

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>Lathe Chuck 1 and Chuck 2 cable installation</u>
<u>Section 4</u>	<u>Install CNC Files and Edit Table Load and Table Wash Programs</u>
<u>Section 5</u>	<u>Configure CNC and VSC Settings</u>
<u>Section 6</u>	<u>VSC Macro CNC Drivers - Features and Programming</u>
<u>Section 7</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

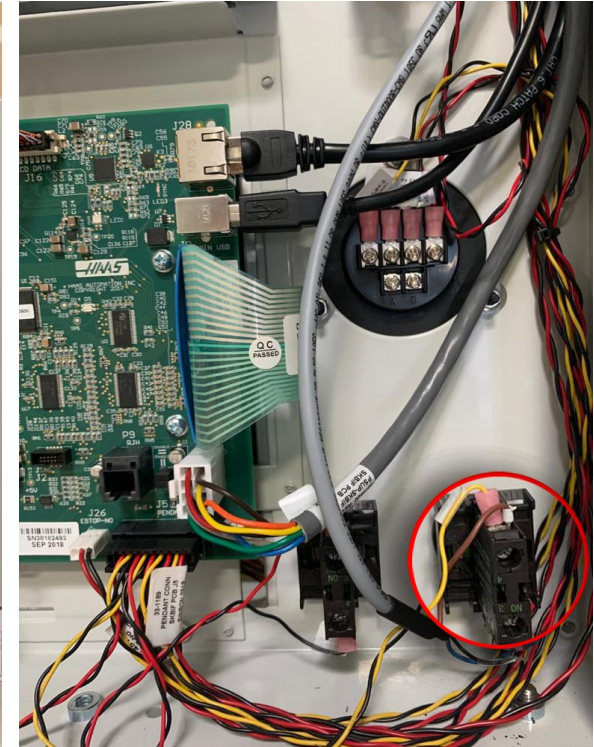
Install Cycle Start Cable

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



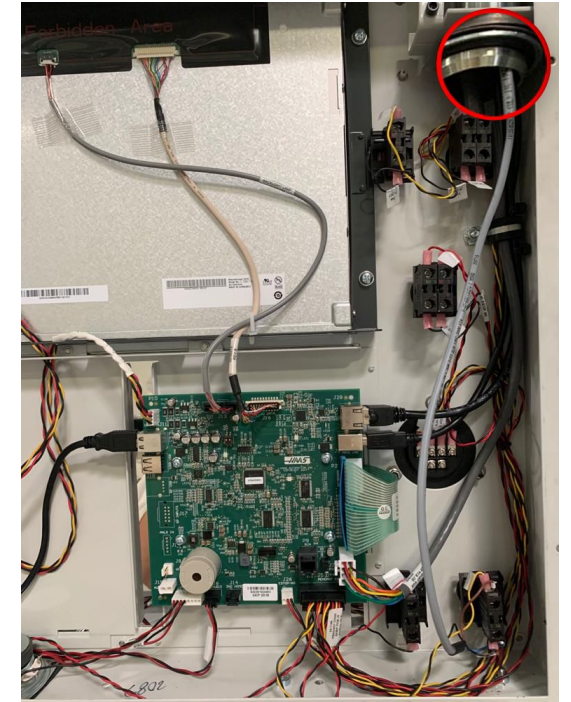
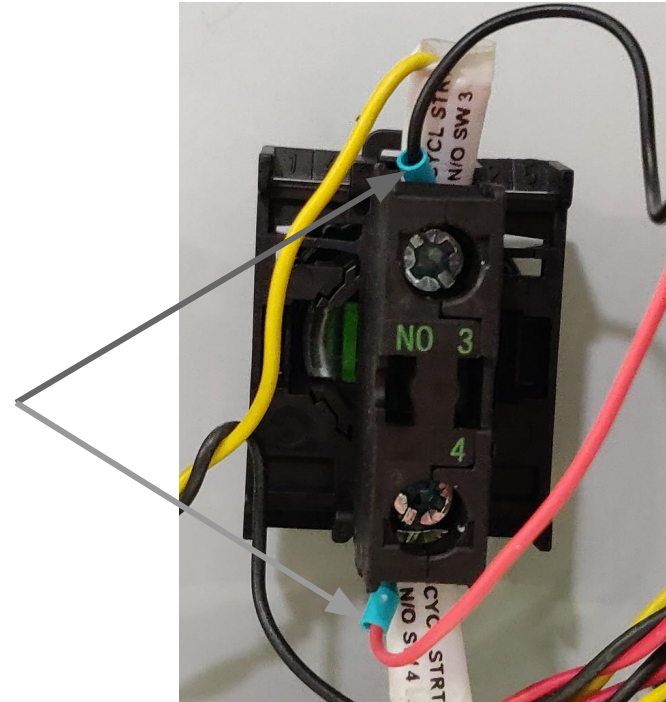
Back of CNC



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*



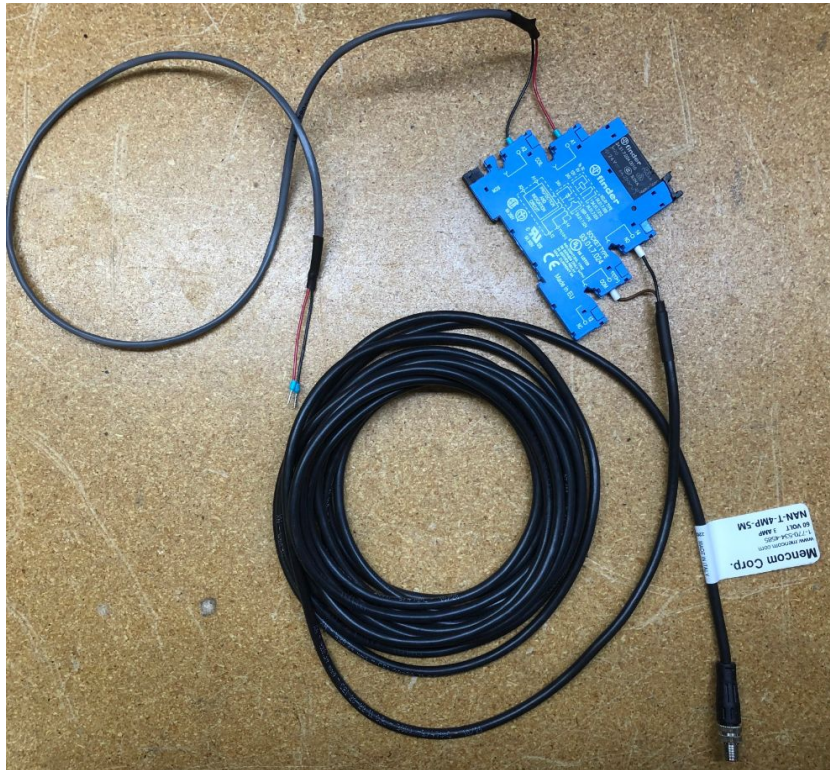
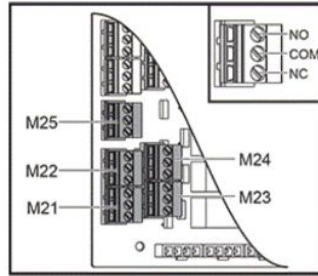
Ethernet Cable



Serial Cable

Connect Generic CNC Driver Cycle End Cable

Main I/O PCB M-code Relays.



Identify CNC End of Cycle Signal on CNC

For Generic Driver applications, 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. These can be used as a user defined m-code pair for turning signals on and off. When one m-code is executed, a 24 volt output is triggered, when the other 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.

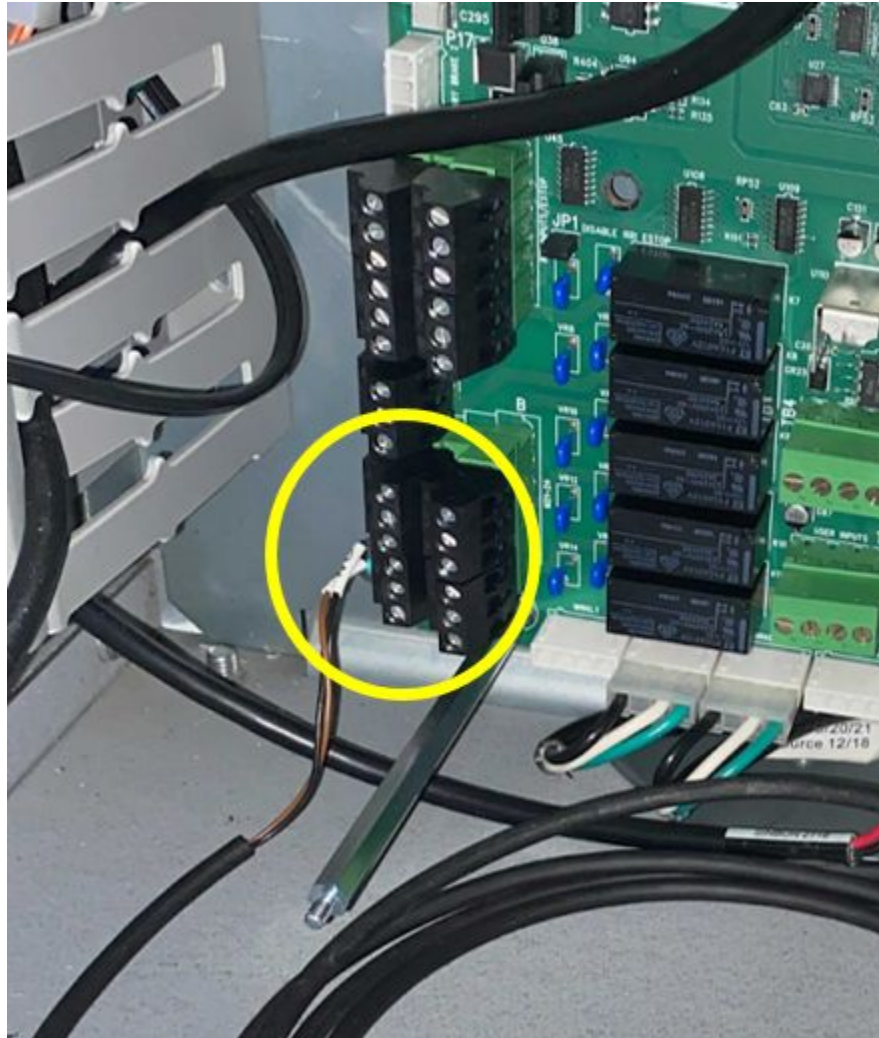
Cycle End Cable

A Cycle End cable is available from VersaBuilt for Generic Driver applications (see image to the left). The cable assembly, VersaBuilt P/N 5010458, is comprised of an M8 4-pin male connector on one end, connected to a relay and flying leads on the other end. Connect the flying leads to an available 24VDC M-Code output. One wire to a 24vdc signal M-code, the other to 0vdc.

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.

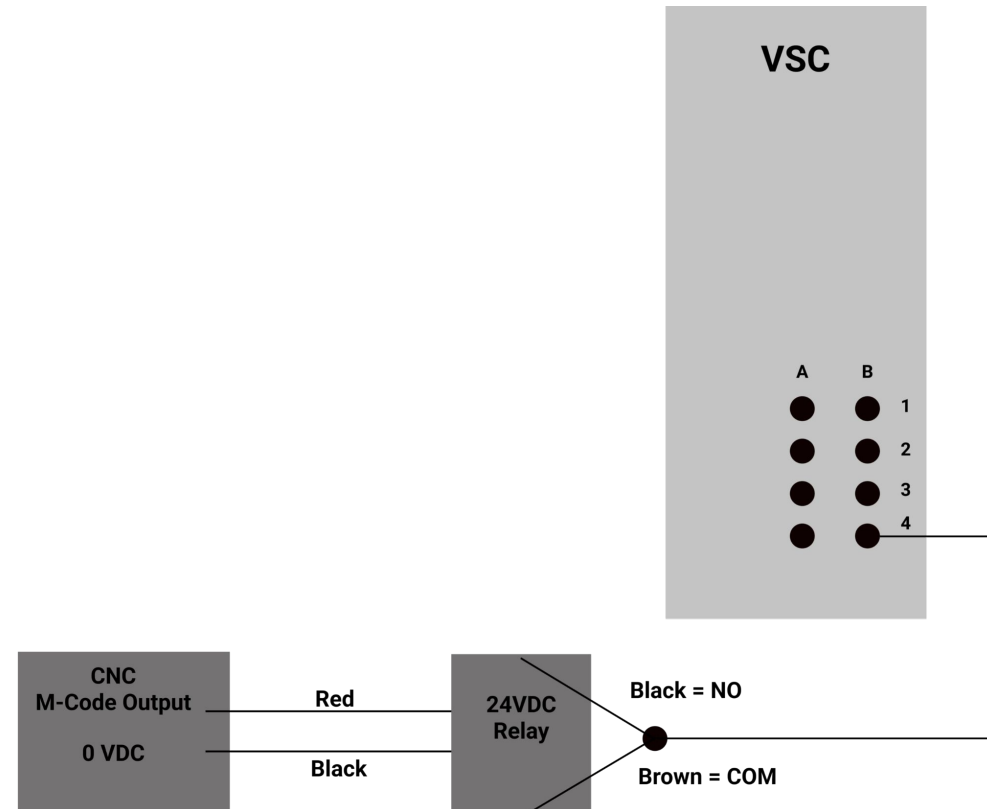
Connect Generic CNC Driver Cycle End Cable



M-Code Connection Example
Haas Relay, M-code controlled

Connect Cable From M-Code connection to VSC

After connecting to M-code terminals at CNC, connect M8 Cable to VSC, position B4, as shown below. To test and/or troubleshoot M-code signalling, set the M-code with the CNC while reviewing the VSC I/O Panel screen, input named "CYCLE_COMPLETE". The input will display as "ON" when the M-code signal is on, the "OFF" when the signal is reset.



Lathe Chuck Cable Installation

(Lathe Version Only)

Section 3

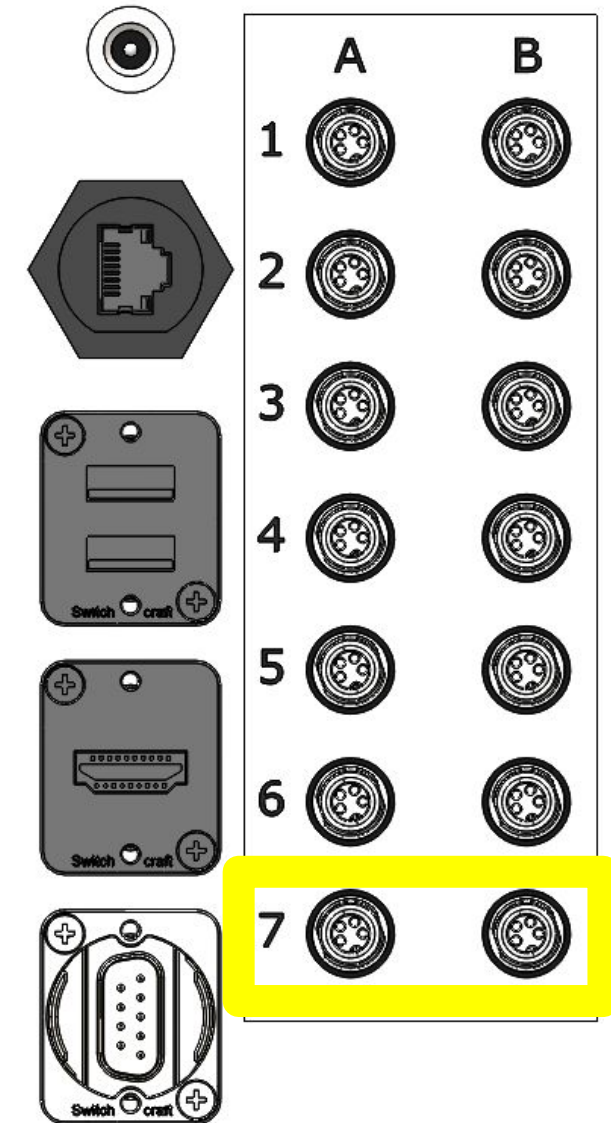
Lathe Chuck Cable Installation

Cables connected to A7 and B7 on the VSC are intended to be tied into either existing foot pedal(s) at the CNC or to CNC I/O to trigger Chuck 1 or Chuck 2 Open/Close

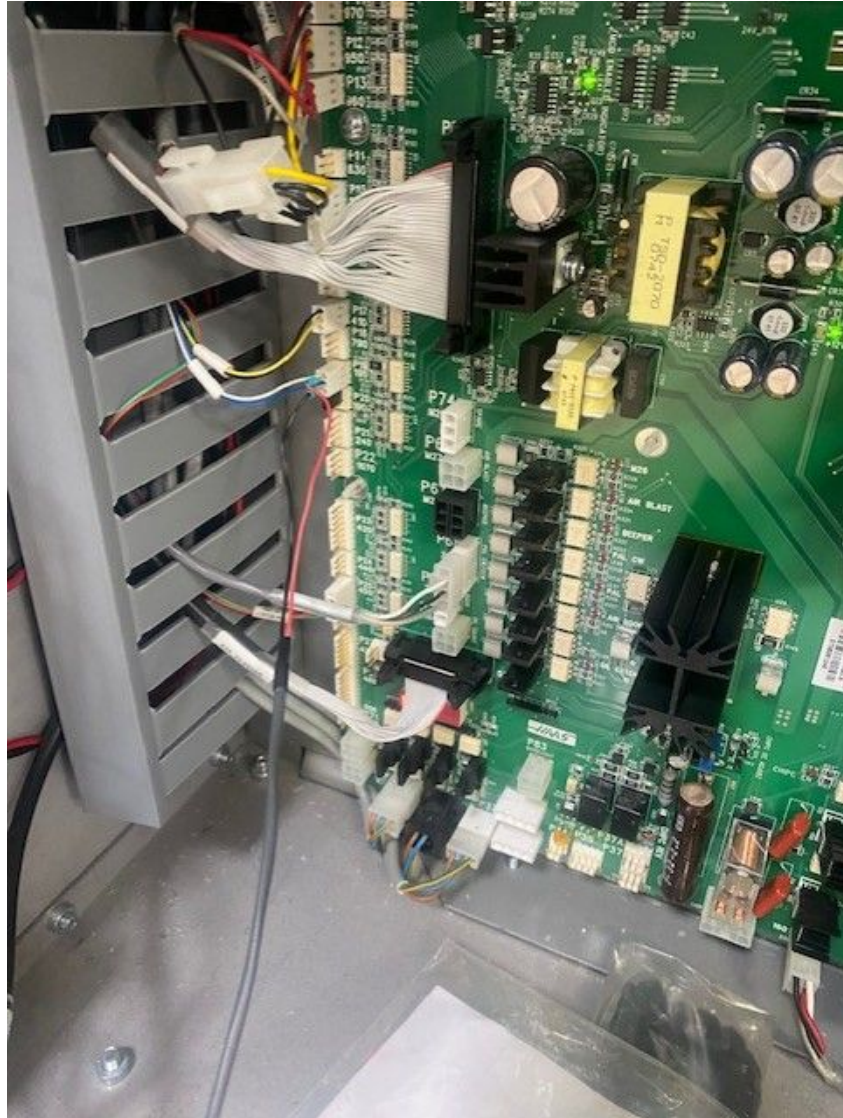
- **A7 = Chuck 1 Actuation**
- **B7 = Chuck 2/Tailstock Actuation**

The following pages provide installation instructions for installing the cable

*Legacy systems are connected to A1 and A2 *if your VCS does not have M8 connectors for A7 and B7, the Chuck cables connect to A1 and A2*



Lathe Chuck Cable Installation



Installation of Chuck Open/Close Cable to CNC Terminal Block:

Chuck Open/Close cable connects to the CNC by way of the chuck actuation terminal blocks in CNC electrical cabinet or the Chuck Foot Pedal. Contact your CNC maker to determine if a chuck control connector is available in the CNC electrical cabinet and how to connect to it.

Connect Cable From Terminal Block connection to VSC

After connecting to chuck actuation terminal blocks at CNC, connect M8 Cable to VSC position A1 for chuck 1 and A2 for chuck 2. To test and/or troubleshoot chuck actuation terminal blocks, set the “CHUCK1_OPEN” to high and set “CHUCK1_CLOSE” in the VSC I/O Panel screen, for chuck 2 repeat with “CHUCK2_OPEN” and “CHUCK2_CLOSE”. Swap states and makes sure the chuck(s) actuate(s).

Lathe Chuck Cable Installation

Installation of Chuck Open/Close Cable to Foot Pedal:

Chuck Open/Close cable connects to the CNC by way of the Chuck Foot Pedal Switch or a connector in the CNC electrical cabinet. Contact your CNC maker to determine if a chuck control connector is available in the CNC electrical cabinet and how to connect to it.

If no chuck control connector is available, the Chuck Open/Close Cable will be wired into the CNC foot pedal switch.

The following pages include images and instructions for connecting the cable to a foot pedal



Lathe Chuck Cable Installation

Foot Pedal Wiring Installation:

Unscrew the fasteners securing the cover to the foot pedal, exposing wire connections.



Lathe Chuck Cable Installation

Foot Pedal Wiring Installation:

Some foot pedals will include a cable gland that will allow the Chuck Open/Close Cable to pass through; if a cable gland is available, use it to pass the cable through.

If a cable gland is not available, a hole can typically be drilled into the foot pedal and a cable gland added.

Add spade connectors (crimp in place) to ends of flying leads of cable.



Lathe Chuck Cable Installation

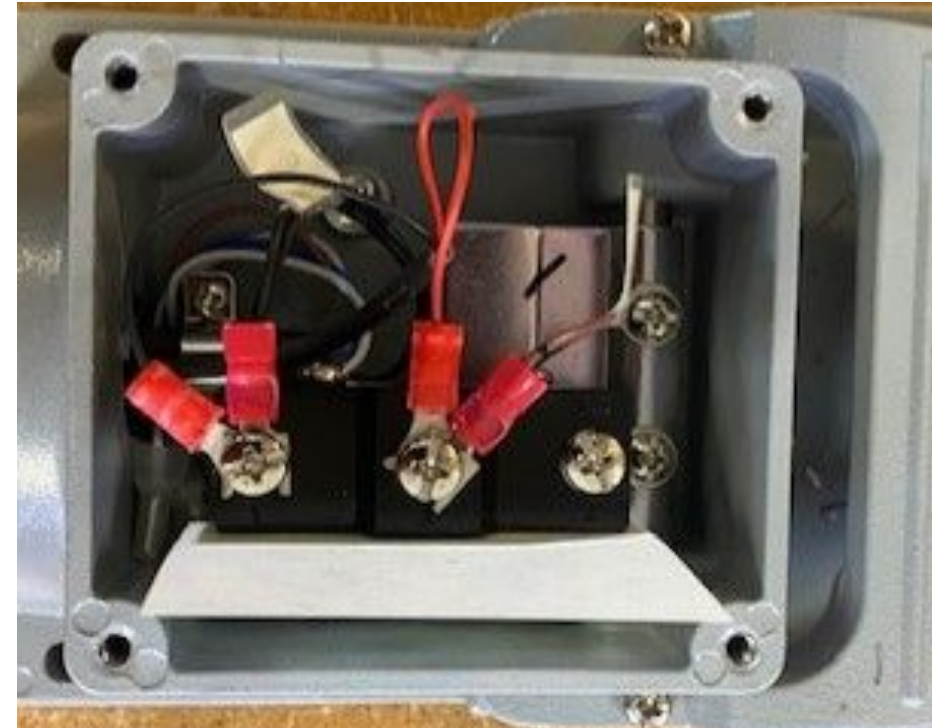
Foot Pedal Wiring Installation:

Most foot pedals will have two terminal connections for a contact switch or a 2-wire proximity switch. If your foot pedal has a different configuration, contact VersaBuilt for further instructions.

Loosen screws attaching the 2 terminals with existing wire connections. Add 2 wires from Chuck Open/Close cable and screw into terminals. **Polarity does not matter.** Reattach foot pedal cover.

Connect Chuck Open/Close to VSC:

- Chuck 1 to A1
- Chuck 2 to A2 *if CNC includes sub-spindle



Install CNC Files and Edit Table Load and Table Wash Programs

Section 4

Install G-code 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 G-code 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	

Edit CNC Table Load Program (Mill Only)

- 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 G-code 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 (Mill Only)

- Edit the 8001 CNC Vise Wash Program
- Note that the program is generic G-code 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  
%
```

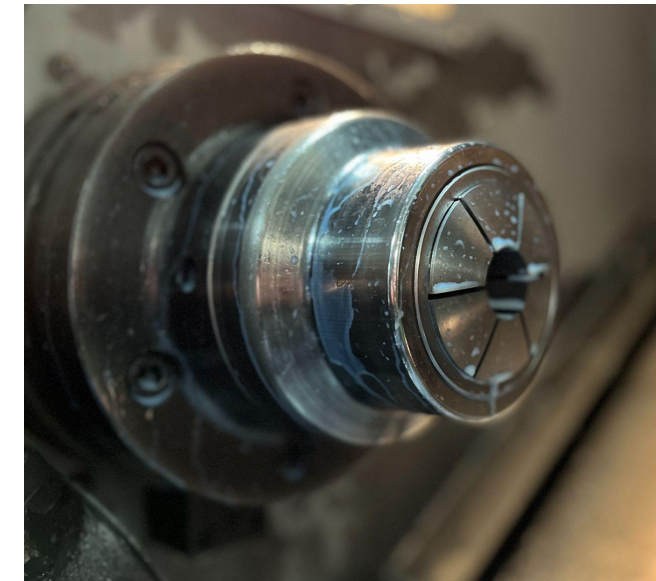
**CNC Wash Program
cleans vises and jaws of
debris/chips between
operations**

Create Chuck Wash Program (Lathe Only)

- Create the 8001 CNC Chuck Wash Program
- Select a specific tool to be called when the CNC Chuck Wash Program runs, should be a short tool that won't collide with parts in the chuck with good coolant flow; may need to use two tools, one with coolant flow to the centerline of the chuck and one with flow to the side of the chuck
- Test the CNC Chuck Wash Program to ensure all chips will be adequately flushed from the chuck and part
- If all chips can't be removed with the part in place, there is an option in the part configuration to run the wash program by itself after the part is unloaded

```
%
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 Chuck Wash
Program cleans chuck
and part of debris/chips
between operations**



Configure CNC and VSC Settings

Section 5

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

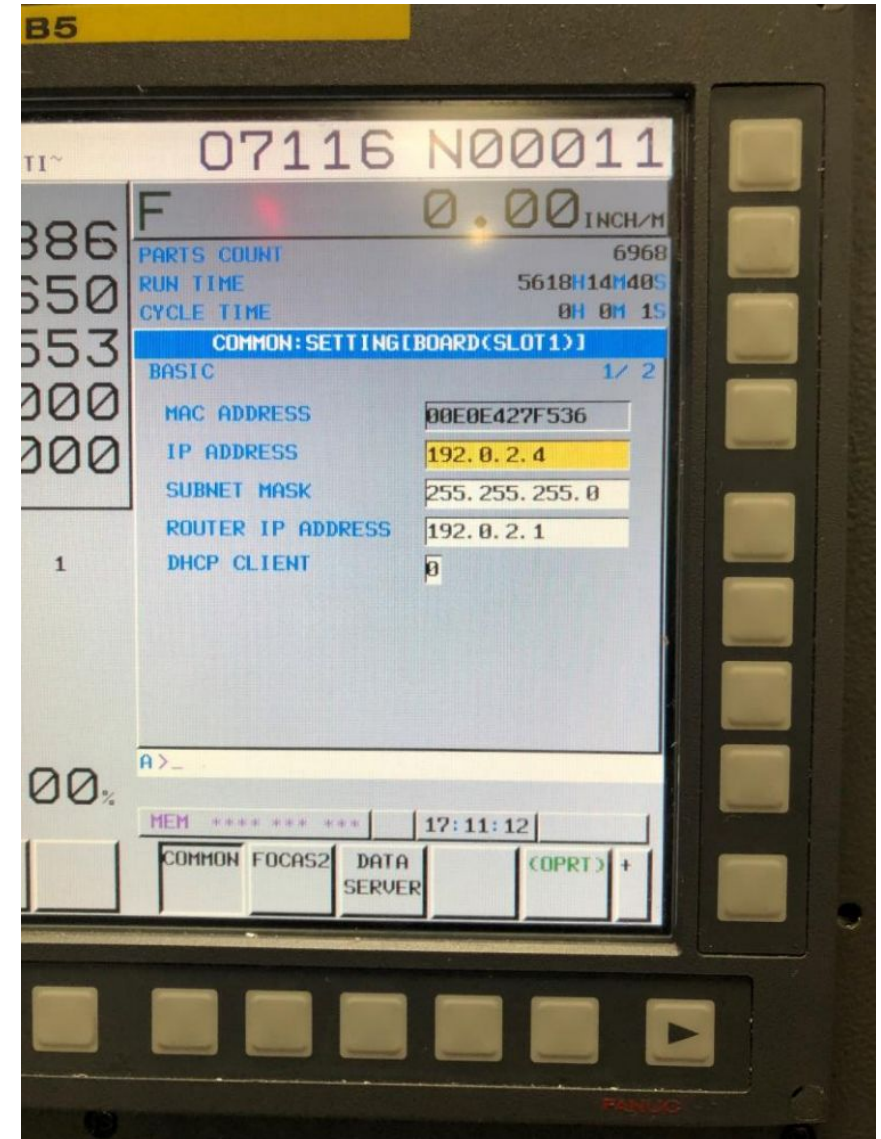
GENERAL		PROGRAM	I/O	CONTROL PANEL	SYSTEM	MAINTENANCE	POWER SETTINGS
RS-232 PORTS							
11	MAX FEED (INCH/MIN)						115200
12	MAX FEED (DEG/MIN)						NONE
13	FEED ACCEL ROUGH						1
14	FEED T CONST ROUGH						XON/XOFF
37	FEED DELTA V ROUGH						8
24	FEED ACCEL MEDIUM						NONE
25	FEED T CONST MEDIUM						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							

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

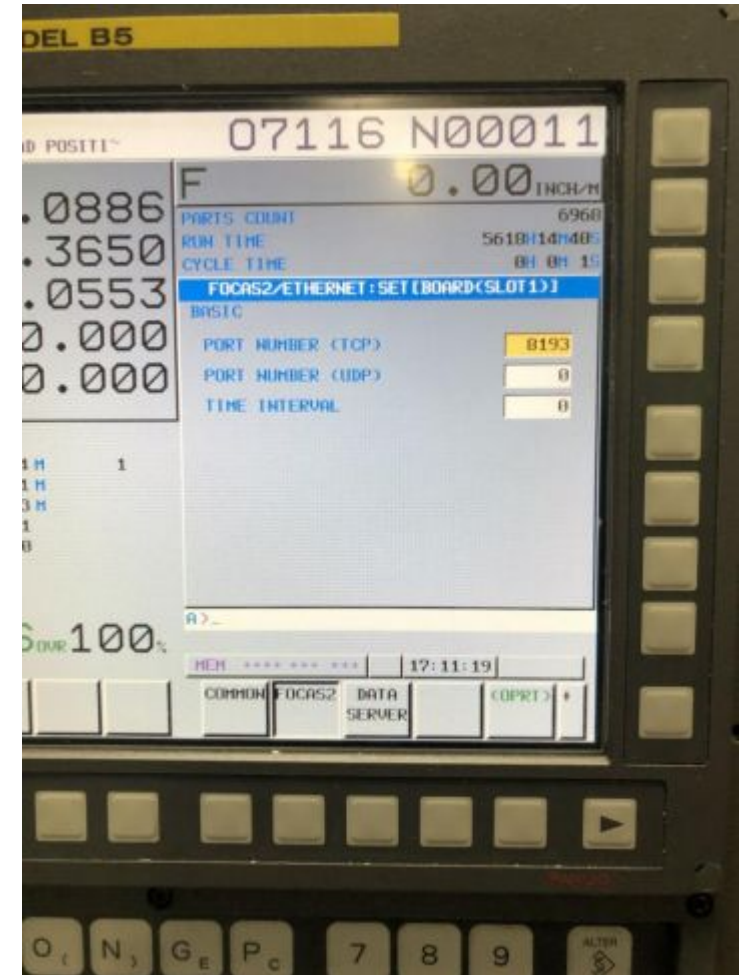
Set the following CNC Network settings:

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



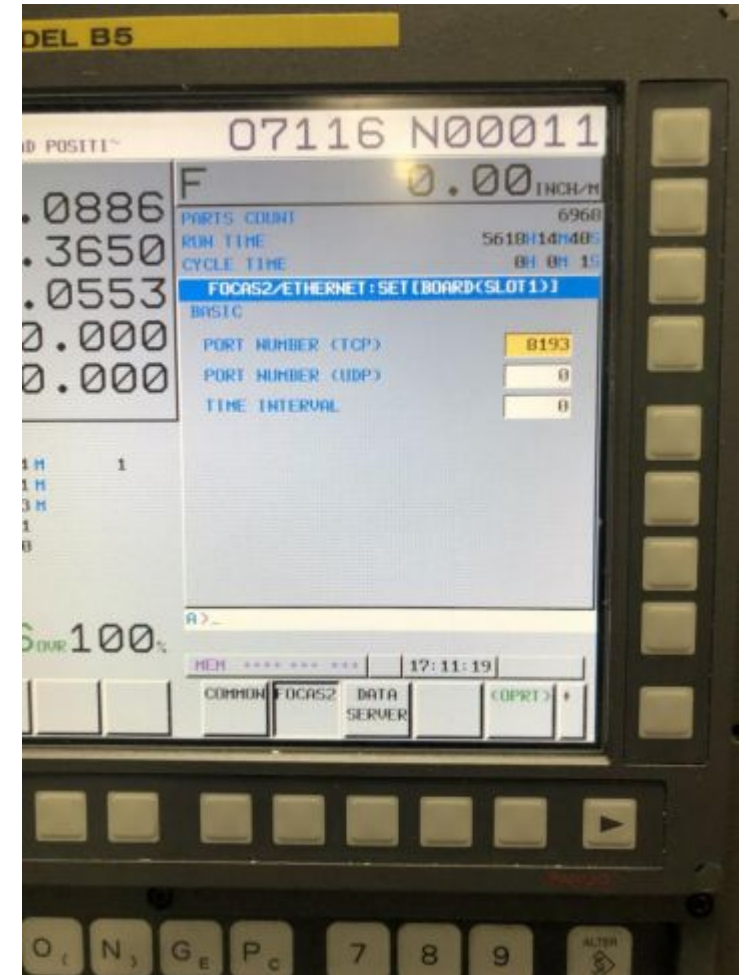
Fanuc CNC Settings

- 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 Macro CNC Drivers Features and Programming

Section 6

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
- 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 G-code 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 G-code 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 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 G-code 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  
%
```


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 (Mill Only)

The VSC Macro CNC Driver allows each of the MutliGrip 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 7

VSC Generic CNC Driver Overview - Lathes

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

- Run the G-code program currently selected in memory
- 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 be able to provide a programmable signal to the VSC after each the turning operation is complete. This is usually accomplished with a user m-code output.

If the part configuration requires a wash program to be run after the part is unloaded, the turning program must set the end of cycle signal and then an M0; after the M0, the wash program can be inserted or called as a sub program. After the wash program, the CNC must set the end of cycle signal followed by an M30. When the CNC stops at the first M0 after the CNC program is complete, the system will open the door, remove the part, close the door and then cycle start the CNC. When the wash program completes, the CNC will have set the end of cycle signal and ended the program with an M30, resetting the program to the beginning. The Lathe Automation System will load in the next part and continue processing.

VSC Generic CNC Driver Overview - Mills

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

- Run the G-code 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 be able to provide a programmable signal to the VSC after each the milling operation is complete. This is usually accomplished with a user m-code output.

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.

VSC Generic CNC Driver Programming Requirements - Mills

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 and can be called at the end of each milling program. For CNCs that do not support subprograms, the Wash and Table Load G-code must be copied and pasted into the end of each milling program.

Edit Mcodes for CNC Configuration

Sample G-code for use with Generic CNC Driver is included on the VersaBuilt USB in the Generic-Macro directory. This G-code 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.

VSC Generic CNC Driver Empty Jaw Blowoff - Mills

Empty Jaw Blowoff Configuration Options Requires Unique G-code Program Flow

If Empty Jaw Blowoff is selected in the part configuration for either one op or two op generic driver parts then the G-code program flow listed in the generic one op and two op processes must be adhered to and the Begin Optional Table Wash Section must be included in the G-code. This optional section is at the end of the one and two op generic processes and the system will not run if Empty Jaw Blowoff is selected in the part configuration and this section of G-code hasn't been edited. The typical G-code naming convention from VersaBuilt for the Table Wash program is 8001, which can be called in a subprogram and would include the table load position after the wash part of the program.

VSC Generic CNC Driver Workflow Recommendation

Save a Unique G-code Program for Each Configured Part

VersaBuilt recommends that the machinist save a unique G-code program for each part configured in the system. The G-code program should be tested to ensure that it works properly with the part configuration in the VSC. The setup sheet for the part should reflect the name of the part configuration in the VSC and the program that must be selected on CNC.

Generic G-code Program Editing - CNC-Specific Programs

```
1  %
2  09000  (Dispatcher Program - Two Op)
3
4  N10
5  (Enter CNC specific code to enable look-ahead)
6  (M98 PXXXX call Op1 as a subprogram or paste in op1 gcode)
7  (Enter CNC specific code to disable look-ahead)
8  (Set Handshake using M code controlable relay)
9  M0 (Wait for robot to complete transfer)
10 (Clear handshake using M code controlable relay)
11 (Enter CNC specific code to enable look-ahead)
12 (M98 PXXXX call Op2 as a subprogram or paste in op2 gcode)
13 (Enter CNC specific code to disable look-ahead)
14 (Set Handshake using M code controlable relay)
15 M0 (Wait for robot to complete transfer)
16 (Clear handshake using M code controlable relay)
17 (Enter CNC specific code to enable look-ahead)
18
19 (** BEGIN OPTIONAL TABLE WASH SECTION ONLY TO BE INCLUDED IF PART CO
20 (Enter CNC specific code to enable look-ahead)
21 (M98 P8001 call Table Wash program ONLY if part configuration uses O
22 (Enter CNC specific code to disable look-ahead)
23 (Set Handshake using M code controlable relay)
24 M0 (Wait for robot to complete transfer)
25 (Clear handshake using M code controlable relay)
26 (Enter CNC specific code to enable look-ahead)
27 (** END OPTIONAL TABLE WASH SECTION **)
28
29 GOTO 10
30 M30
31 %
```

- Edit the one op and two op sample programs to make versions that are specific to your CNC, you will save these programs as your default programs to edit for each part
- The following CNC specific codes needs to be added below the comment of where it is needed:
 - CNC specific code to enable look-ahead
 - CNC specific code to disable look-ahead
 - M-code to set CNC cycle end hand-shake signal
 - M-code to clear the CNC cycle end hand-shake signal
- Add all CNC specific codes under the comments where they are called for and save the program
- Save the updated CNC-specific programs in a location you will remember on your PC for later use

Generic G-code Program Editing - New Part Programs

```
1  %
2  09000  (Dispatcher Program - Two Op)
3
4  N10
5  (Enter CNC specific code to enable look-ahead)
6  (M98 PXXXX call Op1 as a subprogram or paste in op1 gcode)
7  (Enter CNC specific code to disable look-ahead)
8  (Set Handshake using M code controlable relay)
9  M0 (Wait for robot to complete transfer)
10 (Clear handshake using M code controlable relay)
11 (Enter CNC specific code to enable look-ahead)
12 (M98 PXXXX call Op2 as a subprogram or paste in op2 gcode)
13 (Enter CNC specific code to disable look-ahead)
14 (Set Handshake using M code controlable relay)
15 M0 (Wait for robot to complete transfer)
16 (Clear handshake using M code controlable relay)
17 (Enter CNC specific code to enable look-ahead)
18
19 (** BEGIN OPTIONAL TABLE WASH SECTION ONLY TO BE INCLUDED IF PART CO
20 (Enter CNC specific code to enable look-ahead)
21 (M98 P8001 call Table Wash program ONLY if part configuration uses O
22 (Enter CNC specific code to disable look-ahead)
23 (Set Handshake using M code controlable relay)
24 M0 (Wait for robot to complete transfer)
25 (Clear handshake using M code controlable relay)
26 (Enter CNC specific code to enable look-ahead)
27 (** END OPTIONAL TABLE WASH SECTION **)
28
29 GOTO 10
30 M30
31 %
```

- Start with a one op or two op CNC-specific generic G-code program created in the previous step
- Under each comment requesting a subprogram call or to paste in code for Op1, Op2 or Table Wash program, enter the required subroutine call or cut and paste in the G-code to accomplish the required task
- The program can be tested manually; at each M0 stop, perform the task the robot would perform. For a two op program, the part will need to be moved from the Op1 vise to the Op2 vise
- Once tested, reset the CNC to end the program cycle
- After testing and validating the program, save the program and reference the program in the part setup sheets

Appendix Supported Fanuc Controls

Robot2CNC supports Fanuc CNC controls that support the Focas 2 protocol. The following controls are supported:

- 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

VERSABUILT ROBOTICS