

Pocket NC V2 User's Manual

November, 2021

English

Original Instructions



© 2019 Pocket NC Company

All rights reserved. No part of this publication may be reproduced, stored in a retrieval system, or transmitted, in any form, or by any means, mechanical, electronic, photocopying, recording, or otherwise, without the written permission of Pocket NC Company. No patent liability is assumed with respect to the use of the information contained herein. Moreover, because Pocket NC Company strives constantly to improve its high-quality products, the information contained in this manual is subject to change without notice. We have taken every precaution in the preparation of this manual; nevertheless, Pocket NC Company assumes no responsibility for errors or omissions, and we assume no liability for damages resulting from the use of the information contained in this publication.







Revision History

First Release: November 2021 (QR)

Limited Warranty Certificate

Please read the following carefully. It affects your rights as a purchaser of this product.

Pocket NC Company's warranty obligations and liability for all hardware products sold by Pocket NC Company in the fifty United States, the District of Columbia, and U.S. Territories are limited exclusively to the terms set forth below (this "Warranty").

Pocket NC Company warrants our hardware products against defects in materials and workmanship under normal use for a period of one year from the date of delivery to the original purchaser (the "Warranty Period").

If a hardware defect arises, and a valid claim is received within the Warranty Period, at our sole option and to the extent permitted by law, POCKET NC COMPANY will either: (1) Repair the hardware defect at no charge, using new or refurbished replacement parts; (2) Exchange the product with a product that is new or reconditioned by POCKET NC COMPANY or that has been manufactured from new or serviceable used parts and is functionally equivalent to the original product; (3) Refund the purchase price of the product.

A repaired or replacement product assumes the remaining warranty of the original product or 6-months (180 days) from the date of replacement or repair, whichever is longer.

When a product is exchanged, any replacement item becomes your property and the replaced item becomes POCKET NC COMPANY property. In order to receive a refund, the product for which the refund is provided must be returned to POCKET NC COMPANY and becomes the property of POCKET NC COMPANY.

Exclusions from and Limitations to Warranty

This Warranty applies only to hardware products manufactured by or for POCKET NC COMPANY. The Warranty does not apply to any non-POCKET NC COMPANY hardware products or any software, even if packaged or sold with POCKET NC COMPANY hardware. Manufacturers, suppliers, or publishers other than POCKET NC COMPANY may provide their own warranties to the end user purchaser. Software distributed by POCKET NC COMPANY with or without the POCKET NC COMPANY brand name (including, but not limited to system software) is not covered under this Limited Warranty. Refer to the Terms and Conditions for the software for details of your rights with respect to its use.



POCKET NC COMPANY does not warrant that the operation of the product will be uninterrupted or error-free. POCKET NC COMPANY is not responsible for personal injury or property damage arising or resulting from failure to follow instructions relating to the product's use, storage, or handling.

This Warranty does not apply to: (1) Damage caused by accident, abuse, misuse, neglect, flood, fire, earthquake, or other external causes; (2) Damage caused by operating the product outside the permitted or intended uses described by POCKET NC COMPANY; (3) Damage to a product or part caused by or resulting from a modification to alter its functionality or capability without the written permission of POCKET NC COMPANY; (4) Cosmetic damage, including but not limited to scratches, dents, and ordinary wear and tear.

This Warranty gives you specific legal rights, and you may also have other rights which vary from state to state. POCKET NC COMPANY's responsibility for hardware defects is limited to the repair or replacement service, or to refund, as described in this Warranty and determined by POCKET NC COMPANY in its sole discretion.

Disclaimer of Warranties; limitation of liability

Except as explicitly provided in this Warranty, we do not guarantee, represent, or warrant that your use of our service or any POCKET NC COMPANY product will be uninterrupted, timely, secure, or error-free. We do not warrant that the results that may be obtained from the use of the service or the product will be accurate or reliable.

The products delivered to you by POCKET NC COMPANY are (except to the extent otherwise expressly stated by us in this Warranty) provided "AS IS" and "AS AVAILABLE" for your use, and, except as expressly set forth in this Warranty, without any representation, warranties, or conditions of any kind, either express or implied. POCKET NC COMPANY further disclaims all implied warranties or conditions of merchantability, merchantable quality, fitness for a particular purpose, durability, title, and non-infringement.

Some states restrict or do not allow limitations on warranties, which may affect your rights, and this Limited Warranty will be adjusted to maintain compliance with such state limitations or restrictions.

No POCKET NC COMPANY reseller, agent, or employee is authorized to make any modification, extension, or addition to this Warranty and the terms in this Warranty shall constitute the entire agreement between you and POCKET NC COMPANY concerning the matters discussed herein. If any term is held to be illegal or unenforceable by a court of competent jurisdiction, the legality or enforceability of the remaining terms shall not be affected or impaired and shall remain in full force and effect.

Except as provided in this Warranty and to the extent permitted by law, POCKET NC COMPANY is not responsible for direct, indirect, special, incidental, or consequential damages resulting from any breach of warranty or condition or any other reason, or under any other legal theory,



including but not limited to loss of use; loss of revenue; loss of actual or anticipated profits (including loss of profits on contracts); loss of the use of money; loss of anticipated savings; loss of business; loss of opportunity; loss of goodwill; loss of reputation; loss of, damage to, or corruption of data; or any indirect or consequential loss or damage howsoever caused, including the replacement of equipment and property, any costs of recovering, programming, or reproducing any program or data stored or used with POCKET NC COMPANY products and any failure to maintain the confidentiality of data stored on the product. This is true even if POCKET NC COMPANY is advised of the possibility of such damages.

POCKET NC COMPANY's liability for actual damage from any cause whatsoever will be limited to the greater of 1) \$2,000 or 2) the amount you paid for the product that caused the damage. This limitation of liability will not apply to claims for bodily injury, or damage to real property or tangible personal property for which POCKET NC COMPANY is legally liable. In no event will either party be liable for any damage caused by the other party's failure to fulfill its responsibilities under this Limited Warranty.

Some states do not allow for the foregoing damages cap and exclusion or limitation of incidental or consequential damages, so the above limitations may not apply to you.

Obtaining Warranty Service

Please access and review the online help resources referred to in the documentation accompanying this hardware product before requesting warranty service. If the product is still not functioning properly after making use of these resources, please contact POCKET NC COMPANY at info@pocketnc.com.

Help us help you. You must provide reasonable assistance to POCKET NC COMPANY in diagnosing issues with your product and follow POCKET NC COMPANY's warranty processes.

POCKET NC COMPANY restricts service to the United States of America and the District of Columbia and U.S. territories. For qualifying warranty service, POCKET NC COMPANY will send a prepaid shipping label to enable you to ship the product in its original packaging to POCKET NC COMPANY's repair service location for service. If the original packaging is not kept, you will be charged for a replacement POCKET NC COMPANY product packaging box to be sent to you to enable you to ship the product to POCKET NC COMPANY's repair service location. Upon your receipt of the replacement product, the original product becomes the property of POCKET NC COMPANY and you agree to follow instructions, including, if required, arranging the return of original product to POCKET NC COMPANY in a timely manner.

Service options, parts availability, and response times may vary. In accordance with applicable law, POCKET NC COMPANY may require that you furnish proof of purchase details and/or comply with registration requirements before receiving warranty service. Please refer to the according documentation for more details on this and other matters on obtaining warranty service.



POCKET NC COMPANY will maintain & use customer information in accordance with the POCKET NC COMPANY Privacy Policy. The contents of your product will be deleted and the storage media reformatted in the course of warranty service. Your product will be returned to you configured as originally purchased, subject to applicable updates.



Customer Feedback

If you have concerns or questions regarding this User's Manual, please contact us on our website, www.pocketnc.com. Use the "Contact Us" link to send your comments to a customer service representative.

You also can find an electronic copy of this manual and other useful information on our website under the "Help" tab.

Join Pocket NC owners online and be a part of the greater CNC community at these sites:



Google Groups: <https://groups.google.com/forum/#!forum/pocket-nc>



Facebook: <https://www.facebook.com/pocketnc>



Twitter: <https://twitter.com/pocketnc>



Instagram: https://www.instagram.com/pocket_nc/



YouTube: <https://www.youtube.com/c/pocketnc>



Declaration of Conformity

Product: 5 Axis Desktop CNC Mills

Physical:

Pocket NC
119A Gold Miner Lane
Belgrade, MT 59714

Mailing:

Pocket NC
1051 Springbrook Avenue
Bozeman, MT 59718

We, Pocket NC Company, hereby declare that the following products,

Pocket NC V2-10 and V2-50, 5 axis desktop CNC mills

fulfill all of the relevant requirements of

- EC Machinery Directive 2006/42/EC
- EC EMF Directive 2014/30/EU
- EC Low Voltage Directive 2006/95/EC



How to Use this Manual

To get the maximum benefit of your new Pocket NC machine, read this manual thoroughly and refer to it often. The content of this manual is also available online at www.pocketnc.com.

IMPORTANT: Before you operate the machine, read and understand the Operator's Manual Safety chapter.

Declaration of Warnings

Throughout this manual, important statements are set off from the main text with an icon and an associated signal word: "Danger," "Warning," "Caution," or "Note." The icon and signal word indicate the severity of the condition or situation. Be sure to read these statements and take special care to follow the instructions.

Description	Example
Danger means that there is a condition or situation that will cause death or severe injury if you do not follow the instructions given.	 DANGER: Risk of electrocution. Do not open cabinet.
Warning means that there is a condition or situation that will cause moderate injury if you do not follow the instructions given.	 WARNING: Never put your hands between the tool and the workpiece while in operation.
Caution means that minor injury or machine damage could occur if you do not follow the instructions given. You may also have to start a procedure over if you do not follow the instructions in a caution statement.	 CAUTION: Power down the machine before you do maintenance tasks.
Note means that the text gives additional information, clarification, or helpful hints.	 NOTE: Follow these guidelines to set the tool length offset.



Text Conventions Used in this Manual

Description	Text Example
Code Block text gives program examples.	G00 G90 G54 X0. Y0.;
A Control Button Reference gives the name of a control key or button that you are to press.	Press [CYCLE START] .
A File Path describes a sequence of file system directories.	<i>Service > Documents and Software >...</i>
A Mode Reference describes a machine mode.	MDI
System Output describes text that the machine control displays in response to your actions.	PROGRAM END
User Input describes text that you should enter into the machine control.	G04 P1.;



Table of Contents

Revision History	4
Limited Warranty Certificate	4
Exclusions from and Limitations to Warranty	4
Disclaimer of Warranties; limitation of liability	5
Obtaining Warranty Service	6
Customer Feedback	8
Declaration of Conformity	9
How to Use this Manual	10
Declaration of Warnings	10
Text Conventions Used in this Manual	11
Chapter 1: Safety	17
Chapter 1.1: Introduction	17
Chapter 1.1.1: Read Before Operating	17
Chapter 1.2: Unattended Operation	19
Chapter 1.3: Modifications to the Machine	19
Chapter 1.4: Safety Decals	19
Chapter 1.5: Mill Warning Decal	20
Chapter 1.6: More Information Online	20
Chapter 2: Startup, Shutdown and User Interface	22
Chapter 2.1: Mill Orientation	22
Chapter 2.2: Software Setup	23
Chapter 2.3: Startup and Shutdown	23
Chapter 2.3.1: Startup	23
Chapter 2.3.2: Shutdown	26



Chapter 2.4: User Interface Overview	26
Chapter 2.4.1: Status Bar	27
Chapter 2.4.2: Setup Page	29
Work Offsets Section	29
Jog Section	33
Tool Offsets Section	36
Chapter 2.4.3: Manual Page	38
Spindle Section	39
Feed Section	42
Jog Section	44
Control Section	44
MDI Section	45
Chapter 2.4.4: Production Page	47
Spindle Section	48
Feed Section	48
DRO Section	48
Control Section	50
G-Code Section	53
Chapter 2.4.5: History Page	56
Message Log Section	57
Chapter 2.4.6: Config Page	57
Client Tab	58
Keybindings Section	58
Units, Audio and Language Sections	60
Server Tab	61
Software Section	61
Machine Config Section	62
Chapter 3: Machine Operation	63



Chapter 3.1: File Management	63
Chapter 3.1.1: Loading Files	63
Chapter 3.1.2: Running Files	65
Chapter 3.1.3: Deleting Files	67
Chapter 3.2: V2-10 and V2-50 Tooling	68
Chapter 3.2.1: Tool Holders (V2-10 Only)	68
Chapter 3.2.2: Tool Holder Care (V2-10 Only)	69
Chapter 3.2.3: Tool and Tool Holder Set Up	69
V2-10	69
V2-50 CHB	70
V2-50 CHK	71
Chapter 3.2.4: Tool Measurement	72
With Tool Probe	73
By Hand	75
Chapter 3.3: Feeds and Speeds	78
Chapter 3.4: Fixturing	82
Chapter 3.4.1: Pocket NC Vise	82
Operation	82
Installation	84
Chapter 3.4.2: ER-40 Collet Fixture	87
Installation	89
Chapter 4: G and M Codes	90
Chapter 5: Maintenance, Storage, and Transport	92
Chapter 5.1 Maintenance	92
Chapter 5.2: Storage	93
Chapter 5.3: Transport	93
Chapter 6: Troubleshooting Common Problems	94
Chapter 6.1: Common Problems	95



Chapter 6.2: Machining/Performance Issues

98

Chapter 6.3: Error Codes

100





Chapter 1: Safety

Chapter 1.1: Introduction



CAUTION: Only authorized and trained personnel may operate this equipment. You must always act in accordance with the Operator's manual, safety decals, safety procedures and instructions for safe machine operation. Untrained personnel present a hazard to themselves and the machine.

IMPORTANT: Read and understand all warnings, cautions, and instructions before operating this machine. All milling machines present hazards from rotating cutting tools, belts and pulleys, high voltage electricity, and noise. When you use CNC machines and their components, you must always follow basic safety precautions to reduce the risk of personal injury and mechanical damage.

Chapter 1.1.1: Read Before Operating



DANGER: Do not enter the machining area any time the machine is in motion.

Severe injury or death may result.

Basic safety:

- Consult your local safety codes and regulations before operating the machine. Contact Pocket NC any time safety issues need to be addressed.
- It is the machine owner's responsibility to make sure that everyone who is involved in installing and operating the machine is thoroughly acquainted with the operation and safety instructions provided with the machine **BEFORE** they perform any actual work. The ultimate responsibility for safety rests with the machine owner and the individuals who work with the machine.
- Use appropriate eye protection while operating the machine. ANSI-approved impact safety goggles are recommended to reduce the risks of eye damage.
- This machine is automatically controlled and may start at any time.
- This machine can cause severe bodily injury.



- As sold, your machine is not equipped to process toxic or flammable material; this can create deadly fumes or suspended particles in the air. Consult with the material manufacturer for safe handling of material by-products, and implement all precautions before you work with such materials.
- When moving the machine, always lift by the base and the L bracket. Never lift by movable components.

Electrical safety:

- The electrical power must meet the required specifications. Attempting to run the machine from any other source can cause severe damage and will void the warranty.
- The electrical panel covers should be in place and fastened to the machine at all times, except when the machine is being serviced. When the machine is on, there is high voltage throughout the electrical cabinet (including the circuit boards and logic circuits) and some components operate at high temperatures; therefore, extreme caution is required.
- Never service the machine with the power connected.

Operation Safety:

- Do not operate the machine unless all body parts and clothing are clear of the work area. Rotating cutting tools can cause severe injury. When a program runs, the mill table and spindle head can move rapidly at any time, and in any direction.
- [POWER] is the black switch located on the front of the machine. When you switch [POWER], the axis motors, spindle motor, and gear motors all stop. While [POWER] is in the off position, both automatic and manual motion is disabled. Use [POWER] in case of emergency.
- Check for damaged parts and tools before operating the machine. Any part or tool that is damaged should be properly repaired or replaced by authorized personnel. Do not operate the machine if any component does not appear to be functioning correctly.
- Keep your hands away from the tool in the spindle when the machine is in operation.



DANGER: Improperly clamped parts or oversized parts may be ejected with force.

Follow these guidelines while you work with the machine:



- Normal operation – Keep clear of the work area of the machine while the machine operates.
- Part loading and unloading – An operator ensures the machine is not in operation, completes a task, clears the work area of the machine, then presses [START/PAUSE] (the green button on the machine which starts automatic motion).
- Tool loading and unloading – A machinist enters the machining area to load or unload tools. Exit the area completely before automatic movement is commanded (for example, [PAUSE]).
- Maintenance / Machine Cleaner – Switch [POWER] to off position.
- Do not operate the machine with the table loaded with over 1.5 pounds of material.
- If using material larger than the work volume of the machine, ensure that the movements in the program will not cause a crash.

Chapter 1.2: Unattended Operation

The Pocket NC is not designed for unattended operation.

As it is the machine owner's responsibility to set up the machine safely and use best practice machining techniques, it is also their responsibility to manage the progress of these methods. The machining process must be monitored to prevent damage if a hazardous condition occurs.

For example, if there is the risk of fire due to the material machined, then you must install an appropriate fire suppression system to reduce the risk of harm to personnel, equipment and the building.

Chapter 1.3: Modifications to the Machine

DO NOT modify or alter this equipment in any way. Pocket NC must handle all modification requests. Modification or alteration of any Pocket NC machine without authorization could lead to personal injury and mechanical damage, and will void your warranty.

DO NOT use liquid coolant on this machine. The electronics are not waterproof and liquids are likely to create a short circuit.

Chapter 1.4: Safety Decals

To help ensure that CNC machine dangers are quickly communicated and understood, hazard symbol decals are placed on Pocket NC machines in locations where hazards exist. If decals become damaged or worn, or if additional decals are needed to emphasize a particular safety point, contact Pocket NC.



Note: Never alter or remove any safety decal or symbol.

Each hazard is defined and explained on the general safety decal, located at the side of the machine. Review and understand each safety warning and familiarize yourself with the symbols before operation.

Chapter 1.5: Mill Warning Decal

This is an example of a general mill warning decal in English.

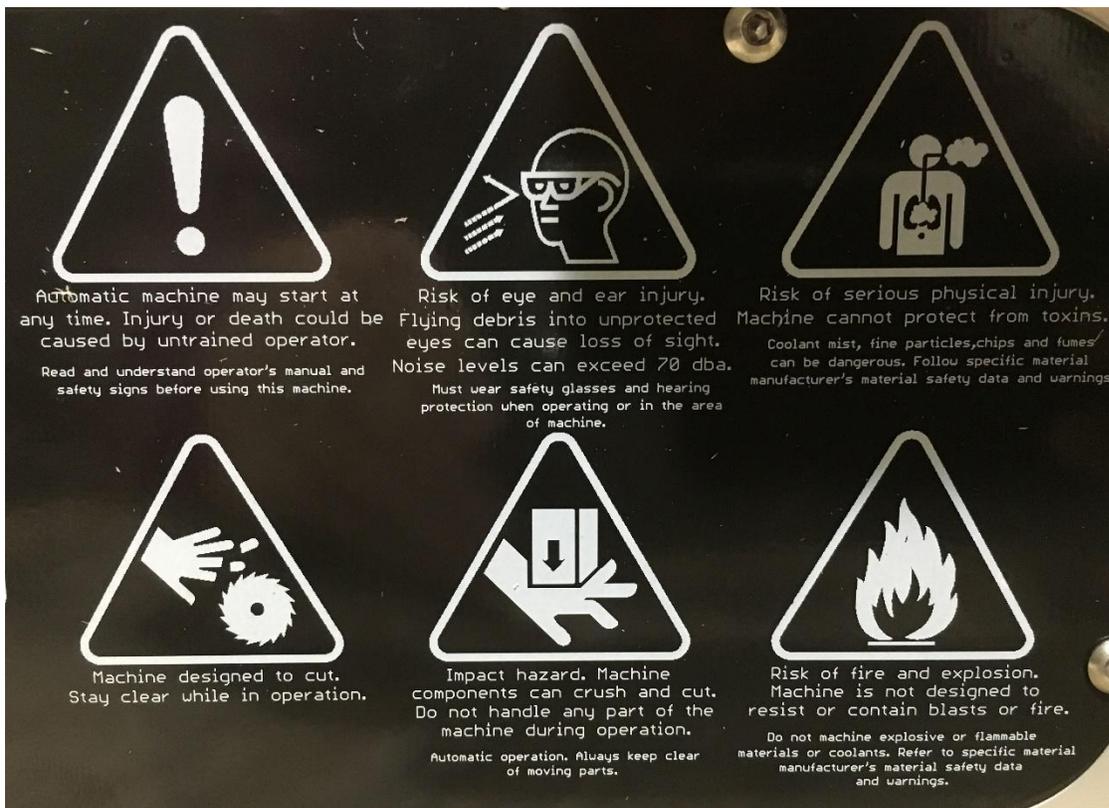


Figure 1 Mill Warning Decal Example

Chapter 1.6: More Information Online



For updated and supplemental information, including tips, tricks, maintenance procedures, and more, go to www.PocketNC.com and select the Help>Resources.

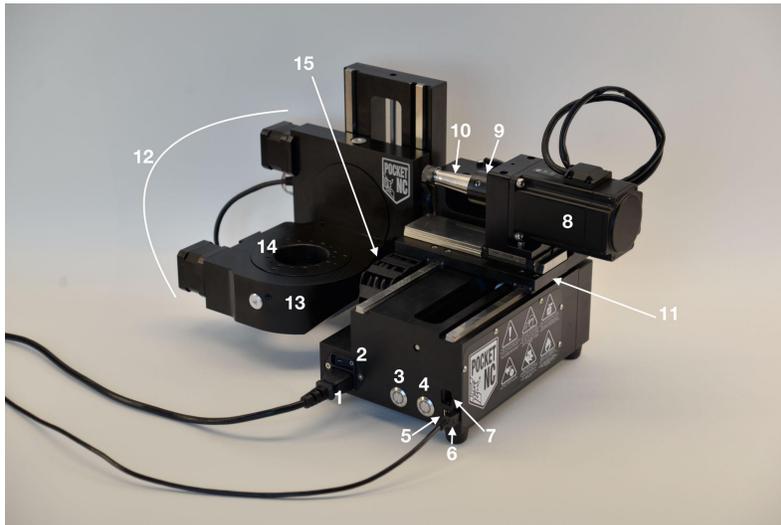


Chapter 2: Startup, Shutdown and User Interface

The purpose of this chapter is to introduce users to the Pocket NC's physical components as well as its user interface.

Chapter 2.1: Mill Orientation

The following figures show some of the standard and optional features of your Pocket NC. Note that these figures are representative only; your machine's appearance may vary depending on the model and installed options.



- | | |
|---------------------------------|---|
| 1. Power cord and plug | 8. Spindle motor |
| 2. Power button | 9. Spindle collar |
| 3. E-stop button | 10. Tool holder |
| 4. Cycle/Start and Pause button | 11. X carriage |
| 5. Standard USB port | 12. Trunnion assembly |
| 6. Mini USB port | 13. B-table housing (part of trunnion assembly) |
| 7. Ethernet port | 14. B-table (part of trunnion assembly) |
| | 15. X cable chain |



Chapter 2.2: Software Setup

Software needed: web browser (Chrome, Firefox, Safari) No internet connection is required

Hardware needed: computer/Chromebook, USB to mini USB cable

Chapter 2.3: Startup and Shutdown

Download and install one of the following web browsers:

- Google Chrome
- Firefox
- Safari

Chapter 2.3.1: Startup

To begin the startup process, follow these steps:

1. Connect the machine to your computer using the USB cable.
2. Turn on your machine.
3. Wait about 30 seconds for the machine to fully boot before attempting to connect to it.
The red, E-stop button will blink when the machine is fully booted.
4. Open your web browser and type the Pocket NC's IP address, "192.168.7.2" for Windows and "192.168.6.2" for Mac, into the search bar.

Upon startup, the user interface will open to the PRODUCTION page. From this page users have access to all of the buttons necessary for machine initialization.



The screenshot displays the KINETIC CONTROL software interface. At the top, there are navigation tabs: SETUP, MANUAL, PRODUCTION (highlighted), HISTORY, and CONFIG. The KINETIC CONTROL logo is in the top right corner. The main interface is divided into several sections:

- SPINDLE:** A circular gauge showing RPM at 0. It includes sub-gauges for PRESSURE (0.09 MPa), LOAD (0.0%), and TEMPERATURE (22.7 °C). A WARM UP button is located below the gauge.
- FEED:** A circular gauge showing IN/MIN at 0. It includes sub-gauges for POWER (0 W), CHIP LOAD (0.0000 in), and SURFACE SPEED (0.00 ft/min).
- DRO:** A section for Digital Read Out with a Position dropdown and a HOME ALL button. It shows coordinates for X (0.0000 in), Y (0.0000 in), Z (0.0000 in), A (0.000 deg), and B (0.000 deg).
- CONTROL:** A section with buttons for CYCLE/START and FEED HOLD. It includes a STEP button and an OPTIONAL STOP toggle. Below these are four sliders for MAX VELOCITY (100%), MAX RAPID (100%), FEED RATE (100%), and SPINDLE RATE (100%).
- G-CODE:** A section showing "NO FILE LOADED" and a file selection icon.

At the bottom of the interface, there is a status bar. On the left, it says "CONNECTED" and lists G17, G20, G49, G54, G90, G94, M5, M9, M429. In the center, a large red button reads "E-STOP". On the right, it shows "IDLE" and "00:00:00.0" along with several status icons.

To begin initializing the machine, begin by clicking on the large, red E-STOP button at the bottom of the page. Doing so will provide power to the machine's motors and prepare them for movement. Once the E-stop is deactivated, its button will change to RESET. The same behavior can be observed when pressing the E-stop button on the front of the machine.

Next, after ensuring that the machine is clear of anything it may collide with during movement, including hands and fingers, click the HOME ALL button. This will start the machine's homing process, sending each axis searching for its home switch. The machine will home the Z axis first, then X and Y together and then A and B together.



Once the machine is homed, the HOME ALL button will no longer have the triangular warning symbol next to it and, if a V2-50 machine is being used, the WARM UP button will no longer be greyed out.



If the machine is a V2-10, it is now initialized and ready to use. If the machine is a V2-50, a spindle warm up sequence must be run. To do so, ensure that the air pressure to the spindle is 20 psi or higher as read by the gauge in the upper left corner of the page and then click the WARM UP button. The machine will begin the spindle warm up sequence it determines is necessary based on the amount of time elapsed since its last spindle warm up. A notification will pop up in the lower right hand corner displaying the warm up sequence it is going to run.

The screenshot displays the KINETIC CONTROL interface with the following sections:

- Navigation:** SETUP, MANUAL, PRODUCTION (active), HISTORY, CONFIG.
- SPINDLE:** Pressure gauge (30.85 psi), RPM (0), Temperature (78.2 °F), and a WARM UP button.
- FEED:** Chip Load (0 IN/MIN), Surface Speed (0.00 ft/min), and a WARM UP button.
- DRO:** HOME ALL button and position values: X (2.5000 in), Y (2.5000 in), Z (-0.0000 in), A (0.000 deg), B (-0.000 deg).
- CONTROL:** CYCLE/START, FEED HOLD, STEP, and OPTIONAL STOP buttons. Sliders for MAX VELOCITY, MAX RAPID, FEED RATE, and SPINDLE RATE, all set to 100%.
- G-CODE:** NO FILE LOADED.
- Footer:** CONNECTED G17 G20 G49 G54 G90 G94 M5 M9 M429, RESET button, MDI 00:00:01.4, and status icons.

Chapter 2.3.2: Shutdown

When you are done operating the machine you can simply shut off the power button on the Pocket NC and unplug the machine from the computer.

The computer may generate an error message warning you to eject the BeagleBone before unplugging it. You may eject the BeagleBone from the computer if desired, but it is not necessary.

Chapter 2.4: User Interface Overview



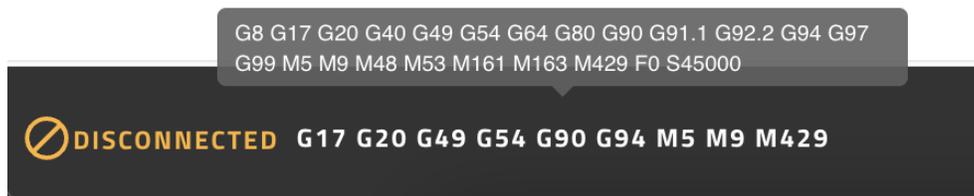
Chapter 2.4.1: Status Bar

The status bar at the bottom of the screen is where users can get information about the active status of the machine. This bar is always visible, no matter what page of the user interface a user is on.



On the far left of the status bar is the connection status. This is where users can check to see whether a connection has been made with the machine. When connected this text will be green and read “Connected”, when disconnected the text will be yellow and read “Disconnected”.

Next to the connection status is a list of g and m codes that are currently active on the machine. To see all of the active codes users can hover their mouse over this section of the status bar and the rest of the codes will show up.



In the center of the status bar is the E-STOP and RESET button. As mentioned before, this button shows up as E-STOP when the machine is first turned on and/or when the E-stop button on the front of the machine has been pushed and is blinking red. It becomes RESET after it or the E-stop button on the front of the machine has been pressed.

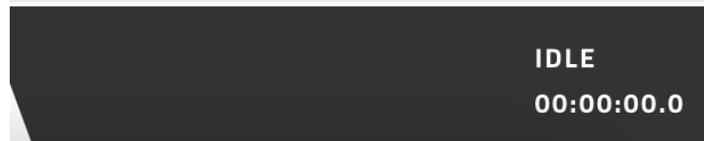
It is important to note that when the machine is in E-stop, there is no power to any of the machine’s motors and when the machine is taken out of E-stop all axes must be homed before the machine can be used. This is different behavior than the RESET button; when the RESET button is pressed the machine simply stops whatever it is doing. This includes any axis or spindle movement that is happening at the time. The RESET button will also automatically take the machine back to the beginning of the program so it should not be used to pause the machine.



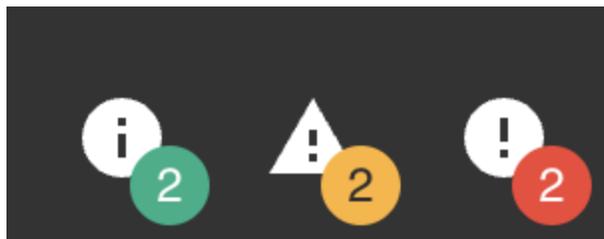
E - STOP

RESET

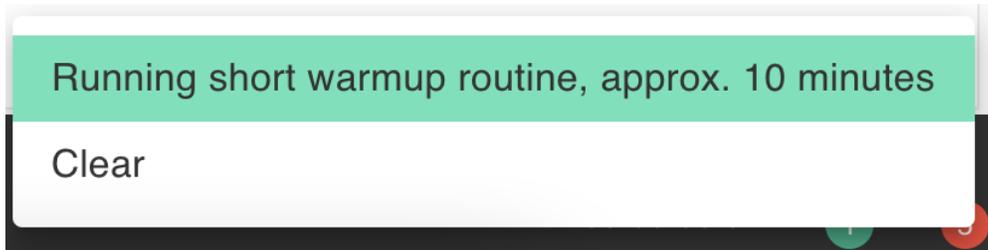
To the right of the center button is the machine's runtime clock. This is where users can see whether the machine is Idle, Running or Paused. It is also where users can view how long a program has been running. This clock will also show how long the last program took to run when the machine is in an idle state.



Lastly, on the far right side of the status bar are the message indicators. This space will be clear of indicators when there are no messages to be read but as messages pop up, indicators will show, along with how many new messages there are.



The "i" indicator represents information messages, the triangular exclamation mark indicator represents warning messages and the circular exclamation mark indicator represents errors. Each of these indicators can be clicked on to read their associated messages and then clear them.



After a message has been cleared from one of these indicators, it can still be read on the HISTORY page that will be discussed in more detail later in this guide.

Chapter 2.4.2: Setup Page

The SETUP page is where users can find all the tools and functionality needed for setting up the machine to run a specific part or program. Each section of this page is outlined below.

Work Offsets Section

The WORK OFFSETS section allows users to set a Work Coordinate System (WCS) origin to a location other than the machine's center of rotation and then assign that location to a work coordinate system g-code (G54 - G59.3).

WORK OFFSETS

	DRO		Absolute Position		G54	G92	No Active Tool
X	2.5000 in	=	2.5000	-	0.0000	0.0000	
Y	2.5000 in	=	2.5000	-	0.0000	0.0000	
Z	-0.0000 in	=	-0.0000	-	0.0000	0.0000	0.0000
A	0.000 deg	=	0.000	-	0.000	0.000	
B	-0.000 deg	=	-0.000	-	0.000	0.000	

The values in the DRO column represent the current DRO accounting for any offsets that have been applied.



WORK OFFSETS

	DRO		Absolute Position		G54	G92	No Active Tool
X	0.2308 in	=	0.2308	-	0.0000	0.0000	
Y	1.3000 in	=	1.3000	-	0.0000	0.0000	
Z	-3.2550 in	=	-3.2550	-	0.0000	0.0000	
A	0.000 deg	=	0.000	-	0.000	0.000	
B	-0.000 deg	=	-0.000	-	0.000	0.000	

Set X DRO via G54

New DRO value

SET CANCEL

After selecting SET, the row for that axis will update with new values to represent the current position of the axis as it applies to the work coordinate system that is active.

WORK OFFSETS

	DRO		Absolute Position		G54	G92	No Active Tool
X	0.0000 in	=	0.2308	-	0.2308	0.0000	
Y	1.3000 in	=	1.3000	-	0.0000	0.0000	
Z	-3.2550 in	=	-3.2550	-	0.0000	0.0000	0.0000
A	0.000 deg	=	0.000	-	0.000	0.000	
B	-0.000 deg	=	-0.000	-	0.000	0.000	

The G92 drop down menu allows users to Zero the DRO in relation to the G92 offsets as well as clear, suspend or restore G92 offsets. The editable fields below this drop down menu function the same as the ones to their left. Using G92 offsets is not as common as G5x offsets and a thorough understanding of how they work should be obtained before attempting to use them.



WORK OFFSETS

	DRO		Absolute Position		G54				Active Tool
X	0.0000 in	=	0.2308	-	0.2308	-	0.0000	-	
Y	1.3000 in	=	1.3000	-	0.0000	-	0.0000	-	
Z	-3.2550 in	=	-3.2550	-	0.0000	-	0.0000	-	0.0000
A	0.000 deg	=	0.000	-	0.000	-	0.000	-	
B	-0.000 deg	=	-0.000	-	0.000	-	0.000	-	

Lastly, the Active Tool column of this section will show what tool number is active based on recent g-code execution or selection of the tool in the tool table below. When a tool is active, its tool length offset will show up in the editable field of this column. This value can be altered using the same method as the other editable fields, but caution should be taken to make sure there is a full understanding of what effects the change will have on other pieces of the machine setup process.

WORK OFFSETS

	DRO		Absolute Position		G54		G92		TL0: Tool1
X	2.2692 in	=	2.5000	-	0.2308	-	0.0000	-	
Y	2.5000 in	=	2.5000	-	0.0000	-	0.0000	-	
Z	-3.2345 in	=	0.0000	-	0.0000	-	0.0000	-	3.2345
A	0.000 deg	=	0.000	-	0.000	-	0.000	-	
B	-0.000 deg	=	-0.000	-	0.000	-	0.000	-	



Jog Section

The JOG section of the SETUP page allows users to move the different axes of the machine in selected increments.

Note: If the user selects metric units the jog section and DRO will read in millimeters, not inches as shown below. The jog increments will also be in millimeters.

The screenshot shows the JOG section interface. At the top, there is a "Position" dropdown menu, a green "HOME ALL" button, a question mark icon, and a hamburger menu icon. Below this are five rows representing axes: X, Y, Z, A, and B. Each row has a colored header (X is green, others are grey), a numerical value, a unit, and a home icon. The X-axis value is 2.5000 in, with the last zero highlighted by a green box. The Y-axis value is 2.5000 in. The Z-axis value is 0.0000 in. The A-axis value is 0.000 deg. The B-axis value is 0.000 deg. At the bottom, there is a green minus button, a green box containing "0.0001 in" with a dropdown arrow, and a green plus button.

Axis	Value	Unit	Home Icon
X	2.5000	in	Home
Y	2.5000	in	Home
Z	0.0000	in	Home
A	0.000	deg	Home
B	0.000	deg	Home

At the top of the JOG section is the readout selector. This dropdown menu allows users to select what values they would like to have displayed in the DRO portion of the JOG section. The “Position” option shows the axes’ position with the work offsets included. The “Actual Position” shows the axes’ position without the offsets included, also known as machine coordinates.

This screenshot is similar to the previous one, but the "Position" dropdown menu is open, showing two options: "Position" and "Actual Position". The "Actual Position" option is highlighted. The X-axis value is still 2.5000 in, with the last zero highlighted by a green box.



Next to the readout selector is the HOME ALL button. This button does exactly as it says, when clicked the machine's axes will begin searching for their home switches. As mentioned before, the machine will home Z first, then X and Y together and then A and B together.

Taking up most of the JOG section is the DRO (digital readout). Each axis row has 2 functions, displaying the position of the axis and providing a way to jog the axis by decimal place.

To jog an axis using the readout, users can select the decimal place they wish to jog by and then click and drag the computer mouse up or down to change the number in increments of one. Touch screen users can do the same with their finger. Users with trackpads can simply click on the number once and then scroll. For example, if a .2" move of the Z axis is desired, the first decimal place would be selected and then scrolled up 2 increments.

JOG

Position ▾ **HOME ALL** ? ☰

X	2 . 5 2 0 0 in	🏠
Y	2 . 5 0 0 0 in	🏠
Z	- 0 . 2 0 0 0 in	🏠
A	0 . 0 0 0 deg	🏠
B	0 . 0 0 0 deg	🏠

− 0.1 in ▾ +

At the end of each axis row is a small house symbol. This button performs the homing sequence for the specified axis.

Below the DRO is a set of buttons that provide an alternate jogging option. The center dropdown menu allows users to select the increment size they would like to jog in. When the increment matches a specific decimal place, it will change the selected digit in the DRO. Custom



increments can be manually entered in the text field. The plus and minus signs on either side of the dropdown menu increment the selected axis positive or negative by the selected increment.

The screenshot shows the JOG control interface. At the top, there is a 'Position' dropdown menu, a green 'HOME ALL' button, a help icon (?), and a menu icon (≡). Below this are five axis position displays: X (2.5200 in), Y (2.5000 in), Z (-0.2000 in), A (0.000 deg), and B (20.000 deg). The Z axis is highlighted in green. Below the Z axis display is a green minus sign button, a text input field containing '0.01 in', and a dropdown menu with the following options: 0.0001, 0.001, 0.01, 0.1, and Continuous.

In the upper right Jog menu, there are additional options that affect jogging.

The screenshot shows the JOG and CONTROL interface. The JOG section on the left has the same layout as the previous screenshot, with the Z axis highlighted. The CONTROL section on the right has a 'MAX VELOCITY' label and a dropdown menu. The dropdown menu is open and shows 'JOG MODE Joint' with a dropdown arrow, 'KEYBOARD JOGGING' with a green toggle switch, and 'CAPTURE MOUSE WHEEL' with a white toggle switch.



Jog Mode allows the user to specify jogging in Joint mode vs Axis mode. Joint mode commands the machine's individual motors. Axis mode jogs the machine in the current cartesian axis. Often, both axis and joint mode will behave the same as axes are, by default, aligned with the motors of the machine. In some cases, they behave differently. For example, when TCPC is active and one or both of the rotary axes are not at 0, jogging the X axis could move multiple motors at once. Furthermore, machines can have software compensation in place to account for very small misalignments of the physical axes, which can result in small movements of Y and Z, when jogging in X, for example. In order to jog with those adjustments in place, the jog mode must be in Axis. When jogging in Axis mode, the machine is not aware of the limits of the joints and can move slightly over the soft limits, which causes the machine to error and movement to be disabled until the offending axis is moved back into its proper range.

Lastly, at the very bottom of the upper right Jog menu, users have the option of turning on keyboard and mouse wheel jogging. Keyboard jogging allows users to continuously jog the machine using the keys on their keyboard. Key bindings can be found on the CONFIG page. Mouse wheel jogging allows users to jog the selected axis by the selected increment using the wheel on their computer mouse without needing the mouse to be over the selected digit (it essentially binds mouse wheel up to the + button and mouse wheel down to the - button).

Tool Offsets Section

The TOOL OFFSETS section, sometimes known as the tool table, is where users can define and record data on the tools that will be used to run a part.

TOOL OFFSETS				
	TLO (in)	Diameter (in)	Flutes	Description
 T 1	5.0000	0.1250	1	
 T 2	5.0000	0.1250	1	
 T 3	5.0000	0.1250	1	
 T 4	5.0000	0.1250	1	
 T 5	5.0000	0.1250	1	

At the far left of each row is the tool measure button. When this button is pressed, the machine will begin the tool measuring process. This includes moving the machine to home followed by bringing the spindle forward and prompting the user to install the proper tool and then press cycle/start. From there the machine will position the tool in front of the tool probe button under the B axis and then finish the probing sequence by pushing the button twice. After the probing



sequence is complete the machine will return to the position it was in when the tool measure button was pressed and the TLO (tool length offset) will be stored in the TLO column.

TOOL OFFSETS

	TLO (in)	Diameter (in)	Flutes	Description
T 1	3.2345	0.1250	1	
T 2	5.0000	0.1250	1	
T 3	5.0000	0.1250	1	
T 4	5.0000	0.1250	1	
T 5	5.0000	0.1250	1	

Next to the tool measure button is the active tool indicator and button. This button can be clicked to activate a specific tool number, turning it black. The tool number that is shown as active here should agree with the tool that is listed as active in the far right column of the WORK OFFSETS section discussed earlier.

The TLO column is where the TLO measurements are stored after a tool has been measured using the machine's integrated tool probe. These editable fields can have values manually input if a user chooses to measure a tool by hand.

The Diameter column allows users to note the diameter of the tool that has been assigned that tool number. This value helps inform the gauge readouts on the MANUAL and PRODUCTION pages and, in some cases, with the right CAM settings, this value can be used to dictate cutter compensation. Information on how to use that functionality should be sourced from the user's CAM provider.

The Flutes column allows users to note how many flutes the tool assigned to that tool number has on it. This field also helps inform the gauge readouts on the MANUAL and PRODUCTION pages.

Lastly, the Description column of the TOOL OFFSETS section allows users to add any additional information about the tool assigned to that tool number. Things like type of tool, coating, manufacturer, etc.



TOOL OFFSETS

	TLO (in)	Diameter (in)	Flutes	Description
T 1	5.0000	0.0625	1	Ball endmill, TiN coated, Harvey Tool
T 2	5.0000	0.1250	1	
T 3	5.0000	0.1250	1	
T 4	5.0000	0.1250	1	
T 5	5.0000	0.1250	1	

Chapter 2.4.3: Manual Page

The MANUAL page is where users can manually manipulate the machine in a variety of ways, including jogging and executing MDI (manually written g-code) commands.

The screenshot displays the 'MANUAL' page of the KINETIC CONTROL interface. At the top, navigation tabs include SETUP, MANUAL (active), PRODUCTION, HISTORY, and CONFIG. The main area is divided into several functional sections:

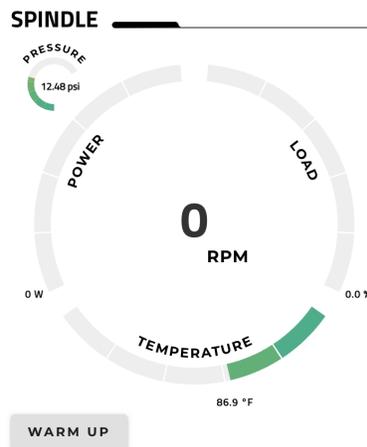
- SPINDLE:** Features three circular gauges: RPM (0), LOAD (0.0%), and TEMPERATURE (0.0%).
- FEED:** Features three circular gauges: IN/MIN (0), CHIP LOAD (0.0000 in), and SURFACE SPEED (0.00 ft/min).
- JOG:** Includes a 'Position' dropdown, a 'HOME ALL' button, and input fields for X (2.5000 in), Y (2.5000 in), Z (-0.2519 in), A (0.000 deg), and B (349.362 deg). A '0.1 in' dropdown and '+'/'-' buttons are also present.
- CONTROL:** Contains four sliders for MAX VELOCITY (100%), MAX RAPID (100%), FEED RATE (100%), and SPINDLE RATE (100%).
- MDI:** Has an 'Enter MDI...' input field containing 'G0 B2000'. Below it is a list of commands: RECENT (G0 B2000), LOG (G0 B2000), and FAVORITES (G0 B0).

At the bottom, a status bar shows 'CONNECTED' on the left, 'RESET' in the center, and 'IDLE 00:00:00.0' on the right.



Spindle Section

Under the spindle section of the MANUAL page users will find gauges and buttons related to the machine's spindle.

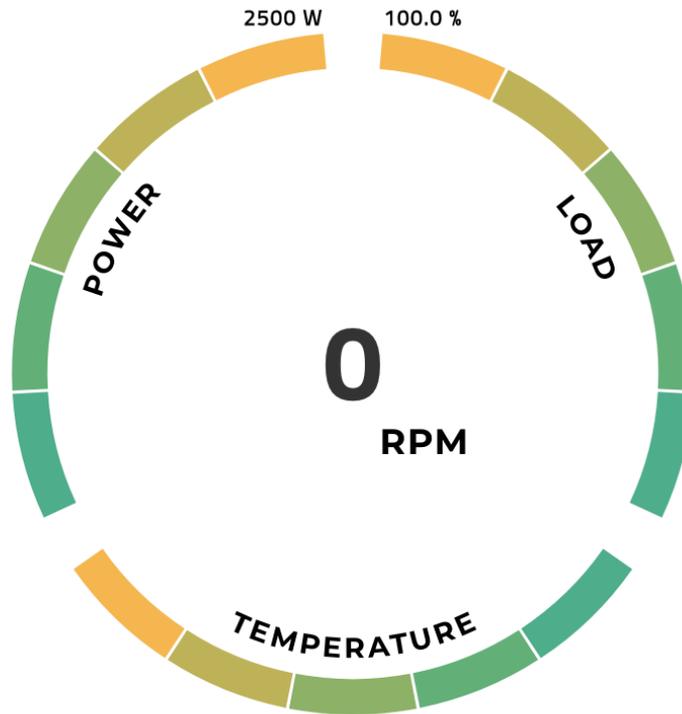


On V2-50 machines there will be an air pressure gauge in the top left corner of the SPINDLE section, this is the gauge that must read a certain pressure before the machine's spindle can be run. This gauge will not be present on V2-10 machines.

In the center of the SPINDLE section is the 4-in-1 radial gauge. The center value is the spindle's actual spindle speed in RPM. The upper left third of the dial is the spindle's power consumption in watts while the upper right third is the load on the spindle displayed in a percentage. And the bottom third of the radial gauge is a temperature of the spindle displayed in the units chosen in the CONFIG page. Note, that on the V2-50, the temperature is read from a sensor mounted to an electronics board so it may not reflect the actual temperature of the spindle, but is used as a safety measure to prevent spindle usage in extreme temperatures. Each portion of the radial gauge will display green towards the bottom of its range and progressively change to orange as values reach the top of its range.



SPINDLE

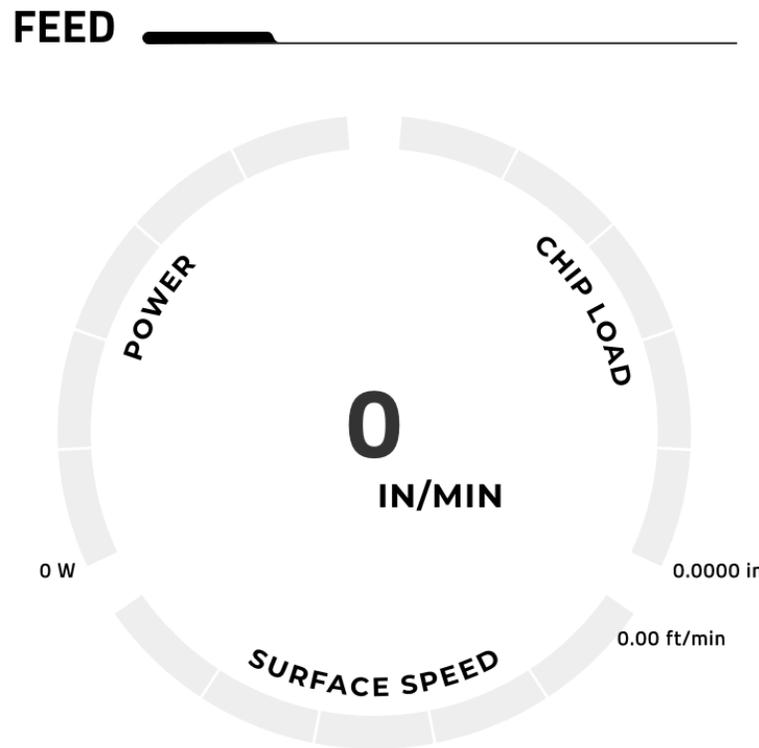


Lastly, at the bottom of the SPINDLE section, there is the WARM UP button. This is the button that was mentioned and used in the Initialization section of this guide. This button is only present on V2-50 machines and is used to warm up the spindle after periods of inactivity. It will be green with a triangular warning sign on it when a warm up is needed and dark grey when a warm up is no longer needed. If it appears "greyed out" it is likely that the machine needs to be taken out of E-stop and/or homed.



Feed Section

The FEED section of the MANUAL page is very similar to the SPINDLE section, it displays the various values related to how fast the machine is moving (typically through a material).



In the center of the 4-in-1 radial gauge is the feed rate readout. This value is displayed in the units that have been selected in the CONFIG page.

The upper left third of the gauge displays the total power that the machine is consuming in watts.

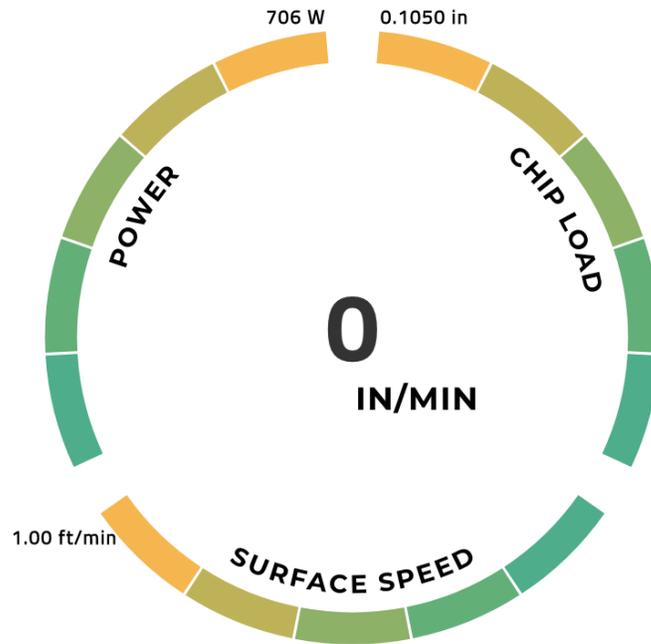
The upper right third of the gauge displays the calculated chip load of the current cut based on the feed rate and the number of flutes defined in the tool table for the active tool.

And lastly, the bottom third of the gauge displays the calculated surface speed of the tool based on the spindle speed and the diameter input in the tool table for the active tool.



Just like the gauges in the SPINDLE section, the gauges in the FEED section change colors as they reach the end of their range.

FEED





Jog Section

The JOG section of the MANUAL page functions the exact same way as it does on the SETUP page. Refer to that section above for a detailed walkthrough.

JOG _____

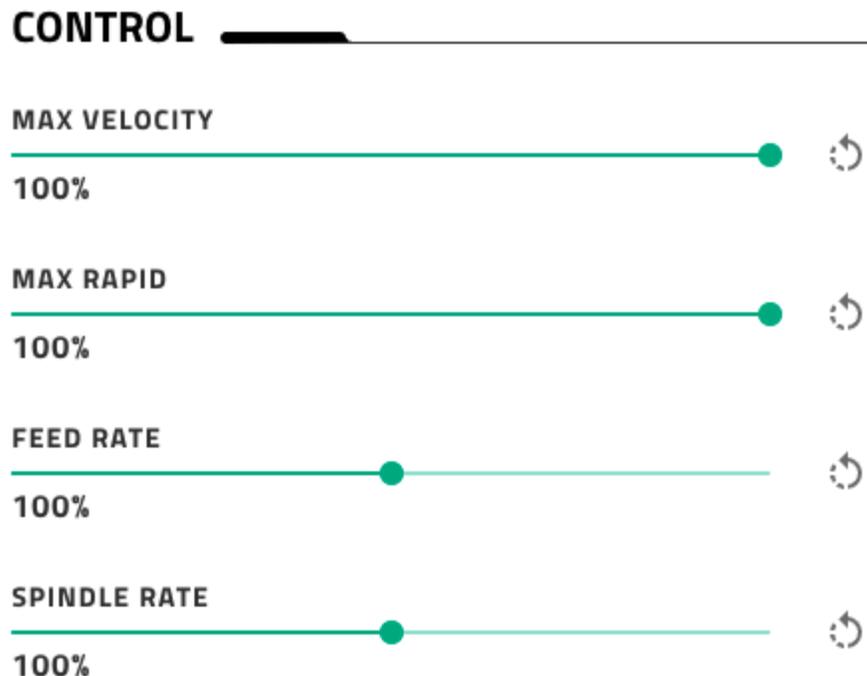
Position ▾ **HOME ALL** ? ☰

X	2 . 5 0 0 0 in	⬆
Y	2 . 5 0 0 0 in	⬆
Z	0 . 0 0 0 0 in	⬆
A	0 . 0 0 0 deg	⬆
B	0 . 0 0 0 deg	⬆

— 0.0001 in ▾ +

Control Section

The CONTROL section of the MANUAL page allows users to override the velocity of commanded moves as well as the spindle speed.



The CONTROL section is made up of four override sliders. Each slider allows users to override the nominal or commanded value of that particular parameter. The percent by which the value has been changed is displayed below the slider and the refresh button to the right of each slider resets it back to 100%. The sliders can be adjusted in 1% increments.

Max Velocity controls the maximum speed at which the tip of the tool can move.

Max Rapid refers to how fast the tool can move during G0 commands specifically.

Feed Rate is how fast the machine is moving the tool during feed rate moves (F values).

Finally, Spindle Rate is the speed at which the spindle is rotating.

MDI Section

The MDI section of the MANUAL page allows users to manually enter lines of g-code to perform various operations or put the machine into certain modes or positions.



MDI

Enter MDI...		
RECENT	M3 S45000	☆
LOG	T3 M6	☆
FAVORITES	M6	☆
QUICK ACCESS		

At the top of the MDI section is the MDI command bar. Here is where users can enter single line g-code commands and then press enter to issue them. Once a command has been entered and issued, it gets stored in one or more of the folders below the command bar.

The Recent folder shows the lines of g-code that have most recently been commanded to the machine. This folder only shows the uniquely different commands that have been issued recently, it does not show the number of times a command has been issued. Any command in this folder can be selected to automatically populate the command bar with it and issue it again.

The Log folder displays each command that has been issued and the order in which it was issued, regardless of whether there are duplicates or not.

MDI

Enter MDI...		
RECENT	M3 S45000	☆
LOG	T3 M6	☆
FAVORITES	T3 M6	☆
QUICK ACCESS	T3 M6	☆
	M6	☆

The Favorites folder is where any commands that have been “starred” in the Recent or Log folders (by clicking on the star to the far right of the command) are stored. In this folder users have additional options like being able to give the command a name (which saves it to the Quick Access folder) and assigning it to a keyboard key. The date the command was added to the Favorites is also displayed here.



MDI

Enter MDI...				
RECENT	Command ↑	Name ?	Keybinding	Date Added
LOG	<input type="checkbox"/>	M3 S45000	Quick Access Name	10/6/2021, 1:03:58 PM
FAVORITES			<input type="text"/>	
QUICK ACCESS				

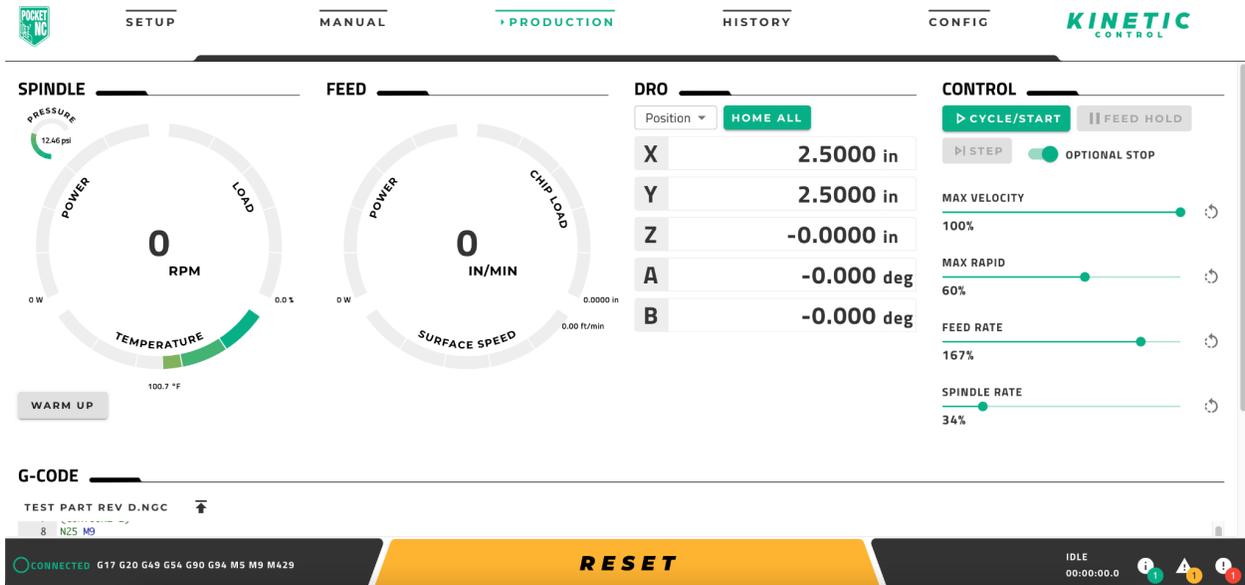
Lastly, the Quick Access folder provides users a way to quickly select and issue commands by clicking on buttons named for the command they perform. For example, instead of searching for “M3 S45000” in the Favorites folder, clicking on it to put it in the command bar, and then pressing enter to issue the command, a user can simply click on the button labeled “Spindle on - 45K” and the command will be issued. As mentioned above, Quick Access buttons are created by naming a command that has been added to the Favorites folder.

MDI

Enter MDI...			
RECENT	SPINDLE ON - 45K		
LOG			
FAVORITES			
QUICK ACCESS			

Chapter 2.4.4: Production Page

The production page is where users have the ability to upload, start, monitor and manipulate the g-code program that will be run. There is no way to manually move the machine on this page.



Spindle Section

The SPINDLE section of the PRODUCTION page is the same as it is on the MANUAL page. Refer to that portion of this guide for a detailed description and walkthrough.

Feed Section

The FEED section of the PRODUCTION page is the same as it is on the MANUAL page. Refer to that portion of this guide for a detailed description and walkthrough.

DRO Section

The DRO section of the PRODUCTION page is very similar to the JOG section that has been discussed previously. The main difference is that this section does not allow users to jog any of the machine's axes. The only way to move the machine's axes from this section is with the Home all button.

The DRO's main function is to provide users a way to monitor the position of the machine's axes relative to the point that corresponds with the selected readout type. The readout type dropdown menu at the top left corner of the JOG section shares two readout options with the JOG section on other pages but also has two additional options, DTG (distance to go) and Velocity.



DRO

Position	HOME ALL
Actual Position	2.5000 in
DTG	2.5000 in
Velocity	
Z	-0.0000 in
A	-0.000 deg
B	-0.000 deg

DTG (distance to go) is a DRO option that allows users to see how far away a programmed move is from its final position. For example, if the program commands a G0 X1.5 move, the DRO (set to DTG) would show X as 1.5 and then slowly count down until the X axis was at 1.5, where the DRO would then read 0.

The Velocity option allows users to see what each axis is reaching for velocity during the moves the machine is making. This allows users to see if the machine is actually able to reach the feed rates that are being commanded.

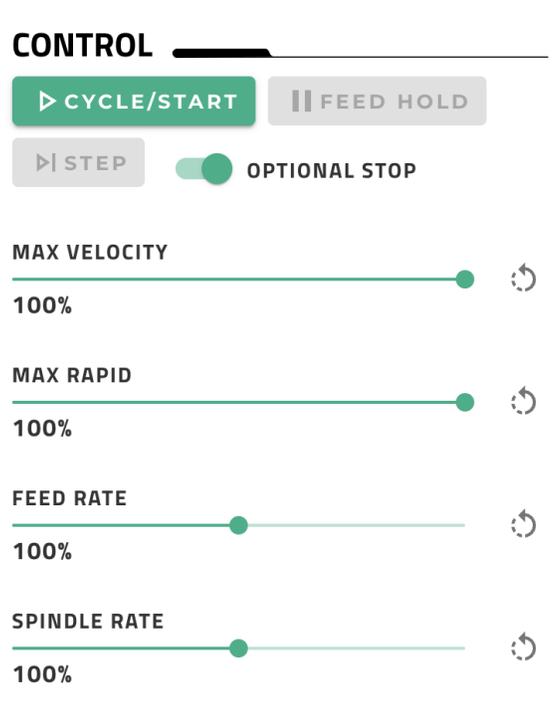


DRO

Velocity ▾	HOME ALL
X	0.0000 $\frac{\text{in}}{\text{min}}$
Y	0.0000 $\frac{\text{in}}{\text{min}}$
Z	0.0000 $\frac{\text{in}}{\text{min}}$
A	0.000 $\frac{\text{deg}}{\text{min}}$
B	0.000 $\frac{\text{deg}}{\text{min}}$

Control Section

The CONTROL section of the PRODUCTION page allows users to start, pause and manipulate the active or running program.



At the top of the CONTROL section there are several buttons that provide users the ability to start and stop the g-code program that is active on the PRODUCTION page. Each of these buttons has a slightly different way of doing this and some of them can be used with manually written g-code as well.

The Cycle/Start button starts the g-code program that is active on the PRODUCTION page and will only start or restart (after a pause) that program, nothing else. This button is mimicked by the green button on the front of the machine.

The Feed Hold button acts as a pause button for the actively running program. It is only available when g-code (manually input or active on the PRODUCTION page) is running. This button will pause the axis movement of the machine but **WILL NOT stop the spindle**. If stopping the axis movements and the spindle is needed, press the RESET button at the bottom of the screen.

Below the Cycle/Start button is the Step button. This button allows users to step through the g-code that is active on the PRODUCTION page one line at a time. The program must be started and then paused (feed hold) before this button is active and available to use.



CONTROL

The CONTROL interface features a top row with a green '▶ CYCLE/START' button and an orange '|| FEED HOLD' button. Below this is a dark green '▶| STEP' button and a green toggle switch labeled 'OPTIONAL STOP'. The bottom section contains four sliders, each with a green knob and a refresh icon to its right. The sliders are labeled 'MAX VELOCITY', 'MAX RAPID', 'FEED RATE', and 'SPINDLE RATE', with '100%' displayed below each.

To the right of the Step button is the Optional Stop switch. This feature allows users to have M1 commands (program pause) in their g-code programs and choose whether those commands are recognized or not. The slider is green when the switch is “on” and the M1 commands will be executed and it is grey when the switch is “off” and the M1 commands will be ignored.

The rest of the CONTROL section is made up of four override sliders. Each slider allows users to override the nominal or commanded value of that particular parameter. The percent by which the value has been changed is displayed below the slider and the refresh button to the right of each slider resets it back to 100%.

Max Velocity controls the maximum speed at which the tip of the tool can move.

Max Rapid refers to how fast the tool can move during G0 commands specifically.

Feed Rate is how fast the machine is moving the tool during feed rate moves (F values).



Finally, Spindle Rate is the speed at which the spindle is rotating.

CONTROL

▶ CYCLE/START **|| FEED HOLD**

▶ STEP **OPTIONAL STOP**

MAX VELOCITY
100%

MAX RAPID
60%

FEED RATE
167%

SPINDLE RATE
34%

The control panel interface includes a title "CONTROL" with a horizontal line underneath. Below the title are four buttons: a green "▶ CYCLE/START" button, a grey "|| FEED HOLD" button, a grey "▶ STEP" button, and a grey "OPTIONAL STOP" button with a green toggle switch that is currently turned on. Below these buttons are four sliders, each with a green knob and a circular refresh icon to its right. The sliders are labeled "MAX VELOCITY" (100%), "MAX RAPID" (60%), "FEED RATE" (167%), and "SPINDLE RATE" (34%).

[G-Code Section](#)

The G-CODE section of the PRODUCTION page is where users can upload and manage the g-code programs that are intended to be run on the machine.



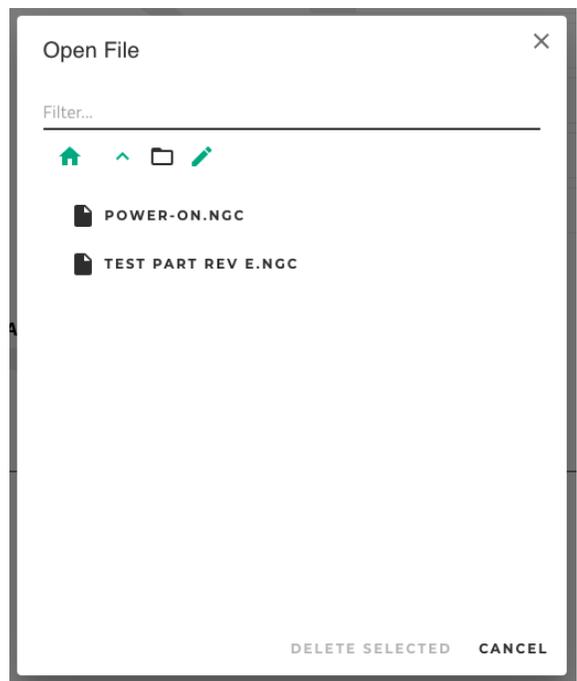
G-CODE

  Test part Rev E.ngc

```
1 %  
2 (AXIS,stop)  
3 (1)  
4 N10 G20  
5 N15 G90 G94 G40 G17 G91.1  
6 N20 G53 G0 Z0.f20  
7 (CONTOUR1 2)  
8 N25 M9  
9 N30 G49  
10 N35 M5  
11 N40 G53 G0 X2 5 Y2 5
```

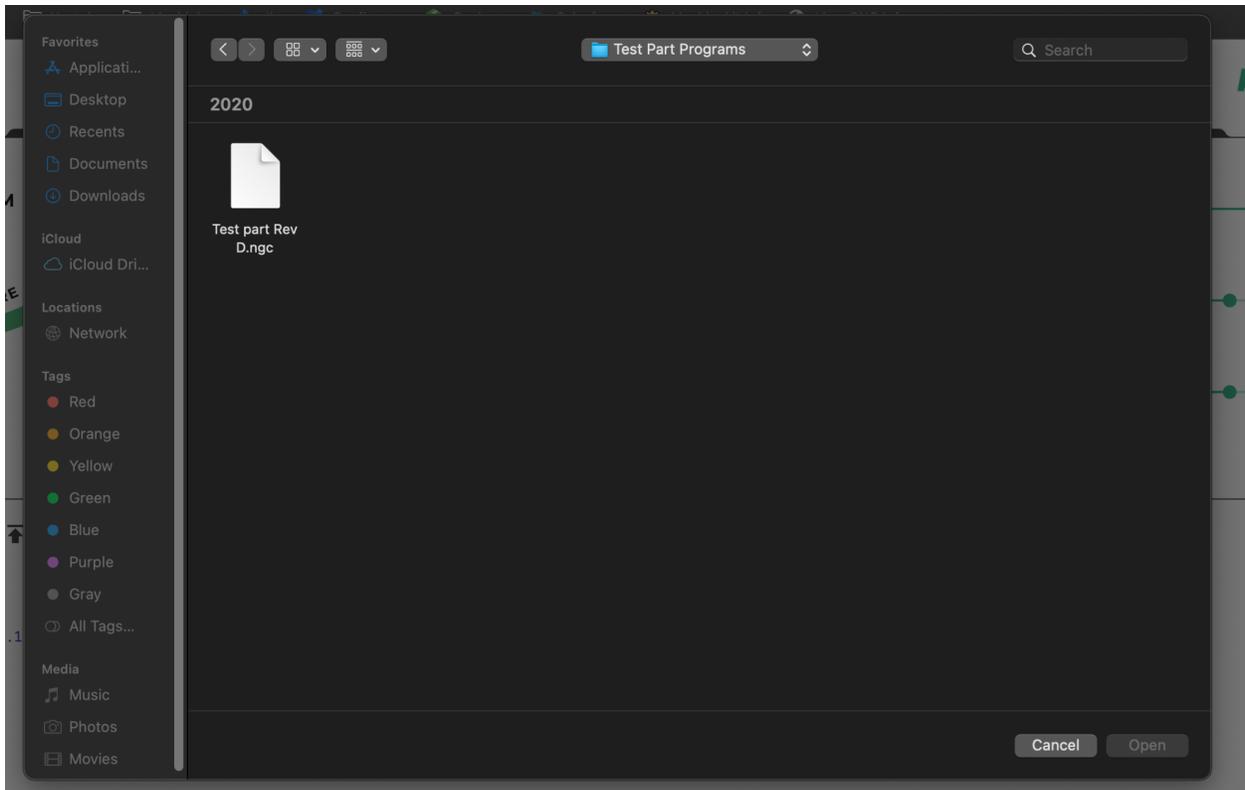
 

At the very top of the G-CODE section there are buttons for opening and uploading g-code programs followed by a label that shows the name of the currently active program. If there is no g-code program active then it will say “No File Loaded”. When the Open File button is clicked (the one with the folder icon), the “Open File” window comes up. From this window users can manage the location of the files that have been uploaded to the machine, delete files or select one to be active.





To the right of the active program text is the Upload File button. This is the button that users can press to open up their computer's file browser and select a file to be uploaded to the machine.



The rest of the G-CODE section is simply a window that shows the actual g-code of the active program. Here users can look through the code and track with the machine as it runs the code. This window can also be helpful when errors arise and the machine stops running, allowing users to see what line and code(s) the machine stopped on.



G-CODE

📁⬆️Test part Rev E.ngc

```
127 x-.02
128 y.02
129 X0 y0
130 x.8 y.8f30
131 g01 x.8 y.8 z-3.0 f15. s10000 m3
132 X.54 y.54 f25
133 Z-3.0
134 Y-.54
135 X-.54
136 Y.54
```

⬇️⬇️

Chapter 2.4.5: History Page

The HISTORY page provides users with a place to review the various errors, warnings and info panels that a user may encounter while operating the machine.

SETUPMANUALPRODUCTIONHISTORYCONFIGKINETIC CONTROL

MESSAGE LOG

🟢 Successfully uploaded Test part Rev D.ngc More >>	22 minutes ago
🔴 File does not exist: /sysroot/home/pocketnc/ncfiles/Test part Rev D.ngc More >>	25 minutes ago
🔴 Failed to initialize file download: Test part Rev D.ngc More >>	25 minutes ago
🟢 Connected to machine.	25 minutes ago
🔴 Spindle air supply pressure below minimum 20 PSI (0.138 MPA). More >>	2 hours ago
🟢 Connected to machine.	3 hours ago
⚠️ Lost connection with machine.	3 hours ago
🔴 Error executing command More >>	4 hours ago
🔴 Error executing command More >>	6 hours ago

🟢 CONNECTED G17 G20 G49 G54 G90 G94 M5 M9 M429RESETIDLE
00:00:00.0



Message Log Section

Under the MESSAGE LOG section, users can see the chronological log of all the messages that have come up since the machine was powered on. Each log can be expanded by clicking on the small “More>>” text that is at the end of the message title.

MESSAGE LOG

- Successfully uploaded Test part Rev D.ngc** << Less 29 minutes ago
10/6/2021, 4:04:01 PM
code 70K
data
notify
message Successfully uploaded Test part Rev D.ngc
filename Test part Rev D.ngc
type info
id FILE_UPLOAD_FINALIZE_TEST PART REV D.NGC
meta
actionEmitters
0 @@rockhopper/RESPONSE/FILE_UPLOAD_FINALIZE/TEST PART REV D.NGC
- File does not exist: /sysroot/home/pocketnc/ncfiles/Test part Rev D.ngc** More >> 31 minutes ago
- Failed to initialize file download: Test part Rev D.ngc** More >> 31 minutes ago

Green messages serve as notifications and confirmation messages, yellow messages are warnings and red messages are errors.

Chapter 2.4.6: Config Page

The CONFIG page allows users to adjust the configuration of both the user interface and the machine.



The screenshot displays the KINETIC CONTROL software interface. At the top, there are navigation tabs: SETUP, MANUAL, PRODUCTION, HISTORY, and CONFIG. The CLIENT tab is selected, showing a sub-tab for SERVER. The main content area is divided into three sections: KEYBINDINGS, UNITS, and AUDIO. The KEYBINDINGS section contains a table with columns for Command and Keybinding, listing various machine functions and their assigned keys. The UNITS section shows settings for Length, Time, Surface Speed, Pressure, and Temperature. The AUDIO section includes a volume slider and an 'Audio Enabled' toggle. The LANGUAGE section shows settings for Language and Dialect. At the bottom, there is a status bar with 'CONNECTED' and a list of machine models, a large 'RESET' button, and an 'IDLE' status with a timer.

Command	Keybinding
Axis-select Jogging	
Jog +	Equal
Jog -	Minus
Next Axis	Quote
Previous Axis	Semicolon
Increase Step Size	BracketLeft
Decrease Step Size	BracketRight

Client Tab

The CLIENT tab is where users have the option to make changes to the way the machine's user interface looks and functions

Keybindings Section

In the KEYBINDINGS section of the CLIENT tab users can bind the keys of their keyboard to certain machine commands.



KEYBINDINGS

Command	Keybinding		
Axis-select Jogging			
Jog +	Equal		
Jog -	Minus		
Next Axis	Quote		
Previous Axis	Semicolon		
Increase Step Size	BracketLeft		
Decrease Step Size	BracketRight		

To set or edit one of the bindings, all a user has to do is click on the button with the pencil symbol on it to open up the edit window. From there the desired key can be pressed and the Set button clicked to save the selection.

The screenshot shows the 'KEYBINDINGS' menu with a list of commands and their current keybindings. The 'Jog -' command is selected, and a dialog box is open with the text 'Keybinding: jog-' and two buttons: 'SET' and 'CANCEL'. The background is dimmed, showing other menu options like 'UNITS', 'AUDIO', and 'LANGUAGE'.

Command	Keybinding		
Axis-select Jogging			
Jog +	Equal		
Jog -	Minus		
Next Axis	Quote		

Keybinding: jog-

SET CANCEL



If a user wishes to unbind a command completely, the button with the trash can symbol can be pressed.

Units, Audio and Language Sections

UNITS 

Length Time Surface Speed Pressure Temperature

Program Units ▼ min ▼ ft/min ▼ psi ▼ F ▼

AUDIO 

Volume

Audio Enabled

LANGUAGE 

Language Dialect

English ▼ Industry ▼

The UNITS section of the CLIENT tab gives users the option to change the units of the various values that are displayed throughout the user interface.

The AUDIO section allows users to change the volume of the feedback noises used throughout the user interface. The UI's audio can be completely disabled using the switch at the bottom of this section.

The LANGUAGE section gives users the ability to change the base language of the user interface as well as change the dialect (lingo) that is used throughout. The "Industry" dialect option will display components of the UI with their common industry name, the "Intuitive" option will display certain components with their more common-language names.



Server Tab

The SERVER tab is where users can update their machine's software and make changes to their machine's back end configuration.

The screenshot shows the SERVER tab interface. At the top, there are two tabs: CLIENT and SERVER, with SERVER selected. Below the tabs, the SOFTWARE section is active, displaying 'Active Version: 0.0.1'. There are three buttons: 'SHUTDOWN SERVICES' (green), 'REBOOT MACHINE' (orange), and 'SHUTDOWN MACHINE' (orange). Below the SOFTWARE section is the MACHINE CONFIG section, which has three sub-sections: 'DOWNLOAD' (with a 'Serial number' input field), 'UPLOAD', and 'RESTART SERVICES'. The 'COMPENSATION TABLES' sub-section is active, showing a table with columns for Position, Forward Trim, and Reverse Trim. The table has two rows, A and B.

	Position	Forward Trim	Reverse Trim
A	-45	-0.043937	-0.043937
B	5	0.004882	0.004882

Software Section

The SOFTWARE section of the SERVER tab is where users will find the ability to update their software. The active version will be shown at the top. If the machine is connected to the internet, the latest version will be shown below the active version. In order to update the software, the main PocketNC and Rockhopper services need to be shutdown, so the first step to updating is to click the Shutdown Services button. Once the services have shutdown, if updates are available, an update button will be shown that can be clicked to download and install the updates. The update process can be interrupted at any time and will not affect the current state of the machine (i.e. beginning the update process and not finishing it, will not leave the machine in a broken state). After the update process completes successfully, the active version will switch to Pending Version, which will become active after rebooting the machine. The PocketNC and Rockhopper services will start automatically after rebooting the machine. The Shutdown Machine button can be clicked to ensure a complete shutdown of all software services before removing power, but it isn't required to do so.



SOFTWARE

Active Version: 0.0.1

SHUTDOWN SERVICES

REBOOT MACHINE

SHUTDOWN MACHINE

Machine Config Section

The MACHINE CONFIG section of the SERVER tab is where users can edit the back end configuration of the machine as well as upload and download the machine's calibration and configuration data.

The screenshot shows the MACHINE CONFIG interface. At the top, there are three buttons: DOWNLOAD, UPLOAD, and RESTART SERVICES. Below these buttons is a table with three tabs: INI OVERLAY, ACTIVE INI, and COMPENSATION TABLES. The INI OVERLAY tab is selected, showing a table with columns for Name, Value, Comment, and Help. The table contains one row for HOME_OFFSET with a value of 2.6073. On the left side of the table, there is a list of joint names: JOINT_0, JOINT_1, JOINT_2, JOINT_3, JOINT_4, POCKETNC, POCKETNC_FEATURES, and TOOL_PROBE. At the bottom right of the interface, there is an EDIT OVERLAY button.

Name	Value	Comment	Help
HOME_OFFSET	2.6073		

The DOWNLOAD button and “Serial number” box allows machine users to download the INI Overlay that is currently active on the machine, this file includes configuration and calibration data.

The UPLOAD button allows users to upload an INI overlay to the machine. And the RESTART SERVICES is what makes the new INI overlay active. This button is also used to make any edits to the INI overlay active.



At the bottom right corner of the MACHINE CONFIG section is the button EDIT OVERLAY. This button opens up the ability to edit the INI overlay by adding a plus and trash button on the left hand side and making the fields in the middle of the screen editable.

MACHINE CONFIG

DOWNLOAD Serial number UPLOAD RESTART SERVICES

INI OVERLAY ACTIVE INI COMPENSATION TABLES

Name	Value	Comment	Help
HOME_OFFSET	2.6073		

JOINT_0
JOINT_1
JOINT_2
JOINT_3
JOINT_4
POCKETNC
POCKETNC_FEATURES

SAVE DISCARD

Editing the INI overlay is not something a user should do without a full understanding of the effects it could have on the machine or having consulted a Pocket NC service and support team member.

Chapter 3: Machine Operation

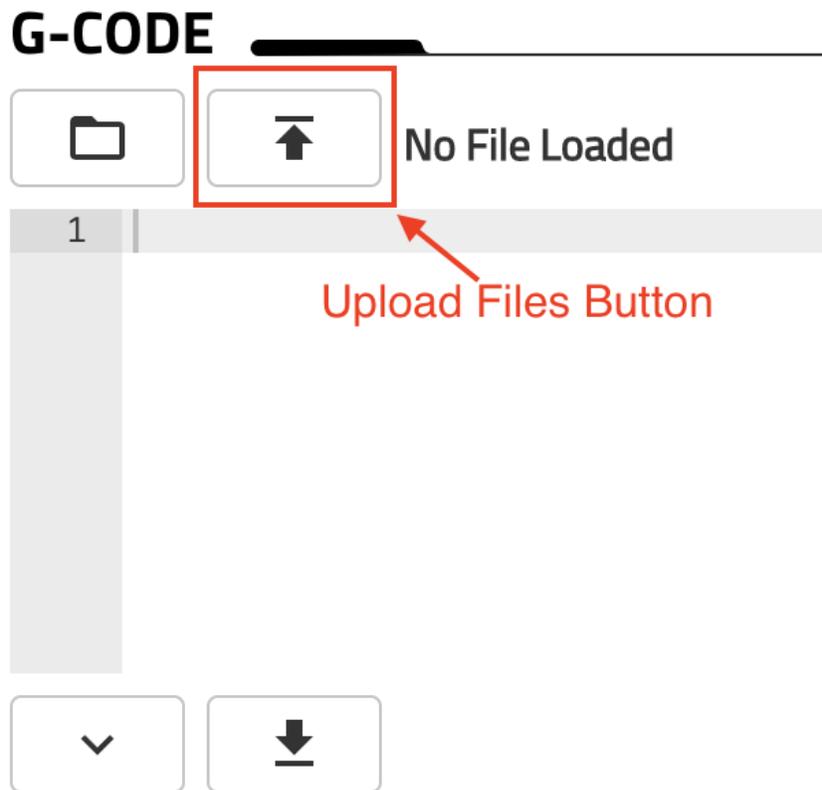
This chapter covers the basics of operating the Pocket NC V2 machines.

Chapter 3.1: File Management

Once you have created a gcode program in CAM software like Fusion 360 or Mastercam, you will need to transfer it to the machine in order to run it.

Chapter 3.1.1: Loading Files

Upload a gcode file to the BeagleBone by clicking on the “Upload Files” button.



Select your .ngc file from the file directory on your computer or USB flash drive plugged into your computer or the machine.

After the file has been successfully uploaded, the file name will appear and be available for selection under the Open File button.





Once a gcode program has been selected, the file name will show up in the G-CODE section, to the right of the upload button. Individual lines of the code will also show up, indicating that the code is active. The user can view the file before running it and while it runs using this interface.

G-CODE

Test part Rev E.ngc

```
1  %
2  (AXIS,stop)
3  (1)
4  N10 G20
5  N15 G90 G94 G40 G17 G91.1
6  N20 G53 G0 Z0.f20
7  (CONTOUR1 2)
8  N25 M9
9  N30 G49
10 N35 M5
11 N40 G53 G0 X2 5 Y2 5
```

Chapter 3.1.2: Running Files

Once a program is active, it can be run by clicking the green “Cycle/Start” button at the top of the PRODUCTION page.



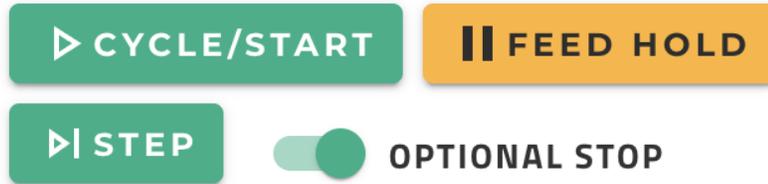
CONTROL

The control panel features several interactive elements: a green "CYCLE/START" button with a play icon, a grey "FEED HOLD" button with a pause icon, a grey "STEP" button with a right-pointing arrow icon, and a toggle switch for "OPTIONAL STOP" which is currently turned on. Below these are four sliders for "MAX VELOCITY", "MAX RAPID", "FEED RATE", and "SPINDLE RATE", each with a "100%" label and a refresh icon.

If you have an M0 (or M1 with optional stop on) command at the start of your code, the program will pause. Click the CYCLE/START button in the UI or push the green, blinking start/pause button on the machine to resume the program. The user may pause the program at any point by clicking the FEED HOLD button or pushing the green start/pause button on the machine. The user can completely stop the program by clicking the RESET button at the bottom of the screen.

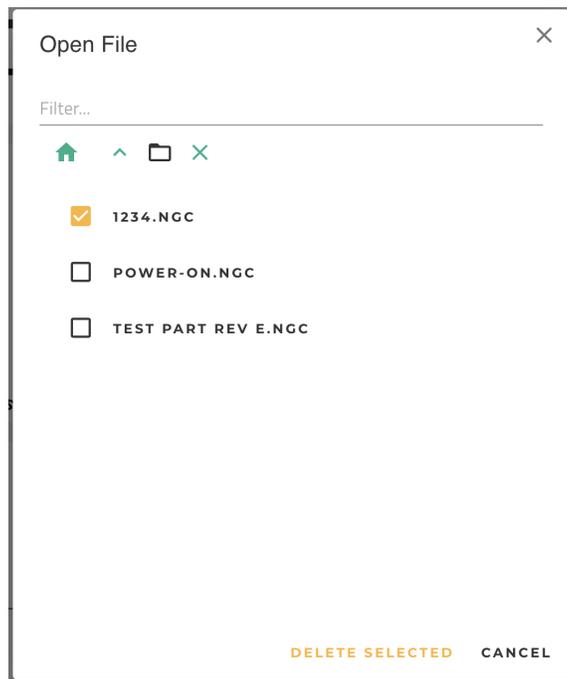


CONTROL



Chapter 3.1.3: Deleting Files

To delete a file from the machine users can click the Open File button then click the pencil at the top of the window. From there the file can be selected by clicking on the box next to it and then it can be deleted by clicking DELETE SELECTED at the bottom of the window.





Chapter 3.2: V2-10 and V2-50 Tooling

This section describes tool management of the Pocket NC: commanding tool changes, loading tools into holders, setting CNC tool offsets. The following is written for V2-10 machines. V2-50 users will follow the same steps except for those related to the tool holders.

Chapter 3.2.1: Tool Holders (V2-10 Only)

The Pocket NC V2-10 requires a specific type of tool holder. Each tool holder has a taper that matches that of the Pocket NC spindle collar. Tool holders are interchangeable and allow for the use of multiple tools with minimal effort between tool changes. Three types of tool holders exist, the short, standard tool holder, the long, extended tool holder, and the edge finding tool holder.

The standard and extended tool holder both use an ER11 collet system for holding tools. The collets are available in Standard and Metric sizes ranging from $\frac{1}{8}$ inch to $\frac{5}{16}$ inch and from 3 mm to 6 mm.

The edge finding tool holder uses set screws to retain the tool. It is only compatible with $\frac{3}{8}$ inch diameter tool shanks.





Chapter 3.2.2: Tool Holder Care (V2-10 Only)

Make sure to keep the tool holders clean. Take special care to remove debris and oil from the inside taper before installing a collet or tool and the outside taper before installing the tool holder in the spindle. Use a soft, lint-free cloth and isopropyl alcohol to clean the tool holders.

Put a thin coat of light oil on the tool holders and collets if they are going to be stored for a long period of time, this will help prevent rust. Remove the oil before using them.

Before operating the spindle, make sure that all of the set screws are tightened evenly and snugly and that the collet nut is tightened securely.

Chapter 3.2.3: Tool and Tool Holder Set Up

V2-10

Each tool holder consists of 4 components; tool holder, ER11 collet, collet nut and end mill tool or other. Tooling is secured within a tool holder using a standard ER11 collet and collet nut.

First install the tool holder in the Pocket NC spindle collar and fasten with 3 M4 set screws. Do not tighten down one set screw completely without starting the other screws. Doing this may cause the tool holder to be pushed off center so that it will not spin concentrically with the spindle. Instead, start each of the set screws and screw in only until each screw starts to touch the tool holder. Then go back around and incrementally snug the set screws down.

Install the appropriate collet for your tool shank diameter into the collet nut. Push the collet in until it snaps behind the retaining ring. It should not fall back out of the collet. It will still be able to wobble some. The user may either install the tool in the collet now or partially install the collet nut onto the tool holder then insert the tool.



After the tool is installed in the collet, spin the collet nut onto the tool holder by hand. Make sure that it is not cross threaded. Adjust the tool stickout, how far the tool sticks out of the collet, to the desired length. Under no circumstances may the butt end of the tool **not** extend through the entire length of the collet. The tool will not be retained safely if the collet does not grip the shaft fully.

Tighten the collet nut with the provided 17mm spindle wrench. Use the provided 3mm hex driver in one of the spindle set screws to keep the spindle and holder from rotating with the collet nut.

When changing to a new tool holder simply loosen the three set screw in the spindle collar, give the tool holder a bump with the handle of the 3mm driver, and then pull the holder from the spindle collar.



V2-50 CHB

Pocket NC V2-50 CHB machines do not use tool holders or collet nuts. The CHB collet threads directly into the spindle and is clamped down on the cutting tool by a lever that draws it back into a taper.

Users can remove the CHB collet by positioning the lever to the open position, installing a cutting tool or dowel pin and then using the provided wrenches to unthread the collet from the spindle. A new collet can then be installed in the reverse order.

The cutting tool can be installed anytime a CHB collet has been fully installed and the spindle lever is in the open position. To clamp the tool into the spindle the spindle lever simply needs to be moved to the lock position.



V2-50 CHK

Pocket NC V2-50 CHK machines do not use a tool holder but do utilize a collet nut. The end of the spindle is threaded for a collet nut that accepts the CHK collet. Threading the nut onto the spindle and tightening it seats the collet, clamping down on the cutting tool.

Users can remove the CHK collet by simply unthreading the collet nut from the end of the spindle and then removing the collet from the spindle taper. This can be done using the provided wrenches to hold the spindle still. The CHK collet can be replaced by doing the reverse.

The cutting tool can be inserted into the collet anytime before the collet has been fully tightened down and then once the preferred tool stickout is set the collet nut should be tightened down completely.



Chapter 3.2.4: Tool Measurement

Before the tool can be used to cut material it must be measured so that the Tool Length Offset can be determined.

The tool length offset (TLO) is the distance between the Z home position and the machine origin when the tip of the tool is a specified distance from the B-table. Basically, this tells the Pocket



NC where the tool is in space so that the correct compensations can be made to the program in order to cut the part accurately and successfully.

The Pocket NC V2 is equipped with a tool probe that will automatically measure the tool length offset for most tools. However, some tools will still have to be measured by hand. This section will explain how to use the automatic tool probe as well as how to measure the tool length offset by hand.

With Tool Probe

Do not run the probe cycle for any tool with a diameter larger than .25” or that is sticking out more than 1.5 inches from the end of the extended tool holder. Doing so will cause a crash and may result in damage to your machine and/or a broken tool.

The tool table is found in the TOOL OFFSETS section of the SETUP page.

TOOL OFFSETS					
	TLO (in)	Diameter (in)	Flutes	Description	
 T 1	5.0000	0.1250	1		
 T 2	5.0000	0.1250	1		
 T 3	5.0000	0.1250	1		
 T 4	5.0000	0.1250	1		
 T 5	5.0000	0.1250	1		

The “TLO” column displays the tool length offset that is set for each tool number. All the tools are set to 5.000 inches by default. The “Diameter” column displays the tool diameter. Note that this is the diameter of the cutter not the diameter of the shank. All the tools are preset to 0.125 inches by default. The user can add a description of the tool in the “Description” column.

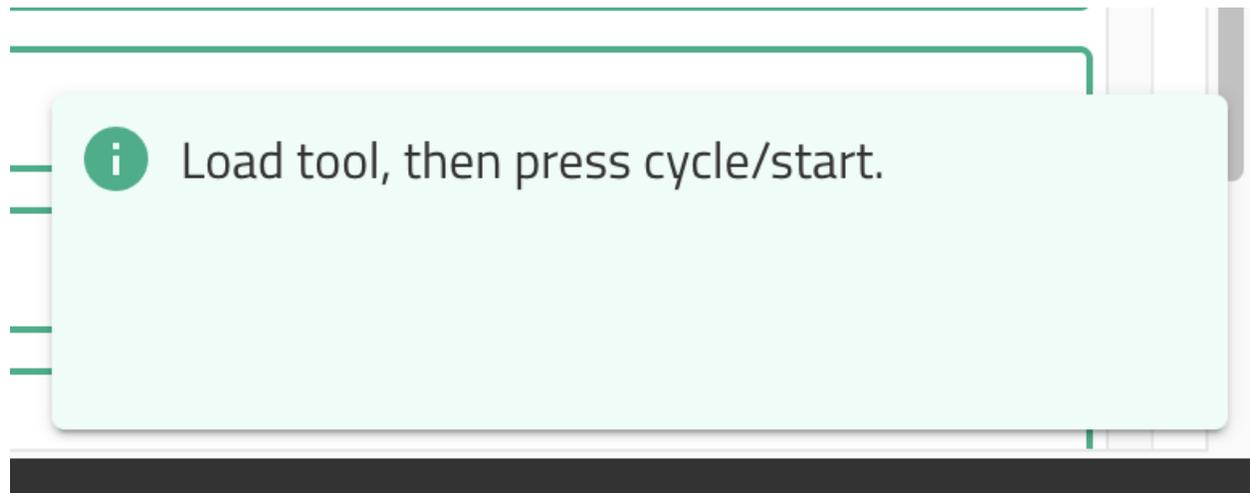
To measure the tool, begin by installing the tool you wish to measure into the spindle of the machine using the steps discussed in Chapter 3.2.3 above. Make sure the tool is securely fastened before moving on.

Click on the green tool probe button next to the tool that you wish to set the tool length offset.



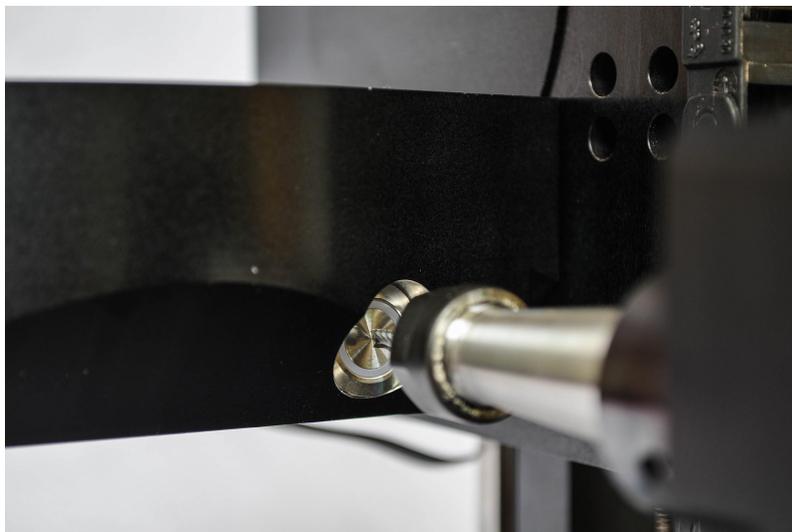


The machine will begin the tool probing routine by bringing the spindle to the front of the machine and pausing. The following prompt will appear in the bottom right corner of the screen.



Double check that your tool is installed correctly then push the green, blinking CYCLE/START button on the front of the machine to run the probe cycle.

The first move in the cycle is to move the Z axis as far back as possible to prevent a collision. Then the Y axis will raise up to give the Z axis access to the tool probe button, which is located under the B-table housing.



The tool/Z axis will advance until it triggers the tool probe button. It will then retract a short distance, then approach at a lower speed until the button triggers again.



After the cycle is finished, the machine will return to the position it was in when the cycle was started. Note that this could cause a collision with the part if you are swapping out a shorter tool for a longer tool in the middle of a gcode program. Adjust your g-code appropriately to avoid this type of collision by retracting to a safe distance before a tool change. The Pocket NC post processor for most CAM softwares will automatically send all axes to their home positions before prompting a tool change.

After the probe cycle has finished, the tool length offset will be updated in the tool table.

		TLO (in)
	T 1	-3.0823
	T 2	5.0000
	T 3	5.0000

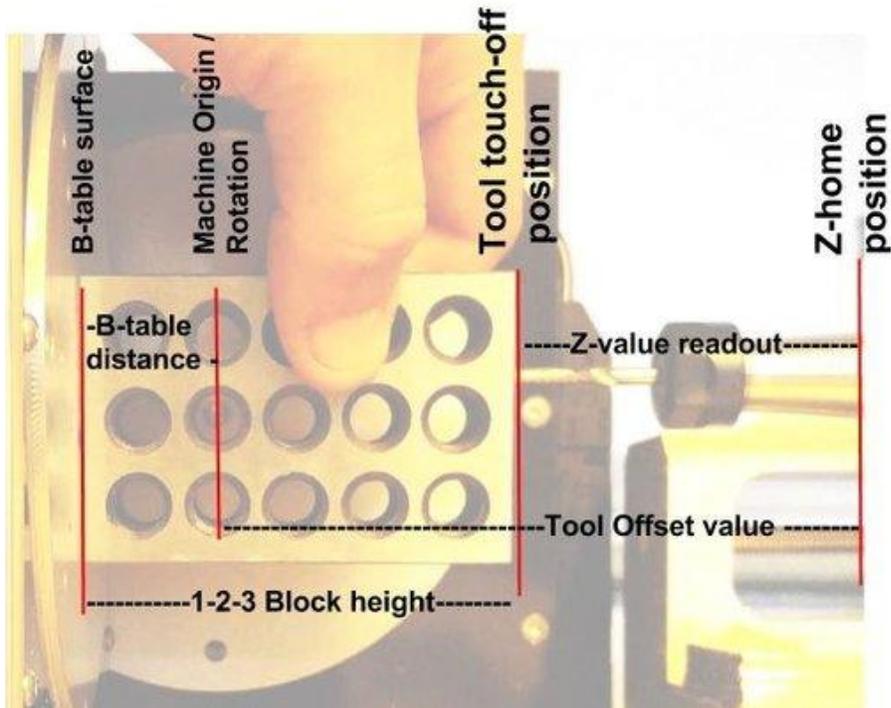
By Hand

If the automatic tool probe cannot be used for a tool, you can measure the tool length offset by hand.

You will need to know your machine's B-Table Offset. This is the distance from the B-Table to the center of A axis rotation. This number is around 0.8xx inches. This value is different for each machine and is included in the paperwork that comes with the machine. If you have misplaced yours, please email us at info@pocketnc.com and we can find it for you.



You will also need a 1-2-3 block. Any 1-2-3 block can be used for this procedure. They can usually be purchased on Amazon fairly inexpensively.



To perform the measurement process, perform the following steps:

1. Home all axes on the machine.
2. Clear any existing offsets by clicking on the “G54” and “G92” drop down buttons and selecting “Clear Current System” from the drop down list.



WORK OFFSETS

	DRO		Absolute Position				
X	-1.1952 in	=	-1.7000	-	-	0.0000	DRO
Y	2.5000 in	=	2.5000	-	0	0.0000	DRO
Z	0.0000 in	=	0.0000	-	0	0.0000	DRO
A	0.000 deg	=	0.000	-	0	0.000	DRO
B	0.000 deg	=	0.000	-	0	0.000	DRO

Zero DRO
Clear Current System
Use G54
Use G55
Use G56
Use G57
Use G58
Use G59
Use G59.1
Use G59.2

G92 v No Active Tool

3. Jog the A axis to 90 degrees by entering the command “G90 G0 A90” into the MDI command bar on the MANUAL page and pressing enter.

MDI

Enter MDI...

GO G90 A90

4. Clean off any debris on the B-table. Hold the 1-2-3 against the B-table with the long edge of the block parallel to the tool. Make sure that the 1-2-3 block is flush with the B-table and that you have it against the center part of the B-table (the raised part that rotates), not the B-table housing (the part that doesn't rotate).

5. Jog the Z in the negative direction until the tip of the tool is close enough to the B axis table that the 1-2-3 block will not fit between the two.

6. In increments of 0.01” or less, jog the Z axis positive until the 1-2-3 block slides between the tool and the table

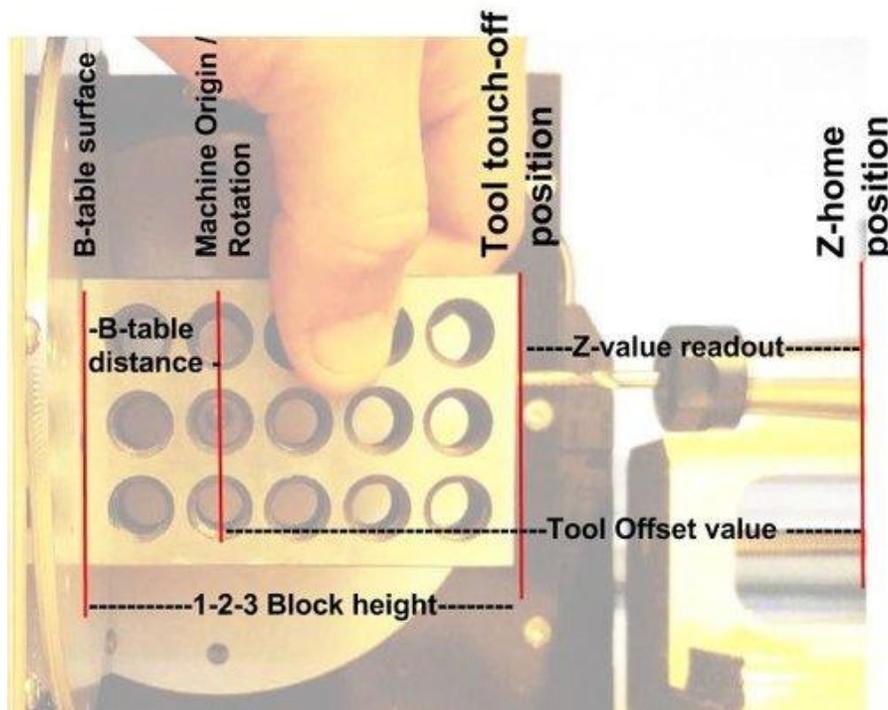
7. Repeat step 6 with smaller increments until the tool position is measured past 3 decimal places



8. Use the current Z position that is displayed in the DRO or JOG section to calculate the tool offset using the equation below:

$$-(1-2-3 \text{ block height}) + (\text{B-table Offset}) + (\text{Z value readout}) = (\text{Tool Length Offset})$$

Example: $-(3.000)+(0.8861)+(-0.9131)=(\text{Tool Offset}) = (-3.027)$



9. Click in the appropriate “TLO” box for the tool you wished to provide a tool length offset for. Enter the tool length offset that you calculated. Make sure that the sign is correct. Update the tool diameter if needed.

Chapter 3.3: Feeds and Speeds

The Pocket NC mill removes material using a tool with a series of small rotating cutting edges. A tool with 2 cutting edges will make two cuts for each revolution. The material that is removed is referred to as a chip. The thickness of a chip that the Pocket NC is capable of producing will vary depending on what material is being removed and what cutter is being used.

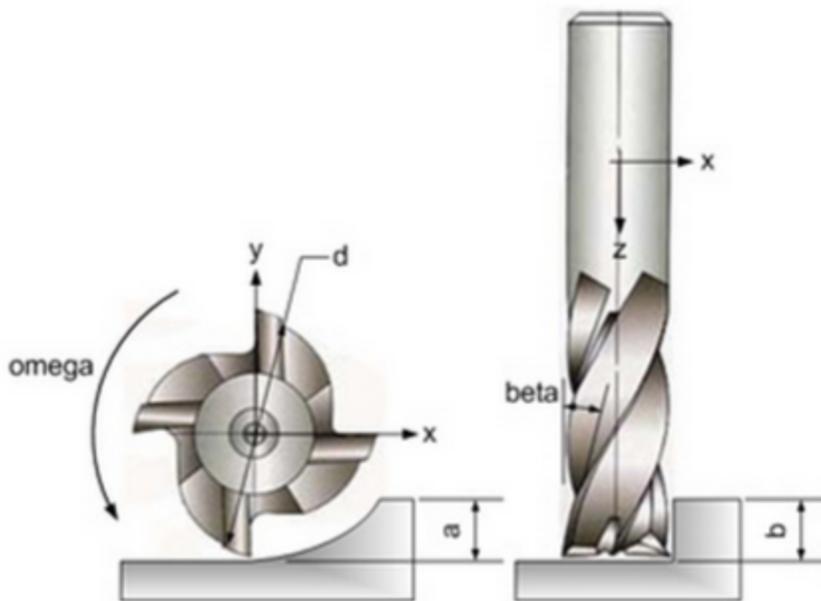
The thickness of a chip(also called the chip load) is adjusted using two variables, feed and speed. Feed refers to the linear rate in inches/minute or millimeters/minute that the cutting tool



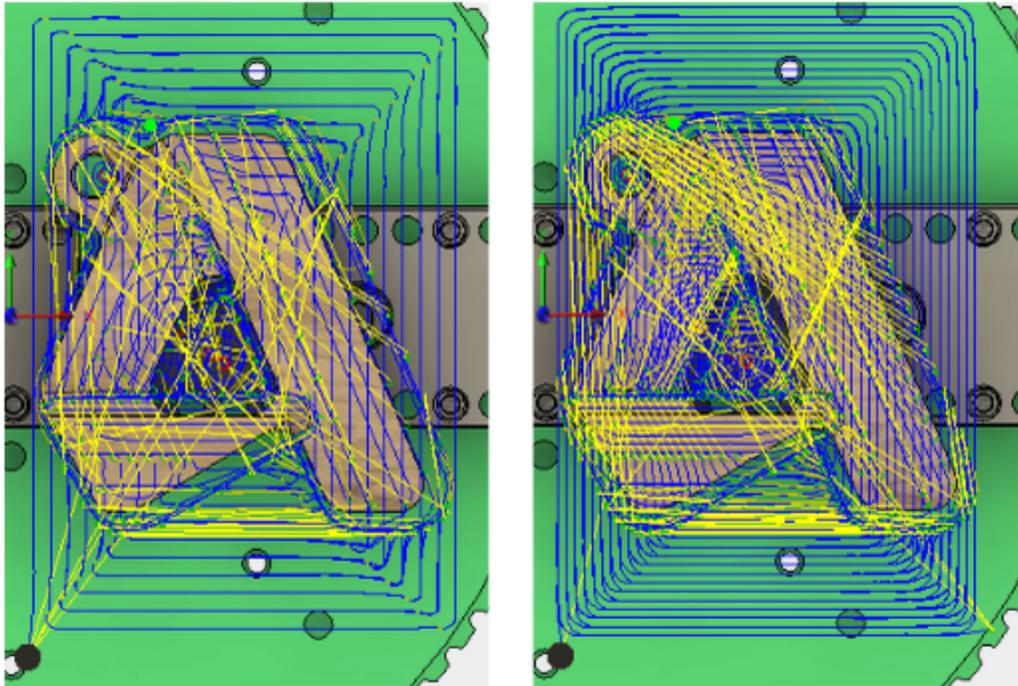
moves through a material. Speed refers to the rotational speed in RPM (revolutions per minute) of the cutting tool. To increase chip load, speed can be decreased or feed can be increased.

Every material has a different ideal chip load. There are many resources available in print and online that list suggested chip loads for many materials.

For example, assume that the ideal chip load for a given material is 0.001". This means, with a spindle speed of 10,000 RPM and a linear progression of 0.002" per revolution (2 cutting edges X 0.001") gives a feed of 20 inches per minute. It is important to keep in mind that the calculated chip load is only a guideline or starting point, the user should adjust the speed and feed to what sounds good and feels good. Feed and speed can be adjusted on the fly using the Pocket NC user interface.



Machining also has two more variables, side load and depth of cut. These two variables will change the width and length of the chip being removed from the material. Dimension "a" in the image above represents the amount of material removed with the cutting tool in the X/Y direction or side load. Dimension "b" in the image above represents the amount of material removed with the cutting tool in the Z direction, also known as the depth of cut.



Above, the image on the left shows a tool path with a side load of 0.08" and the image on the right shows a tool path with a side load of 0.04". If the material being removed was soft like wax or wood, the tool path on the left would be much more appropriate as the machine will be able to handle that amount of side load. If the tool path from the image on the right was used with the same material, the user would have spent twice as much time and worked the spindle half as hard.

Posted below and in the User Resources of the Pocket NC website is a table with general guidelines for the 4 variables discussed above, in a number of materials.



	Speed (RPM)	Feed per tooth (in)	Depth of Cut	Side Load
Copper	8,500	0.0005	90%	10%
Aluminum	8,500	0.0008	90%	10%
Hard Plastic	8,500	0.001	80%	50%
Soft Plastic	8,500	0.0015	70%	60%
Hard Wood	8,500	0.001	70%	60%
Soft Wood	10,000	0.002	50%	70%
Wax	10,000	0.003	50%	70%

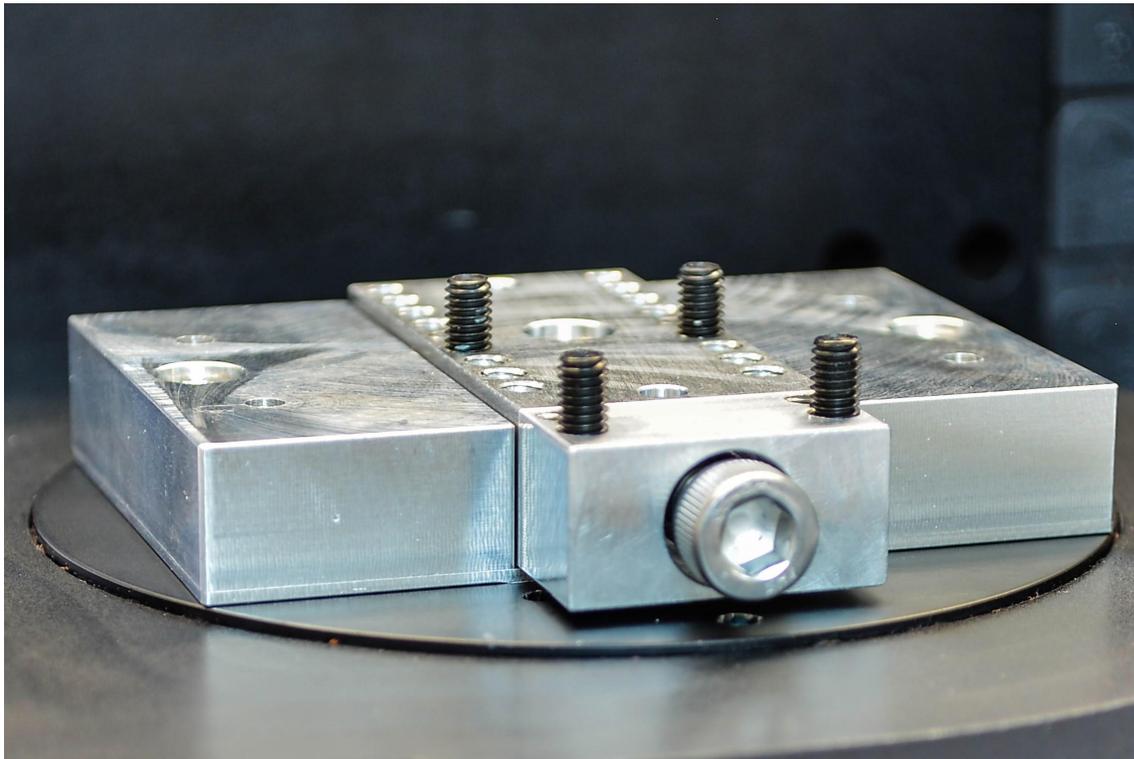
Note that side load and depth of cut will be expressed as a percentage of tool diameter and feed will be represented as feed/tooth. For example, the end mill pictured above has 4 cutting teeth. To calculate feed, use the following equation.

$$\text{Feed in IPM (inches per minute)} = (\text{feed in inches per tooth}) \times (\# \text{ of cutting teeth}) \times (\text{spindle speed in RPM})$$



Chapter 3.4: Fixturing

Chapter 3.4.1: Pocket NC Vise

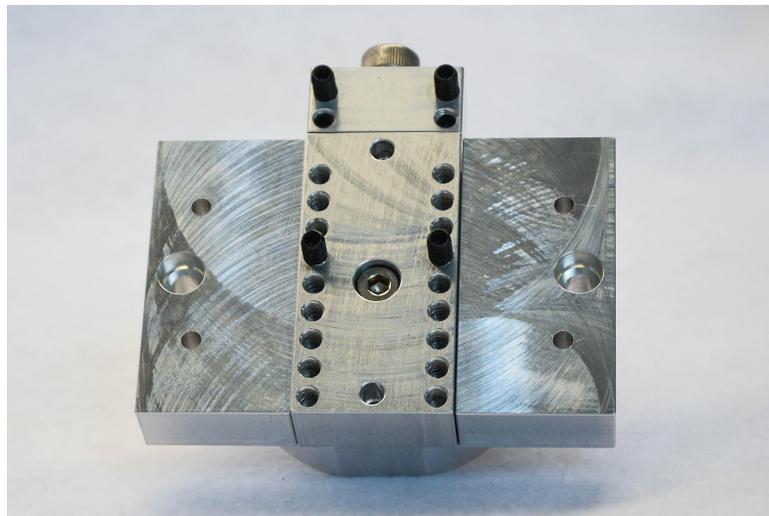
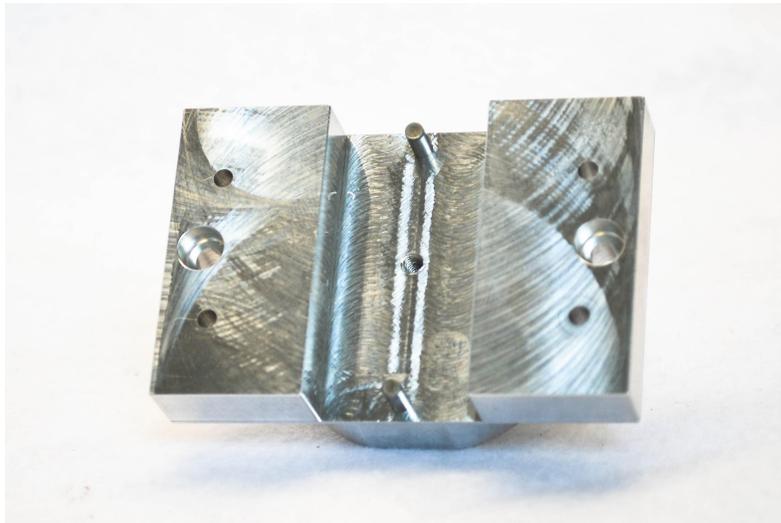


Operation

The Pocket NC vise was designed to be super adjustable and have excellent grip. While it does not look like a traditional vise, the function is the same. Dowel pins and/or set screws are used in place of jaws and are completely interchangeable. M4 set screws work like gripper jaws for roughing material and will work for a hardness of aluminum and anything softer. Dowel pins can be used for light finishing work. The pins can be adjusted in increments of 5 mm. The movable jaw has a travel of 6mm with 6+ mm of thread engagement.



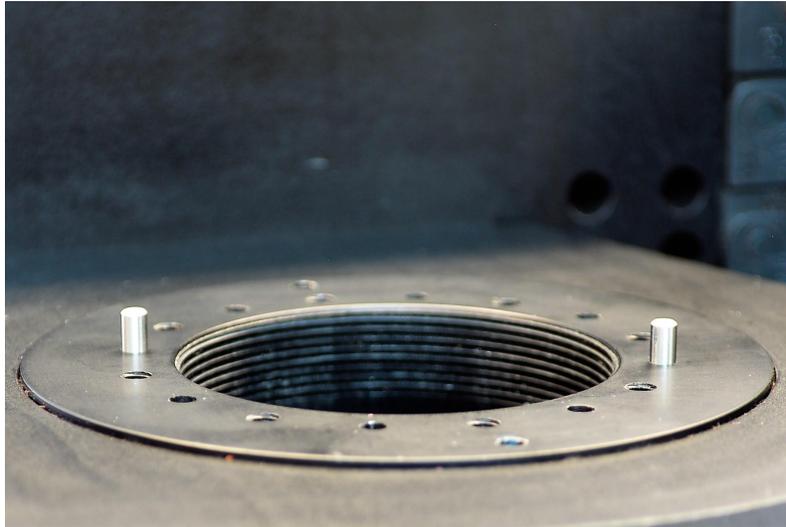
Mounting the vise to the PocketNC V2 mill requires an adapter to bridge the hole in the center of the B-table.



The adapter mounts onto the B-table of the PocketNC using two $\frac{1}{8}$ inch dowel pins and two M4 x 8 fasteners. The adapter can be installed in any of 8 possible positions on the B-Table.

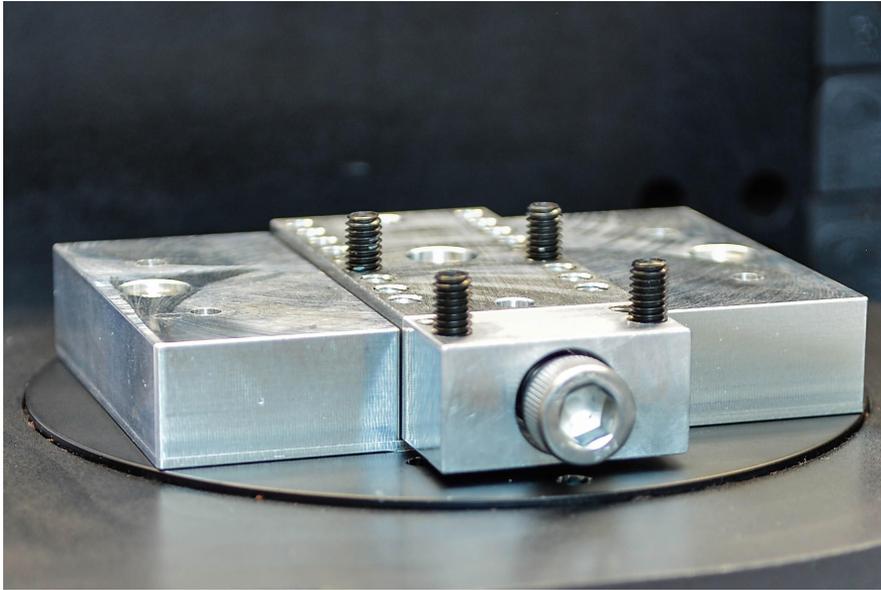
Installation

Place two $\frac{1}{8}$ inch dowel pins directly across from one another.

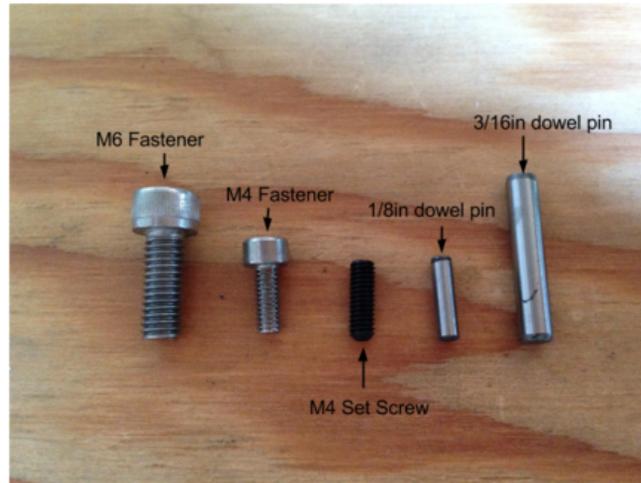


Place the adapter onto the B-table, lining up the holes in the adapter with the dowel pins. You may need to wiggle the adapter as you work it down into place. Make sure that you get the adapter all the way down so that it is flush on the B-table. Install the M4 x 8 fasteners to fix the adapter in place. Tighten the fasteners only until snug, do not overtighten. Overtightening will damage the threads on the B-table.

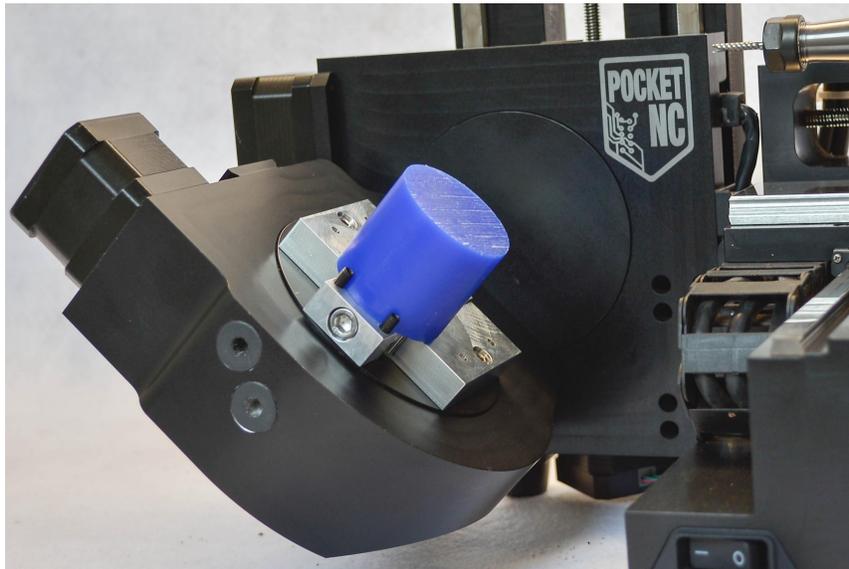
Make sure that the bottom of the vise and the bottom and sides of the adapter slot are clean of debris. Place two $\frac{1}{8}$ inch dowel pins into the dowel pin holes on the bottom of the fixed jaw of the vise. Place the vise into the adapter and wiggle it into place so the bottom of the vise is flush to the mating surface in the vise adapter. Install the M4 x 10 screw, tighten only until snug, do not overtighten. Overtightening will destroy the threads machined into the adapter. A good gauge of an appropriate level of tightness is to tighten the screw with a 3mm hex wrench only to the point that the wrench starts to flex.



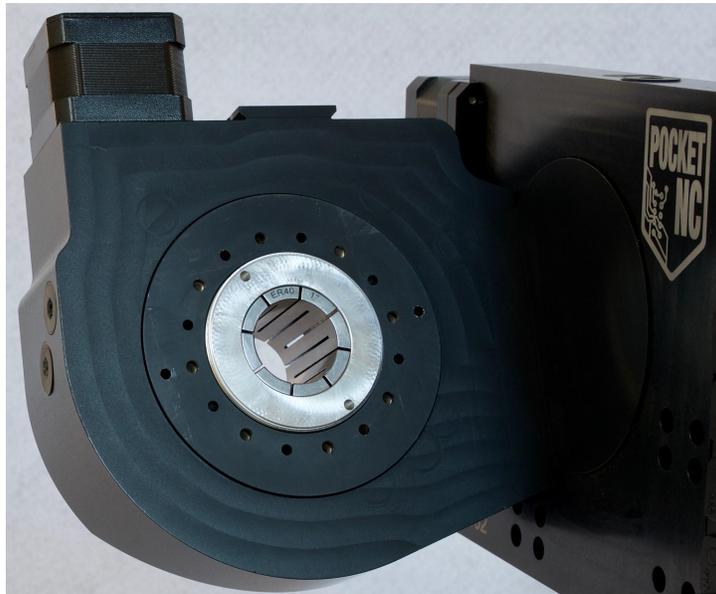
The Pocket NC vise uses 3 different types of off-the-shelf components, socket head cap screw, set screws and dowel pins. You will notice that the fasteners and set screws are metric and the dowel pins are english. We have two reasons for the mix and match. The most common part that will get lost is the $\frac{1}{8}$ inch dowel pins, they tend to roll off desks and slip out of hands but because they are english units, a replacement or even longer versions can be purchased at a local hardware store. The second reason for this mix, M4 fasteners have a minor diameter that is so close to $\frac{1}{8}$ inch that an M4 internal thread can be machined to also accommodate an $\frac{1}{8}$ inch dowel pin. This helps the user easily switch between roughing parts and finishing parts.



Below is a picture of round wax stock being held in the Pocket NC vise.

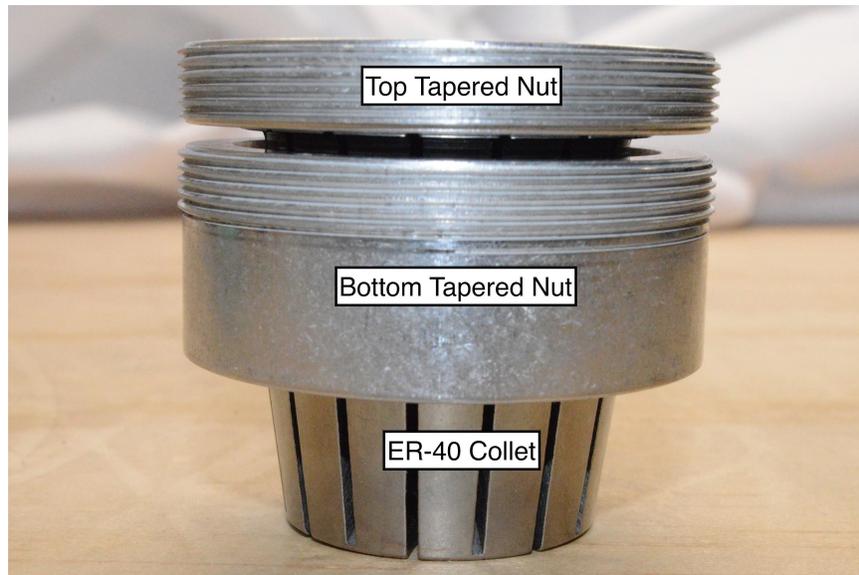


Chapter 3.4.2: ER-40 Collet Fixture



Another way that stock can be mounted on the Pocket NC V2 is using the ER-40 collet fixture. This is a particularly useful way to hold round stock, especially because the through-hole design allows the user to mount longer parts than could be held in the vise.

The ER-40 collet fixture consists of three parts, an ER-40 collet (available in a range of internal diameters), a bottom tapered nut, and a top tapered nut. They fit together as shown in the image below.



Installation

To install the ER-40 collet fixture and the stock it will hold, begin by threading the bottom nut into the B-table. Start the nut by hand to make sure that it is not cross-threaded, then use the provided spanner tool to tighten the bottom nut until it hits the end of the threads.

Next, insert the collet into the bottom tapered nut.

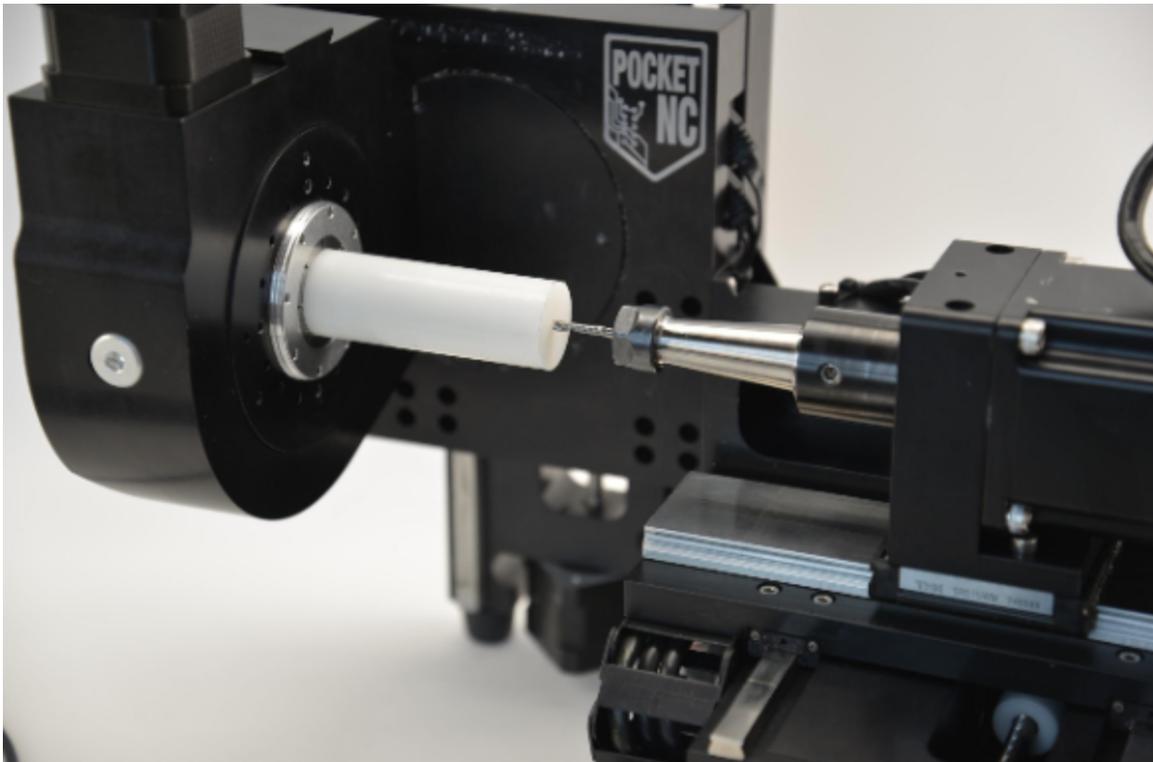
Finally, thread the top collet nut into the B-table by hand, just enough to get the threads started. Place the stock into the collet. Tighten the collet by hand until the stock is secure, then use the provided spanner tool to tighten the nut.

DO NOT OVERTIGHTEN! You will damage the B-table drive system if you use excessive force when tightening the collet nuts. The maximum torque that should be used to tighten the collet is



12 inch pounds (1.36 Nm). The tool provided by Pocket NC is the only recommended way to tighten these nuts. Do not use a standard spanner wrench.

Below is an example of stock being held in the Pocket NC using the ER-40 Collet Fixture.



Chapter 4: G and M Codes

See the following tables for the G and M Codes supported by Pocket NC.

For more information on G and M codes, go to these links:

<http://linuxcnc.org/docs/html/gcode.html> and <http://linuxcnc.org/docs/html/gcode/m-code.html>.

G code	Description	G code	Description
G00	Rapid positioning	G56	Work coordinate offset 3



G01	Linear interpolation	G57	Work coordinate offset 4
G02	Circular interpolation clockwise (CW)	G58	Work coordinate offset 5
G03	Circular interpolation counterclockwise (CW)	G59	Work coordinate offset 6
G04	Dwell - as a separate block only	G59.1	Work coordinate offset 7
G05	Cubic Spline	G59.2	Work coordinate offset 8
G05.1	Quadratic Spline	G59.3	Work coordinate offset 9
G05.2	NURBS Block	G61	Exact Path Mode
G05.3	NURBS Block	G61.1	Exact Stop Mode
G10 L1	Set Tool Table	G64	Path Bending
G10 L2	Set Coordinate System	G73	Drilling Cycle with Chip Breaking
G10 L10	Set Tool Table	G80	Fixed cycle cancel
G10 L11	Set Tool Table	G81	Drilling cycle
G10 L20	Set Coordinate System	G82	Spot-drilling cycle
G17	XY-plane designation	G83	Peck-drilling cycle (deep hole drilling cycle)
G18	ZX-plane designation	G85	Boring cycle, Feed Out
G19	YZ-plane designation	G86	Boring cycle, Spindle Stop, Rapid Move Out
G17.1	UV-plane designation	G89	Boring cycle, Dwell, Feed Out
G18.1	WU-plane designation	G90	Absolute dimensioning mode
G19.1	VW-plane designation	G91	Incremental dimensioning mode
G20	Imperial units of input	G90.1	Absolute Arc Distance Mode
G21	Metric units of input	G91.1	Incremental Arc Distance Mode
G28, G28.1	Go to Predefined Position	G92	Coordinate System Offset
G30, G30.1	Go to Predefined Position	G92.1	Reset Coordinate System Offsets
G40	Cutter radius compensation cancel	G92.2	Reset Coordinate System Offsets
G41	Cutter radius compensation - left	G92.3	Restore Axis Offsets
G42	Cutter radius compensation - right	G93	Feed Rate Mode, Inverse Time Mode
G43	Tool length offset	G94	Feed Rate Mode, Units Per Minute Mode
G43.1	Dynamic Tool Length Offset	G95	Feed Rate Mode, Units per Revolution Mode
G49	Cancel Tool Length Compensation	G96	Spindle Control Mode
G53	Machine coordinate system	G97	Spindle Control Mode
G54	Work coordinate offset 1	G98	Return to initial level in a fixed cycle



G55	Work coordinate offset 2	G99	Return to R-level in fixed cycle
-----	--------------------------	-----	----------------------------------

M Code	Description
M00	Program Pause
M01	Program Pause
M02	End of program(usually with reset, no rewind)
M03	Spindle rotation normal (CW for R/H tools)
M04	Spindle rotation reverse (CCW for R/H tools)
M05	Spindle stop
M06	Manual Tool Change
M30	Program end (always with reset and rewind)
M48	Feedrate override cancel OFF (deactivated)
M49	Feedrate override cancel ON (activated)
M50	Feed Override Control
M51	Spindle Speed Override Control
M52	Adaptive Feed Control
M53	Feed Stop Control
M60	Automatic pallet change (APC)
M61	Set Current Tool Number

Chapter 5: Maintenance, Storage, and Transport

Chapter 5.1 Maintenance

All of the electronics, motors and bearings on the Pocket NC are fairly well covered so debris on top of them should not affect functionality. Although, it is recommended that users regularly clean the machine using a vacuum cleaner with an extension and/or soft brush attachment to get into the crevices.



Wipe the machine with a soft, lint-free cloth to remove fine debris and dust. Isopropyl alcohol is recommended for removing oil or grease.

There are no parts that need oiling or greasing. Applying oil or grease to the lead screws or other components will cause debris to stick to them and may result in impaired function and increased wear.

Chapter 5.2: Storage

If you will be storing your Pocket NC for an extended period of time or corrosion is likely for other reasons, a small amount of light oil may be applied to the lead screws and other steel components in order to prevent rust.

Store the Pocket NC in a dry location, protect the wires from excessive UV exposure and avoid operating the machine immediately after large temperature changes that could cause condensation build up inside the machine. If possible store the Pocket NC in a watertight container or a plastic bag and use desiccant to prevent moisture buildup.

Chapter 5.3: Transport

If you will be transporting your Pocket NC, especially if it is shipped as freight, install the shipping bolts to help prevent damage to the axes in case of an impact during shipment. If possible, retain and reuse the original materials used to ship your Pocket NC. They have been carefully designed to protect the machine.



Chapter 6: Troubleshooting Common Problems

Our goal at Pocket NC is to produce an excellent product at an affordable price so that people can turn their ideas into reality. We hope that using our product is a straightforward process and the machine functions perfectly. But, we know that things don't always go perfectly. Hopefully this troubleshooting guide can help you resolve some of the problems that come up.

If your problem is not covered here, or if you need some more information, please [contact us](#).



Chapter 6.1: Common Problems

Problem	Possible Solutions
<p>I can't connect to my Pocket NC or the user interface is not loading correctly on my computer.</p>	<ol style="list-style-type: none"> 1. Check that the Pocket NC is plugged in and is powered on. The red LED on the E-stop button will light up when the machine is on. The user interface is ready to boot when the red LED starts blinking. It can take up to 2 and half minutes to fully boot the Pocket NC. 2. The computer must be connected to the mini-USB port on the mill, not the standard USB port. 3. The mini USB may not be fully plugged into the machine. The pocket is deep so the plastic around the plug may have to be trimmed to fit. 4. The machine may not be fully booted. Wait 1 minute after turning on the machine before trying to connect. 5. The user interface may not load properly in every browser. Try using Chrome, Firefox, or Safari 6. Try rebooting the machine: power off the machine, unplug all the cords, wait about a minute for the capacitors to drain then plug it back in and restart. 7. Contact Pocket NC at service@pocketnc.com
<p>The Pocket NC is unresponsive, nothing happens when I tell the Pocket NC to move.</p>	<ol style="list-style-type: none"> 1. Check that the Pocket NC is plugged in and that the power switch is on (the - symbol on the power switch). The Pocket NC's electronics can run off of computer power, but the motors can't. 2. Make sure that the Pocket NC is not in E-stop. When the machine is in E-stop the red button on the front of the machine will blink and the button in the center of the status bar at the bottom of the user interface will be red and read E-STOP.



	<p>Push the E-stop on button on the front of the machine or click the E-STOP button in the UI to enable the motors.</p> <ol style="list-style-type: none"> 3. Check that the computer is still connected to the Pocket NC with the USB cord. 4. Try rebooting the machine: power off the machine, unplug all the cords, wait about a minute for the capacitors to drain then plug it back in and restart. 5. Contact Pocket NC at service@pocketnc.com
<p>One of the axes on my Pocket NC is not working</p>	<ol style="list-style-type: none"> 1. Try rebooting the machine: power off the machine, unplug all the cords, wait about a minute for the capacitors to drain then plug it back in and restart. 2. Check the electrical connections for that axis. Look for damage and debris. 3. Contact Pocket NC at service@pocketnc.com
<p>The Pocket NC motors are noisy when the machine is running</p>	<p>Particularly at low feed rates, the stepper motors can produce quite a bit of noise. This is due to the electrical interference in the driver circuit.</p> <p>Increasing the feed rate usually decreases the sound.</p>
<p>The Pocket NC is not cutting in the location that I expected.</p>	<ol style="list-style-type: none"> 1. Check if the coordinate system of your machining setup in CAM is set up so that the positive Z axis points at the spindle of the Pocket NC, the positive X axis points at the A table, and the positive Y axis points straight up from the B table. 2. Check that your work offsets are set correctly relative to your workpiece.
<p>I can't find/load my G-code file</p>	<ol style="list-style-type: none"> 1. Check if the selection set for the file type includes your file extension (file type) for example you can't find machining files with a .nc extension if the selection set is looking for only .ngc files 2. The file must be a text file that the Pocket NC is



	<p>capable of reading. Generally these will have an extension like .ngc, .nc, or .cnc but any machinable file should work.</p>
--	--



Chapter 6.2: Machining/Performance Issues

Problem	Explanation	Possible Solution/Things to Try
My tool chatters during the cut.	Vibration causes the cutter to not cut smoothly. This vibration is due to the natural frequency of the tool/spindle.	<ol style="list-style-type: none"> 1. The feed and speed, and/or step-down and step-over need to be adjusted. See the feeds and speeds tutorial for more information. 2. Turn the spindle speed up. 3. Adjust the feedrate, usually moving slower will be smoother, but this is not always the case 4. Increase the step-down, taking a deeper cut into the material 5. Increase or decrease the step-over, the width of the cut. Typically the ideal step-over is about 60% of tool diameter.
Spindle sounds like it is slowing down during the cut.	The spindle power is insufficient for the amount of material being removed.	<ol style="list-style-type: none"> 1. Decrease the feed rate in 5% steps until the spindle stops bogging down 2. Decrease the step-down and step over in your toolpaths. 3. Switch to a tool with more cutting flutes.
The drive/lead screws squeak.	The end of the lead screw is touching the frame of the machine when it turns.	<ol style="list-style-type: none"> 1. Make sure that the screw and the hole are free of debris. 2. If the noise is minor it is



		<p>best to let it wear into position. The noise should decrease with time</p> <ul style="list-style-type: none">- Use a very small amount of general purpose synthetic grease to lubricate the tip of the screw.- Do not grease the whole screw.- Do not put grease where it will contact the anti-backlash nuts.- Grease will collect debris and may cause increased wear on your machine.
--	--	--



Chapter 6.3: Error Codes

Error Code/Message	Explanation	Possible Solutions
<p>Near Line XX Linear Move Exceeds Joint X's Positive/Negative Limit</p>	<p>This error means that the toolpath/G code is telling the machine to go past one its travel limits</p> <p>Joint 0 = X axis Joint 1 = Y axis Joint 2 = Z axis Joint 3 = A axis Joint 4 = B axis</p> <p>This error is often caused by a tool length offset being set incorrectly or by a part that is outside the Pocket NC's work envelope.</p> <p>The move that is causing the error is located on or near the line of the program indicated</p>	<ol style="list-style-type: none"> 1. If the error is on joint 2 (Z axis), check if your tool length offset is set correctly. See chapter 3.2.4 for information on tool setup. 2. If the error is for exceeding the negative limit of joint 2 and the tool length offset is correct, increase the tool stickout or switch to a longer tool holder. 3. If the error is for exceeding the positive limit of joint 2 and the tool length offset is correct, decrease the tool stickout or switch to a shorter tool holder. 4. Check if your toolpath is asking the machine to exceed its limits of travel. Open your code in a text editing program and use the search bar (CTRL + F) to search for any values that are greater than the limits of travel for the Pocket NC mill. <ol style="list-style-type: none"> a. The X limits are -2.00 to 2.55 inches



		<ul style="list-style-type: none"> b. The Y limits are -2.4 to 2.6 inches c. The Z limits are -3.45 to 0.10 inches* d. The A limits are -25 to 135 degrees e. The B limits are -9999 to 9999 degrees <p>*you have to add the tool length offset to the Z axis value</p> <ul style="list-style-type: none"> 5. Adjust the location of the part/stock so that it is within the Pocket NC's work envelope 6. Decrease the retract height on the toolpaths if the part is close to the limits of the Pocket NC's travel. 7. Oversized parts may be able to be fixtured and cut one side at a time instead of all in one program.
<p>14 Unexpected Real Time Delay on RT Thread 1</p>	<p>This error means that there has been a delay in the Beaglebones processor.</p> <p>It does not usually cause any issues with machine operation, but can in some rare cases.</p>	<ul style="list-style-type: none"> 1. This error can be caused by a temporarily dropped connection between the computer and the Pocket NC mill or by a program that is so large and complicated that the Beaglebone's processor cannot keep up. 2. Check that the



		<p>connection between the computer and the machine is good.</p> <p>3. Try slowing down the feed rate or reducing the file size of the G code file by breaking it into more than 1 program or by loosening the tolerances in the toolpath and reposting it.</p>
--	--	--