

VERSABUILT ROBOTICS



VSC/CNC Communication, G-Code and Programming Manual

Software Version 1.2.6 Copyright VersaBuilt 2025
Updated 2025-12-10

Table of Contents

<u>Section 1</u>	<u>Safety Warnings</u>
<u>Section 2</u>	<u>CNC File Installation and Editing</u>
<u>Section 3</u>	<u>Configure Haas/Fanuc CNC Settings</u>
<u>Section 4</u>	<u>VSC Macro Driver - Haas & Fanuc Focus 2.0 CNC Controller Communication</u>
<u>Section 5</u>	<u>VSC Standard Driver - Non-Haas & Non-Fanuc Handshake Communication</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 Automation System Safety Manual before proceeding

CNC File Installation and Editing

Section 2

Install G-code Files on CNC

- Locate VersaBuilt CNC Files
 - USB Drive provided with each Automation System or Download from the VersaBuilt Resources webpage (<https://www.versabuilt.com/resources/>)
- Load the files 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
 - 8000 - Table Load Program (used with Mill only)
 - 8001 - Table Wash Program (used with Mill only)
 - 9000 - Dispatcher Program *in CNC memory during automation
 - 9001 - Subprogram of Dispatcher
 - 9002 - Subprogram of Dispatcher
 - 9003 - Subprogram of Dispatcher
 - 9004 - Subprogram of Dispatcher
 - 9123 - Subprogram of Dispatcher

SOFTWARE DOWNLOADS

Robot2CNC Files

Mill Automation System Files

Lathe Automation System Files

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

Notes:

1. Edit 8000 & 8001 programs *see following pages
2. Do not edit 9000 programs
3. Program numbers can be revised, if needed

Edit CNC Table Load Program (Mill Only)

- Edit the 8000 CNC Table Load Program to reference the Table Load position found in In the Installation Manual and Calibration Manual
- 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  
%
```

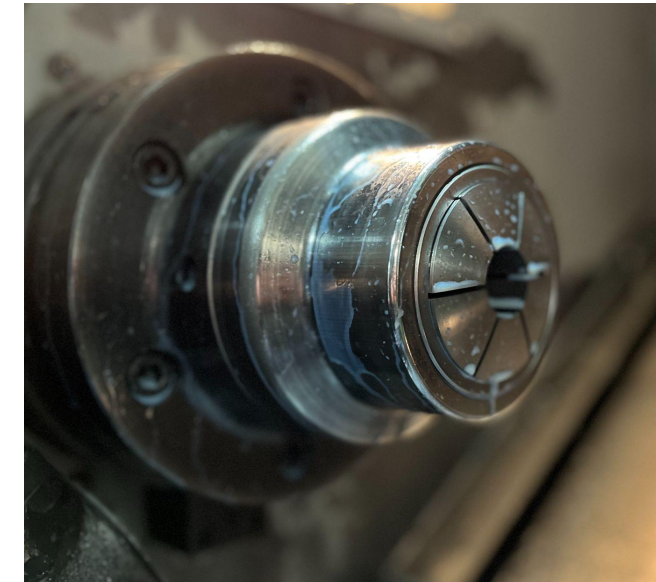
**CNC Wash Program
cleans vises and jaws of
debris/chips between
operations**

Create Chuck Wash Program (Lathe Only)

- Create the 8001 CNC Chuck Wash Program or end all programs with a wash routine
- 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 Haas/Fanuc CNC Settings

Section 3

CNC Communication

This and the next section include details for CNC settings and G-Code program requirements for communication between the automation system and the CNC. The following controls have the option for “VSC Macro CNC Driver” communication, with program selection, cycle start, G-code vise control, etc.

- **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 Focas 2 protocol 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

For all other CNC controllers, refer to the section titled “VSC Standard CNC Driver Features and Programming” for setting up a 2-way handshake communication with Cycle Start and Cycle End.

Haas NGC CNC 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	Alias Codes	
Wired Connection		Wireless Connection		Net Share	Haas Connect
Wired Network Information					
Host Name	HaasCNC1234567		DHCP Server	*	
Domain			IP Address	192.168.2.3	
DNS Server			Subnet Mask	255.255.255.0	
Mac Address	00:C0:08:88:47:7A		Gateway	*	
DHCP Enabled	OFF		Status	UP	

Haas Legacy CNC 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													

Haas Legacy CNC Settings

Parameter: Common

22 Enable Macro: 1

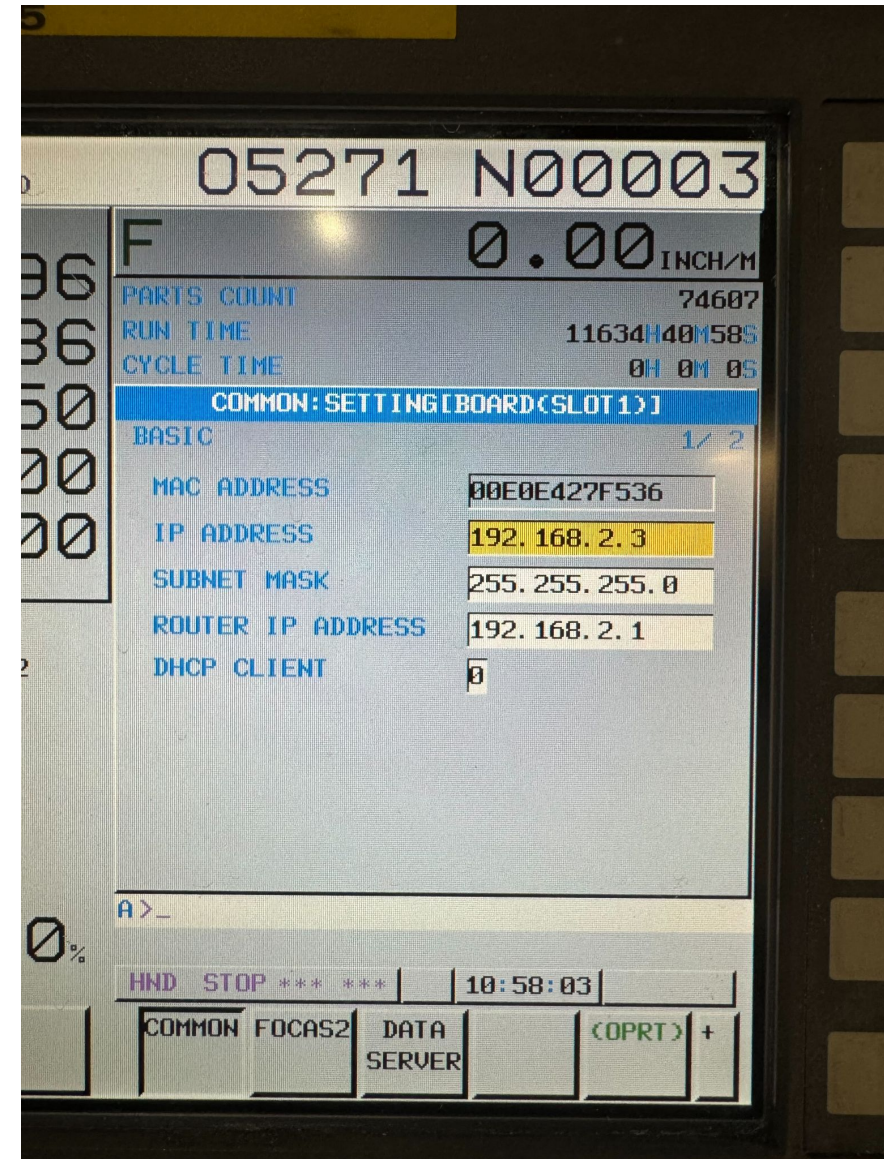
COMMON	SOFTWARE	INPUT INVERT	SYSTEM	AXIS	COMPENSATION
COMMON SW 1 (PARAMETER 57:22) 1063037585					
SPIND. TEMP MONITOR					0
ENABLE ROT & SCALNG					0
ENABLE DNC					1
UNUSED					1
ENABLE GROUND FAULT					1
M19 SPINDLE ORIENT.					0
ENABLE MACRO					1
INVERT SKIP					0
HANDLE CURSOR					1
NEGATIVE WRK OFFSET					1
TRANS OIL LOW PRESS					1
QUICK CODE					1
OILER ON/OFF					1
INV BUSS PWR FAULT					1
SPINDLE ENCODER #2					0
UNUSED					0
Parameter 22 ENABLE MACRO - Enables macro functions.					

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

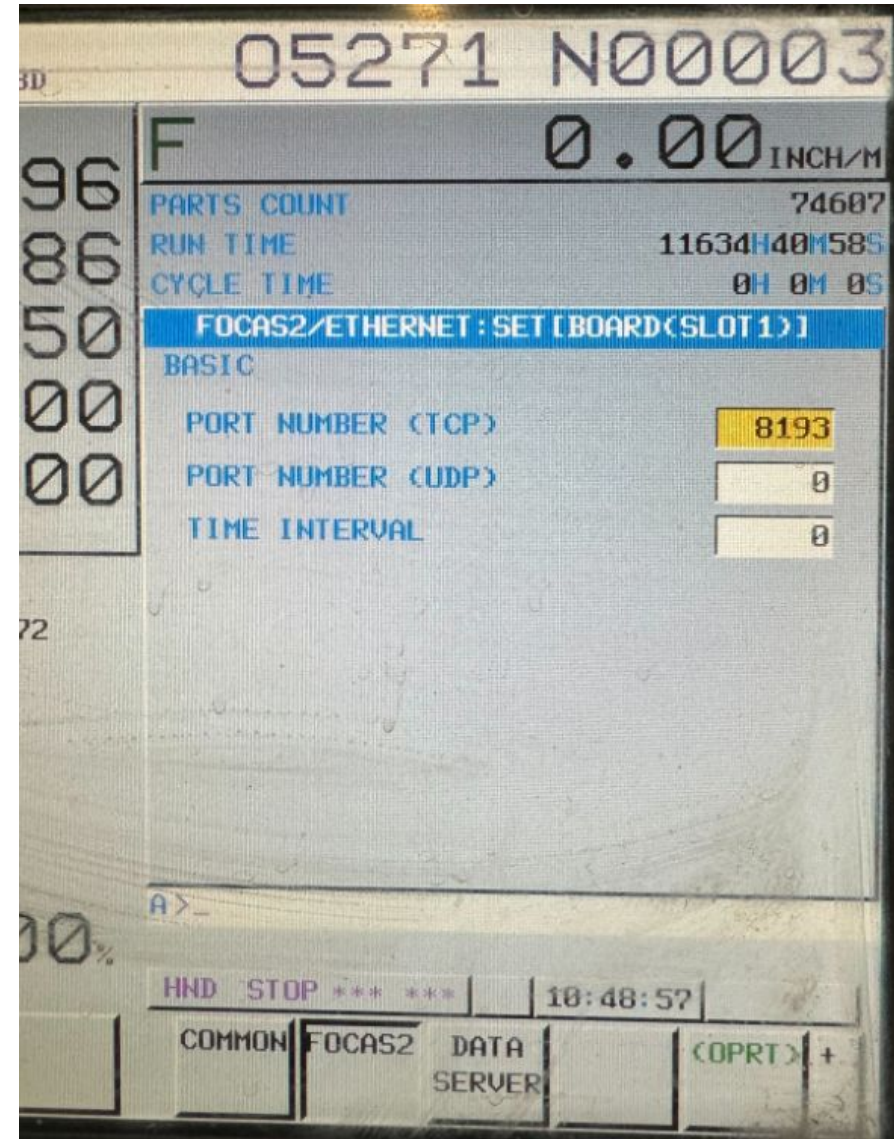
Set the following CNC Network settings:

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:
 - Door close on Cycle Start
 - Door Open on Cycle End

VSC Macro Driver (Haas/Fanuc Controls) Features and Programming

Section 4

CNC Communication Driver Overview

The VersaBuilt System Controller (VSC) includes two types of drivers to control a CNC during automation processing:

- Standard Driver (Handshake Communication)
- VSC Drivers (Haas/Fanuc Control Communication)

The Standard CNC Driver features and programming is documented in the next section.

The VSC 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 Focas 2 protocol 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

VSC Macro CNC Driver Supported CNC Controls

The following programs are included with the Automation System on USB or can be found on the VersaBuilt website, resources page:
<https://www.versabuilt.com/resources/>

Load programs in CNC Control **programs can be renamed if there are numbering conflicts*

- 8000 - Table Load Program (position for robot load and unload) **provided program is sample*
- 8001 - Wash Program (cleaning chips and debris from vises and jaws between operations) **provided program is sample*
- 9000 - Dispatcher Program (loaded in memory at start of automation)
- 9001 - Subprogram of Dispatcher
- 9002 - Subprogram of Dispatcher
- 9003 - Subprogram of Dispatcher
- 9004 - Subprogram of Dispatcher
- 9123 - Subprogram of Dispatcher

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 Programming Requirements

Table Load and Wash Programs

A Table Load and Wash Program must be present in the CNC control. By default the programs are:

- Table Load program = 8000
- Wash Program = 8001

Sample programs are provided and are available at <https://www.versabuilt.com/resources/> and 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  
%
```

G-Code Vise Control (Mill Only)

The VSC Macro CNC Driver allows each of the MultiGrip 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
 - C06. exhausts pressure from all vises*
 - C07. pressurizes all vises*
- 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.
- *Exhaust Vises: G65 P9002 D01.C06. (Device number doesn't matter, all vises exhaust)
- *Pressurize Vises: G65 P9002 D01.C07. (Device number doesn't matter, all vises pressurized)

Z-push with CNC spindle*

Use exhaust command before spindle push down with z-pushdown tool. After the vise pressure is exhausted, push part down with spindle using the z-pushdown tool. Use pressurized command to pressurize the vises and resume machining operation.

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

VSC Standard CNC Driver (Handshake) Features and Programming

Section 5

VSC Standard CNC Driver Overview - Lathes

The VSC Standard 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 Standard CNC Driver Overview - Mills

The VSC Standard 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 Standard 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 M-codes for CNC Configuration

Sample G-code for use with Standard 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 Standard 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 Standard 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.

VSC Standard CNC Driver

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 0
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

VSC Standard CNC Driver

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 controllable relay)
9  M0 (Wait for robot to complete transfer)
10 (Clear handshake using M code controllable 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 controllable relay)
15 M0 (Wait for robot to complete transfer)
16 (Clear handshake using M code controllable relay)
17 (Enter CNC specific code to enable look-ahead)
18
19 (***) BEGIN OPTIONAL TABLE WASH SECTION ONLY TO BE INCLUDED IF PART C
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 controllable relay)
24 M0 (Wait for robot to complete transfer)
25 (Clear handshake using M code controllable 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

VERSABUILT ROBOTICS