Servo Control Users Manual

Index Designs CNC, LLC
21720 Marilla St.
**Chatsworth, CA 91311**
www.indexdesignscnc.com
818-280-8120

Introduction

This manual is intended to provide a description of the Index Designs Servo Control for rotary tables.

The Index Designs Servo Control system features a USB interface that provides a fast and simple way to add a semi-4th axis to any CNC or run as a stand alone indexer. Capable of running a single program, the Servo Control executes indexed moves and sends non-rotary data machine data via DNC. In DNC mode, the Servo Control automatically inserts and sends M function data to the CNC eliminating the user from having to insert them in a program and/or write custom print statements. This eliminates the need to write and coordinate multiple programs between an indexer and a CNC. With almost unlimited memory, the complexity of sub-routines, line repeat, sub-program calls, and nesting can be avoided. Even with applications that do not need a rotary table, the Servo Control is useful to load, save and DNC large programs to almost any CNC with a serial port. Individual serial settings can be saved for quick and easy connection to any number of machines. A menu driven interface with function keys allows easy operation for program entry and other control editing functions. The Servo Control can be programmed from the front panel to rotate the spindle in either direction in step sizes from 0.001˚ to 30,000˚.

Navigating the Menus

There are four arrow keys on the front panel of the Servo Control. There is an Up and a Down arrow key to move the cursor on the menu up or down. The Left and Right keys can also be used to navigate across the menus. The cursor is typically shown bracketing items with > and < symbols. Press the Enter key to select the function pointed to by the cursor brackets.

```
>Run Program<
 Jog
 Edit Program
 Load|Save|Dir          ↓
```
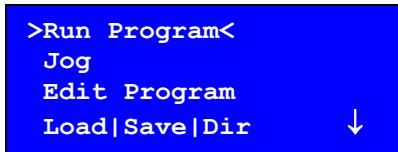
There is an arrow on the right side of the display that will indicate a direction in which more menu items are available. "Run Program" is shown above. It is at the top of the menu and additional menu items are available below "Load|Save|Dir". Pressing the Down arrow will scroll the menu downward where additional selections will be available.

Selecting the Menu key will return the display back to the main menu. The Menu key can be used to cancel out of functions or can be used to return to the main menu.

Getting Started with the Servo Control

1. Power on the Servo Control with the main power switch located at the rear of the indexer box.
2. The display will show the copyright information and version number for about 6 seconds.
3. A "Press Start button to reset Motor Amplifier Fault" message will be displayed.   To enable the axis power, release the Emergency Stop button and press the green Start button to reset the fault.
4. Cursor to "Jog" press Enter to be in jog mode. Use the arrow keys to move the rotary table to the desired cold start position. Press Menu to return to main menu.
5. Cursor to down to "Cold Start" menu choice to begin the rotary zeroing procedure. Press the green Start button. The rotary will attempt to find the zero mark. A message will be

displayed that the rotary is trying to find the motor index mark. When complete, the display will return back to the main menu. (Note, if a CS offset is set, the Servo Control will give the option to press Start to move to the offset stored or press Menu to not move to the offset).

USB Disk Key Format

All USB keys must use FAT32 format.

USB Disk Key Removal

It is important to not remove the USB key when it is in use, otherwise corruption of the USB disk key will occur. A blinking light on a USB key typically indicates that files are being used and that it can not be removed. Only remove the USB disk key from the Servo Control unit when the Main menu is being displayed and the USB disk key is not blinking.

Main Menu

Run Program

Selecting "Run Program" will put the Servo Control in a mode whereby a program can be run from the internal memory. The active program number will be executed. Each line can be manually executed by pressing the green Start button.

```
Pos:36.000 G90 L001
N1  90.000  F100.000
N2  45.000  F40.000
STOP
```

Jog

In Jog mode, the rotary table can be rotated both directions. Also, a specific location can be entered in the MDI portion of the Jog function display. The rotary will rotate to the position entered after pressing the Start button.

```
Jog:160.000     .001
                 ↑
             .1← →-.1
 MDI:           ↓-.001
```

Edit Program

In Edit mode, the active program in internal memory can be edited. (See page 14 for programming details.) The arrow keys are used to move to each NC value in the program. In each program, there are 5 values in a single block of code. The N word is the program line number. The F word indicates the Feed rate value for the step in degrees per second. The S word value determines the step size measured in degrees of rotary table movement. The G word dictates the function or mode for the program. Finally, the L word indicates the loop count value of how many times to repeat a line of code. Some of these NC words will take on different meanings with some G codes like G96 or G98.

```
N1    F160.000    G90
      S90.000     L1
N2    F           G90
#2    S-45.000    L1
```

N = Line number
F = Feed rate in degrees per second
S = Step size in degrees of movement
G = Function or mode
L = Loop count

The currently selected internal program number is shown in the lower left hand corner of the editor screen. In the example above, the current program selected is #2.

Each program can have up to 100 N words.

If a value is blank, it is considered to have no value. For example, a blank S word value will result in no motion when running.

To change a value, use the arrow keys to move the cursor to the value to be changed. Type in a new value. Pressing the Enter key or moving the cursor to another location will load the value into the program.
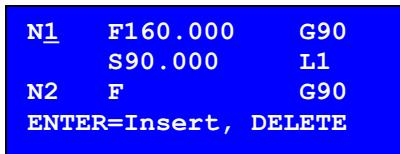
```
N1    F160.000     G90
      S90.000      L1
N2    F            G90
ENTER=Insert, DELETE
```

To insert a line of code, cursor to a N word value. A sub-menu will be shown that will allow the line to be deleted or another line to be inserted just before the cursor position. If a new line is desired before the cursor position, press the Enter key. To delete a line where the cursor is positioned, press the Delete key.

Load|Save|Dir

In the "Load|Save|Dir" mode, a new Servo Control program can be loaded into internal memory, it can be saved or a new one loaded.
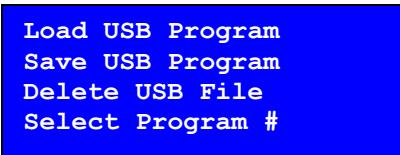
```
Load USB Program
Save USB Program
Delete USB File
Select Program #
```

To save the current program in memory to the USB disk, select the "Save USB Program" option. The display will show ">[type file]<". Pressing Enter will allow the user to name the file that will be saved on the USB key. Use the arrow keys to scroll through the letter/numbers available. Press the left/right arrow keys to change the cursor location on the file name. Do not exceed the 8.3 file format. File names cannot exceed 8 characters and the extension cannot exceed 3 characters. Once satisfied with the file name, press Enter. The program file will be saved to the inserted USB key for future use and safe storage.

To load a program from a USB key into internal memory, select "Load USB Program". Scroll through the available programs by using the up/down arrow keys. If the file name is known, it can be typed at the ">[type file]<" prompt.

The active program can be selected from those stored in local memory by choosing the "Select Program #" option from the menu. This will then show the available programs that can used from the internal memory of the Servo Control.
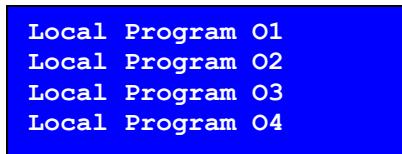
```
Local Program O1
Local Program O2
Local Program O3
Local Program O4
```

DNC

With the "DNC" selection, a program to run the rotary table will either be drip fed (DNC) to the CNC and Servo Control, or a program will be received from the CNC. See pages 17 and 18 for more DNC details.

```
DNC to CNC
DNC from CNC
Mid-tape search
Preparatory data
```

Setup Menu Selection

The "Setup" selection allows settings to be established for the attached rotary table and RS-232 serial communication with a CNC. Highlight the desired parameter option from the list. Press the Enter key to select the highlighted option.

Set CS Offset

This function will set a home offset from the cold start position. Select "Set CS Offset" to store the current position. This offset will be retained after power down of the Servo Control. To restore to this position, jog the table to the cold start mark and select "Cold Start" from the menu. After cold starting the table at the cold start zero marker, if a CS offset was previously set, the Servo Control will give an option to press Start button to move to the home offset stored position. The Menu key can be pressed to cancel and not make the home offset move. A CS offset can only be applied immediately after cold start. Do not set another CS offset if you have already moved to the stored home offset position. To set a new offset from cold start zero, Redo cold start, press MENU to skip the old offset position, Jog to new offset, and set the new offset.

Baud Rate

The "Baud Rate" menu is used to select a common communication rate of data transfer with a CNC. The baud rate is the number of bits per second the RS-232 serial port will communicate. The data speed must match the CNC.

1200
2400
4800
9600
19200
38400
57600
115200

Data/Parity/Stop

Set the number of Data, Parity and Stop bits used for serial communication. This setting is typically fixed on a CNC. Refer the CNC's manual for the proper combination.

7 data, Even parity and 1 stop bit
7 data, Even parity and 2 stop bit

8 data, no parity and 1 stop bit
8 data, no parity and 2 stop bit

Flow Control

"Flow Control" sets the streaming method of data to and from the CNC. This is typically a fixed method. Refer to the CNC manual to determine which type is used by your CNC. Hardware flow control uses two RS-232 signals (CTS and RTS) to start and stop data transmission. Software flow control uses XON/XOFF characters to start and stop data transmission. Software flow control is the preferred and easiest method to implement. This method is recommended.

When the Software flow control method is used, the most current XON/XOFF received during transmission will be displayed on the right of the display. The display of XON/XOFF characters is useful to troubleshoot communication issues should they arise. Xmodem flow control is another option if the CNC control supports this protocol.

HW CTS/RTS
SW XON/XOFF
XMODEM

End of Block Char

"EOB Char" sets the format for the end of block characters. This parameter value will be the end of block that is used when sending data to the CNC. The most common end of block format used by most PC computers is the carriage return/line feed. The next most common type is the Teletype end of block or line feed/carriage return. When data is received from a CNC, a carriage return/line feed end of block is always used. This allows the file to be viewed when using the USB disk on a computer.

CR LF
LF CR
LF CR CR
LF
CR
None/binary

Use the "None/binary" for controls that do not use text files or a file that does not contain control characters. For example, Mazak has an optional non-G code file system that will not work in text mode. The Servo Control will transfer those files when the binary option is selected.

Start of Tx

This sets the character that will be sent before sending data in the DNC file. The most common character used is the percent, %. Choose the setting as specified in the CNC manual. If the DNC program already has a % at the beginning of the file, choose "none". If needed, choose "%" to add the character. Use the null character for controls that need an ASCII char 0 to be used as the start character.

%

End of Tx

This sets the character that will be sent after sending data in the DNC file. The most common character used is the percent, %. Choose the setting as specified in the CNC manual. If the DNC program already has a % at the beginning of the file, choose "none". If needed, choose "%" to add the character. Use the null character for controls that need an ASCII char 0 to be used as the end character.

%
none

Rotary Ratio

This function sets the rotary table ratio to be used. Choose this setting to match the rotary table being used.

  60:1 VH-5C, VH-6, TR-6
  90:1 VH-8
120:1 VH-11
180:1 VH-15

Brake ON Delay

This function will set the amount of time to wait for the brake valve to turn on and engage the brake. The range is 0.2 to 5 seconds and it will automatically turn ON after every move. When it is set to 0 seconds, the brake will be disabled. It is important that you also set the OFF delay.

Brake OFF Delay

This function will set the amount of time to wait for the brake valve to turn off and disengage the brake. The range is 0.2 to 5 seconds and it will automatically turn OFF before every move. When it is set to 0 seconds, the brake will be disabled. It it important that you set this OFF delay when using the ON delay.

Finish Relay

This sets the normal state of the M-function finishing relay to the CNC. It can be set to be "Normally Open" or "Normally Closed". Typically the setting should be set "Normally Open" for most CNC machines.

G90 Style

When using G90 absolute positioning, this setting determines the style of rotation direction. "Shortest Path" will rotate in the direction to the programmed position that makes the shortest move possible. "Sign Dir." will give you control of the direction of rotation based on the + or – of the programmed position. Starting at 0 and programming A90 will rotate in the positive direction +90 degrees to the absolute position of A90. Starting at 0 and programming A–90 will rotate in the negative direction –270 degrees to the absolute position of A90.

Save Setting

Multiple machine settings can be saved using this function. This feature is useful when connecting the Servo Control to different machines each with unique communication settings. They can be easily recalled for quick start up. When selected, the current Servo Control setup parameters can be saved as a file using a name chosen by the operator. Typically, this name is associated with a specific machine. The created parameter files will be saved on the USB key in the INI directory.

```
>[type file]<
..<DIR>
Mazak
Fanuc
```

Load Setting

Use this function to load a previously saved machine setup for easy recall. The parameter files saved in the INI directory on the USB key will be displayed. If multiple USB keys are used, copy the INI directory to all the USB keys used.

The Servo Control does not support long Windows type file names. Use a DOS type 8.3 file format with the Servo Control. 8.3 filenames have at most eight characters, optionally followed by a "." and a file name extension of at most three characters.

Update Software

Use this function to update the firmware on the Servo Control. Only use software that has been provided directly from Index Designs. Note that some updates may require the indexer box to be returned to Index Designs for updating.

CNC DNC | Load | Save

In this mode, the Servo Control can be used without a 5C or rotary table connected. Files can be sent or received from a USB key and CNC machine, including DNC operation. The first set of menus will be shown as below.

CNC to USB

Choose this option to send files from a CNC to a USB key in the Servo Control. A screen will be displayed that will allow the name of the file to be typed by the operator or chosen automatically by the Servo Control. e.g. a file name such as CNC16.TXT will be automatically generated by the Servo Control.*   If the operator desires to give a specific name to the file, press Enter to view file names already on the USB that can be over written or allow the operator to give a unique name to the file. You can easily rename the files on the USB using a computer.

*Automatic File Name Selection: All files saved to the USB use a file name with the following structure: CNC####.TXT The number will be the next number in the sequence of the files found on the USB disk key. If the largest sequence numbered file name number found is CNC15.TXT, then the next file name created will be CNC16.TXT.

If you wish to overwrite an existing file, select one of the files from the list. A warning will appear as shown below.

```
File exists, do you
want to overwrite
the file?      Enter = yes
               Delete = no
```
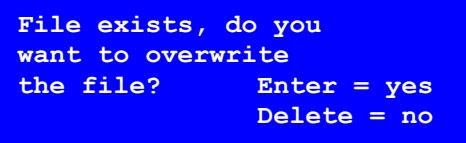
Confirm that you want to replace an existing file by selecting "yes". Selecting "no" will take the operator back to the main menu.

Using the [type file] function

Directory listings on the Servo Control always have a [type file] option in the directory list. This function allows the operator to insert characters to name files. When selected in "CNC to USB" mode, a user may specify the name of the file on the USB that is being saved. The Scroll function is used to increment to another letter or number. Use the arrow keys to accept characters and move to the next character in the file name. Press Enter once satisfied with the file name. A character can be deleted by pressing the Delete key. In the example below, the Servo Control will save a file on the USB key named 110808.NC. Press Enter, then press the Start button to prepare the USB to receive a file from the CNC. To "Cancel", press the Menu key to return to the main menu without creating a file. Once Start has been pressed, a file send operation (punch) from the CNC should be initiated. Press and hold the Menu key to cancel this operation.

```
File Name:
110808.NC
```

USB to CNC Menu Selection

With this menu selection, a file on the USB disk key can be sent directly to a CNC. The first files shown on the display are files on the USB disk key from the last selected directory. Use the arrow keys to highlight a particular file, highlight [type file] to type in a file name or select a new sub directory to display. Press the Enter key to select the desired file.

```
[type file]
CNC16.TXT
>MOLD.TXT<
NEXTJOB<DIR>
```

A new screen will appear showing the file to be sent along with "Start to Begin", "Menu to Cancel", and "Enter to View" displayed. Pressing the Enter key will result in the first few lines of the program in the file being displayed on the screen. This will give the operator a snapshot of the file being sent to ensure the correct one has been selected. Before pressing the Start button, prepare the CNC control to receive a file. When the CNC is ready to accept a file, press the Start button. Press the Menu key to cancel this operation and return to the main menu.

```
to CNC: MOLD.TXT
START to Begin
Menu to Cancel
Enter to View
```

After pressing Start, the CNC will receive the data file selected. In the example above, file data contained in MOLD.TXT will be sent to the CNC. When the transmission is complete, the

- 10 -

display will return to the main menu. For some machines, the operator may have to select "Cancel" to return to the main menu after transmission of a file.

DNC* Menu Selection

With "DNC" selection, a file on the USB disk key can be sent directly to the CNC while the control is in "Automatic" mode. This function provides a way to run a program off the USB disk key. The first files shown on the display are files on the USB disk key from the last selected directory. Use the arrow keys to highlight a particular file, highlight [type file] to type in a file name, or select a new sub directory to display. Press the Enter key to make the selection. See pages 17,18 for more details.

(DNC, Direct Numerical Control, may not be available on all CNCs. Check with the machine's user manual to determine if this feature is available on the CNC control).

DNC Mid-tape Start

This menu selection allows for mid-tape starts if the operator wishes to start sending data midway through a program. Use the Mid-tape start option to do a block search within the file selected. Use the arrow keys to add text for where mid-tape should occur.

```
Midtape:
N101
```
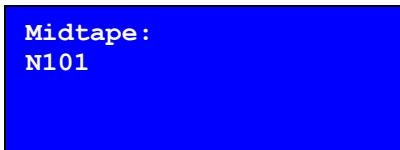
In the example above, the Servo Control will search for N101 and begin DNC operation at the file location where N101 is found. Once the operator is satisfied that the text desired is on the display, press Menu.

Scroll to the "DNC to CNC" menu. Enter this mode and scroll using the arrow keys to find the desired file. Note: An N word can be added to a single line of the file in order to establish a starting block number if one doesn't already exist.

DNC with Preparatory Data

Use the "Prep-data" option from DNC Options to insert text prior to file data that has been selected for DNC. For example, the operator might want to make a tool change prior to running a file. To do this, the operator would enter the appropriate NC codes to make a tool change like T4M6.

```
Preparatory Data:
T4M6
```
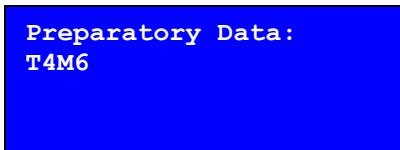
The mid-tape or preparatory text can be typed using the left and the right arrow keys to position the character cursor. To select letters that don't exist on the keypad, use the up and down arrow keys. Alternatively, numbers can be typed in using the keypad adjacent to the display. Use the left and right arrow keys to accept characters and move the cursor to the next desired location. Press the Enter key when finished typing the desired search string. Characters can be deleted by pressing the Delete key.

Once satisfied with the data that should be sent ahead of the data file, press the Menu key. Scroll the cursor to "DNC to CNC". Select the desired file and press Enter. The preparatory data will be sent to the CNC without modifying the contents of the file. In the example, T4M6 will be sent to the CNC followed by the rest of the file.

The Preparatory Data and Mid-Tape start can be used together to quickly resume operation in DNC mode without having to modify DNC files.

Cold Start

Use this selection to initialize the rotary table at the desired zero position on the table. Use the Jog mode before hand to adjust the rotary table position to the desired starting 0 degree position. The mark on the spindle and housing should be close to each other before starting the cold starting procedure. When the marks are close, start the cold start by pressing the Start button, the Servo Control will locate to the zero position on the table based on the encoder marks of the servo motor. The alignment of the marks on the spindle and the housing indicate a successful cold start position.

To set an offset after cold starting, jog to the desired home offset. On the set up menu, choose "Set CS Offset". This offset will be retained by the Servo Control after power off. This message will be displayed after cold starting:

```
     Press START to
   set the CS Offset,
  Press MENU to cancel
```
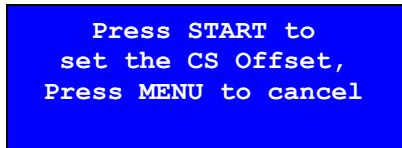
The rotary table will move to the offset position after Start is pressed. To cancel, press the Menu key.

Zero Set

Use this selection to set the Servo Control's 0 degree reference point to the current position. In jog mode, rotate the device using the arrow keys or by directly inputting the location using MDI (manual data input) to the desired location. Press and hold the Zero Set key for approximately 5 seconds. The location displayed on the screen will change to zero indicating a successful zero set. This reference will stay in effect until it is either changed again or the unit is put through the Cold Start procedure. This will <u>not</u> be saved as your CS offset.

Zero Return

Press the Zero Return key to rotate the rotary table back to the zero position without having to use the JOG mode. This key is inactive while editing and running programs.

TR-6 Settings

A/B CNC Orientation

When using 5th axis heads, the face plate axis name is important for CAM programming. The TR-6 can be mounted to your CNC table in either orientation, and the cable connections at the indexer box need to match. **If the orientation is different from the factory settings, you will need to edit the indexer INI settings with a PC text editor.** There are axis specific settings that are only available in the INI file, and you will need to swap all the A and B settings to match the physical CNC orientation. Always backup the INI settings before making any changes.
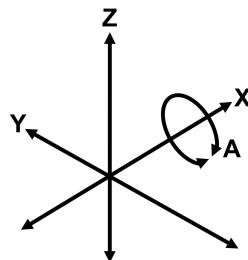
When the rotary face plate is aligned with the CNC X axis, the rotary is considered A axis.
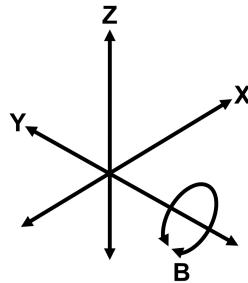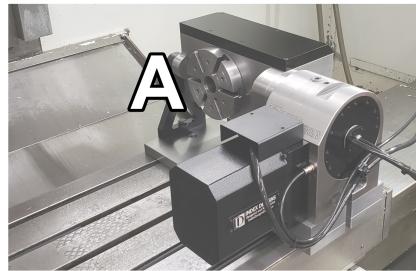
When the rotary face plate is aligned with the CNC Y axis, the rotary is considered B axis.

Software Axis Limits

The TR-6 does not have hardware axis limits when rotating the trunnion body. The axis limits are set by software in reference to the Cold Start position, and they will only be active after a completed CS. They are user defined in the indexer INI settings with a PC text editor. It is important that the trunnion axis limits match the physical CNC orientation in the INI file. It is possible for the trunnion axis to rotate 360° without using limits, and it is very important that you do not continue to rotate in a single direction because this will cause the trunnion motor cable to over-twist causing damage.

A/B Air Brake

The TR-6 uses a single air valve for both brakes, so the A and B brake air supply will be tied together. See the manual for "Brake ON delay" and "Brake OFF Delay" operation.

Indexer Operation

Feed Rate

The feed rate is programmed in degrees per second. When entering an F value in a program, the rotary table will rotate at this programmed speed on the current line and every line after it until a new F word is entered. When a new speed is desired, enter an F value on or before a move command. The F value will remain modal (in effect) until an new value is programmed.

Use F500 for maximum speeds, the control will use the highest allowed speed for your motor. Speeds may need to be lowered for heavy parts and fixtures.

Step

Range = +30000.000 to −30000.000

The step is programmed degrees. The current coordinate system will determine whether a move will occur or not. There are two coordinate systems, absolute and incremental.

G90 Absolute Coordinate system

Step values in the absolute coordinate system will move to that value relative to the home position on the rotary table. The home position is the position that is considered 0 degrees. Moves larger than 359.999 will be translated to a value in the range of 0.000 to 359.999 degrees. Direction of rotation will be determined by "G90 Style" in the "Setup" menu.

G91 Incremental Coordinate system

Step values in the incremental coordinate system will move the number of degrees in the value of the S word. All moves are relative to the current position of the rotary table.

G codes

G28	Return to home position
G81	B axis continuous rotation in the negative direction
G82	B axis continuous rotation in the positive direction
G83	A axis continuous rotation in the negative direction
G84	A axis continuous rotation in the positive direction
G85	Cancels continuous rotation
G86	Turn CNC relay ON
G87	Turn CNC relay OFF
G88	Return to home position
G89	Wait for remote input
G90	Absolute coordinate system
G91	Incremental coordinate system
G92	Pulse CNC relay and wait for remote input
G93	Pulse CNC relay. During "DNC to CNC" G93 sets DPS feed rate programming.
G94	Pulse CNC relay and run next L number of steps without waiting for the CNC completion relay (M??).

	During "DNC to CNC" G94 sets 1/T feed rate programming.	
G95	Return from G96 subroutine	
G96	Subroutine call or jump to line in S word. The program execution will be redirected to the line number that is stored as a value in the S word. The G95 code is used to return to the line after the G96 coded line. The G96 is not nestable (only one maybe used at a time).	
G97	Delay 0.1 seconds multiplied by L word	
G98	360 degree division (circle division), L word will be the number of equal sized divisions of a complete rotation to make into steps. After completion, the rotary table will be back at the original starting location.	
G99	End of program	
G100	For testing only: Automatic operation and endless cycling	

Loop Values (L words)

Values in the L word are typically used to control the number of times to repeat a line of code. For example, a program to move the rotary table 25 degrees three times could be accomplished using the code below:

N1 G91
N2 S25.000 L3

Other uses of the L word include using them in conjunction with G94, G98, and the G97. The L word value is used in these special function G codes as a parameter for the function.

Example Programs

The following program will rotate the rotary table 90 degrees and then to 135 degrees using the absolute coordinate system, G90:
N1 F100.000 G90
N2 S90.000
N3 S135.000

The following program will rotate the rotary table 45 degrees and then another 60 degrees using the incremental coordinate system, G91:
N1 F100.000 G91
N2 S45.000
N3 S60.000

The following program will make ten 36 degrees moves on the rotary by using the circle division function:
N1 G98 L10 F100.000

The following program will make ten 36 degrees moves on the rotary by using the L value to loop:
N1 G91 S36.000 L10 F100.000

DNC from CNC operation procedures

- Communication with the CNC will require the provided RS-232 "straight thru" DB25 cable and M function cable.

On the indexer control:
1. Select "CNC DNC | Load | Save" from the menu.
2. Select "DNC from CNC".

On the CNC:
3. Your CNC program will need to have serial macro statements for every AB rotary move.
4. Press START on the CNC and the program should begin to run.

DNC from CNC - Macro Statements

The Servo Control NC data blocks will be sent from the within the CNC program using an RS-232 serial macro statement. If you need to start your CNC from <u>mid-program</u>, you must re-establish a <u>fresh</u> "DNC from CNC" connection <u>after</u> your CNC is ready to start at the mid-program code line.

Examples of lines to be inserted are as follows.

<u>Fadal</u>

#WAIT	CNC to wait for all previous lines to be completed before execution
#SPRINT "R" *	*Use only at the start of your program to reset any data in DNC memory
#SPRINT "A90. F100 G91"	Send the code inside the quotes, no double G codes per line
M21	One M code start for <u>every</u> #SPRINT line, except for "R" reset

<u>Fanuc / Haas / Yasnac / Mitsubishi / Mazak</u>

POPEN	CNC to open the RS-232 port for sending data
DPRNT [R] *	*Use only at the start of your program to reset any data in DNC memory
DPRNT [A90. F100 G91]	Send the code inside the brackets, no double G codes per line
G4 P1000 *	*CNC dwell before M code may not be needed
M??	One M code start for <u>every</u> DPRNT line, except for [R] reset
PCLOS *	CNC to close the RS-232 port, * or it can be left open at all times

DNC to CNC operation procedures

- Communication with the CNC will require the provided RS-232 "straight thru" D25 cable and M function cable.

On the indexer control:
1. Insert your USB disk.
2. Select "CNC DNC | Load | Save" from the menu.
3. Select "DNC to CNC".
4. Select the desired DNC file from your USB disk.
5. The screen will show "DNC: <filename>" and "Wait or press START".
6. Wait. Do not press START yet.

On the CNC:
7. Enter the DNC mode.
8. Press START on the CNC and the program should begin to run.
9. If it does not start automatically, press START on the indexer control.

Note: In software handshaking mode, the Servo Control will wait for a start command before sending the file. If the CNC is placed into DNC mode before setting up the Servo Control, the start command will be sent prematurely. The Servo Control will miss the start file command and will wait indefinitely in a wait state unless the user presses the Menu key.

DNC to CNC - "A" move Recognition

The Index Designs Servo Control features a G code recognition technology that separates G codes meant for the indexer and those for other machines. Standard machine data is sent to the CNC in DNC mode. When an indexer move is recognized, M function commands are self-generated and sent to the machine serially to activate machine contact closure for machine and CNC coordination. For CNC feedback, the Servo Control dynamically generates its own internal M functions for indexer moves and closes a contact as feedback to a CNC machine tool. This auto insertion multi-axis DNC functionality makes multi-machine code easy to code and read by using 4th axis-like programming. Burdensome print statements are eliminated. In addition, the complexity of generating multiple programs and coordinating between them can be avoided. No loading of a program into CNC or indexer is required. Load files on a USB key with A move style programming and begin operation.

Example code to drill two holes, one hole at 0 degrees and another at 90 degrees.

```
G0 G17 G40 G90 G98
T01 M06
A0.0 F50.0 G90        "A" move data kept by indexer box and replaced with M21 for CNC
G81 Z-.25 F4 .R.01       Drill first hole
A90.0                 "A" move data kept and replaced with M21. Move A 90 degrees
Y0                      Drill second hole
G80                     Cancel drill cycle
A0.0 G28              "A" move data kept and replaced with M21. Send A axis home
G28                     All other axes home
M30
```

When selecting "DNC to CNC", the Servo Control will send the data to the CNC while keeping the rotary motion blocks for itself. The CNC will run in DNC mode receiving the NC blocks as they are drip feed from the Servo Control. The Servo Control will detect A axis rotary motion blocks automatically and remove these blocks from the drip feed data and replace it with M21 indexing commands. The automatically generated M21 commands will synchronize the CNC and the Servo Control box. If the CNC uses a different code other than M21 to close the Servo Control contact, they can be changed. See the INI section to change the M code. All codes to be used by the indexer must be on the same line beginning with an A move. Only program data can be sent in DNC; no tooling or fixture data.

.INI file

The INI file can be edited to make changes to the defaults for the Servo Control unit. To create this file, select SETUP menu from the main menu. Select SAVE SETTING to save the INI to the USB. Enter the file name to be used or use the default. Edit the values in a computer text editor. The file created should have the following contents:

All menu choice #s are from the top of the list down, <u>starting at 0</u>.

BAUD=5	Speed of communication, needs to match CNC baud
FLOW=1	Streaming method of data
EOBDELAY=0	Pause at the end of a block of data
EOB=0	End of block character
STARTTX=1	Character that will be sent before sending data in the DNC file
ENDTX=1	Character that will be sent after sending data in the DNC file
BITS=0	Number of Data, Parity and Stop bits used for serial communication
STARTRX=0	How the Servo Control will initiate receiving a file
DIRECTION=0	Rotary direction
RATIO=1	Rotary motor to gear ratio
COMMENT1=*	DNC from CNC comment character used by CNC, used to ignore comment data
COMMENT2=(	DNC from CNC comment character used by CNC, used to ignore comment data
INDEX=M21	M code sent back to the CNC. The number depends on the CNC setup.
AOFFSET=0	A axis Cold Start offset from setup menu
A-LIMIT=0	Negative limit units in .001 of a degree from Cold Start, -180000 = -180 deg
A+LIMIT=0	Positive limit units in .001 of a degree from Cold Start, 180000 = 180 deg
ACS0=0	Fixed Cold Start offset
A0=0	Compensation and survey in quadrants
A90=0	"
A180=0	"
A270=0	"
A-0=0	"
A-90=0	"
A-180=0	"
A-270=0	"
BOFFSET=0	B axis Cold Start offset from setup menu
B-LIMIT=0	Negative limit units in .001 of a degree from Cold Start, -180000 = -180 deg
B+LIMIT=0	Positive limit units in .001 of a degree from Cold Start, 180000 = 180 deg
BCS0=0	Fixed Cold Start offset
B0=0	Compensation and survey in quadrants
B90=0	"
B180=0	"
B270=0	"
B-0=0	"
B-90=0	"
B-180=0	"
B-270=0	"
BRAKEONDELAY=10	Delay in .10 second units, 0 is off, the range is .20 to 5 seconds
BRAKEOFFDELAY=10	Delay in .10 second units, 0 is off, the range is .20 to 5 seconds
FINISH=0	Normal state of the M-function finishing relay to the CNC
FINISHTIME=500	Time in ms needed for the CNC to recognize the finish relay.
G90=0	Style of moving to G90 position. Shortest Path or Sign Dir.

- 19 -

The parameters BAUD to RATIO can be set directly from the SETUP menu, or the number after the = sign can be changed in the INI file and then loaded. The number corresponds to the position in the list for that parameter, from the top of the list down, starting at 0.

The COMMENT1= and the COMMENT2= should be edited off line on a computer. The characters after the = sign will set the default character(s) used by the CNC to identify comments in the CNC G code. The defaults are * and (.

The INDEX= sets the CNC code to close the indexer contact closure for indexing the program on the indexer. Typically, this is set to M21. If this is not the M code used by the CNC, it can be changed by editing and loading the INI file with the correct code after the INDEX=. This parameter is primarily used by the "DNC to CNC" feature of the Servo Control unit.

Indexer Electrical Connections

M-Function

We use industry standard M-function relay, wiring and signal protocols.

The indexer finish relay is normally open by default. This can be changed in the menu setup.
  The indexer finish relay is closed for 500ms. If the CNC needs a longer time,
  increase the INI settings FINISHTIME value in a PC text editor.

Pin definitions on the circular DIN connector:

- **Only touch Black +24 volts to White to test cycle start.**

Pin 1	Red	Servo Control Finished relay
Pin 2	Green	Servo Control Finished relay
Pin 3	Black	CNC Start Relay (+24 Volts out from Servo Control)
Pin 4	White	CNC Start Relay (Cycle Start on Servo Control)

The CNC M-function start signal will be wired to the N/O relay coil.

Use a voltage meter to measure across the Black and White relay terminals:

  24vDC when the relay is <u>open</u>.

  0vDC when the relay is <u>closed</u> (Indexer will start).