CNC ZEUS CONTROL
OPERATORS GUIDE v1.79

This document is believed to be accurate, however changes to the software may not be reflected in this document and therefore should be used for a guide only.

CNC ZEUS must first be properly configured, if you have not already done this, please refer to installation and setup guide.

The preferred configuration would include axis limit switches and external feed-rate override controls. This could be enhanced further with the addition of a manual pulse generator and a GS2 AC drive controller for spindle RPM.

MACHINE HOME

HOMING the machine sets the machine zero position and is required each time it is started.
First set mode to MDI by selecting F6, then pressing the Home key, the axis home pop up windows appears. Each axis will home individually by selecting the letter of the axis to home or by pressing enter, home all axis.

When homing all axis by pressing enter in the home window, the "Z" axis is sent home first, followed by "X" and "Y" axis simultaneously.

With good quality axis limit switches, machine home will typically repeat within .0005 inches and is useful for continuing a work in progress.

Properly exiting the control by pressing SETUP then "X" or selecting EXIT CONTROL, saves all previous tool and work offsets and will be restored on next startup.

When no limit switches are used, the machine must be moved to the desired machine zero location, then, by pressing F6 (MDI) and home, the home manual window will appear. By pressing enter in the home manual window, the machine position is set to zero at the current position.

JOG

To JOG an axis to a desired location, activate jog mode by pressing F9 (JOG MODE). Pressing JOG (F9) while in jog mode will toggle between Rapid jog mode and Feed jog mode.

Rapid jog mode is set to the maximum rate defined in parameters and is affected by the manual rapid override control. Feed jog mode sets the jog buttons to the last programmed feed and is affected by the manual feed override control.

Jog can be locked in feed mode by pressing the "L" key while in jog mode. Lock jog mode will lock the direction of the last axis direction key pressed and will stop feeding either by pressing the last direction key again, or the "L" key or Space Bar or "Esc".

Selecting the appropriate key can then move the desired axis.
"X" = left arrow or right arrow
"Y" = up arrow or down arrow
"Z" = page up or page down
"A" = insert or delete

Note: JOG in feed mode uses the last currently set feed-rate, if the machine was just started the feed-rate may be zero and jog will use the minimum feed rate.

The feed-rate can be changed in jog mode by pressing "+" to increase or "-" to decrease. After increasing the override to 200% the feed-rate is then incremented by the value set in parameters feed inc/dec setting.
Selecting MDI can alternately change the feed-rate, by entering the desired feed-rate (Fxx), where xx is feed in inches per minute.

HANDLE

HANDLE mode is the preferred method to move the machine when setting tools. Although some external hardware is required, the benefits of easy movement and rapid accurate positioning can save time especially during tool setup.

NOTE: handle operation in X100 mode can easily move a machine in excess of 250 IPM! If your machine cannot achieve these speeds, then the handle X100 mode should be disabled or adjusted in the handle parameter settings to suit.
 Handle operation is set by selecting F10 (HANDLE) then selecting on the pulse generator the desired axis and step size. Rotation of pulse generator effects machine movement in direction and speed of the pulse generator.
Maximum speed of the pulse generator is limited by the maximum feed rate parameter setting.
CAUTION: moving the handle faster than the maximum feed rate parameter setting can accumulate stored movement! Either the handle parameter settings or the maximum feed rate parameter settings should be changed to prevent stored movement.

FILE ACCESS

FILE operations are selected by pressing F1 (FILE), this displays the working directory set by file parameters in alphabetical order.
Navigation is by arrow keys as well as page up / down if the listing is larger than a single page.

F1 selects and loads the highlighted file.

Control + "S" saves the currently loaded file with a new name. If no path is given the file is saved to the current directory path otherwise it will be saved to the path given in the file name.

Example: in the save as new file pop up window type (A:newfile.nc) then enter to save the currently loaded file to the A drive as newfile.nc then, reload it as the current file. The file-listing directory remains unchanged.

Control + "D" deletes the highlighted file.

Control + "C" changes the working directory and displays a new list of files for the new directory.
Example: in the change directory pop up window type "A:" then enter, to display a list of files on the A drive.

SAVING POINT DATA

G31 is the skip cutting command and is similar to the G1 linear feed command with the exception that it is only operational on the line it is executed (not model) and execution is halted by a high transition on pin 10 or "A" axis limit switch input.
Typically the "A" rotary axis limit is not used and can be used for digitizing probe input.

M33 at anytime in during program execution or manually pressing the "P" key while in JOG MODE or HANDLE MODE will output the current position to the DATA POINT FILE if enabled.

Saving data points to a file by use of the M33 command or by pressing the "P" key in jog mode or handle mode is enabled in the Setting Parameters Screen. A valid file name must be given and will be created in the default file directory location when the parameter "SAVE POINT DATA" is set to ENABLED. WARNING! any previous file with the same name as the "SAVE DATA FILE NAME" will be OVER WRITTEN when the save point data parameter is set to enabled.
Data collection start time and date is recorded at the time the first data point is saved and again when the machine is reset (Esc is pressed or M30 in program). If additional data points are sent then a new time and date will be appended to the point data file and point will be saved until the next machine reset.
To open the point data file for editing the "SAVE POINT DATA" parameter must be set to disabled to first close the file.

POSITION SETTING BY G92

Setting of the master zero point location can be achieved by several methods.
(1) After locating the machine by use of MDI, JOG or HANDLE operation to a desired location. The location can then be set to zero by opening the POSITION table (F4) and

highlighting an entry in the G92 row then pressing ("X" then ENTER) will input the value from the absolute machine position required to make the current X location zero.

The "/" key can be used during position setting to calculate the center of the new point and the previous set point.

(1) Move the X-axis to the edge of work and set to zero. (Position screen then X then ENTER)
(2) Move X to other edge of work then while in position screen, press (X / enter).

X-axis zero is now set to the midpoint of the current position and the previous point.

X zero may also be set at anytime in MDI by entering "G92 X0".
X position may also be PRESET to any location in MDI by entering "G92 Xx.xx", presetting the X-axis to the value of x.xx.
EXAMPLE: (G92 X2.25) presets the X-axis to 2.25.
Any axis may be set in the same manner.
G54-G59 are additional work offsets used to shift the working zero in relationship to the master zero location set by G92.

WORK OFFSET

WORK OFFSETS are accessed by pressing F4 (WORK-OFF) and can be changed anytime, even while in auto block operation.
CAUTION: Work offsets take effect immediately. Changing the active work offset while in auto block operation may result in undesired movement! Care should be taken to change only unused offsets or avoided altogether during auto block operation.

G54 is the default offset and is used at startup.
Any work offset G54-G59 can set the current position to zero by highlighting the desired offset and pressing the axis to zero followed by enter.
EXAMPLE: with G54 highlighted, (X + enter) inputs the correct value to set X-axis to zero from the zero already set by G92.
Additionally, entering 2.00 in G54 X-axis would move X-axis zero 2.00 from the zero location set by G92.
Using this example, up to six work locations can be set, all based from the main zero set by G92. Changing the setting of G92 X value then by .1 would then shift all work locations G54-G59 by .1 also.
Program execution can then select the desired work point zero by using G54-G59 commands.
EXAMPLE: during program execution, (G55 X3 Y1) would move the machine x-axis to 3. and the y-axis to 1 using both work offsets G92 and G55.

G54-G59 are model and stay in effect until changed or until reset is performed M30 or (Esc).

TOOL OFFSET

TOOL length and radius offsets are accessed by F3 (TOOL) and can be changed anytime, even while in auto block operation.

CAUTION: Tool length offsets become effective on any block with Z-axis movement and undesired movement may result if active offset is changed during operation.
Care should be taken to change only unused offsets or avoided altogether during auto block operation.

Tool length offsets are used to adjust to the varying lengths of tools and are usually set from the face of the work or a known distance from the face of the work (usually 1") depending on the programming method.
Length offsets are easily entered for either method using the quick entry keys "Z" and "C".
Setting off the work face using a 1" setup block.
(1) With tool in machine, move Z-axis down until setup block just slides between tool and face of work.
 (2) Select F3 (TOOL) and highlight the desired tool location using the arrow keys.
Pressing "Z" then enter inserts the current position into the work offset length.
Tool is now set 1" above work face.

If tool was to be set off face of work then by entering –1"(size of setup block) in the input box and pressing "C" for calculate, adds –1" to the current Z-axis position and inputs the value.
Tool is now set to work face.

Radius offsets are entered by highlighting the desired tool and radius column typing the radius value into the input box then entering.

Small values can be added or subtracted to either length or radius values using the "Add" or "Subtract" keys.
Input a value into the input box, then highlighting the desired tool (length or radius) press "A", the input box is added to the selected tool value or "S" and the input box is subtracted from the selected tool value.

If the control is exited properly (setup exit control), all tool settings will be saved and will be restored on restarting.

PROGRAM OPERATION

When a program is loaded operation can be executed in either single block mode or auto block mode as selected by the F7 or F8 keys by the enter key.
When auto block is selected, the program will be run continuously until M30 (end of program) or (M00) program stop is reached.
When single block is selected, the program is executed one block at a time or in the case of canned cycles, one step of the canned cycle.

Anytime during program execution the program can be halted with the space bar which initiates FEED HOLD and resumed again by the enter key.
Feed rate override can be effected with the "+" or "-" keys to adjust the feed rate up to 200% of the programmed feed rate. If the feed increase ("+") key is continued to be pressed then the programmed feed rate will be increased by increments set in the feed rate parameters settings and will stay in effect until the program reaches another feed command.

RESET is initiated by pressing the "Esc" key and terminates all program activity and resets all default G codes also stops the spindle and coolant pump.

PROGRAM EDIT mode is activated by the F2 key while in either auto block or single block mode and program is not in operation.

Program edit is a quick onscreen editor and is not intended for major program editing. The editor has a quick find feature that is activated by pressing Crtl + "F" while in edit mode. Entering a search string in the find window searches from the current cursor position to the end of the program for the string entered.

When exiting program edit by pressing the "Esc" key, the edited program is saved and reloaded.

While in program edit, Crtl + S can also be used to save the edited program to a new name, the program will then be saved and reloaded with the new name.

SUB PROGRAM CALLS

Sub program calls using M98 Pxxxx are made to the address Oxxxx located somewhere in the currently operating program, usually past the end of program "M30" so as to prevent program execution from continuing into the sub program inadvertently.

Example:

G90 G0 X12 Y2 S1200

```
G43 H1 Z1. M3
G81 Z-.15 R.1 F5
M98 P204
G0 G28 Z
M30

O204
X4.25
X2.25
X1.
Y1
X-2 Y-1
Y-2
M99
```

LINE REPEAT

Repeat line command "L" is useful for executing the same move several times which is sometimes the case in subprogram calls or drilling operations.

Example:

```
G90 G0 X6 Y0 S1200
G43 H1 Z1. M3
G81 Z-.15 R.1 F5
G91 X –1 L5          Drill 5 more holes at X-1" spacing
G90 X6 Y1
G91 X-1 L8           Drill 8 more holes at X-1" spacing
G90 G0
G28Z
M30
```

Repeat line can also be used on sub program calls.

Example:

```
G90 G0 X-4 Y-2 S1200
G43 H1 Z1. M3
G81 Z-.5 R.1 F8
G90 X-4 G91 Y.5 M98 P204 L4
G90 G0
G28 Z
```

M30

O204
X.5
X.6
X.25
X2
M99

DIGITIZING PROBE

The example program digitizes a 2 x 2 area in .1 increments using an incremental backup of .25 on the "Z" axis from last contact point to help save time.

EXAMPLE PROGRAM:

(PROBE A 2X2 AREA)

```
G90 G00 X0 Y0     (START POSITION X0 Y0)
G43 H1 Z0         (SET LENGTH OFFSET)
M98 P100 L20      (CALL SUB P100 20 TIMES)
G90 G28 Z0        (SET ABSOLUTE AND ZERO RETURN Z)
M30               (END PROGRAM)

O100              (SUB PROGRAM 100)
M98 P110 L20      (CALL SUB P110 20 TIMES)
G90 G0 Z0         (SET ABSOLUTE AND MOVE TO Z0)
X0 G91 Y.1        (MOVE X0 SET INCRMENTAL AND MOVE Y.1)
G90 M99           (SET ABSOLUTE AND RETURN)

O110              (SUB PROGRAM 110)
G90 G31 Z-2 F5    ( SET ABSOLUTE AND MOV Z-2 AT FEED OF 5)
M33               (SAVE POSITION TO POINT DATA FILE)
G91 G0 Z.25 X.1   (SET INCREMENTAL AND BACKUP Z.25)
X.1               (MOVE X.1)
M99               (RETURN)
```


HOLE MILLING

G12 and G13 allow circular milling to be performed in a single command line
Example:

G12 I.5 F5   (Moves X axis .5 then does a CW circle with a .5 radius then .5 back to circle center.

G13 is similar but performs a CCW circle.
 A parameter of Pxx added to the line causes the circle portion to repeat xx times.

EXAMPLE:
G13 J.5 P2 F5   (Moves Y-axis .5 does CCW circle two times and returns to center position.)

A Z move can be added to the line and will take affect only during the circle portion of the command.

EXAMPLE:
G12 J.5 Z.2 F5   (Moves Y-axis .5 does CW circle with Z move then returns to center.)

If the Z-axis movement is to be repeated with the "P" parameter then the move must be incremental (91).

EXAMPLE:
G91 G12 J.5 Z.04 P4 F5 (Z-axis moves during the circle repeats .04 each revolution.)

Cutter compensation can be applied and is automatically cancelled on the return to center.

EXAMPLE:
G91 G41 D1 G12 J.5 Z.04 P4 F5 (Moves Y .5 with cutter compensation and does four circles moving the Z .04 per revolution then cancels cutter comp and returns to center.)

G-CODE LIST

"*" Indicates current G and M codes in Ver 1.71

*G00 positioning (rapid traverse)
*G01 linear interpolation (feed)
*G02 circular interpolation CW
*G03 circular interpolation CCW
 G04 dwell
 G07 imaginary axis designation
 G09 exact stop check
 G10 offset value setting
*G12 Hole milling cycle CW

*G13 Hole milling cycle CCW
*G17 XY plane selection
*G18 ZX plane selection
*G19 YZ plane selection
*G20 input in inch
*G21 input in mm
 G22 stored stroke limit ON
 G23 stored stroke limit OFF
 G27 reference point return check
*G28 return to reference point
 G29 return from reference point
 G30 return to 2nd 3rd & 4th ref. point
 G31 skip cutting
 G33 thread cutting
*G40 cutter compensation cancel
*G41 cutter compensation left
*G42 cutter compensation right
*G43 tool length compens'n + direction
*G44 tool length compens'n - direction
*G49 tool length compensation cancel
 G45 tool offset increase
 G46 tool offset decrease
 G47 tool offset double increase
 G48 tool offset double decrease
 G50 scaling OFF
 G51 scaling ON
 G52 local coordinate system setting
*G54 work conditions system 1 select
*G55 work coordinate system 2 select
*G56 work coordinate system 3 select
*G57 work coordinate system 4 select
*G58 work coordinate system 5 select
*G59 work coordinate system 6 select
 G60 single direction positioning
 G61 exact stop check mode
 G64 cutting mode
 G65 custom macro simple call
 G66 custom macro modal call
 G67 custom macro modal call cancel
 G68 coordinate system rotation ON
 G69 coordinate system rotation OFF
*G73 peck drilling cycle
*G74 counter tapping cycle
 G76 fine boring
*G80 canned cycle cancel
*G81 drilling cycle, spot boring

*G82 drilling cycle, counter boring (dwell)
*G83 peck drilling cycle
*G84 tapping cycle
*G85 boring cycle
 G86 boring cycle
 G87 back boring cycle
 G88 boring cycle
*G89 boring cycle (dwell)
*G90 absolute programming
*G91 incremental programming
*G92 programming of absolute zero pt
 G94 per minute feed
 G95 per revolution feed
 G96 constant surface speed control
 G97 constant surface speed control cancel
*G98 return to initial point in canned style
*G99 return to R point in canned cycle

 M Codes

*M00 program stop
*M01 optional stop
*M02 end of program (no rewind)
*M03 spindle CW
*M04 Spindle CCW
*M05 spindle stop
 M06 tool change
 M07 mist coolant ON
*M08 flood coolant ON
*M09 coolant OFF
*M30 end program (rewind stop)
*M50 hold machine operation until a low to high transition on A axis limit input
*M33 save current PGM position to file
*M98 call sub-program
*M99 end sub-program