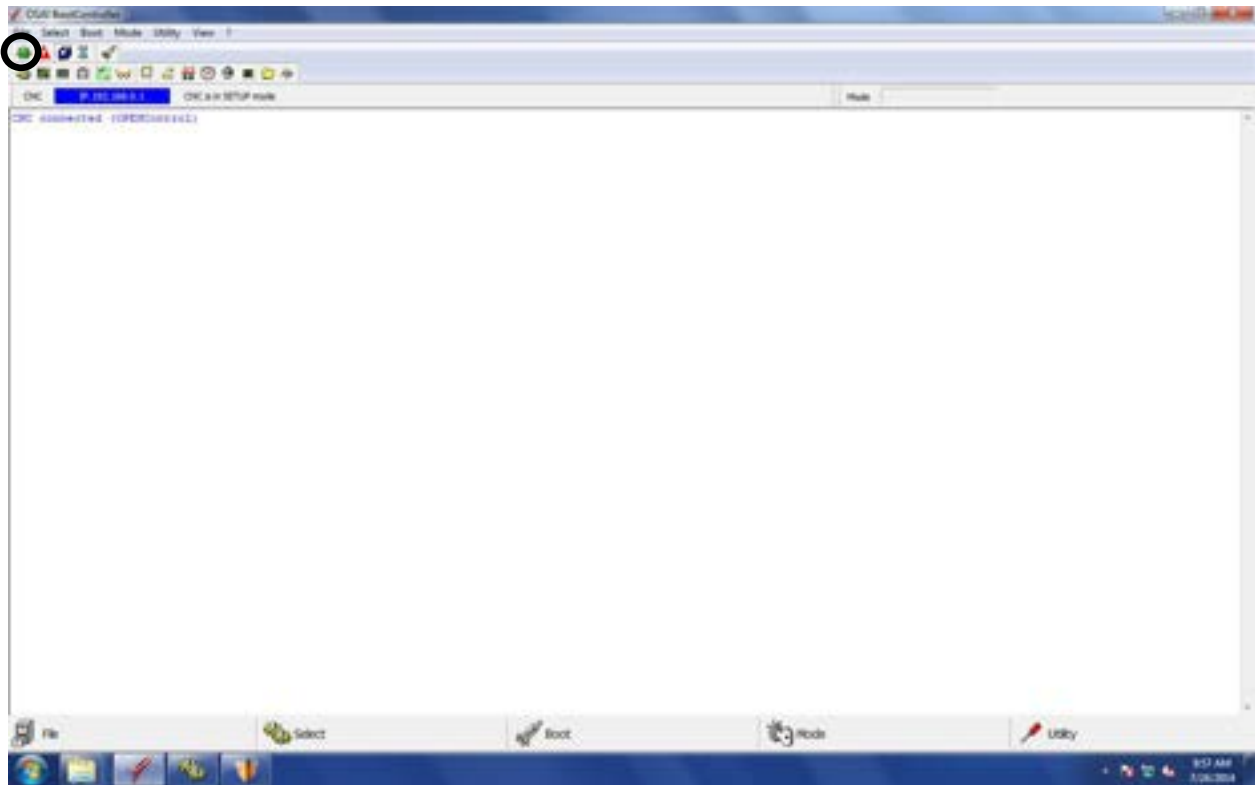
14. Click “No” when prompted with the message “The CNC needs to be rebooted to complete the installation. Press OK to reboot now, CANCEL if you want to manually reboot later.”



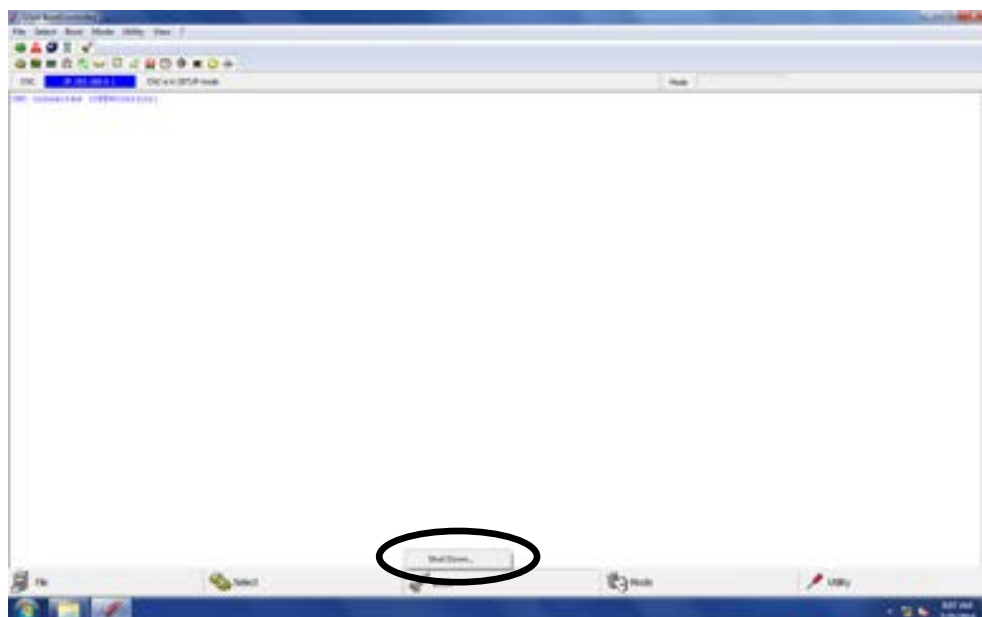
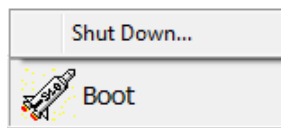
15. Click on the rocket symbol to access the boot control window.



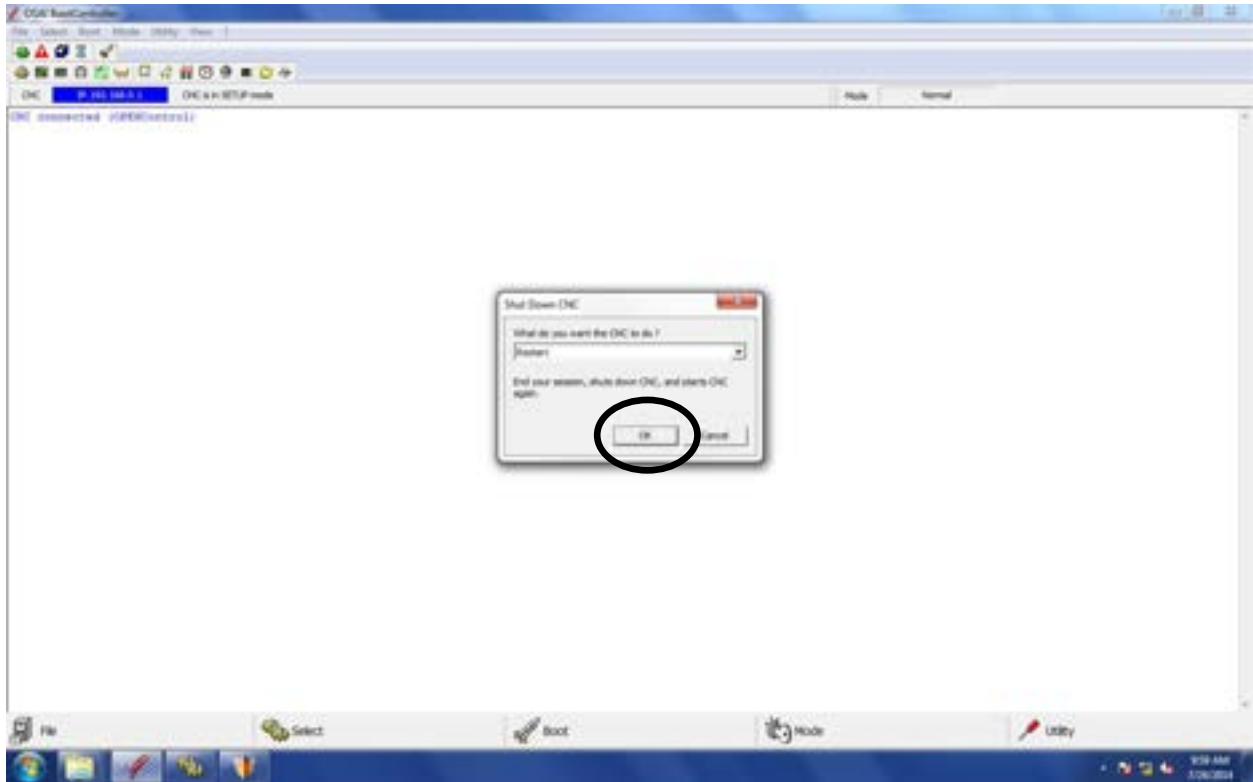
16. Click on the Normal mode button.



17. Click Boot then click Shutdown.



18. Click “OK” to restart the controller.

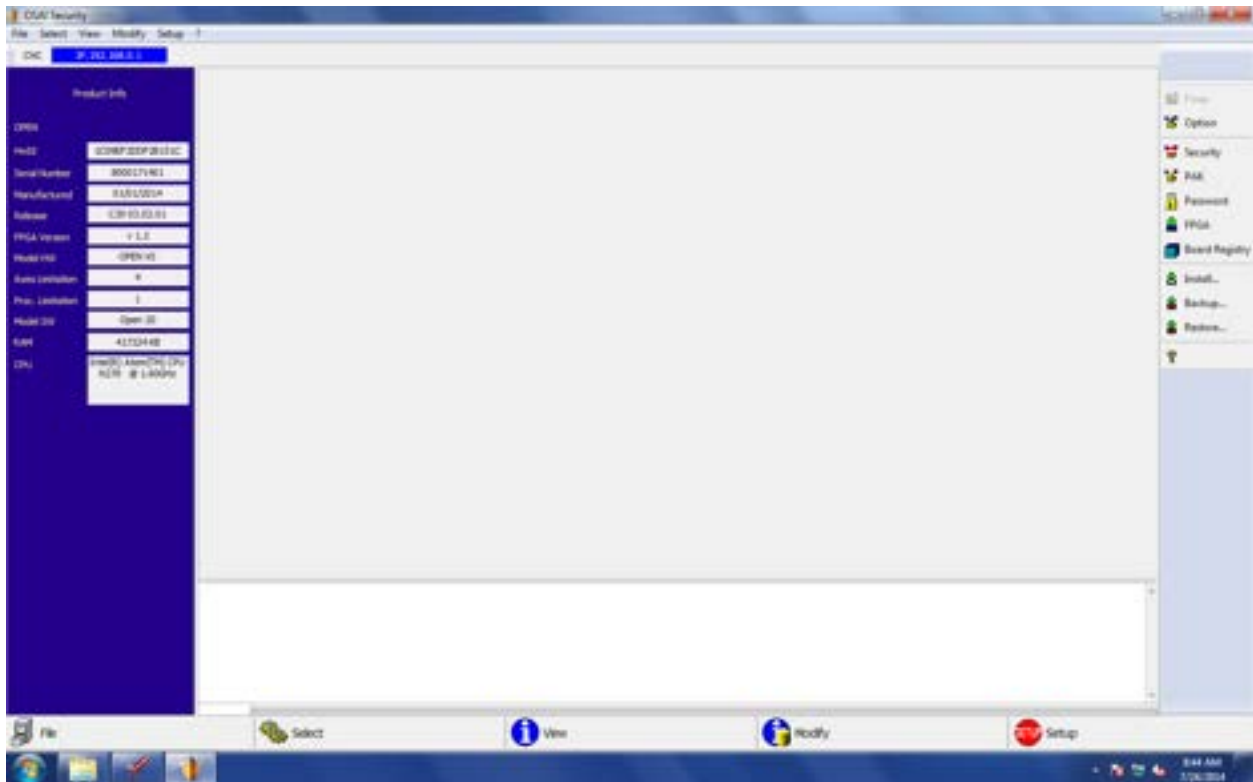


19. Wait for the controller to restart. The system recovery is now completed.

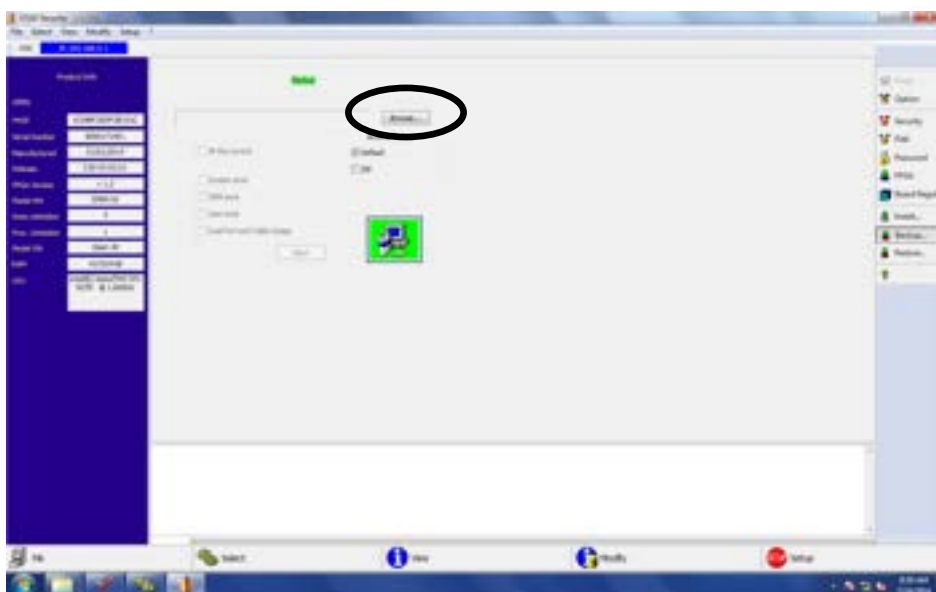
System Backup

System backup saves the system parameters into a file which can be used later to restore the system parameter.

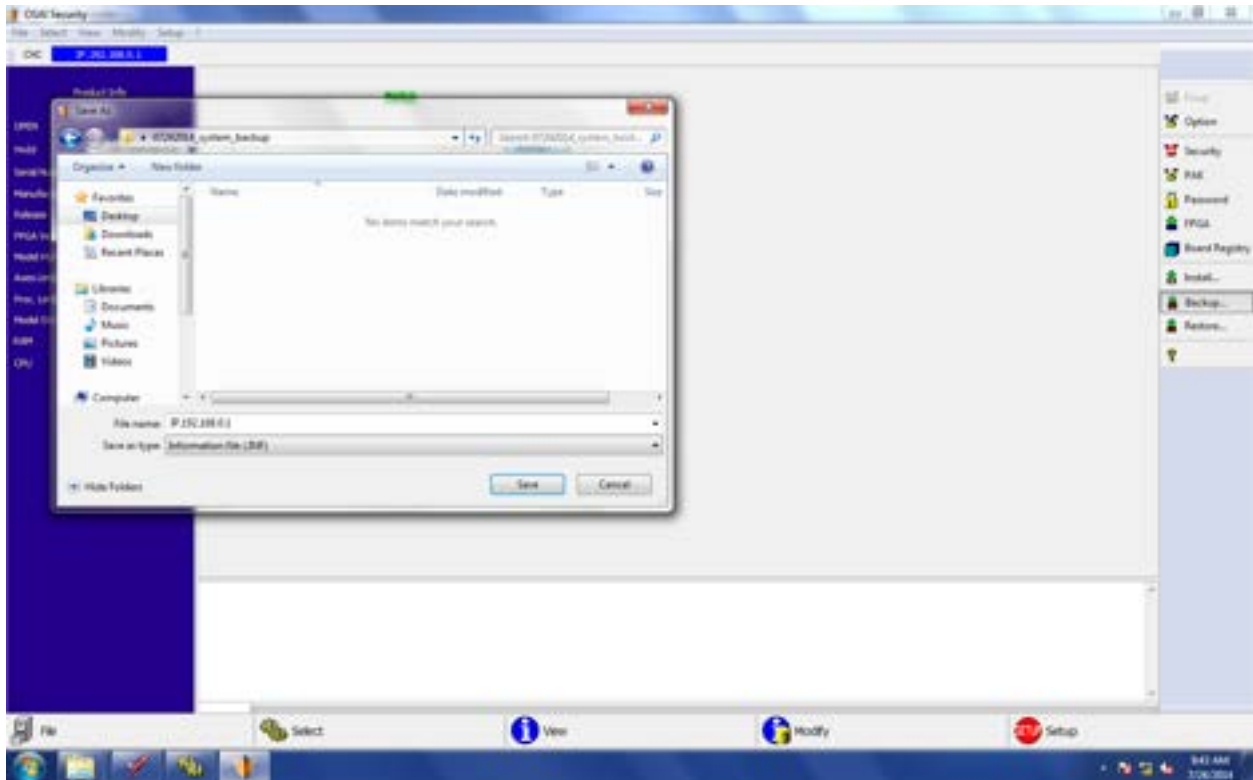
1. Follow the system recovery procedure to access the security window.



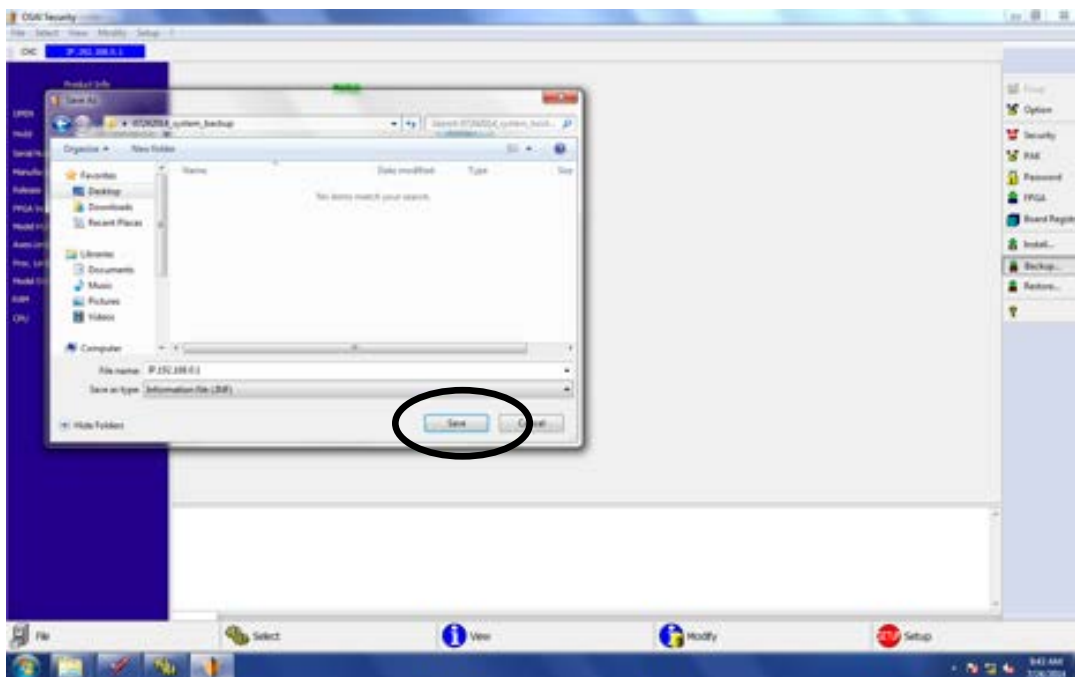
2. Click on Backup



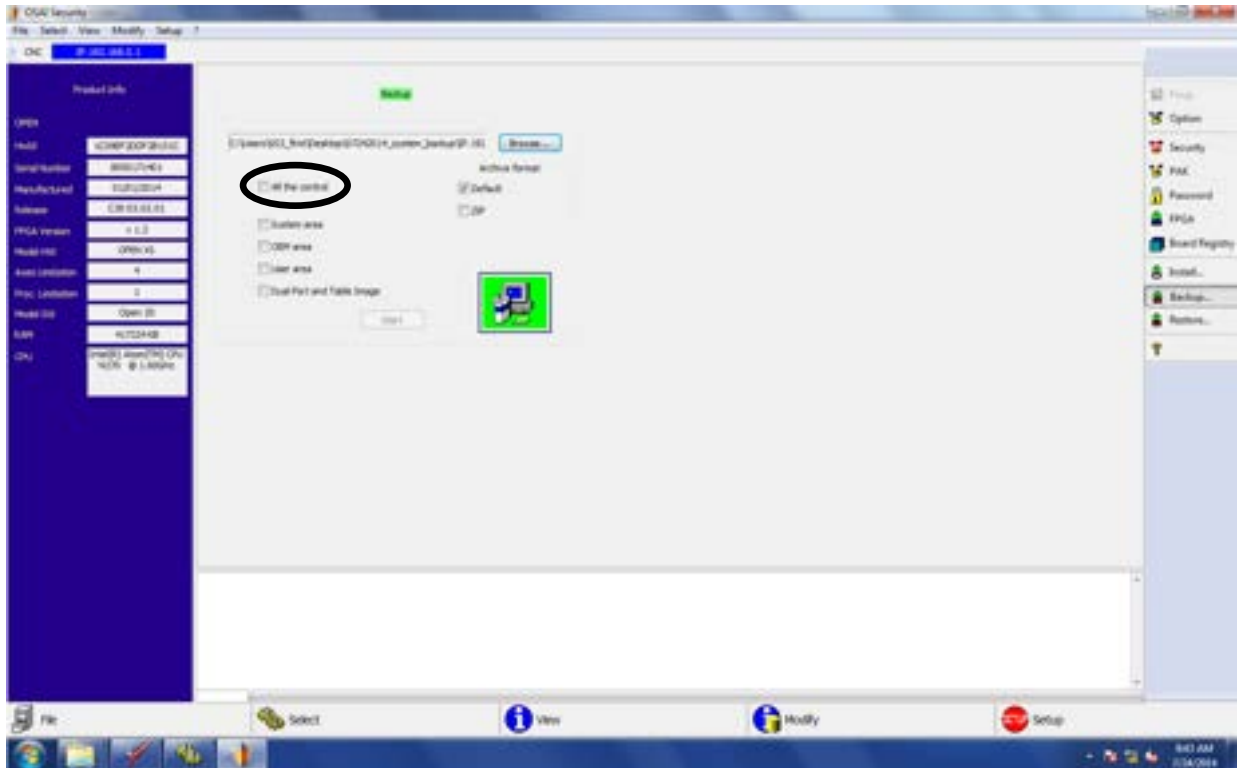
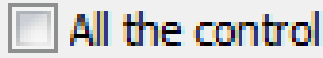
- Click on Browse and navigate to the desired folder to put the backup files. Note that the program will generate more than one file. The recommended format for the folder name is "07252014_CNCBackup", where "07252014" is the current date.



- Click ok to confirm the folder selection



5. Click “All the control” checkbox to check it.



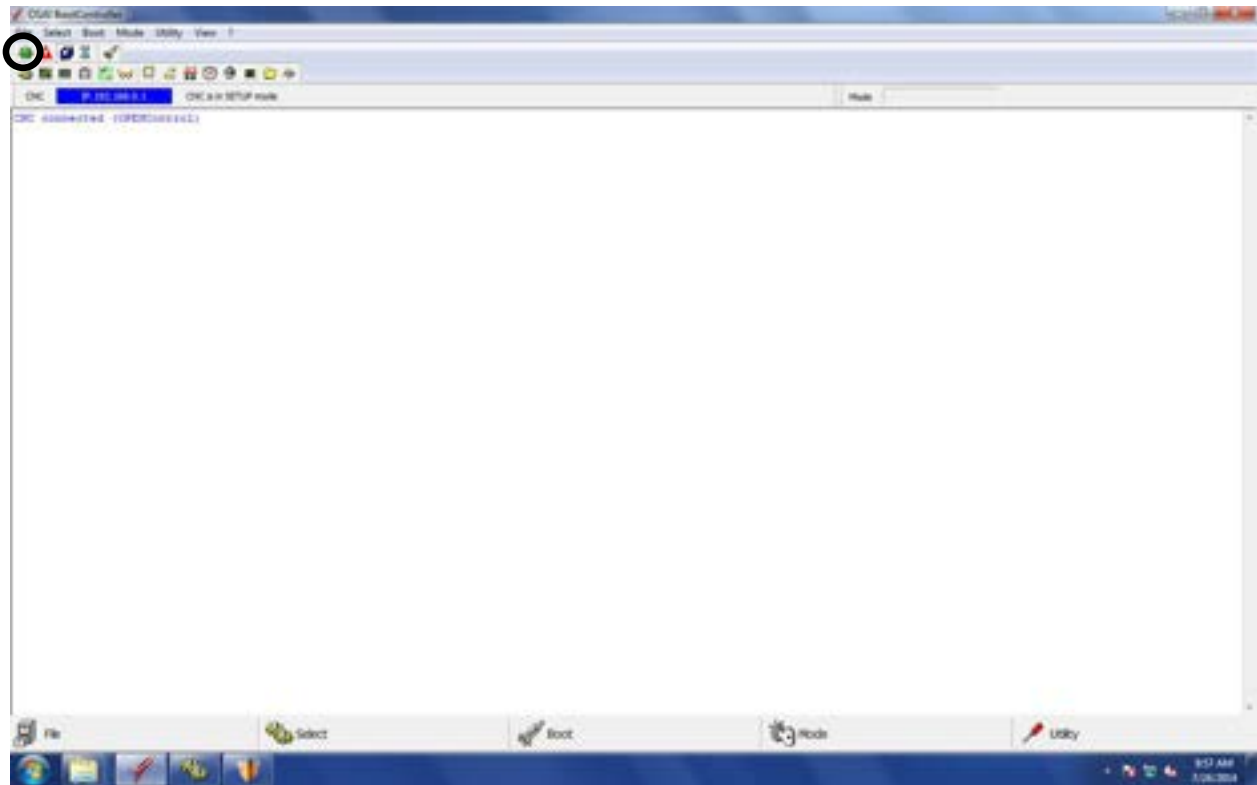
6. Click “Start” and wait for the recovery process to end.



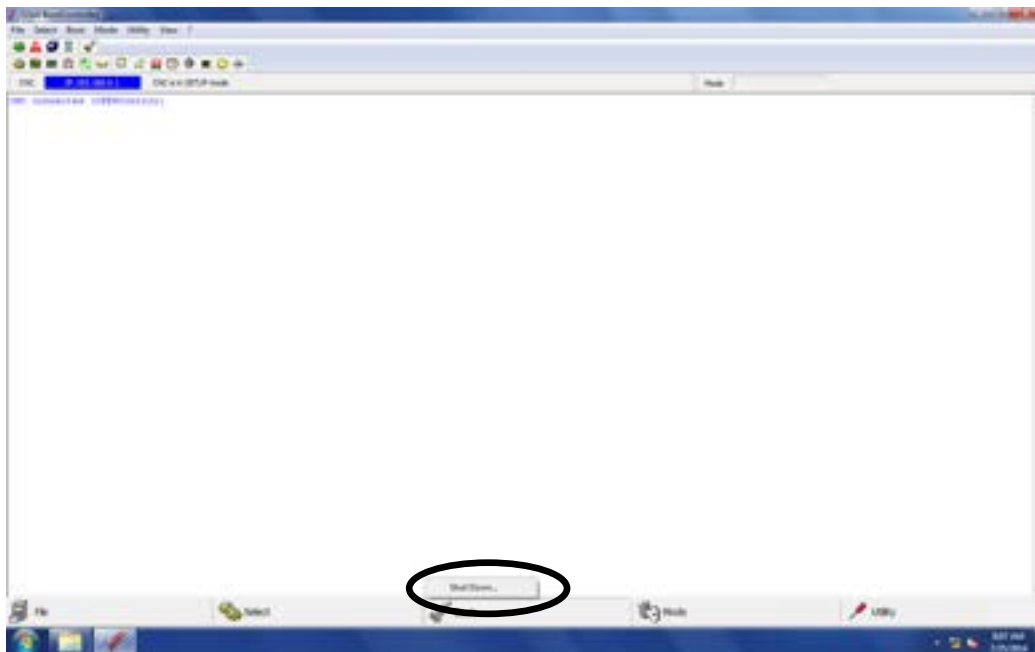
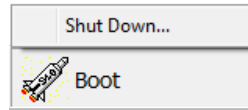
7. Click on the rocket symbol to access the boot control window.



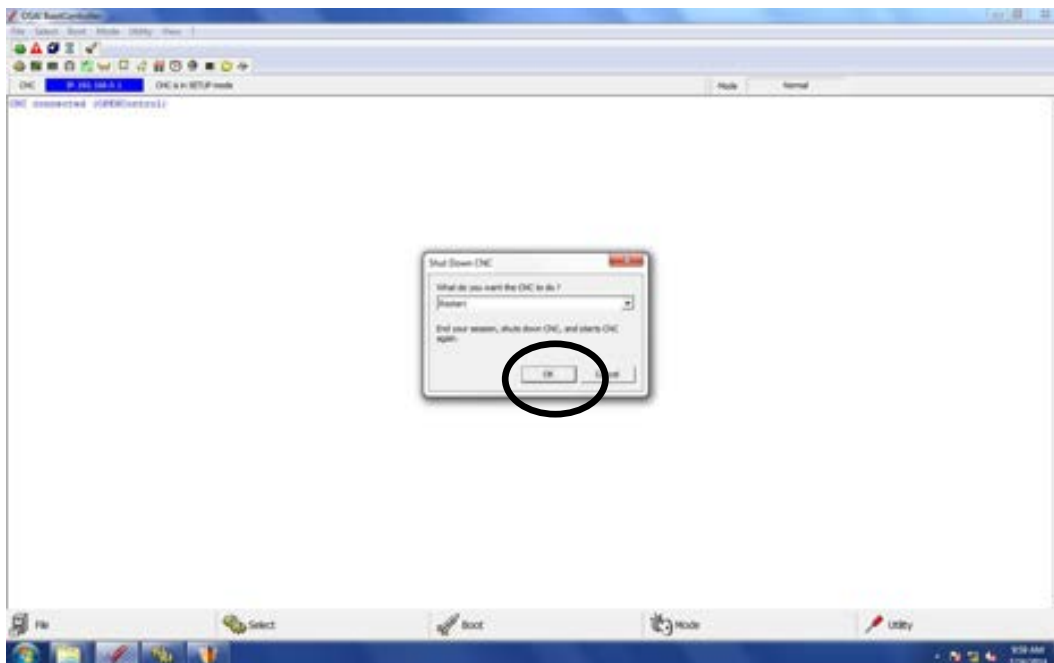
8. Click on the Normal mode button.



9. Click Boot then click Shutdown.



10. Click "OK" to restart the controller.



11. Wait for the controller to restart. The system recovery is now completed.

Appendix

MDI Command list

The MDI command box can be used to execute all G-Code, M-Code, and controller specific functions. Attached is a list of useful

(UAO, 1) – activate origin 1. Replace the number 1 with the number 2 to 9 for other origins. For example, enter (UAO,2) to activate origin 2.

M6T1 – Do a tool change to tool 1. Replace the number 1 to a number from 2 to 8 to change to the other tools. For example, use M6T2 to change to tool 2. This also activates the tool length offset of the specified tool.

M6T0 – put away current tool in spindle

X0Y0 – move the spindle to the current origin.

M401 Automatic touch off. E.g. “M401 T2” command will automatically touch off tool 2. The specified tool must already be in the spindle when executing this command

G-Code List

G00 Rapid axes positioning
G01 Linear interpolation
G02 Circular interpolation CW
G03 Circular interpolation CCW
G04 Dwell at end of step
G09 Deceleration at end of step
G10 Circular interpolation for three points in space
G12 CCW circular interpolation in space
G13 CW circular interpolation in space
G14 Linear interpolation with dynamic rapid parameters
G16 Defined interpolation plane
G17 Circular interpolation and cutter diameter compensation in the XY plane
G18 Circular interpolation and cutter diameter compensation in the ZX plane
G19 Circular interpolation and cutter diameter compensation in the YZ plane
G27 Continuous sequence operation with automatic speed reduction on corners
G28 Continuous sequence operation without speed reduction on corners
G29 Point-to-point mode
G33 Constant or variable pitch thread
G40 Disables cutter diameter compensation
G41 Cutter diameter compensation - tool left
G42 Cutter diameter compensation - tool right
G70 Programming in inches
G71 Programming in millimetres
G79 Programming referred to machine zero
G80 Disables fixed cycles
G81 Drilling cycle
G82 Spot-facing cycle
G83 Deep hole drilling cycle
G84 Tapping cycle
G85 Reaming cycle
G86 Boring cycle
G89 Boring cycle with dwell
G90 Absolute programming
G91 Incremental programming
G92 Axis pre-setting without mirror
G98 Axis pre-setting with mirror
G93 Inverse time (V/D) feedrate programming
G94 Feedrate programming in ipm or mmpm
G95 Feedrate programming in ipr per revolution or mmpr
G96 Constant surface speed in fpm or mpm

G97 Spindle speed programming in rpm

G72 Point probing with probe ball radius compensation

G73 Hole probing with probe ball radius compensation

G74 Probing for theoretical deviation from point without probe ball radius compensation

G99 Deletes G92

M-Code List

Some M-Code might only apply to a certain machine configurations.

M03	Spindle On
M05	Spindle Off
M06	Tool change. E.g. "M06T2" changes to tool 2.
M13	Drilling block rotation on
M15	Drilling block rotation off
M20	Main spindle tool grip
M21	Main spindle tool release
M27	Extend tool changer cylinder
M28	Retract tool changer cylinder
M30	End program
M48	Dusthood up
M49	Dusthood down
M52	Spindle piston up
M53	Spindle piston down and activate Z offset
M54	Spindle piston down and activate X,Y,Z offset
M63	Extend tool on drilling block. E.g. "M63 T11" extends tool 11 and use its offset. E.g. "M63 T12/T11,T13,T15" extends tool 11,12,13, and 15 while using offset for tool 12.
T0	Retract all tool on drilling block
M130	Vacuum table on
M131	Vacuum table off
M401	Automatic touch off. E.g. "M401 T2" command will automatically touch off tool 2. The specified tool must already be in the spindle when executing this command
M405	Auto load material.

Controller Specific Functions

(UAO, 1) – Activate origin 1. Replace the number 1 with the number 2 to 9 for other origins. For example, enter (UAO,2) to activate origin 2.

(DLY, 3) – Delay the program for 3 seconds. Replace the number 3 with any integer to delay the specified number of seconds. E.g. “(DLY,5)” delays the program by 5 seconds.

Sample G-Code

The characters after semicolon on each line are comments and are not read by the machine.

```
G00                ; rapid mode
G17                ; Circular interpolation and cutter diameter compensation in the XY plane
G27                ; Continuous sequence operation with automatic speed reduction on corners
G40                ; Disables cutter diameter compensation
G70                ; Programming in inches
G80                ; Disables fixed cycles
G90                ; Absolute programming
G79 Z0             ; Programming referred to machine zero, move z axis to zero.
T0                 ; retract tools on drill block
(UAO,1)            ; use origin offset 1
M63 T11           ; extend tool 11 on drill block
M13 S12000        ; turn on drill block rotation
G00 X2 Y2         ; rapid move to X2 Y2
G00 Z3
G01 Z-0.25 F300    ; Feed move to Z-0.25
G01 Z3
M15               ; turn off drill block rotation
T0                 ; retract tool on drill block
M6T2              ; Tool change to tool 2 on main spindle
M3 S12000         ; turn on the spindle to 12000RPM
G00 X2 Y2
G00 Z3
G01 Z-0.25
G01 Z3
M05               ; turn off spindle
M30               ; end of G-code program
```


**For technical support, contact Laguna Tools at
1-800-234-1976**

or

**email Laguna Tools customer service at
customerservices@lagunatools.com
Please input machine type in subject line.**

LAGUNA

2072 Alton Parkway. Irvine, CA 92606
Ph: 800.234.1976 | www.lagunatools.com

Laguna Tools is not responsible for errors or omissions.
Specifications subject to change. Machines may be shown with optional accessories.

© 2018, Laguna Tools, Inc. LAGUNA® and the LAGUNA Logo® are the
registered trademarks of Laguna Tools, Inc. All rights reserved.