

# Xseries 32/64bit solutions – Xlink-64

## *Installation and Operations Manual*

# **DRAFT ONLY**

### Inspiration Systems

37 Shropshire Avenue, Hillcrest 5086  
South Australia

Intl Tel: +61 8 8266 5708 Fax +61 8 82665955  
Within Australia: (08) 8266 5955 Fax (08) 82665955  
Email: [support@softCNCware.com](mailto:support@softCNCware.com)

## Contents

Software Installation.....	3
Relocating the Database.....	4
Parameters – Definitions of the variables for ISO.....	5
Adding ISO CNC machines via the parameter page .....	11
Editing Parameters .....	12
Testing your Parameter Entries.....	13
Performing Transfers to the CNC.....	15
Working with the Transfer and Edit Icons.....	16
Identifying a Com Port on the PC.....	18
The Jobbing System .....	19
Using The Jobbing System .....	19
AutoRepeat.....	21

**Note:** The CNC parameter password is **z00insp** ,and it is the same for all machine parameter forms..The master parameter password is **z99insp**.

### Warning:

Reinstalling the software from the CD will delete and replace the database with the database from the CD, in other words you will lose all your entries. Backup your database before reloading the software.

## Software Installation

If you are installing the software from the distribution CD:

1. Place the distribution CD into CD drive.
2. After a short delay the Windows bit size (32 or 64bit) and the dotnet version detection program "NetTest" should start automatically.
3. If the automatic program launch fails, start "File Explorer" or "Windows Explorer". You can do this easily by "right mouse" clicking the Windows Start button and click "File Explorer". In the explorer, locate the "CD/DVD" drive and "double mouse" click the drive. With the contents displayed, double click on the "NetTest.exe" file to execute the program. NetTest displays the currently installed versions of the .net. Now using the same procedure double click on "Setup.exe" file.
4. To meet specific site requirements we often do variations to the basic setup. These "specials" are usually for specific machine folder creation.
5. During software installation you will be able to choose the correct dotnet version\*\*\* and the Windows size 32 or 64bit for the PC. Use the last dotnet version shown in the "NetTest" above. NetTest makes a recommended setting. The top entry "Main Section" in the "Choices" list must be left ticked. The software will now be installed onto the PC and a desktop shortcut will be created.

Installation to early versions of Windows XP is a special case:

If the program "NetTest.EXE" reports an error it generally means that there is no version of dotnet installed on the PC. If this occurs, start Windows Explorer and navigate to the CD drive. On the CD is a program called WindowsInstallerV2.EXE, run this program followed by the program dotnetfx2.exe. The combination of these 2 programs will install dotnet V2 on to your XP system. Now run SetUp.EXE and select dotnet 2.

*\*\*\* .net or dotnet is a Windows mechanism to provide common software libraries for standard software functions. It has the advantage of making programs significantly smaller and functionally more stable. There are many different versions of the .dot net and at the time of writing, the latest version is 4.5. Windows XP typically ships with version 2.0.*

## Relocating the Database

The installation puts the database on the local PC in the C: drive in the database subfolder of a folder with the same name as application. In the main folder is the file DbaseLocn.INI and the location of the database is defined in the first line. To relocate the database to a server, simply copy the database to a location on the server and modify DbaseLocn.INI so it points to this location.

## Parameters – Definitions of the variables for ISO

For 99% of installations the values in the “Master Parameters” can remain unchanged. However if you have older CNCs that use the ISO transfer facility and you believe you will need to use character and or line delay values then before creating machines, from the main menu, select “Parameters” and then select “Master Parameters”. Click on the “Calculate Time Delay Constant”. Enter the Master Parameters Password z99insp and then click on the “Update Master Values”.

From the home page select “ISO Parameters”. There is at least one sample entry, use this as a guide for other machines or you could modify the sample entry.

ISO/EIA RS232 Parameters

CNC ID

Machine Index

CNC Name\*

Description

CNC Model\*

Program Name Extension

Select from the drop down List or you enter specific requirement i.e. XYZ

Com Port  Baud Rate

DataBits  Stop Bits

Parity  Flow Control

Base Folder

Document Folder

Leadin File

Output Line End  Input Line End

Character Delay  Line Delay

Send First %  Use N in Block Numbering  Use Leading 0's in Block Numbering  Use Leadin File

Starting String  Ending String

Switch Box Text  AutoLink Port #

Remove Rcd %'s  Don't Send %'s

Add Text Count  Ending String

Parameters UnLocked

## **Terms Used and their Meanings**

**CNC-ID** each CNC machine has a unique identification number which is generated automatically when you add a new set of CNC parameters. This field cannot be edited.

**CNC Name** this used to identify the CNC to the users and should be the name that users know the machine by.

**Description** this clarifies which CNC you are dealing with and is not used anywhere else in the program.

**Program Extension** Most Cad/Cam systems and workshops use a default filename extension for the CNC programs. We have provided a number of standard extension names, but if the one you are using isn't there just enter the extension you use in the text box. Don't forget the "." Character.

**CNC Model** here you define the make and model of the CNC that these parameter values are for. If the make and model you have, isn't in the list then select "0=None". With this value nothing is added/removed at the beginning, during and the end of a program transfer.

**Com Port** is the hardware device in the PC that handles the transfer of programs between the PC and the CNC. The modern PC can have up to 256 com ports. Determining the correct com port can be tricky. The distribution CD has a utility program call "FindPort.exe" that can determine the current device when a test plug is inserted into a port. Comports come in 2 sizes "9 and 25" pin. The plug on the PC is always a male or a plug where you can see the pins. The female connectors are a printer port or video output (15 pins). Do not attempt to use the printer port with gender changers – it won't work.

*Once you have the correct com port, try receiving a program from the CNC. If you get some data from the CNC then you can start changing the following parameters. If you get nothing change the baud rate first until you see some characters or a complete program if you are in luck.*

**Data bits** is the number of bits (7 or 8) that make up a character. Standard ISO uses 7. So start with this, as it's the defacto standard. If during transfers back to the PC you get odd characters, change this value to 8 and the parity to none.

**Parity** is a technique used to validate each character being transferred. ISO uses even parity. With even parity using 7 data bits the 8<sup>th</sup> bit is added to make each character have an even number of bits. Odd parity, is rarely used,

and changes the 8<sup>th</sup> bit to make the number of bits an odd count. No parity turns the 8<sup>th</sup> bit off, this is used by the ASCII transfer protocol. ASCII appears to be the new standard used in the latest Fanuc controller models.

**Baud Rate** is the speed of transfer and defines the number of bits sent per second. The defacto standard is 4800 or 4800 bits/second. As most characters are made up of 10 bits (1 Start bit, 8 data bits\*, 1 stop bit) the effective character rate is 480 characters/second. **The baud rate set in the PC and in the CNC must be exactly the same.** The Data bits, and the Parity and the Stop Bits can be different to the CNC, you will still get characters transferred, there will be errors but you will still see activity. If the baud rates are different, you will have nothing but errors or no data at all.

**Stop Bits** can be 1 or 2 and in rare cases 1½. We have never seen a CNC setup for 1½. The stop bits are used to synchronize the transfer of data bits. The makeup of a character is 1 Start bit, 8 data bits\*, 1 or 2 stop bit(s). The stop bits also add a small delay between each character. Two stop bits is the standard for Fanuc controls.

**Flow Control** is used to synchronize the transfer of data into the CNC from the PC. Most CNC controls use the DC control characters. This is considered to be a software handshake protocol. The CNC will request characters by sending a Xon character to the PC and request the PC to stop sending by send a Xoff character to the PC. The CTS protocol is considered to be a hardware protocol as the CNC will raise and lower the RTS or "Ready To Send" line that is wired to the CTS or Clear to Send on the PC. When a positive voltage appears on the CTS line of the PC, the PC will send characters. There are also combinations of these two flow control methods.

**Base Folder**, here you define the base folder used on the PC to store your CNC programs. We refer to it as the base folder as you can if you wish to create sub folders below the base folder. The use of a base folder will make it easier for you to backup the programs as it keeps all your program files in the one area.

**Output Line End** in a text file, as used for CNC programs, mostly have a carriage return (CR) and a line feed character (LF) at the end of each line. Again, what a CNC can and will accept is as varied as the number brands and models. A good starting point is the default of CR + LF. Be aware how ever that you may need to change this if there are issues with the transfer.

**Input Line End**, when a program is sent from the CNC to the PC, the CNC more often than not does not send a CR character, just a LF character (10). In some case the CNC doesn't send a LF just a CR (13) character. If you see input activity, but just one long stream of characters then change this parameter to 13.

**Character and Line Delays** are used to slow the stream of characters from the PC to the CNC, in older CNC models. If you are losing characters in the middle of blocks, then introduce a character delay (start with a value of 500) if you are losing characters at the beginning of lines then introduce a Line Delay (start with a value of 500). You can also loose characters because the PC is just too fast. This may sound a bit strange when the baud rates are the same. The problem is the delay between each character in modern high speed PCs. They operate so fast that there is no perceivable delay between each character, thus the need to introduce a delay in the transfer process.

**Send First %**, some CNC controls need to see a % at the start of a transfer, while others don't, this variable allows the control of the % in a program stream of characters. One of the anomalies is that some CNCs insist on punching or sending a % out at the beginning of a program but don't want one sent back at the beginning of a transfer – *figure that out*.

**Use N in Block Numbering** - in the editor you can add and or renumber the blocks or lines of the program. Some CNC's want the N character at the start of the block number some don't.

**User Leading 0's in Block Numbering** Some older CNC's require 4 characters in the block number. This parameter tells the function to pad the number out with 0's

Recent Additions:

**Remove Rcd %** when ticked will remove all received % characters.

**Don't Send %** when ticked will stop the sending of all %s.

**Add Text Count** in combination with **Ending String** :- We have found that some of the early Mazak M32 and Mplus machines with early system software versions have an issue with completing a drip feeding or DNC operation. It

manifests itself when the transfer stops a few blocks before the end of the transfer. The controller then reports a "Tape Reader Error". What in actual fact is happening is that the FIFO buffer read function on the Mazak isn't terminating when the M30 or M02 or M99 is read from the PC even though the parameter for Tape Read ending code is set correctly.

What these 2 parameters allow is the sending of a text string (in "Ending String") to the control a number of times as defined in "Add Text Count".

The string you enter will never be processed by the control as it will read the "M30" type code and stop correctly.

**Starting String/Ending String** these two parameters add the characters you insert in the parameters at the beginning and end of transfers. The characters in the string to be sent are the numerical values for the character. They are in a 3 digit format. i.e. the numerical value for A is 65 so you must enter 065. Each value is separated by a space or comma. A chart of the entry values appears in the next page. Note that all values are 3 characters long.

DEC	CHAR
000	NULL
001	SOH
002	STX
003	ETX
004	EOT
005	ENQ
006	ACK
007	BEL
008	BS
009	TAB
010	LF
011	VT
012	FF
013	CR
014	DO
015	SI
016	DLE
017	DC1
018	DC2
019	DC3
020	DC4
021	NAK
022	SYN
023	ETB
024	CAN
025	EM
026	SUB
027	ESC
028	FS
029	GS
030	RS
031	US

DEC	CHAR
032	SPACE
033	!
034	"
035	#
036	\$
037	%
038	&
039	'
040	(
041	)
042	*
043	0
044	,
045	"
046	.
047	/
048	0
049	1
050	2
051	3
052	4
053	5
054	6
055	7
056	8
057	9
058	:
059	;
060	<
061	=
062	>
063	?

DEC	CHAR
064	@
065	A
066	B
067	C
068	D
069	E
070	F
071	G
072	H
073	I
074	J
075	K
076	L
077	M
078	N
079	O
080	P
081	Q
082	R
083	S
084	T
085	U
086	V
087	W
088	X
089	Y
090	Z
091	[
092	\
093	]
094	^
095	_

DEC	CHAR
096	`
097	a
098	b
099	c
100	d
101	e
102	f
103	g
104	h
105	i
106	j
107	k
108	l
109	m
110	n
111	o
112	p
113	q
114	r
115	s
116	t
117	u
118	v
119	w
120	x
121	y
122	z
123	{
124	
125	}
126	~
127	DEL

## Adding ISO CNC machines via the parameter page

To add a new CNC click the Parameters menu and then select ISO parameters. Now click on the “Create New CNC”\*\*\* button to firstly clear some of the values and to insert some default values in others.

The basic starting points are:

- Enter a name for the CNC, use a name that everyone knows the CNC by.
- Enter a description for the CNC, especially if there are a number of similar CNC’s
- Select the make and model from the drop down list, use the NULL machine type if yours isn’t listed.
- Click on the large dark green button and select the com port to be used for this machine. If there are a number of com ports attached to the PC then use the procedure for locating the correct com port below.
- Select 7 data bits as this is the defacto value for ISO type CNCs.
- Select even parity as this is the defacto value for ISO type CNCs.
- Select a baud rate of 4800 as this the defacto value used by many of the CNC manufacturers
- Select 2 stop bits as this the defacto value for most CNC machines, especially those of Japanese origin.
- Select 01 = Xon/Xoff (DC Ctrl) flow control, as this is the defacto standard for 95% of the standard ISO CNCs. (Ignoring special format controls like the Mazak Mazatrol controls).
- If this a new CNC then click on the “Save New Record” button to save your initial values.
- Click the red “Exit” button at the bottom right of screen to take you back to the home page.

You are now ready to test a download from the CNC to the PC. Refer to the Performing Transfers section below.

**\*\*\* If you are using a USB to RS232 adaptor, the device driver software for the device must be installed and the device needs to be plugged into the PC before clicking the “Create New CNC” button as a list of com ports attached to the PC won’t contain the USB adaptor plus there may be an error generated if there are no other com ports in the PC.**

## Editing Parameters

As above, select Parameters from the home page menu then select the ISO parameters. Using the left arrow for the previous record and right arrow for the next record, move through the CNC entries until the required CNC is being displayed.

The change CNC parameters the password is z00insp, that's zed zero zero i n s p and must be entered to enable changes.

As stated above, use the sample CNC in the machine parameters to get your first machine up and running. The sample is a basic set of values for a Fanuc control and if the target CNC is a Fanuc there is a very good chance that the transfers will work straight away or you may just need to change the baud rate.

*Remember that the parameter book for the CNC and the parameters in the CNC itself can be invaluable for setting the baud rate for the target CNC. Most communication issues with the ISO CNCs are due to an incorrect baud rate being set.*

Modifying an Existing CNC:

- Change the name for the CNC if required.
- Change the description for the CNC.
- Change the make and model from the drop down list.
- Click on the large dark green button and change the selection of the com port to be used for this machine.
- Change the data bits value if required.
- Change the parity if required, generally CNCs use even or none.
- Change the baud rate if required.
- Change the stop bits as required.
- If you have had erroneous characters in the middle of lines then enter a character line delay value, start with 500.
- If you have missing characters at the beginning of lines then add a line delay value. Start with 500.
- Enter the change parameter password.
- Click on the "Update Record" button to save your changes.
- Click the red "Exit" button at the bottom right of screen to take you back to the home page.

You are now ready to test a download from the CNC to the PC with the revised values. Refer to the Performing Transfers section below. Repeat the process above to refine your settings. Remember it's wise to make only 1 or 2 changes at a time. Changing several at a time can confuse the issues.

## Testing your Parameter Entries

From the home page menu select "Transfers", in the top left of the screen is a gold box with a small arrow to its right, click on the arrow and then click on the drop down list entry of the machine to be tested.

The best first test is to see if you can transfer a program from the CNC to the PC. At the near top of page is a row of icons. Click on the red up arrow icon to receive a program from the control. Repeat the capture process of changing parameters, reselecting the CNC, and receiving a program until you see valid data from the CNC control.

When this is working correctly it establishes that the baud rate is correct, and it establishes that the data bits are correct, especially if all characters are correct. If you have a few that are wrong, change the data bits to 8 and the parity to "none".

When you have a good program, visible on the screen, change the program number or name at the top to a name or number you know isn't in the control. Now try to send it back to the CNC. To do this, simply make sure the cursor is at the top of the editor's page and then click on the red down arrow.

If you selected the Xon/Xoff flow control method, the status box will now show the message "Waiting for Start Code". The transfer function now waits for the CNC control to request program data.

Some READ function tips.

- If the transfer starts then stops almost immediately, try deleting the % at the start of the program.
- If the transfer status and progress box reaches say 95% and the CNC is happy with what was sent then there is data at the end of the program in the CNC editor that the CNC doesn't need, just cancel the

transfer. This usually happens with programs generated on Cad/Cam systems and it's generally easier to hit the transfer cancel button than fix the Cad/Cam post processor.

- If the transfer stops prematurely, say half way through, then check the CNC error message for insufficient memory availability in the control, before modifying transfer parameters.
- If you are getting intermittent errors during transfer, introduce a small character delay, start with a value of 5 and increase or decrease the value until you achieve reliable transfers.

## Performing Transfers to the CNC

Before doing any serious transfers of programs between the CNC & the PC you should read the “Testing your Parameter Entries”.

### Reading in Programs from the CNC:

1. Click on the “Transfers” menu from the home page.
2. At the top right of screen is a gold box, in right hand corner of the box is an arrow, click this arrow and click on the entry for the target CNC.
3. Click on the large red up arrow.
4. For a standard program memory load go to the CNC and select the READ function.
5. Enter a program number if necessary.
6. Depending on the CNC control start the Read in process, on a Fanuc this is typically “EXEC”.
7. See “Testing Parameter Entries” below for any issues in reading in a program.

### Punching out a Program from the CNC:

1. Repeat the CNC selection proves above.
2. Click on the Red down arrow.
3. The status text box at the top of the screen should read “Waiting For Data”
4. Go to the CNC and command the CNC to punch out a program.
5. After the program has been sent out from the control, Xlink-64 will display the received data from the CNC. If this doesn't happen check the settings of Xlink-64 using the “Testing your Parameter Entries” procedure.

## Working with the Transfer and Edit Icons

Sending and capturing programs is dealt with in the pages above. DNC or drip feed transfers is identical to a standard transfer except you will probably use the fast load mechanism explained below. The following icons from left to right appear at the top of the transfer page.

### **Xlink-64 – XlinkAdvance- Xlink-IQ ISO Transfer Action Icons** **These are NOT all the Icons – But the most used ones**



Clear Editor Contents



Open File to Left Editor in Monochrome / Fast Mode



Open File to Left Editor Coloured



Open File to Right Editor Coloured



Send Left Edit contents to the CNC (READ)



Load CNC programs from the CNC (PUNCH)



Transfer Left Editor from Cursor Position



Transfer Right editor contents and then Left from Cursor Position



Abort Transfer



*Pause Transfers – Should only be done after Feed Hold – ensures no run on.*



Undo Left Edit changes



Redo Left Edit Undo Changes



Cut Mark Text – Available for both Left & Right Editors



Copy Marked Text – Available for both Left & Right Editors



Paste Text from Copy Text function - Available for both Left & Right Editors



Transfer contents of the right editor to the CNC



Save the contents of the Left or Right editors to a New File.



File Save – Update existing File contents from the editor.



Print the contents of the Editor – Right or Left.



Select Files for Multi Send.



Multi Transfer – Send programs selected in the Multi Send Select function



Convert All Lower Case characters to Uppercase.

Using the: **Find – Replace – Next – Replace All – The cursor is usually placed at the top of the Left editor**

To Find Text:

*In the box next to Find enter the text to be found. Click **Find**. Click - **Next** to repeat the find*

To Replace Text:

*Enter the text to Find in the box as above – Enter the replacement text in the box next to Replace – Click **Replace** – click **Next** to repeat the find & **Replace** to replace new find*

To Replace All:

*Enter the text to find as above – Enter replacement text as above – Click **Replace All**.*

A laminated A5 size page of the icons including images is available. Simply send us an email with your postal address and we will forward a copy.

## Identifying a Com Port on the PC

Possibly the most difficult aspect of setting up machines in the parameters page, is identifying the com port when there is more than one com port attached to the PC.

When you click on the large green button in parameters you will get a list containing all com ports, however not all com ports displayed are RS232 ports, some are modem devices or other USB devices that appear as com ports, and some USB ports are RS232 devices.

In the latest distribution of the software, the CD contains a utility program to find a com port. The distribution also contains a small 9 pin com port testing plug. A diagram of the plug appears below for earlier distribution users who didn't get one of these devices.

To find the port name, simply plug the test plug into the 9 pin port and run "FindPort.exe" Find port will test each port by initializing the port and testing for a response.

There will be a new testing program for verifying the baud rate, more on that when it is completed.

## The Jobbing System

The jobbing system is an essential part of the Xlink-64 systems.

The jobbing system can record most of the details about a job. Each time a user saves a program or group of programs from the CNC control, the user has the option of creating a new job record. When the user opts to create a job record, the filename used to save the program(s) is carried into the add job function along with the type of CNC and the CNC name.

There are a number of variables associated with jobbing, including a freeform area for recording basic details on the job. You can also record up to 21 photos. The first photo of which is an image of the job you are creating. The additional photos can be used for defining clamping, or material positioning or zero positioning etc.

## Using The Jobbing System

There are 2 major facets to any Jobbing system – Adding New Jobs and Finding Existing Jobs.

### Adding Jobs

There are 2 ways you can add a new Job – manually or via the various transfer functions.

The best way to add jobs is via the transfer functions as vital information is carried from the transfer function into jobbing and probably and more importantly the job details are still fresh in the mind of the user.

The user adds details over and above the details carried from the transfer systems, but as a minimum the user must add a job name.

Below are the search facilities provided in the “Find” facility of jobbing. It’s a good idea to have a policy of what are the minimum entries that should be entered in each and every job record. The program makes the user, as a minimum, enter a job name. If you are performing a manual job entry then you can locate the filename used to save the programs via the camera icon next to the filename text box. It is also important to select a machine type and the actual machine that ran the job.

<u>Name</u>	<u>Part</u>	<u>Description</u>
<u>Client</u>	<u>Filename</u>	<u>Work/Program Nos</u>
<u>Drawing</u>	<u>Job ID</u>	<u>Travel ID</u>

## **Finding Jobs**

There are 2 basic ways of finding a job as follows:

1. By scrolling through the grid of jobs and visually identifying the required job.
2. By selecting a search criteria and a value to search for.

In the initial search and in the search result grids you can right mouse click a row and a popup will appear with a brief summary of the entry. Once you have found the job required, you can review the details once you mouse click the required row.

In the details page you have a number options such as:

- Edit the record.
- Delete the record.
- Print the record.
- Activate the transfer from the details in the record. Here the filename is transferred into transfer function that was used to generate the program in the first instance. The CNC is also activated. If you installed a drawing viewer and you specified a drawing in the job, then clicking the icon will activate the drawing viewer.
- If you are manually entering an entry for a Mazak Mazatrol CMT program you can extract the work numbers from the filename specified. The machine model will be used for the extraction process.
- The systems support a job/program quarantine system. In essence if you choose to use the system, every program transferred and saved is stored into a save folder rather than the active folder. The job record for this save is then set as being quarantined. A quarantined job cannot be sent back to the CNC until it is taken out of quarantine by an authorised user. During authorisation the file is moved from the save folder

location into the active folder. This system is an attempt to prevent good programs being overwritten by not so good ones.

A note on the drawing viewer system. The viewer system installer will put the drawing viewer application and its associated files into the "Program Folder" of the PC. The location of the viewer application must be entered into the Master database. The default location of the viewer application is stored in the Master table of the database and will most likely not need changing.

If there are photos and images in the job record, all of these can be viewed once an entry has been selected.

### **AutoRepeat**

This function allows you to run a program directly from Xlink-64 and is sometimes necessary if the program is larger than the available memory in the CNC control.

To activate Auto Repeat function click on the Auto Repeat check box or on the Auto Repeat icon, to the right of the Renumber icon.