

VERSA **BUILT** ROBOTICS



VersaBuilt System Controller CNC Installation and Programming Manual

Table of Contents

<u>Section 1</u>	<u>Safety Warnings</u>
<u>Section 2</u>	<u>Cycle Start and Communication Cable Installation</u>
<u>Section 3</u>	<u>Install CNC Files and Edit Table Load and Table Wash Programs</u>
<u>Section 4</u>	<u>Configure CNC and VSC Settings</u>
<u>Section 5</u>	<u>VSC Macro CNC Drivers - Features and Programming</u>
<u>Section 6</u>	<u>VSC Generic CNC Driver - Features and Programming</u>

Safety Warnings

DANGER: VersaBuilt makes industrial machine tool automation components designed to be operated by trained personnel only. Machine tool automation components may move suddenly and without warning. Serious or fatal crushing injuries can occur from contact with the robot, gripper or vises.

Before deploying VersaBuilt industrial machine tool automation components, a safety risk assessment must be completed in accordance with local, state and/or federal requirements.

VersaBuilt industrial machine tool automation components should only be used by trained operators.

Read and understand the VersaBuilt Mill Automation Kit Safety Manual before proceeding

Cycle Start and Communication Cable Installation

Section 2

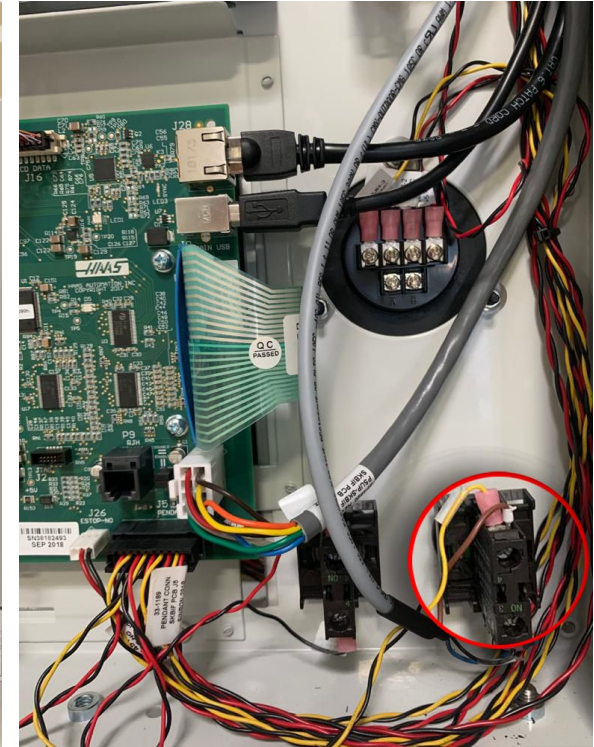
Install Cycle Start Cable

Installation instructions for Cycle Start wiring is CNC make/model/year dependent, with the following instructions provided as a guideline only

- Verify proper Cycle Start wiring installation carefully, consult with a properly trained CNC repair technician if necessary
- Remove the back panel of the CNC control
- The Cycle Start button is generally located in the back right corner of the control; trace the location of the cycle start button by locating the cycle start button on the front of the control and then looking for it in the back of the control



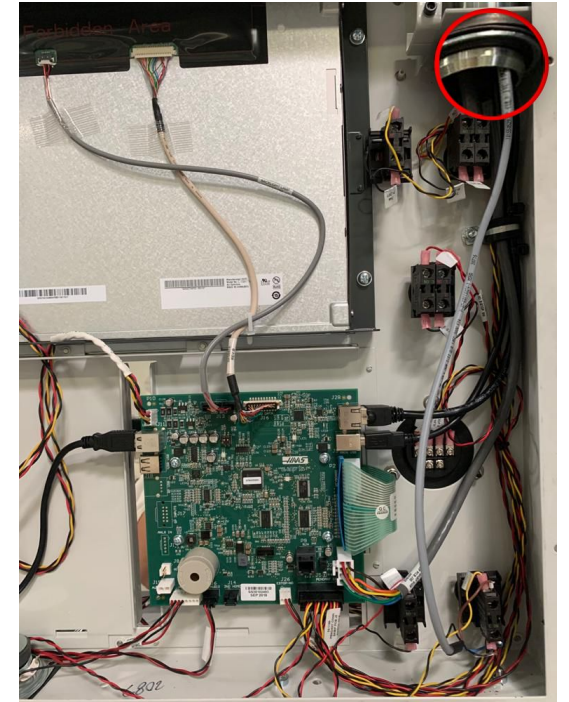
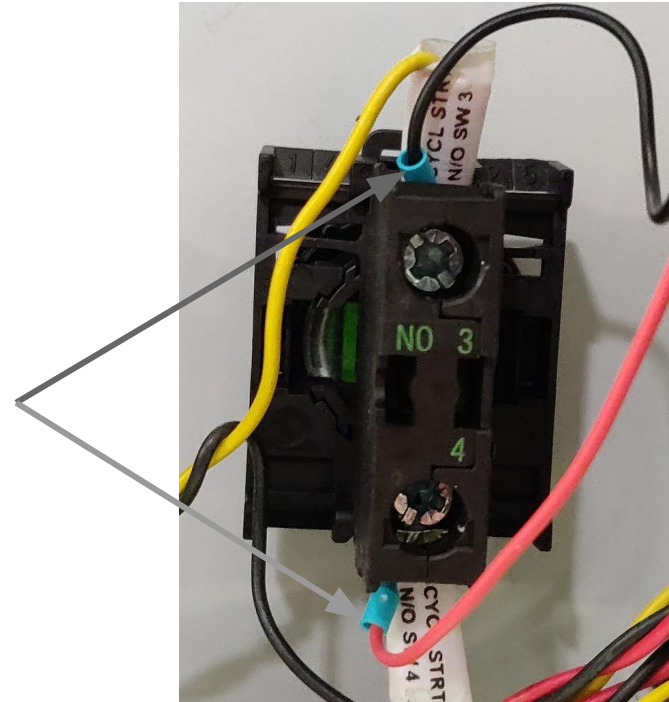
Back of CNC



Back of Cycle Start Button

Install Cycle Start Cable

- The Cycle Start Cable includes two wires that must be attached to the Cycle Start terminals
- The Cycle Start button should have two terminals, loosen the terminal screw and attach one of the Cycle Start Cable wires to each terminal
- Polarity of the wires does not matter
- Route the M8 connector side of the Cycle Start Cabling through the CNC's wire chase and back to the VersaBuilt System Controller (VSC)
- Plug the M8 connector into the Digital I/O **B1** on the VSC



Connect Communication Cable

Haas Legacy

Plug the 25-pin end of the provided serial cable into the CNC, plug the 9-pin end of the cable into the VersaBuilt System Controller (VSC).

Haas NGC, Fanuc or Okuma

Location of the user Ethernet port is machine builder dependent. Review the documentation provided with your CNC to determine the location.

Plug one end of the provided Ethernet cable into the CNC Ethernet port and the other end into the Ethernet switch mounted on the VSC.

***Route cables so they are protected from coolant and chips, anchor as needed using included cable-ties and magnetic loops*

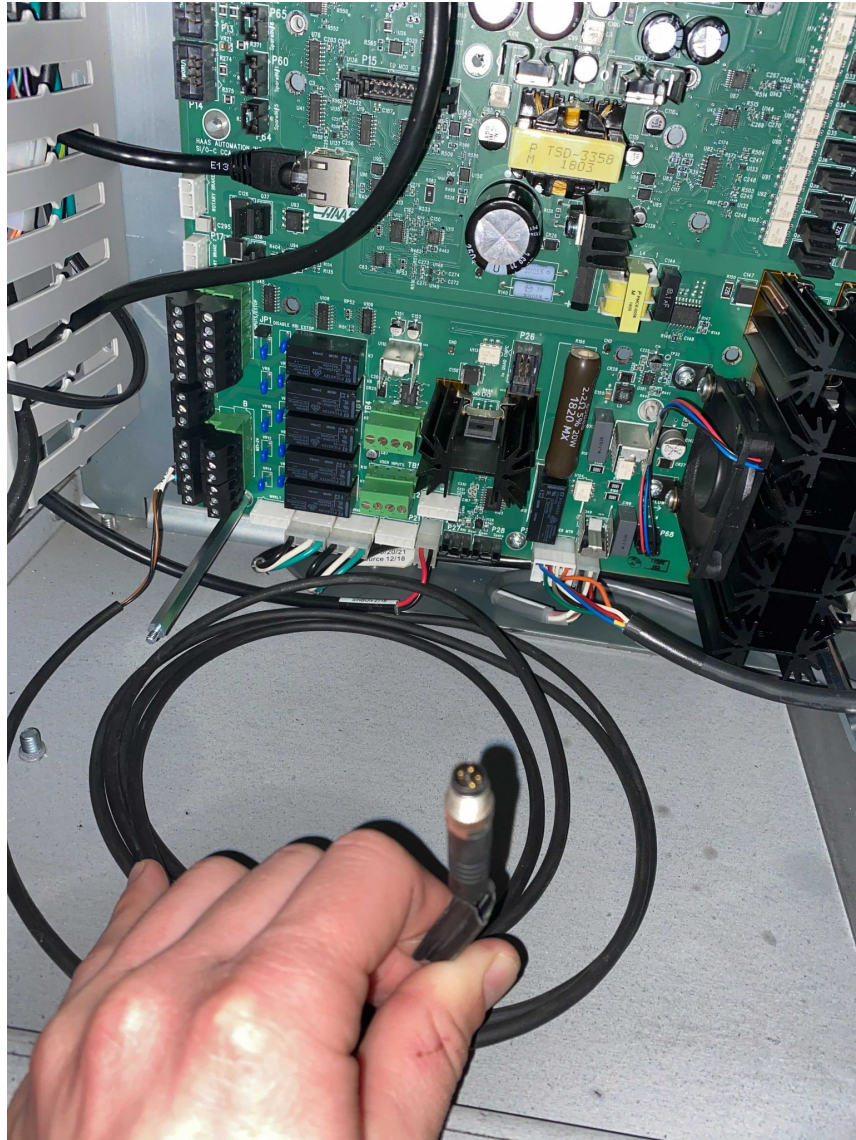


Ethernet Cable



Serial Cable

Connect Generic CNC Driver Cycle End Cable



Identify CNC End of Cycle Signal

The VersaBuilt System Controller (VSC) requires a 24 volt input signal when the CNC cycle is complete. Most CNCs provide one or more m-codes that are tied to a 24 volt output signal. When one m-code is executed, the 24 volt output is triggered, when another m-code is executed, the 24 volt output is reset. Most CNCs provide terminal blocks in the control cabinet to connect the provided Cycle End cable. Check your CNC documentation or your CNC dealer for more information.

Cycle End Cable

A Cycle End cable is provided having an M8 connector on one end and flying leads on the other end. Connect the flying leads blah blah

WARNING

Verify the voltage and the polarity of the CNC cycle end signal before connecting the Cycle End cable. Failure to do so could result in damage to the VSC or the CNC.

Install CNC Files and Edit Table Load and Table Wash Programs

Section 3

Install Gcode Files on CNC

- Insert the VersaBuilt USB Drive into the CNC control
- Navigate to the folder on the USB drive machine the CNC machine control
- Copy all gcode files from the folder onto the CNC control

MEMORY	USB DEVICE	HARD DRIVE	NET SHARE
CURRENT DIRECTORY: USB DEVICE\Haas Legacy\			
(USB DEVICE)			
9004. NC	431	07-24-2020	18:28
9000. NC	513	07-24-2020	18:28
9001. NC	889	07-24-2020	18:35
9002. NC	1617	07-24-2020	18:33
9003. NC	582	07-24-2020	18:31

Okuma Only: Install VersaBuilt vbDispatcher Program

1. Insert VersaBuilt USB drive into the Okuma USB port
2. Navigate to the Okuma directory on the VersaBuilt USB drive
3. Double-click on the setup.exe program
4. Follow the on-screen instructions
5. Click Yes to override User Account Control if prompted

The vbDispatcher program can be started from the Start menu manually or can be started automatically every time the CNC boots by adding a short cut of the vbDispatcher program to the Windows Startup folder



Edit CNC Table Load Program

- Edit the 8000 CNC Table Load Program to reference the Table Load position found in In the Mill Automation Kit - Installation Guide
- Note that the program is generic gcode and should be carefully reviewed and edited before running on your CNC
- Make sure the program includes an M98 P9004 just before the M30

```
%  
O8000(DEFAULT TABLE LOAD POSITION)  
#3000=1(LOAD POSITION NOT SET)  
(DELETE ALARM LINE WHEN SET AND TESTED)  
G0G91G49G40G28Z0 (MAKE SURE Z AXIS IS  
HOME)  
G0G90G53X0.0Y0.0 (SET TABLE LOAD POSITION)  
M98 P9004  
M30  
%
```

CNC Table Load Program positions the CNC table for robot exchange of parts and MultiGrip Jaws

Edit/Test Vise Wash Program

- Edit the 8001 CNC Vise Wash Program
- Note that the program is generic gcode and should be carefully reviewed and edited before running on your CNC
- Select a specific tool to be called when the CNC Vise Wash Program runs, should be a short tool that won't collide with parts on the vise
- Test the CNC Vise Wash Program to ensure all chips will be adequately flushed from the vises
- The Vise Wash Program must end with the CNC in the table load position (positive ack performed by Table Load program)

```
%
O8001(VISE WASH)
(-----ZIG ZAG PATTERN-----)
#3000=1(WASH ROUTINE NOT SET)
(DELETE ALARM LINE WHEN SET AND TESTED)
G0G91G49G40G28Z0(MAKE SURE Z AXIS IS HOME)
T20 M06 (ADJUST PER WASH TOOL LOCATION IN TOOL TURRET)
G90
G54(ADJUST WORK COORDINATE TO MATCH VISE 1)
M08
G00 X-3.0Y3.0
G01X-3.0Y-3.0F400.
X-1.5Y3.0
X-1.5Y-3.0
X0.0Y3.0
X0.0Y-3.0
X1.5Y3.0
X1.5Y-3.0
X3.0Y3.0
X3.0Y-3.0
G55(ADJUST WORK COORDINATE TO MATCH VISE 2)
G00 X-3.0Y3.0
G01X-3.0Y-3.0F400.
X-1.5Y3.0
X-1.5Y-3.0
X0.0Y3.0
X0.0Y-3.0
X1.5Y3.0
X1.5Y-3.0
X3.0Y3.0
X3.0Y-3.0
M09
M98 P8000
M30
%
```

**CNC Wash Program
cleans vises and jaws of
debris/chips between
operations**

Configure CNC and VSC Settings

Section 4

Haas NGC CNC and Autodoor Settings

Settings: Network Tab

Wired Network Enabled	On
Obtain Address Automatically	Off
IP Address	192.168.2.3
Subnet Mask	255.255.255.0

Settings: Settings Tab

131 Auto Door	On*
143 Machine Data Collection	9000

* Only if Haas-brand auto door is installed

Settings			
Settings	Network	Rotary	User Positions
Wired Connection		Wireless Connection	
Wired Network Information		Net Share	
Host Name		DHCP Server	
Domain		IP Address	
DNS Server		Subnet Mask	
Mac Address		Gateway	
DHCP Enabled		Status	
HaasCNC1234567		*	
		192.168.2.3	
		255.255.255.0	
00:C0:08:88:47:7A		*	
OFF		UP	

Haas Legacy CNC and Autodoor Settings

Settings: IO Tab

11	Baud Rate:	115,200
12	Parity Select:	NONE
13	Stop Bit:	1
14	Synchronization:	XON/XOFF
37	RS-232 Data Bits	8
41	Add Spaces RS232	ON
69	DPRNT Leading Sp	OFF
143	Machine Data Col	ON
131	Auto Door	ON*

* Only if Haas-brand auto door is installed

GENERAL		PROGRAM		I/O	CONTROL PANEL		SYSTEM		MAINTENANCE		POWER SETTINGS		
RS-232 PORTS													
11	MAX FEED (INCH/MIN)										115200		
12	MAX FEED (DEG/MIN)										NONE		
13	FEED ACCEL ROUGH										1		
14	FEED T CONST ROUGH										XON/XOFF		
37	FEED DELTA V ROUGH										8		
24	FEED ACCEL MEDIUM										NONE		
25	FEED T CONST MEDIUM										CR LF		
41	FEED DELTA V MEDIUM										ON		
50	FEED ACCEL FINISH										XON/XOFF		
54	FEED T CONST FINISH										4800		
69	FEED DELTA V FINISH										OFF		
70	AUTOFEED-STEP-UP										ON		
143	AUTOFEED-STEP-DOWN										ON		
187	AUTOFEED-MIN-LIMIT										OFF		
155	RIG TAP FINISH DIST										OFF		
156	ROTARY AXIS INCRMNT										ON		
157	CELL SAFE PORT										A		
Setting 11 - Baud Rate Select													

Okuma CNC and Autodoor Settings

Setup a static TCP/IP address configuration on the Okuma CNC using the built-in Windows network setup:

IP Address	192.168.2.3
Subnet Mask	255.255.255.0
Gateway	192.168.2.1
Preferred DNS Server	8.8.8.8

Configuration of the Okuma CNC autodoor varies by model year. Refer to the documentation included with your CNC to configure the Okuma CNC to close the door on Cycle Start and open the door on Cycle End.

Internet Protocol Version 4 (TCP/IPv4) Properties

General

You can get IP settings assigned automatically if your network supports this capability. Otherwise, you need to ask your network administrator for the appropriate IP settings.

☐ Obtain an IP address automatically

☒ Use the following IP address:

IP address: 192 . 168 . 2 . 3

Subnet mask: 255 . 255 . 255 . 0

Default gateway: 192 . 168 . 2 . 1

☐ Obtain DNS server address automatically

☒ Use the following DNS server addresses:

Preferred DNS server: 8 . 8 . 8 . 8

Alternate DNS server:

☐ Validate settings upon exit

Advanced...

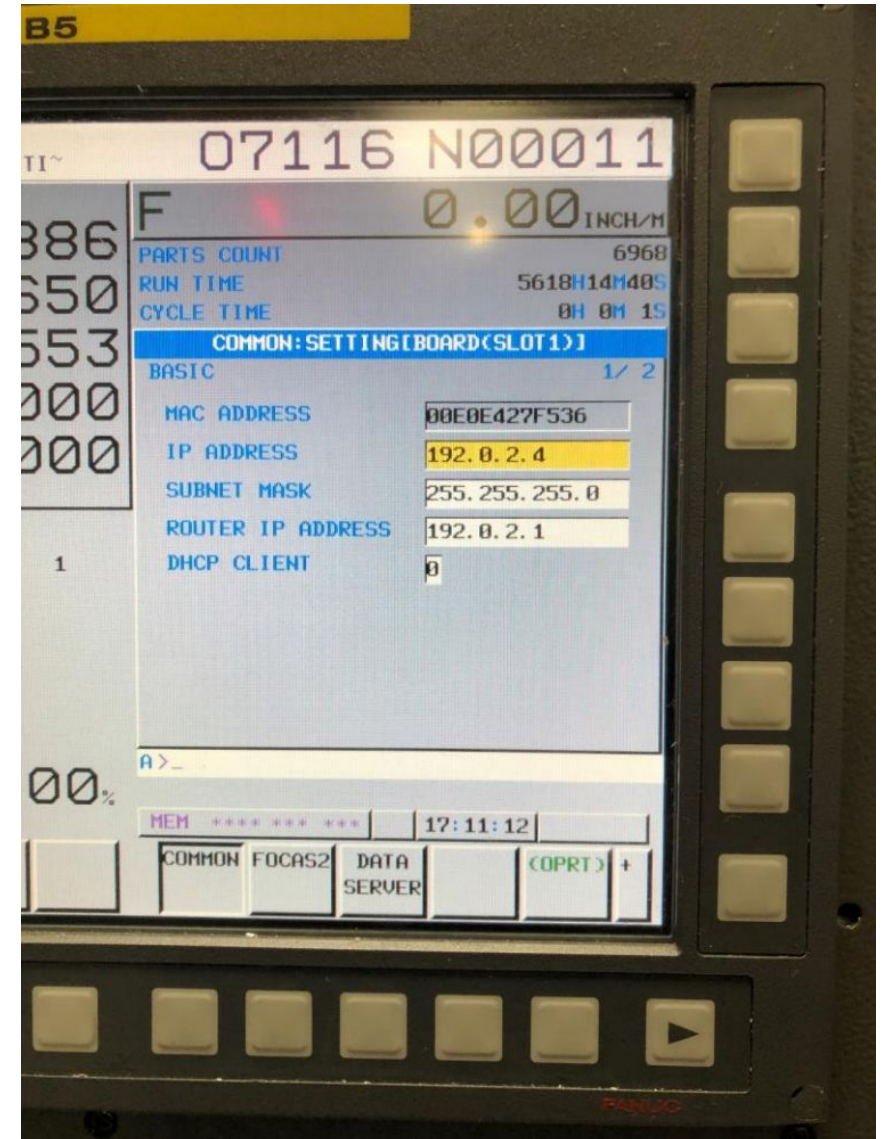
OK Cancel

Fanuc CNC Settings

- On the Fanuc control, press the System button
- Press the right arrow until the Ethernet soft key appears
- Press the Ethernet soft key

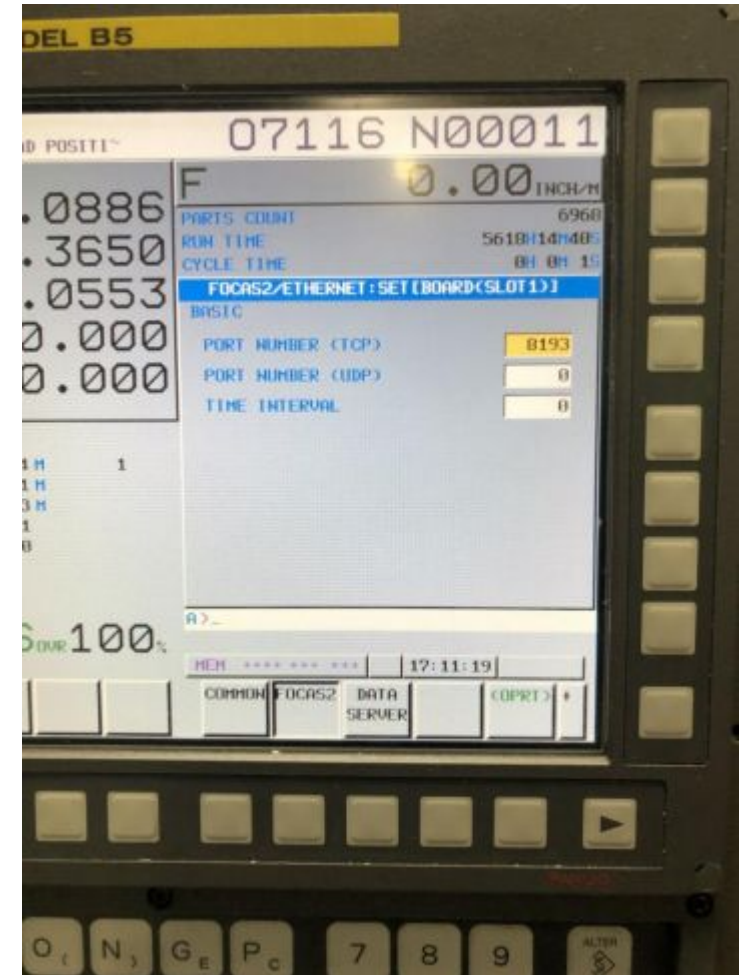
Set the following CNC Network settings:

IP Address	192.168.2.3
Subnet Mask	255.255.255.0
Router IP Address	192.168.2.1
DHCP Client	0



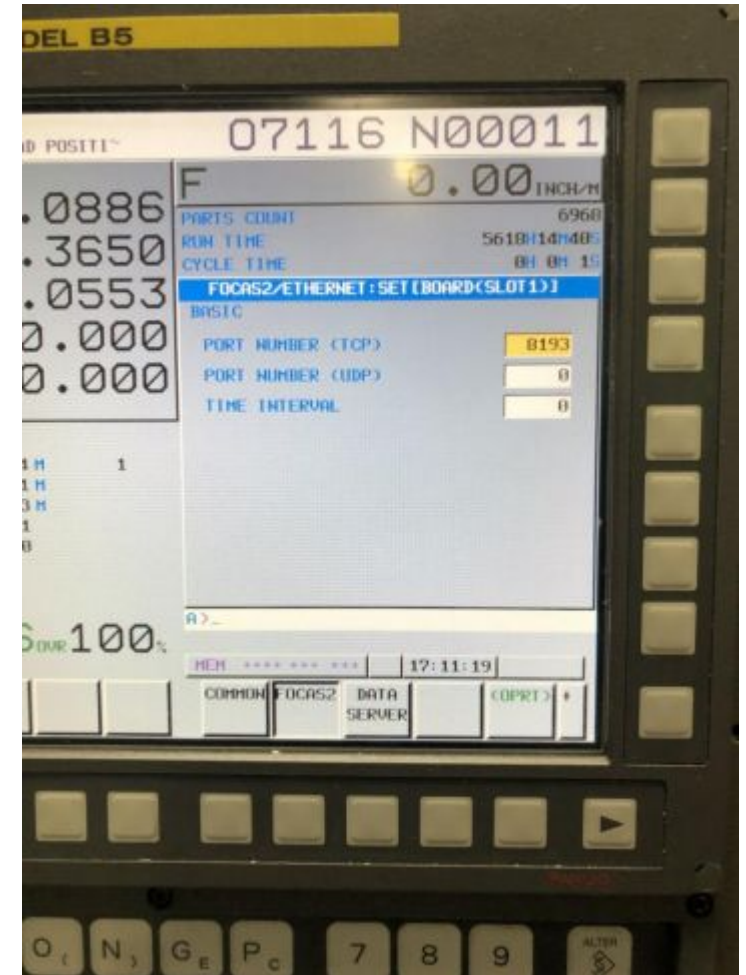
Fanuc CNC Settings

- Press the FOCAS2 soft key
- Set the Focas2 TCP port to 8193
- The other settings can be left alone



Fanuc CNC Autodoor Settings

- Autodoor settings for Fanuc-based CNC controls are machine builder-specific
- Refer to your CNC documentation to configure the CNC autodoor to close on Cycle Start and Open on Cycle End



VSC System Settings

Main System Settings	
Setting	Value
Units	Imperial
Infeed Template	UR10
CNC Dispatcher Program	9000
Default CNC Wash Program	8001
Vises	2
Robot	UR - UR10
Robot IP	192.168.2.2
CNC	Focas
CNC IP	192.168.2.3
IO	I2C
CNC Door Controller	VBX Controlled Autodoor
Vise Mode	IO
Ignore Vise Sensors	true
Ignore Door Sensors	true
Door Wait Time	7000
Remote Support Token	Not yet configured

Edit System Settings

- Navigate to the System Settings page on the VSC
- Press the Edit System Settings button
- Select the CNC corresponding to the type of CNC controlled by the VSC: Haas, Haas Legacy, Fanuc Focas, Okuma or Generic
- If using a CNC Controlled Autodoor: set CNC Door Controller to CNC Controlled Autodoor
- If using a VersaBuilt VersaDoor, set CNC Door Controller to VSC Controlled Autodoor
- If VersaDoor door sensors are installed, set Ignore Door Sensors equal to false and Door Wait Time equal to 0
- If VersaDoor door sensors are not installed, set Door Sensors equal to true and Door Wait Time equal to the greater of the time measured to open or close the CNC door plus one second (door wait time is in milliseconds, 7000 equals 7 seconds)

VSC Macro CNC Drivers Features and Programming

Section 5

VSC Macro CNC Driver Overview

The VersaBuilt System Controller (VSC) includes two types of drivers to control a CNC during automation processing: Generic CNC Driver and Macro CNC Drivers. The Generic CNC Driver features and programming is documented in the next section.

The VSC Macro CNC Drivers allow bi-directional communication between the CNC and the VSC using the CNC's macro variables via a communication link. Bi-directional communication between the CNC and the VSC enables the following features:

- Run any gcode program stored on CNC from VSC
- Verify CNC is ready for automation processing
- CNC and VSC control of vises and vise pressure
- VSC notification of successful or failed CNC program

The VSC starts the CNC by tying into the CNC's cycle start button and using a relay to close the circuit, simulating the effect of pressing the cycle start button. The VSC Macro CNC Drivers detect the end of a signal by first looking for a positive acknowledgement of end of program from the CNC gcode program followed by polling the CNC status for an Idle status.

VSC Macro CNC Driver Supported CNC Controls

The VSC includes macro CNC driver support for following CNC control types:

- Haas Legacy Controls for mills and lathes
 - Must include setting 143 machine data collection
 - Must have user-definable macros option enabled
- Haas NGC Controls for mills and lathes
 - Must have user-definable macros option enabled
- Fanuc Controls for mills and lathes:
 - FANUC Series 30i /31i /32i-MODEL A
 - FANUC Series 31i-MODEL A5
 - FANUC Series 30i /31i /32i /35i-MODEL B
 - FANUC Series 31i-MODEL B5
 - FANUC Power Motion i-MODEL A
 - FANUC Series 0i-MODEL D/F
- Okuma OSP-P300 Mill Controls with ThinkAPI version 1.17.2 or newer

VSC Macro CNC Driver Operational Overview

Dispatcher Program (Haas and Fanuc Controls)

For Haas and Fanuc CNC controls, a dispatcher program is provided. The dispatcher program, program 9000 by default, must be selected in memory prior to starting the automation process. Before execution, the VSC writes to a set of shared macro variables on the CNC. One macro variable is for command control the other macro variable stores parameters. For program execution, the VSC writes to the command macro variable a request for program execution and writes the number of the CNC program to be executed in the parameter macro variable.

When cycle started, the dispatcher program looks to the command macro variable for the CNC program execution command and then reads the parameter variable to determine what CNC program to execute.

During execution, the VSC polls the command variable looking for other requests, such as CNC vise control and for a positive acknowledgement of end of cycle.

VSC Macro CNC Driver Operational Overview

Direct Program Select (Okuma Controls)

For Okuma controls, a vbDispatcher program is provided as an MS Windows executable program. This program must be running in the background before the VSC automation process is started. The vbDispatcher program allows the VSC to read and write macro variables and to select the current CNC program in memory.

Before the CNC is cycle started, the VSC sends a command to the vbDispatcher program to select the required program number.

During execution, the VSC polls the command variable looking for other requests, such as CNC vise control and for a positive acknowledgement of end of cycle.

VSC Macro CNC Driver Programing Requirements

Table Load and Wash Programs

A Table Load and Wash Program must be present in the CNC control. By default the Table Load program is 8000 and the Wash Program is 8001. Sample programs are provided but must be customized for each installation.

The Table Load program positions the CNC table for robot load/unload and is the table position recorded during the calibration procedure. The Table Load program is used at the start of a job and in VSC Recovery mode to position the table for load/unload.

The Wash Program must clean the vises of chips and other debris to ensure the success of the automation process and finish with the CNC in the table load position. The Wash Program must be in the control as a stand-alone program.

VSC Macro CNC Driver End of Program Sequence

Positive Acknowledgement of CNC Program Completion

The protocol between the CNC and the VSC requires any gcode program run by the VSC to positively acknowledge to the VSC successful completion of the program. This is done by writing a value to a macro variable shared between the CNC and the VSC and should be done just prior to program end. For Haas and Fanuc controls a helper sub-program 9004 is provided.

Example:

```
...  
M98 P9004  
M30  
%
```

For the Okuma controls, the value 2 is written to variable VC190. Example:

```
...  
VC190=2  
M30  
%
```


VSC Macro CNC Driver End of Program Sequence

VersaBuilt recommends the Table Load program end with VSC positive acknowledgement followed by M30. Example:

```
...  
M98 P9004  
M30  
%
```

VersaBuilt recommends the Wash Program end with a call to the Table Load program to position the table and provide VSC positive acknowledgement:

```
...  
M98 P8000  
M30  
%
```

VersaBuilt recommends all CNC milling programs run by the VSC end with a call to the Wash Program:

```
...  
M98 P8001  
M30  
%
```

VSC Macro CNC Driver Vise Control

The VSC Macro CNC Driver allows each of the MutliGrip FJ Vises to be opened or closed from within a CNC program using the included CNC subprograms:

- 9002 program is used for vise control
- Call the 9002 program using the G65 command and using D and C parameters:
 - D01. selects vise 1
 - D02. selects vise 2
 - C02. closes the vise
 - C03. opens the vise
- Recommend dwelling 1 second to allow vise to actuate:
 - G04 P1.

EXAMPLES:

- Close Vise 1: G65 P9002 D01. C02.
- Open Vise 1: G65 P9002 D01. C03.
- Close Vise 2: G65 P9002 D02. C02.
- Open Vise 2: G65 P9002 D02. C03.

NOTE: decimals after the numeric values shown are important for proper function

VSC Generic CNC Driver Features and Programming

Section 6

VSC Generic CNC Driver Overview

The VSC Generic CNC Driver can be used with almost any CNC but is limited to the following features:

- Run the gcode program currently selected in memory
- Run one op or two op milling operation automation processes
- VSC notification when a program completes successfully

Before starting the automation process, the operator must select the proper CNC program for the automation process. The CNC must provide an end of cycle signal to the VSC after the milling operation is complete.

For multi-operation automation processes, all milling operations must be in a single program. The end of each milling operation must set the end of cycle signal, then execute an M0 command to pause the CNC at the current location. After the M0, the CNC milling program must reset the end of cycle signal. Single operation and multi-operation programs should end in an M30.

Examples are provided on the following pages for CNC machines that have macros enabled and without macros enabled. Consult your CNC documentation to determine if your CNC has macros enabled.

VSC Macro CNC Driver Programing Requirements

Table Load and Wash Programs

Sample Table Load and Wash programs are provided but must be customized for each installation. The Table Load program positions the CNC table for robot load/unload and is the table position recorded during the calibration procedure. The Wash program must clean the vises of chips and other debris to ensure the success of the automation process and finish with the CNC in the table load position. At the end of each milling operation, the vises must be washed and the CNC in the table load position. For CNCs that support subprogram calls, the Table Load and Wash program can be stored as separate subprograms in the control can called at the end of each milling program. For CNCs that do not support subprograms, the Wash and Table Load gcode must be copied and pasted into the end of each milling program.

Edit Mcodes for CNC Configuration

Sample gcode for use with Generic CNC Driver is included on the VersaBuilt USB in the Generic-Macro directory. This gcode must first be edited to replace the sample mcodes used to set and clear the End of Cycle signal from the CNC with the actual mcodes used in your configuration.

Generic-Macros Gcode Two Op Program Editing

```
1  %
2  O9000      (Dispatcher Program - Two Op - Generic
3  #10000=100  (Op1 Program - Edit)
4  #10001=101  (Op2 Program - Edit)
5  #10002=8001 (Wash/Table Load Program)
6
7  G103 P0.    (Enable Look-ahead)
8  M98 P[#10000] (Op1 Program)
9  M98 P[#10002] (Wash/Table Load Program)
10 G103 P1.    (Disable Look-ahead)
11 M51        (Set Handshake)
12 M0         (Wait for robot)
13 M61        (Clear Handshake)
```

- Edit each milling program and replace the M30 at the end of the file with an M99
- Edit the 9000 gcode program to map Op 1 and Op 2 milling program numbers to the #10000 and #10001 macro variables
- Confirm 8001 Wash/Table Load, Op 1, and Op 2 programs are stored as files in memory; make sure to end programs with the Wash/Table Load
- Select the edited 9000 program as the active program
- Place CNC in Memory Mode

NOTE: If desired, the edited 9000 program can be copied and saved to a new program number so the 9000 program will not need editing before each job

Generic-Macros Gcode One Op Program Editing

```
1  %
2  O9000      (Dispatcher Program - One Op -
3  #10000=100 (Op1 Program - Edit)
4  #10002=8001 (Wash/Table Load Program)
5
6  G103 P0.   (Enable Look-ahead)
7  M98 P[#10000] (Op1 Program)
8  M98 P[#10002] (Wash/Table Load Program)
9  G103 P1.   (Disable Look-ahead)
10 M51        (Set Handshake)
11 M0         (Wait for robot)
12 M61        (Clear Handshake)
13 G103 P0.   (Enable Look-ahead)
14 M30
15 %
```

- Edit the milling program and replace the M30 at the end of the file with an M99
- Edit the 9000 gcode program to map Op 1 milling program number to the #10000 macro variables
- Confirm 8001 Wash/Table Load and Op 1 program is stored as files in memory; make sure to end milling programs with the Wash/Table Load
- Select the edited 9000 program as the active program
- Place CNC in Memory Mode
- Make sure CNC door is open

NOTE: If desired, the edited 9000 program can be copied and saved to a new program number so the 9000 program will not need editing before each job

Generic Gcode Two Op Program Editing

```
1  %
2  O9000      (Dispatcher Program - Two Op - Generic Macro Disabled)
3
4  G103 P0.   (Enable Look-ahead)
5  (Insert Op1 Program) (Op1 Program)
6  (Insert Wash Program) (Wash/Table Load Program)
7  G103 P1.   (Disable Look-ahead)
8  M51        (Set Handshake)
9  M0         (Wait for robot)
10 M61        (Clear Handshake)
11 G103 P0.   (Enable Look-ahead)
12 (Insert Op2 Program) (Op2 Program)
13 (Insert Wash Program) (Wash/Table Load Program)
14 G103 P1.   (Disable Look-ahead)
15 M51        (Set Handshake)
16 M0         (Wait for robot)
17 M61        (Clear Handshake)
18 G103 P0    (Enable Look-ahead)
19 M30
20 %
```

- In a text editor, open a copy of the Generic Two Op gcode program, Table Load, Wash, Op1 and Op 2 milling programs
- In the Op1 milling program, begin text selection after the %OXXXX program number down to but not including the M30 at the end of the program and copy the selection
- In the Generic Two Op gcode program, paste the Op 1 program text below the line that begins with:
(Insert Op 1 Program)
- Repeat the text selection, copy and paste steps for the Wash, Table Load and Op 2 programs (note Wash and Table load will be pasted below the Op 1 and Op 2 programs)
- Select the edited 9000 program as the active program
- Place CNC in Memory Mode

NOTE: If desired, the edited 9000 program can be copied and saved to a new program number so the 9000 program will not need editing before each job

Generic Gcoe One Op Program Editing

```
1  %
2  O9000      (Dispatcher Program - One Op - Generic Macro Disabled)
3
4  G103 P0.   (Enable Look-ahead)
5  (Insert Op1 Program) (Op1 Program)
6  (Insert Wash Program) (Wash/Table Load Program)
7  G103 P1.   (Disable Look-ahead)
8  M51        (Set Handshake)
9  M0         (Wait for robot)
10 M61        (Clear Handshake)
11 G103 P0.   (Enable Look-ahead)
12 M30
13 %
```

- In a text editor, open a copy of the Generic One Op gcode program, Table Load, Wash and milling programs
- In the milling program, begin text selection after the %OXXXX program number down to but not including the M30 at the end of the program and copy the selection
- In the Generic One Op gcode program, paste the Op 1 program text below the line that begins with:
(Insert Op 1 Program)
- Repeat the text selection, copy and paste steps for the Wash, Table Load programs
- Select the edited 9000 program as the active program
- Place CNC in Memory Mode

NOTE: If desired, the edited 9000 program can be copied and saved to a new program number so the 9000 program will not need editing before each job

VERSABUILT ROBOTICS