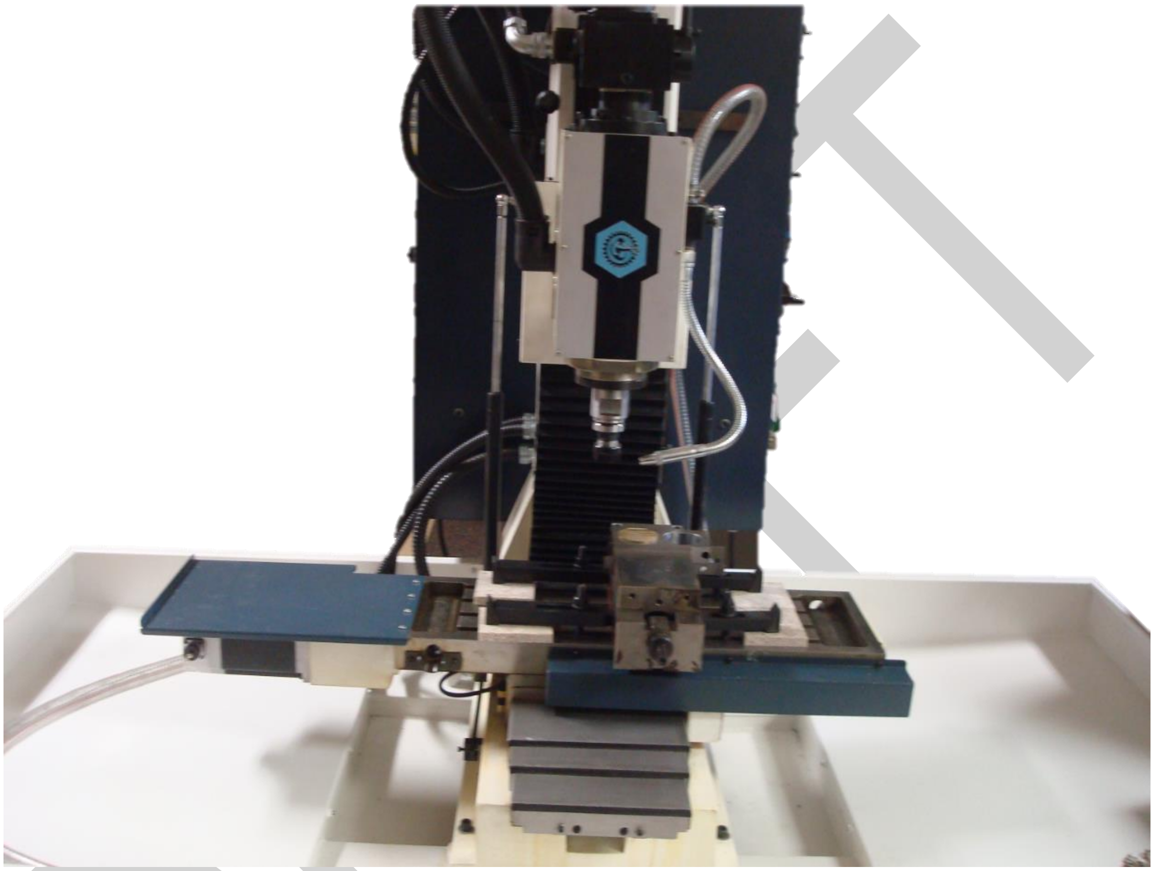
Novakon Pulsar Operator’s Manual



Novakon International Corporation

Sales: 905-258-0366
Tech: 905-258-0566
FAX: 905-258-0633
www.novakon.net
sales@novakon.net
support@novakon.net

29 Skagway Avenue
Scarborough, ON M1M 3T9

Revision 1.00

4/7/2014

THANK YOU FOR YOUR ORDER

Congratulations on your purchase of a Novakon Pulsar CNC Mill. With proper setup and maintenance, your machine should provide many years of quality work and enjoyment.

This manual covers general instructions regarding machine set-up, operation, maintenance and troubleshooting for the Novakon Pulsar CNC Mill.

IMPORTANT

Carefully read this instruction manual and any accompanying instruction manuals before operating your Novakon CNC Mill. Instruction manuals should be kept in a safe place where they are always easily accessible for reference while operating the Pulsar CNC Mill.

While this manual has been compiled to give detailed description and usage of the Novakon Pulsar CNC Mill, changes are possible due to continuous design and development efforts.

Remember safety comes above all else. Carefully read, follow and understand the safety information outline in chapter 2 of this manual and always let common sense be your guide.

SUGGESTIONS / COMMENTS

We are interested in any suggestions and comments you might have to improve our products, Operator's Manual and services. Feel free to contact Novakon International Corporation with your suggestions and comments by e-mail to sales@novakon.net.

Check the www.novakon.net website periodically for the latest updates and revisions to this manual.

While every precaution has been taken in the preparation of this manual, Novakon International Corporation shall not have any liability to any person or entity with respect to any loss or damage caused or alleged to be caused directly or indirectly by the instructions contained in this manual or by the computer software and hardware products described in it. The information provided is on an "as is" basis.

Windows 7, Windows XP and Windows 2000 are registered trademarks of Microsoft Corporation. Mach3 is a registered trademark of ArtSoft Corporation. If other trademarks are used in this manual but not acknowledged, please notify Novakon International Corporation so this can be remedied.

Table of Contents

1 CUSTOMER INFORMATION 1-1

2 SAFETY GUIDELINES 2-1

OVERVIEW 2-1
GENERAL GUIDELINES FOR SAFE OPERATION 2-2
SAFETY CHECK PRIOR TO APPLYING POWER TO THE MILL 2-2
ELECTRICAL SAFETY 2-3
BASIC POINTS OF SAFETY 2-3
CLOTHING AND SAFETY 2-4
SAFETY ASPECTS related to TOOLs, FIXTURES, etc. 2-4
SAFETY ASPECTS RELATED TO TOOLING 2-5
SAFETY ASPECTS RELATED TO MAINTENANCE 2-5

3 INSTALLATION AND SETUP 3-1

RECOMMENDED TOOLS FOR ASSEMBLY 3-1
MACHINE PARTS 3-2
RECEIVING THE PULSAR CRATE 3-3
REMOVING THE MILL FROM ITS SHIPPING CRATE 3-4
SET-UP CLEARANCES & CONDITIONS 3-8
POWER REQUIREMENTS 3-9
INSTALLING THE COOLING TRAY 3-9
LEVELLING YOUR MACHINE 3-11

4 COMPONENT IDENTIFICATION 4-1

5 MACHINE CONTROL (Mach3) 5-1

SCREENS 5-1
INTRODUCTION TO CNC SYSTEMS 5-2
COMPONENTS OF A CNC MACHINING SYSTEM 5-2
HOW MACH3 FITS IN 5-3
STARTING MACH3 5-4
TYPES OF OBJECTS ON SCREENS 5-6
USING BUTTONS AND SHORTCUTS 5-6
DATA ENTRY TO DRO 5-7
JOGGING 5-7
MANUAL DATA INPUT (MDI) 5-9
TEACHING 5-10
WIZARDS – CAM WITHOUT A DEDICATED CAM SOFTWARE 5-12

RUNNING A G-CODE PROGRAM 5-15
TOOL PATH DISPLAY 5-15
HARDWARE REQUIREMENTS 5-17
SPINDLE AND COOLANT CONTROL 5-20
CONFIGURING MACH3 FOR YOUR MACHINE AND DRIVES 5-20
DEFINING THE SETUP UNITS 5-31
CHOOSING AN ACCELERATION VALUE 5-33
OTHER CONFIGURATION 5-38
MACH3 CONTROLS AND RUNNING A PART PROGRAM 5-49
FEED CONTROL FAMILY 5-55
PROGRAM RUNNING CONTROL FAMILY 5-56
WORK OFFSET AND TOOL TABLE CONTROL FAMILY 5-59
USING WIZARDS 5-64
LOADING A G-CODE PART PROGRAM 5-66
EDITING A PART PROGRAM 5-67
MANUAL PREPARATION AND RUNNING A PART PROGRAM 5-67
COORDINATE SYSTEMS, TOOL TABLE AND FIXTURES 5-69
MACHINE COORDINATE SYSTEM 5-69
WORK OFFSETS 5-71
DRAWING LOTS OF COPIES - FIXTURES 5-75
PRACTICALITIES OF "TOUCHING" 5-76
G52 & G92 OFFSETS 5-77
TOOL DIAMETER 5-80
CUTTER COMPENSATION 5-80
MACH 2 G AND M-CODE LANGUAGE REFERENCE 5-83
INTERPRETER INTERACTION WITH CONTROLS 5-88
TOOL FILE 5-88
THE LANGUAGE OF PART PROGRAMS 5-88
FORMAT OF A LINE 5-91
MODAL GROUPS 5-98
G CODES 5-100
BUILT-IN M CODES 5-125
MACRO M-CODES 5-128
OTHER INPUT CODES 5-128
ERROR HANDLING 5-129
ORDER OF EXECUTION 5-130

6 RIGID TAPPING 6-1

7 GLOSSARY 7-1

8 MAINTENANCE 8-1
LUBRICATION AND DAILY MAINTENANCE SCHEDULE 8-1
LUBRICATION SYSTEM 8-3
PERIODICAL MAINTENANCE TASKS 8-4
9 LEVER DRAW BAR (LDB) 9-1
10 TOOL HOLDING 10-1
OVERVIEW 10-1
USING THE LDB 10-1
TOOL HOLDERS 10-1
11 WARRANTY 11-1
NOVAKON WARRANTY 11-1
12 PULSAR Mill SPECIFICATIONS 12-1
PULSAR MILL DIMENSIONS 12-3
AC SPINDLE MOTOR 12-4
OPTIONAL X, Y & Z-AXIS AC SERVO MOTORS (MODEL 90ST-M04025) 12-5
SERVO CONTROLER 12-9
SPINDLE SERVO CONTROLLER PARAMETER SETTINGS 12-10
FOURTH AXIS Dimensions (Six Inch) 12-11
FOURTH AXIS STEPPER MOTOR 12-12
13 WIRING SCHEMATICS AND MACHINE PARTS 13-1
OVERVIEW 13-1
ELECTRICAL CONTROL PANEL 13-2
OPTIONAL FOURTH AXIS 13-3
NOVAKON BOB Rev 0 13-4
PULSAR STEPPER WIRING DIAGRAM 13-1
HEAD ASSEMBLY 13-2
LEVER DRAW BAR ASSEMBLY 13-3
HEAD AND SPINDLE ASSEMBLY 13-5
UPPER HEAD ASSEMBLY 13-6
LOWER HEAD ASSEMBLY 13-7
SERVO X-AXIS ENCODER CABLE ASSEMBLY 13-8
COOLANT NOZZLE ASSEMBLY 13-9

1 CUSTOMER INFORMATION

Please record the information below about your Novakon Pulsar CNC Mill. Having this information readily available will save time if you need to contact Novakon for questions, service, accessories, or replacement parts.

____

Pulsar Serial Number:

____

Purchase Date:

____

Delivery Date:

Upgraded Items (Check all that were included at the time of Purchase)

- ☐ Built in Computer
- ☐ Fourth Axis Stepper Drive Module
- ☐ Six Inch Rotary Table
- ☐ Extended Warranty
- ☐ Lever Draw Bar
- ☐ Automatic Tool Changer
- ☐ Miscellaneous Tooling

____

____

____

We look forward to a long working relationship with you, and thank you again for putting your trust in Novakon International Corporation.

2 SAFETY GUIDELINES

OVERVIEW

1) This manual describes general operational techniques and safety procedures. The Pulsar includes various safety devices to protect the operator and the machine. However, these cannot cover all aspects of safety. Therefore, the operator must thoroughly read and understand the content of this manual before operating the machine. The operator should also take into consideration these and other aspects of safety related to his/her particular environmental conditions, materials and tools.

2) Operating the Pulsar in accordance with the manufacturer's instructions, will provide you with reliable service. However, with machines of this nature, serious accidents can occur due to improper or careless operation. It is mandatory that you read this manual and other documentation to become thoroughly familiar with CNC machines prior to operating the Novakon Pulsar. Machine operations which are not documented in this manual, should be considered potentially dangerous. Do not perform undocumented machine operations before consulting Novakon for advice.

3) Novakon emphasizes that it is the sole responsibility of the operator to perform all operations using the Novakon Pulsar Mill in a safe manner.

4) This manual tries to give you guidance on safety precautions/techniques in using the Novakon Pulsar Mill. We accept no responsibility for the performance of this machine or any damage or injury caused by its use. In other words, it is your responsibility to ensure that you understand the implications of what you design and build, and to comply with any legislation and codes of practice applicable to your country or state.

GENERAL GUIDELINES FOR SAFE OPERATION

1) This machine has various mechanical and electrical safety devices to protect the operator and the machine. The safety devices include interlock devices and emergency stop switches. However, all machine tools are potentially dangerous, and computer controlled machines are potentially more dangerous because CNC machines start, stop and move automatically. Therefore, it is extremely important that you are aware of the machine's moving parts, chips projectiles and fluid while operating the machine.

2) Neither the manufacturer nor its representative or dealers can assume responsibility for any mishaps, damage or personal injury, which may occur because of improper operation or from failure to observe the safety precautions mentioned in this manual.

3) Do not under any circumstances attempt to operate this machine prior to reading and understanding this manual. Neglecting these instructions and warnings can cause serious injury to you and/or damage to the machine.

4) Familiarize yourself with the position of the movable EMERGENCY STOP BUTTON on the machine so that you can press it immediately from any position in case of an emergency.

5) Use extreme care when engaging fellow workers in conversations and running the machine at the same time. Do not proceed to the next step without informing the other personnel that you are about to do so.

6) Warm up the spindle and axis motion before running the machine in automatic mode.

SAFETY CHECK PRIOR TO APPLYING POWER TO THE MILL

1) Learn the control functions of the machine before operating it.

2) Make sure that all safety covers are fitted and electrical boxes are closed and secured before the power switches are turned ON.

3) Check to make sure that the cutting tool will clear the table, fixture, vise and clamps.

4) Make sure to anchor all items placed on the machine's table before starting the machine.

5) Learn to use the correct spindle speed, feed and depth of cut suitable for the work piece and material. Do not operate the spindle above the rated speed of the accessories mounted in it. Replace worn tools prior to a milling operation. Make sure that the tool length to diameter ratio is proper to prevent chatter. Make sure the Lever Draw Bar (LDB) is properly adjusted and the tool holders are tightened properly before actual cutting operations.

ELECTRICAL SAFETY

1) There are high voltage terminals in the electrical control panel, motors, junction boxes and other equipment. When the power supply is on, take extreme care to avoid contact with these components. After the power has been switched off, high voltage remains in various electrical components. Prior to touching any component, carefully check for voltage with a multi-meter or equivalent instrument to make sure that any residual voltage has dissipated.

2) Check all electrical cables for damage prior to applying electrical power to the machine.

3) Always disconnect the main plug from the electrical source and/or turn OFF the circuit breaker when the machine is not in use.

4) Shut down Mach3 and turn off the computer prior to switching off the power to the Novakon CNC Mill.

BASIC POINTS OF SAFETY

1) To prevent incorrect operation of the machine, carefully check the position of switches before operation. If in doubt, consult this manual or a Novakon technician for advice.

2) Always position light sources to shine away from the operator’s eyes.

3) Do not use compressed air to blow chips away from the spindle, parts, the machine or the floor around the machine.

4) All work platforms used around the machine should be sturdy, safe and include anti-slip surfaces.

5) Always be mentally alert, well rested, sober, and never under the influence of drugs that can affect the safe operation of the machine. Do not operate the machine if you suffer from dizziness.

6) Avoid unnecessary touching of the operator controls while the machine is running.

7) Keep the area around the machine free of oil/coolant, chips, debris and other obstructions.

8) Remove chips as often as necessary to prevent them from over accumulating in the machine.

9) Use an exhaust fan to control smoke and toxic fumes generated during machining operations. Always wear a protective mask when machining items that can create a toxic atmosphere.

10) Do not attempt to measure the work piece while the machine is running.

CLOTHING AND SAFETY

1) Always keep safety in mind. Wear eye protection at all times. Do not wear long sleeve shirts, loose or baggy clothes, neckties, wristwatches, rings, jewelry, etc., when operating this machine. Tie back long hair to prevent entangling with rotary tools.

2) Do not operate the machine if any machine guards, interlocks and other safety devices have been removed or any of these safety items are not functioning correctly. Never run the machine with the electrical cabinet open.

3) Do not use gloves when typing on the computer keyboard.

4) Do not handle chips, cutters and coolant with bare hands.

SAFETY ASPECTS RELATED TO TOOLS, FIXTURES, ETC.

1) Take time to properly secure fixtures, work piece and tools.

2) Let the machine and spindle come to a complete stop before accessing the machine, parts, tools or spindle.

3) Use the legs not the back for lifting. Use a hoist or other lifting device to move heavy items.

4) Use proper tools for the job.

5) Always use gloves when loading or unloading work pieces. Use the proper tools and/or the wash down hose when removing chips from the work area to protect your hands from sharp chips and burns generated during machining operations..

6) Do not remove chips when the machine is in operation. Lockout the machine by initiating the Mach3 E-Stop button before removing chips, fixtures, parts and cutting tools. Stop all machine operations before cleaning the machine or any of the peripheral equipment. The movable manual E-Stop Button is a true emergency stop button and shoud not be used in place of the Mach3 E-Stop button.

7) Always use proper cutting tools and work holding clamps suitable for the work and within the specifications of the machine. Do not exceed the machine table rating of 500 pounds. The maximum weight rating includes all items placed on the table, including fixtures, vises, clamps, parts, etc.

SAFETY ASPECTS RELATED TO TOOLING

1) Tools and miscellaneous equipment should be kept away from the moving parts of the machine.

2) Exercise caution when using fixtures, vises and parts that extend beyond the work table. These items could interfere with other machine parts or the machine enclosure.

SAFETY ASPECTS RELATED TO MAINTENANCE

1) If any components or safety covers are to be removed, first switch off the electrical circuit breaker or disconnect the main electrical plug.

2) Only qualified personnel should use and/or perform maintenance on the machine. The operator and programmer should be thoroughly familiar with the machine.

3) Keep the machine well lubricated and clean.

4) Do not modify the machine in any way that will affect safety.

5) Do not paint, soil, damage, modify or remove any of the safety nameplates. If the detail becomes illegible or if the nameplate is lost, obtain a replacement from Novakon and mount it in the original position.

6) In case of machine crash, do not operate it again until the cause and any damage have been evaluated and corrected.

7) Do not remove or adjust switches to increase axis travel beyond the machine specifications.

3 INSTALLATION AND SETUP

RECOMMENDED TOOLS FOR ASSEMBLY

Tin Snips

Philips Head Screw Driver

Ratchet Set

Caulking Gun

Clear or White Silicone Based Caulk

Machine Level

Fork Lift / Engine Hoist / Pallet Jack

Screw Clamps

Heavy Duty Chain

Chain Rings With Nut Locks

220 VAC Plug to suit your wall socket if Mill is not wired directly into electrical junction box

Safety Goggles

Safety Gloves

Extra Helpers

The Novakon Pulsar CNC Mill is shipped mostly assembled. Final assembly consists of installing four sheet metal coolant tray parts using twenty-two screws and four nuts, and installing the four adjustable cabinet support pads. The parts kit includes an extra screw and nut.

Figure 3-3

Figure 3-2

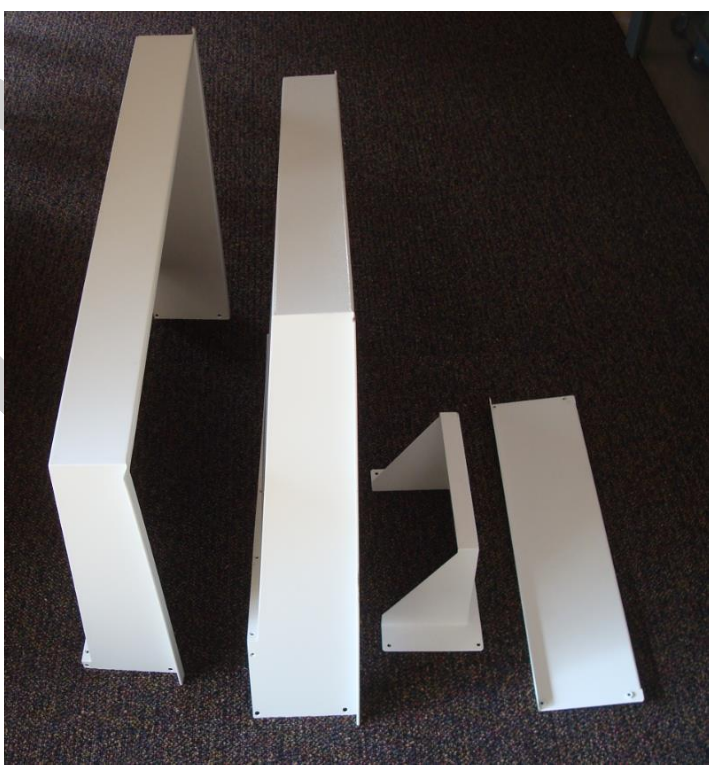


Figure 3-1

A forklift, loading dock or tail gate delivery is required when receiving the Pulsar crate from the trucking company.

Figure 3-4

1) A forklift provides the most desirable way to unload the Pulsar crate from the delivery truck as well as a means to lift the CNC Mill during the assembly process.

2) If you do not have access to a forklift and will be using an engine hoist to lift the Pulsar, it is suggested that the Pulsar crate be set on blocks by the delivery company. This will allow the engine hoist to slide under the skid on which the Pulsar is mounted. Figure 3-5 shows that the Pulsar crate would require a fork truck to lift as an engine hoist could not be slid under the pallet.

Figure 3-5

REMOVING THE MILL FROM ITS SHIPPING CRATE

1) Move the large crate into an area approximately 5 yards squared and with a minimum 85” height clearance.

2) The crate is held together by metal straps, nails and screws. Use tin snips to cut the metal band securing the crate. Remember the bands are under tension. Wear proper eye protection and gloves when cutting the bands.

3) Locate the side panel that is secured with screws. This panel allows access to the front of the Pulsar CNC machine. Remove all screws along the sides, top and bottom of this panel.

Figure 3-6

Figure 3-7

Figure 3-8

4) Remove the panel.

Figure 3-9

Figure 3-10

5) Remove screws from the bottom of the three remaining panels

Figure 3-11

6) Tilt the remaining three sides and top of the crate away from the Pulsar CNC Mill.

Figure 3-13

Figure 3-12

7) Remove the plastic wrap from the Pulsar Mill. Figure 3-14 and Figure 3-15 shows the Pulsar Mill and the shipping pallet placed on blocks to allow an engine hoist to roll under the pallet.

Figure 3-14

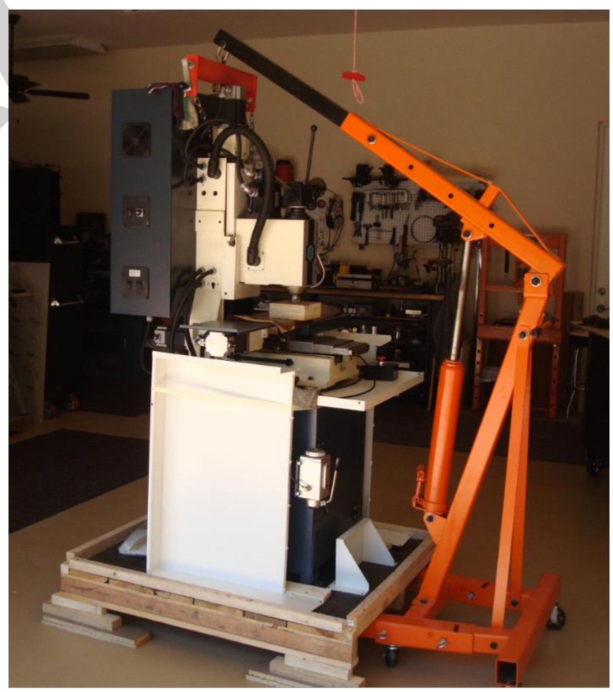


Figure 3-15

8) Remove the Four coolant tray pieces from the shipping palet. Next, remove the four bolts and nuts that secure the Pulsar CNC Mill to the shipping pallet. A forklift is the most desirable means to lift the Pulsar Mill. If a forklift is not available, an engine hoist can be used. The Pulsar CNC Mill can be lifted by using the single eyelet located on the top of the mill head between the spindle motor and the electrical control cabinet. Use a fork truck or engine hoist to lift the Pulsar Mill. Remove the palet from under the mill.

NOTE: Use appropriately sized chains and/or heavy-duty tow straps when lifting the machine from the pallet. Lifting capacity of the chains and tow straps should be three times the weight of the object you are lifting. The Pulsar CNC Mill that you are lifting, weighs at least 900 lbs. Before utilizing the chains and/or towing straps, examine them for cracks or metal fatigue that could reduce the lifting capacity.

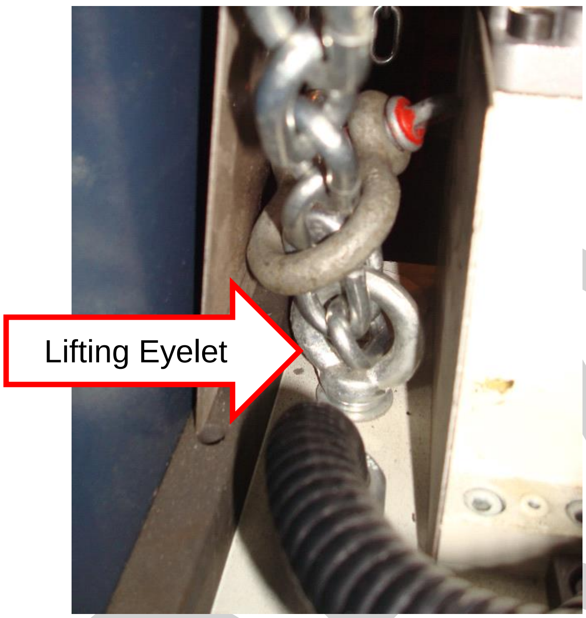


Figure 3-17

Figure 3-16

Screw the four leveling feet to the base of the stand. Use a Phillip screw driver to adjust the feet to the appropriate height for the machine to be level. (See leveling your machine on page 3-11)

Figure 3-19

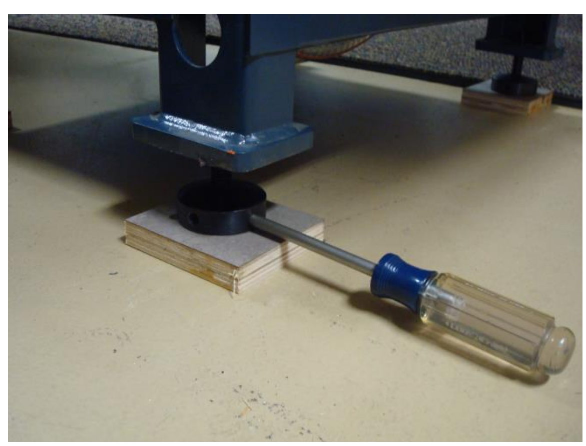


Figure 3-18

SET-UP CLEARANCES & CONDITIONS

When considering the permanent location for your Novakon PULSAR Mill, the following should be taken into consideration:

1) The machine should be installed on a flat surface so that the machine does not rock or slide during operation.

2) This location should be considered the machine’s permanent location.

3) Make sure your Pulsar CNC Mill is level. Improper installation and an unleveled machine can cause both numerical error and loss of precision in your machining operation.

4) The maximum temperature of your shop or working environment should not exceed 125 F or 45 C.

5) Humidity levels should not exceed 90%.

6) Set up the bed mill stand so you have plenty of working space. Leave at least 3 to 4 feet of clearance on the operating side of the machine. Note the suggested clearances of the other sides of the machine (Figure 3-13).

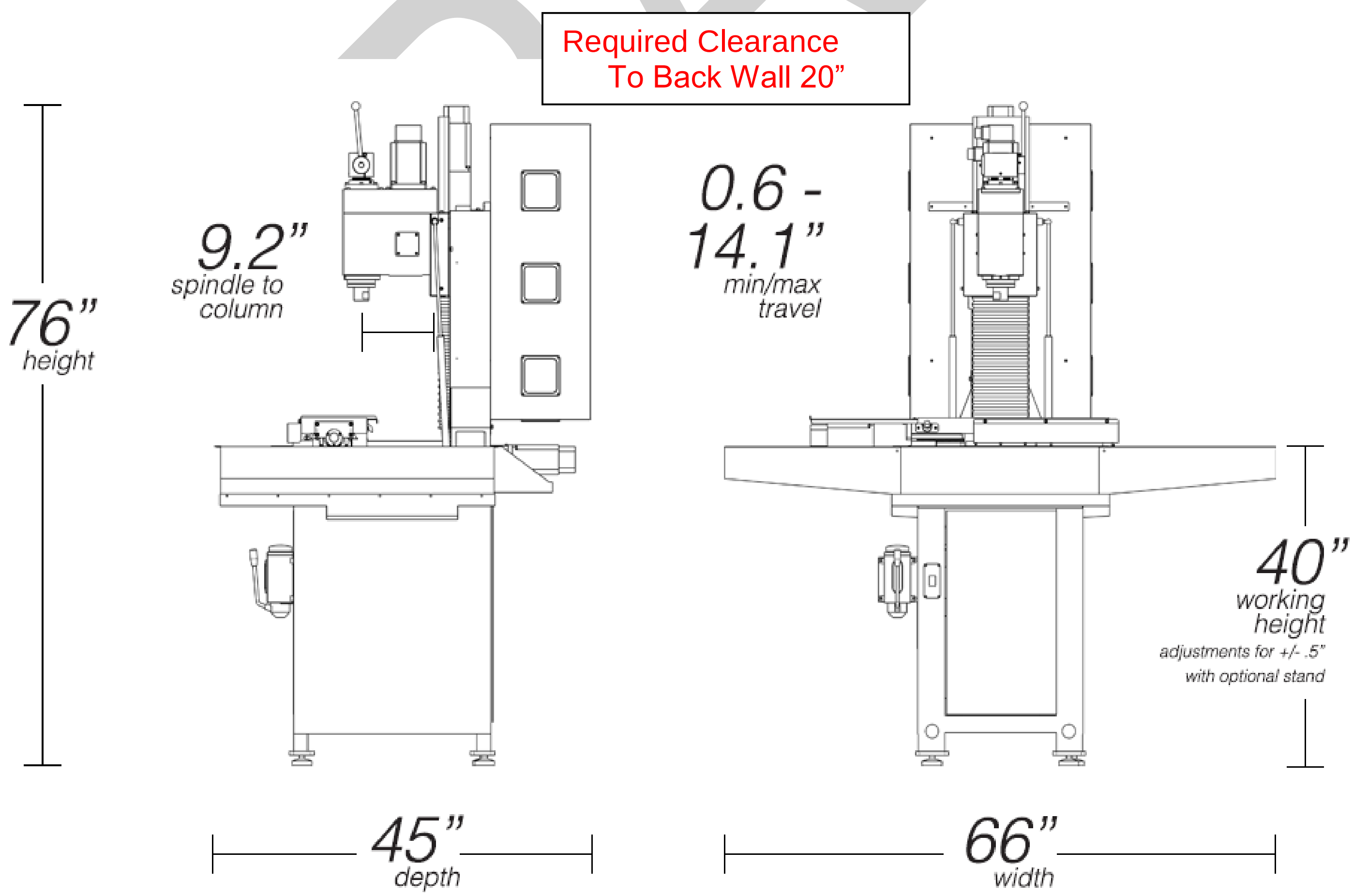


Figure 3-20 Machine Dimensions and Clearances

POWER REQUIREMENTS

	VOLTAGE	AMPS	PHASE
PULSAR MILL	220 VAC	15	SINGLE

CAUTION

Novakon recommends that you run your machine on a dedicated 20-amp circuit breaker and have your 220 VAC electrical work performed by a certified electrician. Install an appropriate male electrical plug on the end of the electrical cable or wire the cable directly into an electrical junction box.

INSTALLING THE COOLING TRAY

1. The cooling tray consists of assembling four sheet metal parts using twenty-two screws and four nuts. The Pulsar CNC Mill includes an extra screw and nut.

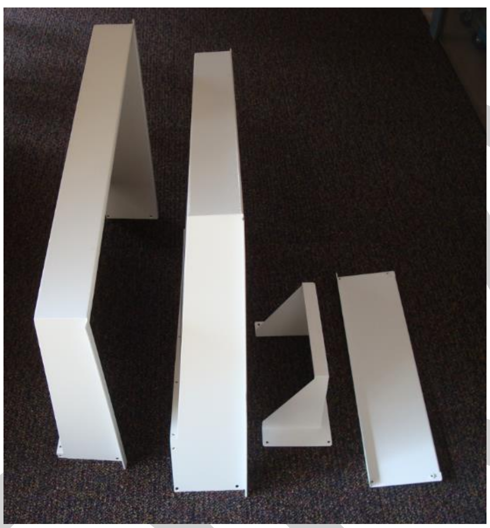


Figure 3-21

Figure 3-22

2. Caulk all cooling tray seams using a clear or white silicone caulking.

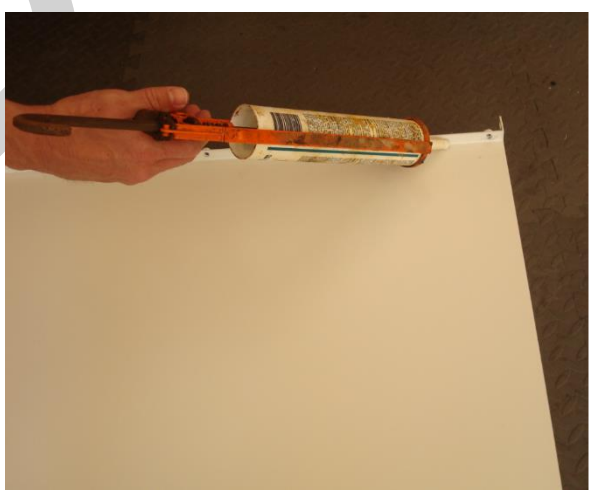


Figure 3-23

3. Assemble the left, front and back portions of the cooling tray at the same time. This task will be much easier with the aid of an additional helper. Do not fully tighten all of the screws until all four sheet metal portions of the coolant tray have been installed.

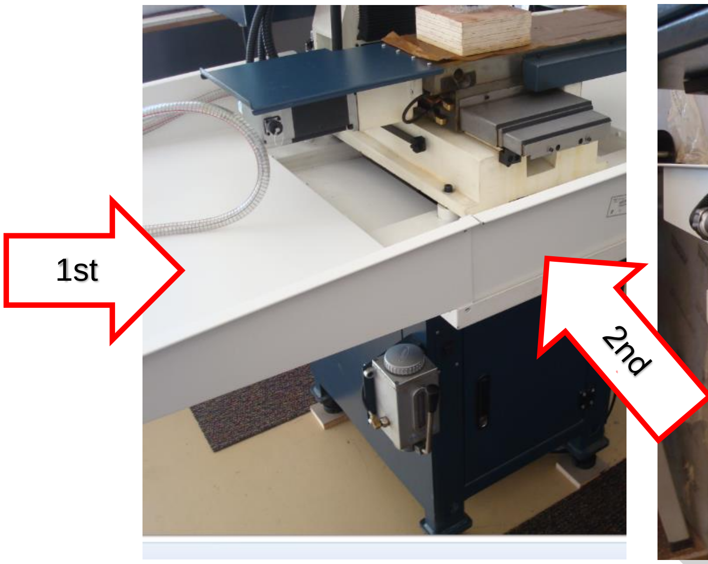


Figure 3-24

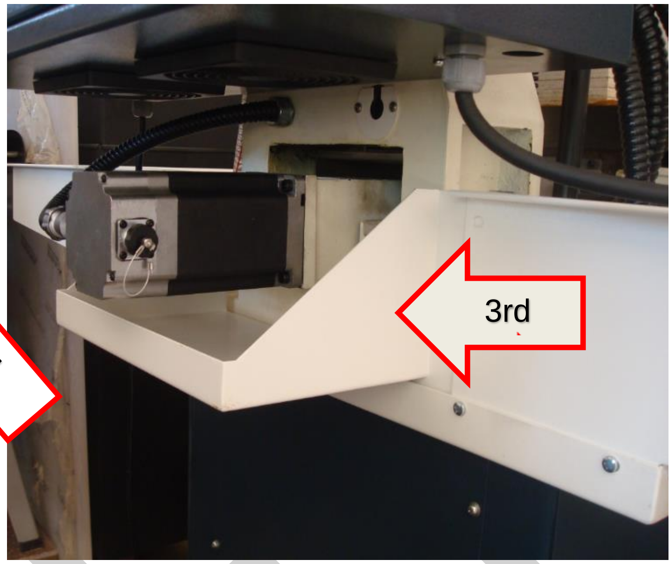


Figure 3-25

4. Install the right side coolant tray.

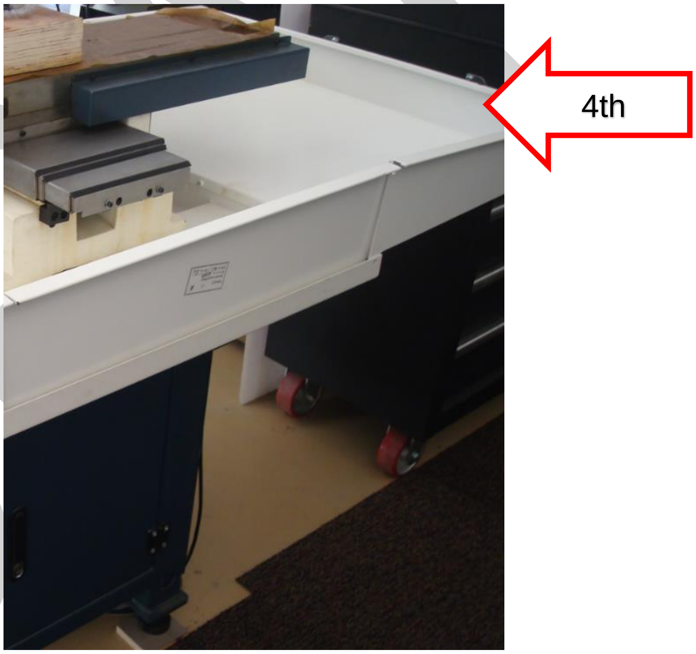


Figure 3-26

5. Tighten all of the screws to complete the coolant tray assembly.

LEVELLING YOUR MACHINE

The Pulsar Mill stand should be located in a vibration-free area on a floor that is designed to support the full weight of the stand, machine, fixtures, accessories, and materials. The treaded mounting holes in the base of the machine are 10 mm.

Before operating the Pulsar Mill, it is crucial to level it for best results.

1. Find a relatively flat place for the installation of the Pulsar Mill and place the mill in this spot taking note of the suggested clearances (Figure 3-20 Machine Dimensions and Clearances on page 3-8).

2. Position the X & Y-axes slides as close to the mid-stroke position as possible. For leveling purposes, the position of the Z-axis does not matter.

   NOTE: This is done after you have hooked a computer monitor, keyboard and mouse to your bed mill.

3. Make sure the stand's four corner feet or leveling pads are touching the ground at the same time. If not, adjust the feet to make sure the machine stand does not physically rock.

4. Once all the feet or leveling pads are touching the ground and the machine is not rocking, place a level on the mill table parallel to the Y-axis (Figure 3-27). Adjust the left and/or right side feet, up or down, to center the level bubble. By adjusting these feet, you are tilting the machine sideways around the Y-axis of the machine.

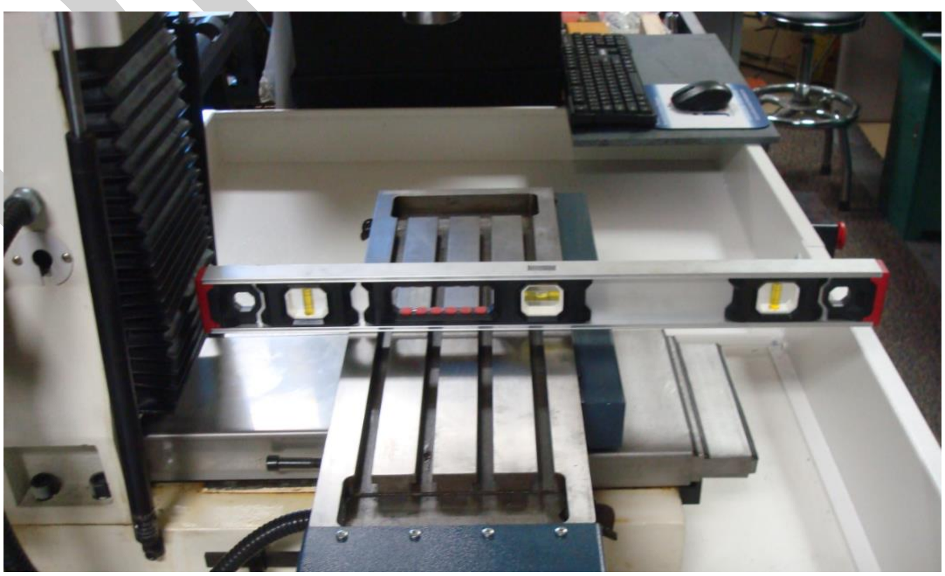


Figure 3-27 Parallel to Y-Axis

5. Now, place the level parallel with the X-axis (Figure 3-28). Adjust the front and back side feet up or down to center the level bubble. By adjusting these feet, you are tilting the machine sideways around the X-axis of the machine.

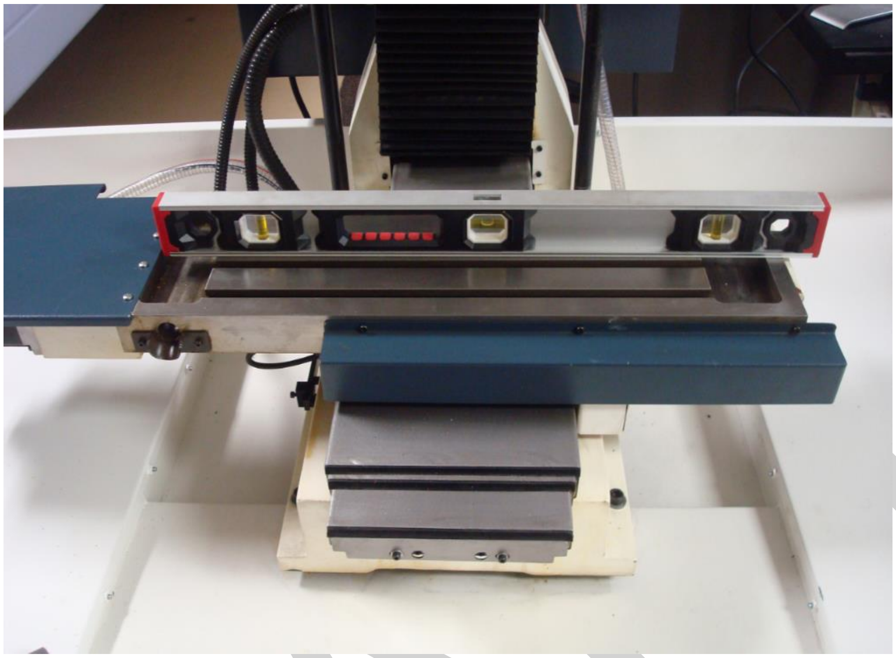


Figure 3-28 Parallel to X-Axis

6. Once both adjustments are completed, move the table in the X and Y directions several times to assure that the Pulsar Mill is level and solid on the floor.

7. Check to confirm that the machine is still level by repeating step 6. If the machine appears to be unleveled, repeat steps 2 through 6.

NOTE: This leveling procedure should be rechecked every 6 months to ensure machine leveling has not changed during machining operations.

4 COMPONENT IDENTIFICATION

This chapter will help you to familiarize yourself with the major components and functions of your machine.

1) **Main and Driver Power Switches –** These switches turns on/off the 220 VAC electrical power supply to the Power Transformer which supplies 50VAC to the Pulsar Mill, servo spindle motor, and stepper or servo axis motors.

Figure 4-1 Main and Driver Power Switches

2) **Fourth Axis Power Switch –** This switch turns on/off the 50 VAC electrical power supply to the optional fourth axis. The main and driver power switches must be turned on before 50 VAC is available to the fourth axis power switch.

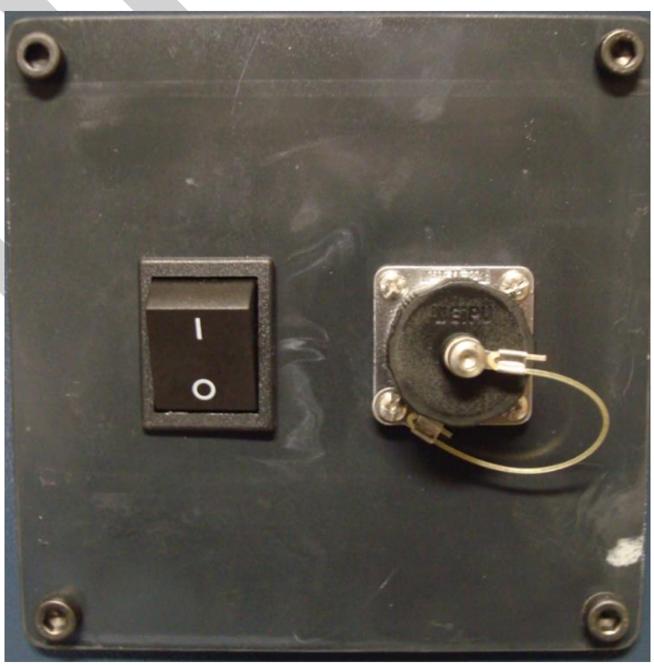


Figure 4-2 Fourth Axis Switch

3) **E-Stop Button –** This big red button is in an enclosure with a magnetic back and should be mounted in a location which is easily and safely reachable should an emergency shutdown of the mill be necessary. This button will immediately shut down all movement of the Pulsar Mill table and spindle. Once the E-stop button is pressed, power can be restored to the mill by turning the E-Stop button clockwise. You must also reset the Breakout Board (BOB) by pressing the reset button on the BOB (Figure 4-4).

Figure 4-4

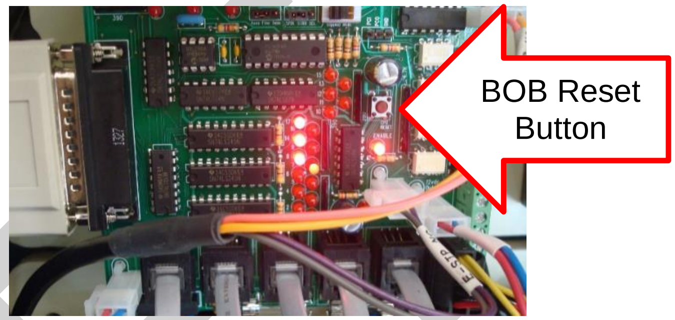


Figure 4-3

4) **Green PC Start/Reset Button–** Pressing this button starts or resets the optional built in computer.

USB Ports – There are four USB ports located on the side of the electrical cabinet.

Monitor Receptacle – This receptacle is used to connect computer monitor using a nine pin video cable.

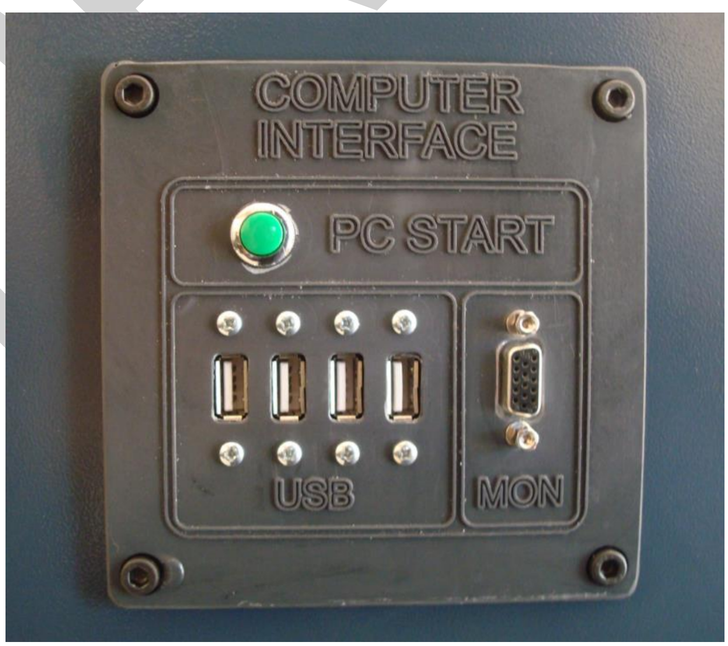


Figure 4-5

5) **Lever Draw Bar -** The LDB supports and "draws" quick change tooling into the ¾" R-8 collet of the Novakon Pulsar Mill. The function of the drawbar is to locate the tool holder accurately in relationship to the main spindle. The clamping and unclamping of the tools is achieved by pulling the LDB handle down while inserting or releasing quick change tooling.

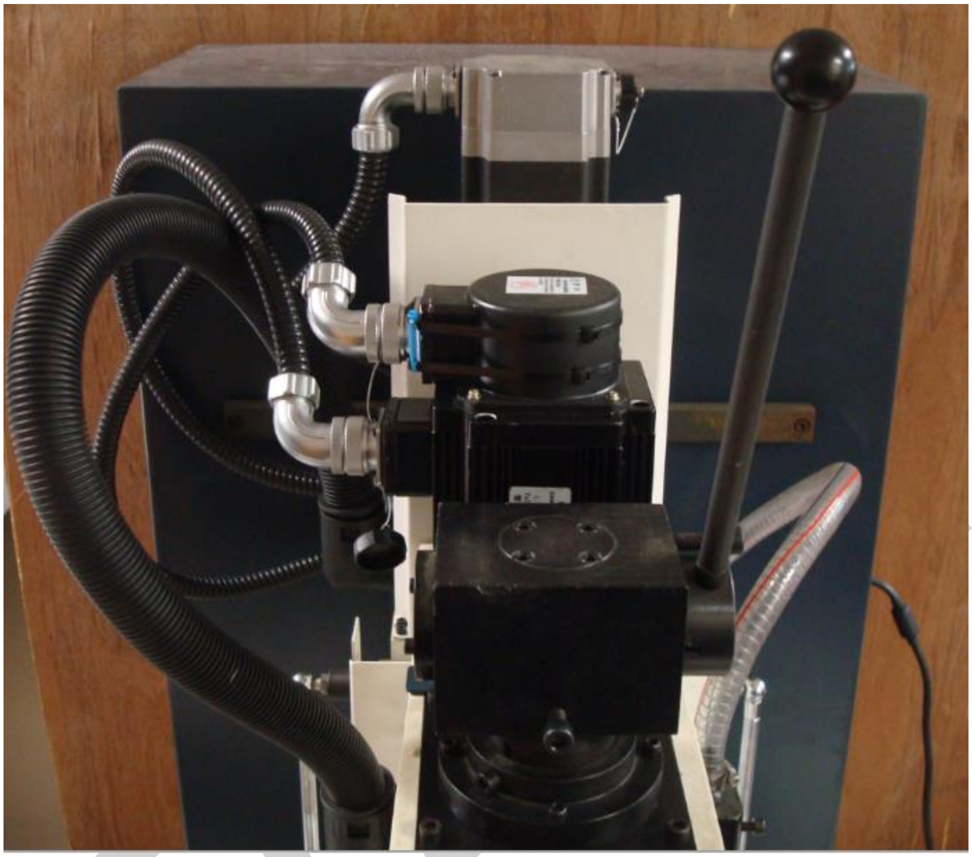


Figure 4-6

6) **Wash Down System –** The Pulsar CNC Mill includes a dedicated pump, hose and nozzle for clearing the coolant tray and mill of debris.

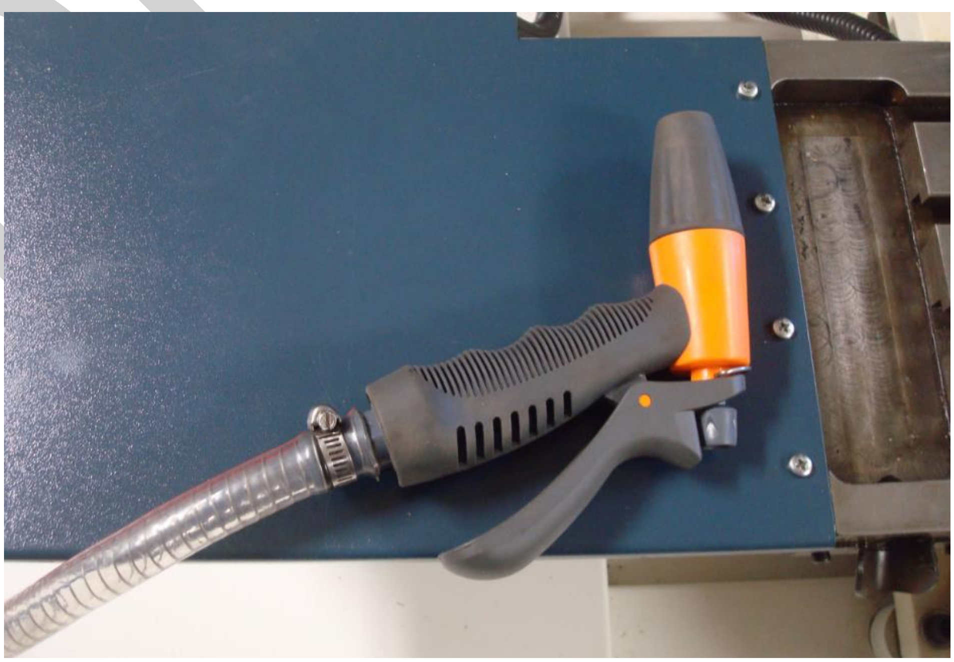


Figure 4-7

7) **Coolant Nozzle and Flex Hose –** The Pulsar CNC Mill includes a coolant nozzle attached to a metal flex hose. The metal flex hose is more stable and more durable than a plastic flex hose

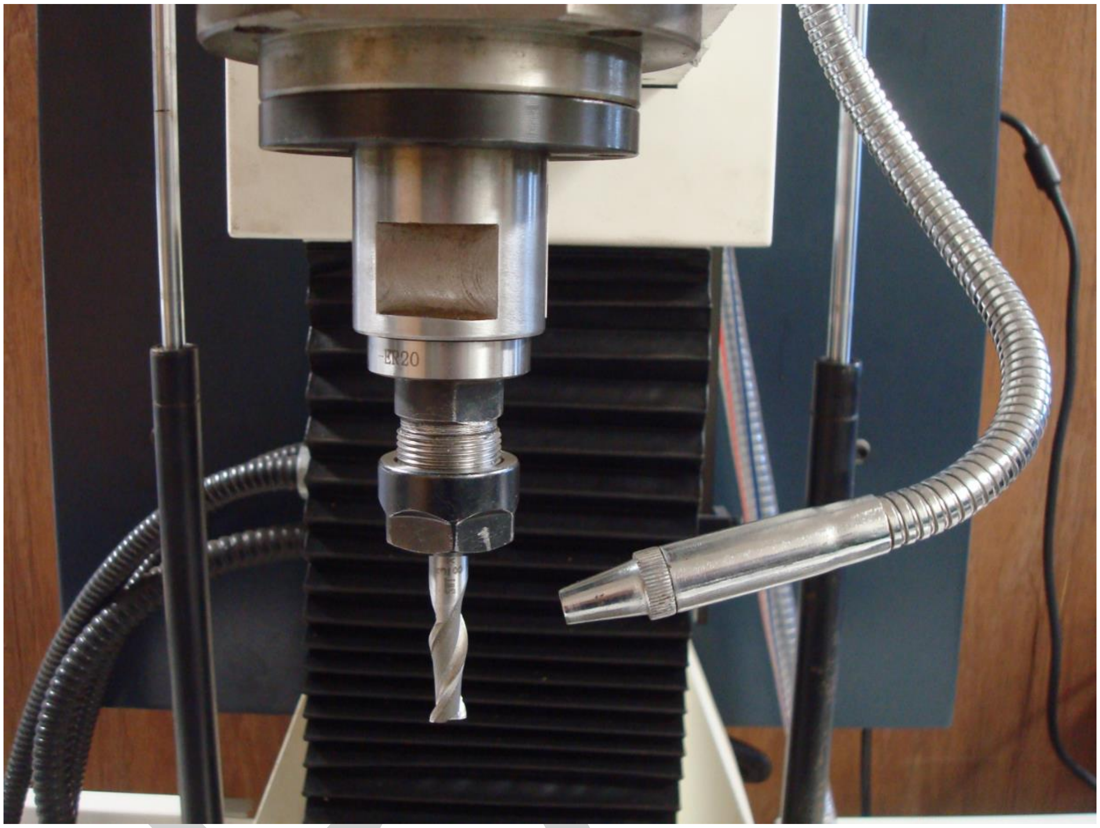


Figure 4-8

8) **Chip Drawer –** The Pulsar CNC Mill includes a chip drawer to collect debris from the milling process. Figure 4-9 shows the chip tray with a user installed filter media.

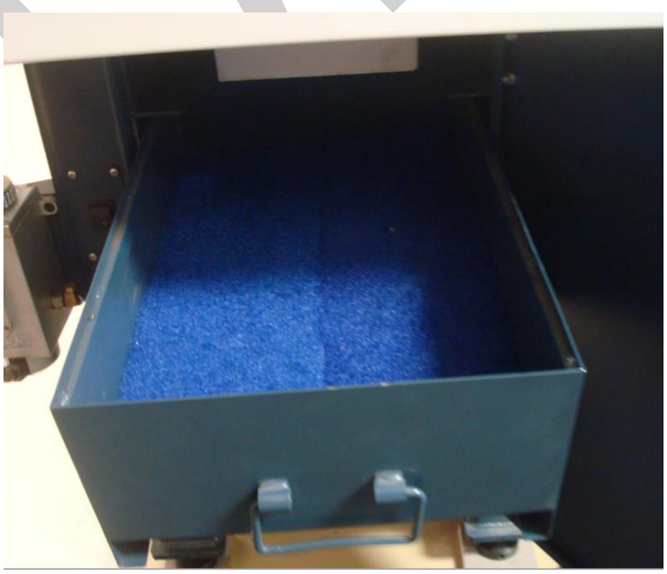


Figure 4-9

Figure 4-10

9) **Coolant Drawer -** Figure 4-11 shows the large coolant storage drawer with two dedicated 90 watt pumps. The left pumps provides fluid for the wash down system and the right pump supplies coolant to milling operation.

Figure 4-11

Figure 4-12

10) **Rigid Tapping Switch -** Figure 4-13 shows the location of the Rigid Tapping Mode Switch on the BOB. Slide the switch to the ON position to implement Rigid Tapping.

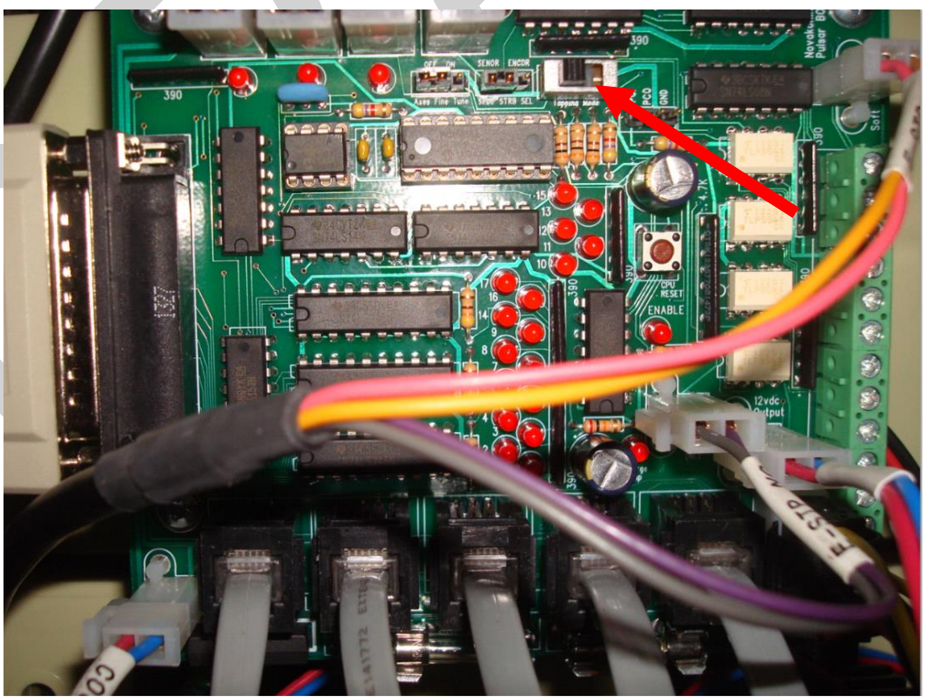


Figure 4-13 Rigid Tapping Selection Mode Selection Switch

11) **Manual Oiler -** This single stroke lubrication system lubricates the ball screws, and dove tail X-axis, Y-axis and Z-axis slides. To lubricate the Pulsar Mill, pull the lever of the one shot lube system away and down from the machine. The number of pumps required will depend on your shop environment. Enough pumps should be administered until a thin layer of lubrication is present on the slides.

Figure 4-14 Manual Oiler

12) **Optional Fourth Axis Rotary Table -** The optional fourth axis rotary table is available directly from Novakon International.

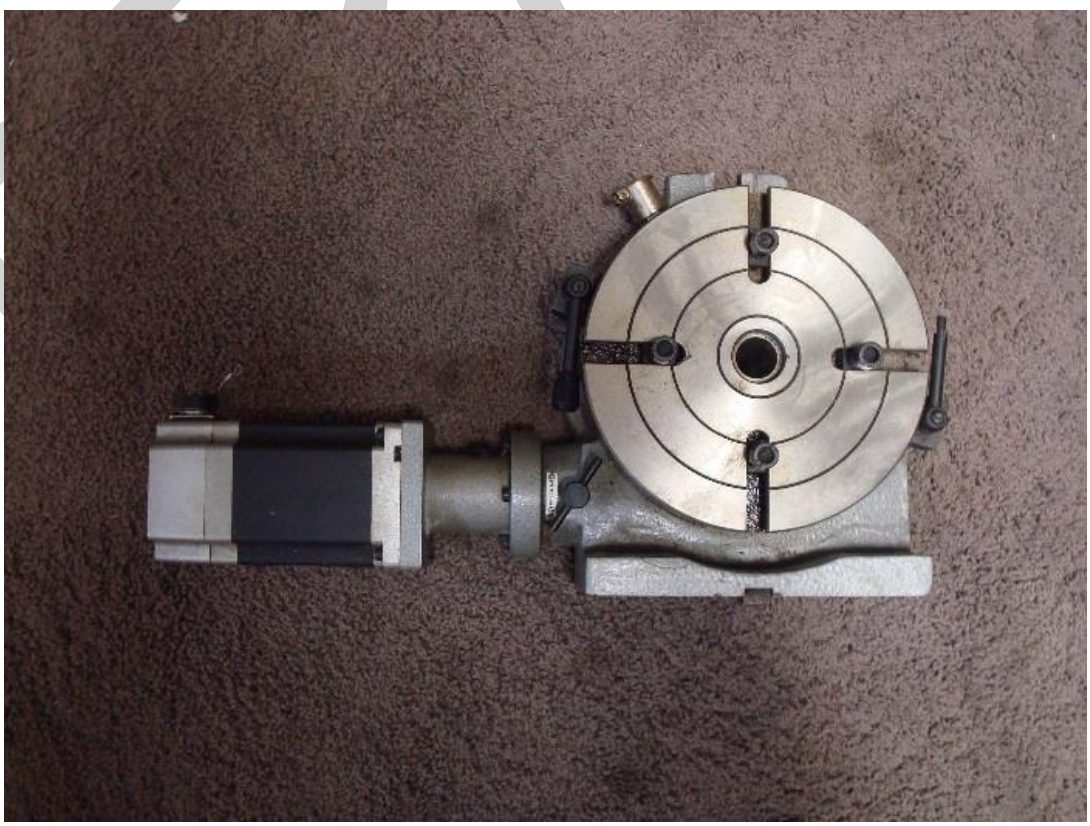


Figure 4-15 Optional Fourth Axis Rotary Table

5 MACHINE CONTROL (Mach3)

SCREENS

The optional computer is built into the Pulsar Mill electrical cabinet and has a demo copy of Mach3 installed on its hard drive. Mach3 is the interface software used to control the Pulsar Mill. It is suggested that you try out a "dry run" of Mach3. You can “pretend" to machine and learn a lot without actually running the Pulsar Mill.

Mach3 is designed so that it is very easy to customize its screens to suit the way you work. This means that the screens you see may not look exactly like those in this manual. Types of objects on screen

You will see that the Program Run screen is made up of the following types of objects:

- Buttons (e.g. Reset, Stop Alt-S, etc.)
- DROs or Digital Readouts. Anything with a number displayed will be a DRO. The main ones are of course, the current positions of the X, Y, Z, and A-axes
- LEDs (in various sizes and shapes)
- G-code display window (with its own scroll bars)
- Tool path display (blank square on your screen at the moment)
- There is one further important type of control that is not on the Program Run screen: MDI (Manual Data Input) line

Buttons and the MDI line are your inputs to Mach3. DROs can be displays by Mach3 or can be used as inputs by you. The background color changes when you are inputting.

The G-code window and Tool Path displays are for information from Mach3 to you. You can however, manipulate both of them (e.g. scrolling the G-code window, zooming, rotating and panning the Tool Path display).

INTRODUCTION TO CNC SYSTEMS

This section introduces you to Mach3. The Pulsar Mill requires a computer numerically controlled (CNC) system. Mach3 is one of the most widely used PC based CNC software on the market. Novakon has installed a demo copy of this software to its optional built in computer. Mach3 will run for an unlimited period in a demonstration version. As demonstration software, it places a few limitations on the speed, the size of job that can be undertaken, and the specialist features supported. When you purchase a license, you can "unlock" the demonstration version you have already installed and configured it to offer full software functionality. Information for purchasing a Mach3 license can be obtained from the following website:

http://www.machsupport.com/purchase.php

The Mach3 demo version is limited to 500 lines of code when running the Pulsar Mill. A licensed copy of Mach3 is only required on the computer which is actually running the Pulsar Mill. If a computer is not connected to the Pulsar Mill and is used to test code, use Mach3 in Simulation mode - it will remove the 500 line limit. Simulation mode can be accessed by uninstalling (or not installing) the Parallel Port Driver.

If you are not familiar with Mach3, you will need to read this documentation! Mach3 is a complex piece of software. You will not be successful if you simply start trying to "get it to work." While that approach may be appropriate for some software, it is not appropriate for Mach3. Save yourself a lot of aggrevation by taking the time to work through this chapter. An online discussion forum for Mach3 can be found at www.machsupport.com.

COMPONENTS OF A CNC MACHINING SYSTEM

The main components of a CNC system are shown in Figure 5-1. These are:

1) **Computer Aided Design/Computer Aided Manufacturing (CAD/CAM) program**. The part designer uses the CAD/CAM program to generate an output file called a *part program*. The part program, often written in "G-Code," describes the machine steps required to make the desired part. You can also create a G-Code program manually.

2) **File transfer medium** such as a USB flash drive, floppy disk or network link, transfers the output of the CAD/CAM program to a Machine Controller.

3) **Machine Controller** reads and interprets the part program to control the tool which will cut the work piece. Mach3 running on a PC, performs the Machine Controller function and sends signals to the Drives.

4) **Drives**. The signals from the Machine Controller are amplified by the Drives so they are powerful enough and suitably timed to operate the motors driving the machine tool axes.

5) **Machine**. The machine's axes are moved by screws, racks or belts which can be powered by servo or stepper motors.

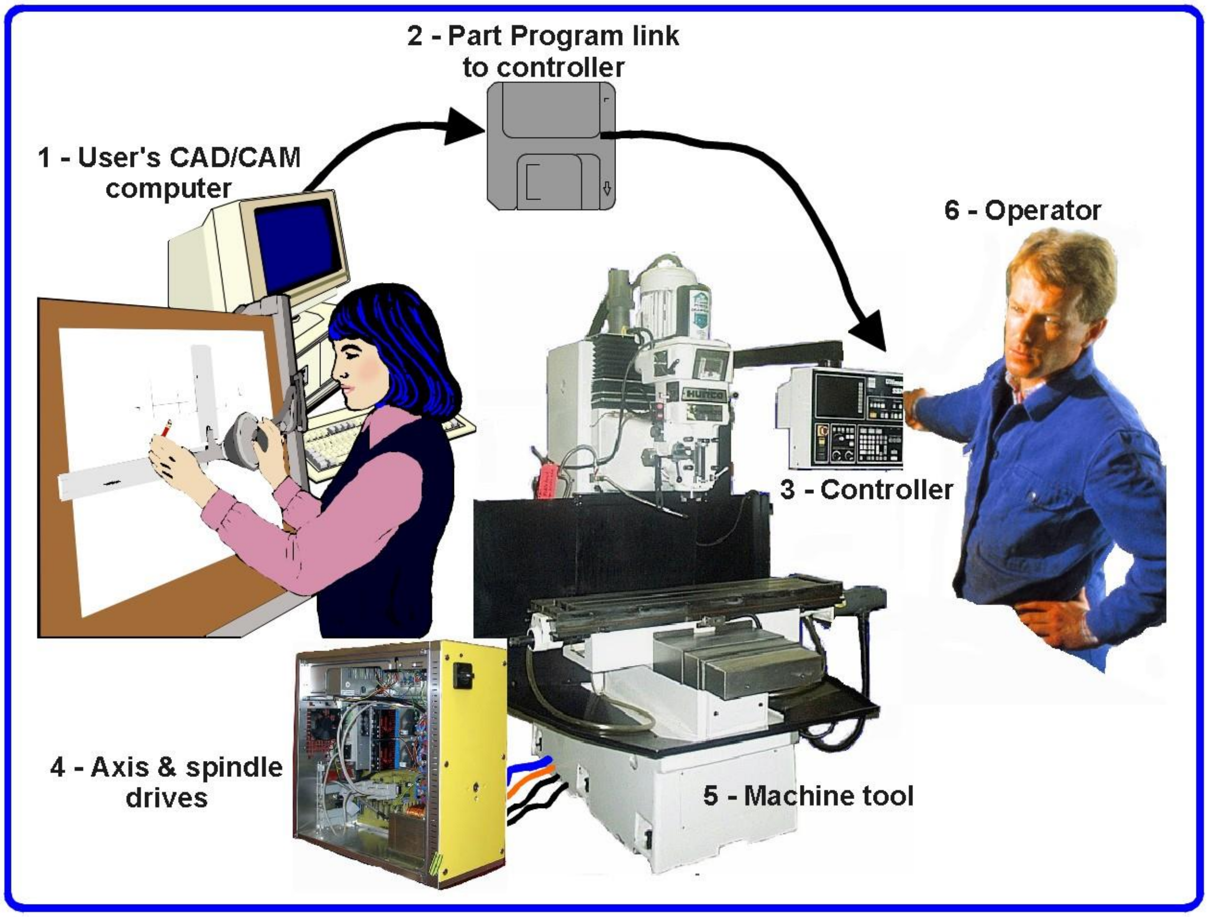


Figure 5-1 MAIN PARTS OF A CNC SYSTEM

HOW MACH3 FITS IN

Mach3 is a software package that runs on a PC and turns it into a very powerful and economical Machine Controller to replace (3) Controller (Figure 5-1). Mach3 and its parallel port driver communicate with the Pulsar Mill's hardware through a parallel (printer) port connected to the optional built in PC. Mach3 is used to control the Pulsar Mill by generating step pulses and direction signals to perform the steps defined by a G-Code part program and sending them to the parallel port.

You are now ready to try a "dry run" of Mach3. It will be easier to understand how to use Mach3 to control the Pulsar Mill after you have experimented a bit with the software. You can "pretend" to machine and learn a lot even without running the Pulsar Mill.

Double-click the Mach3 Loader icon (Figure 5-2) to display the current Mach3 profiles (Figure 5-3). Highlight the "Pulsar-Stepper Axis-PBOB" profile and select the "OK" button to display the Mill Program Run screen (Figure 5-4).

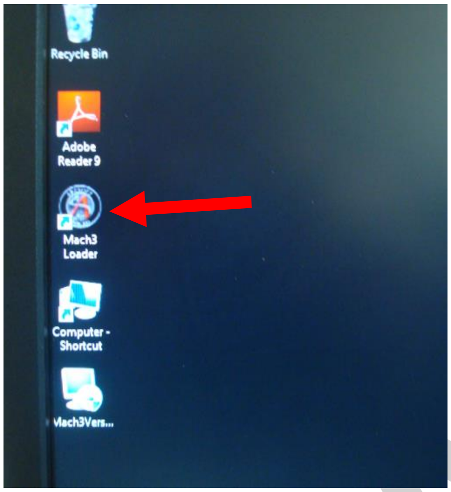


Figure 5-3

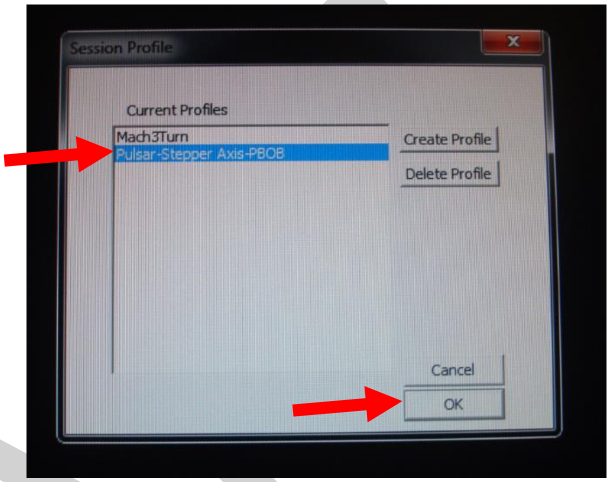


Figure 5-2

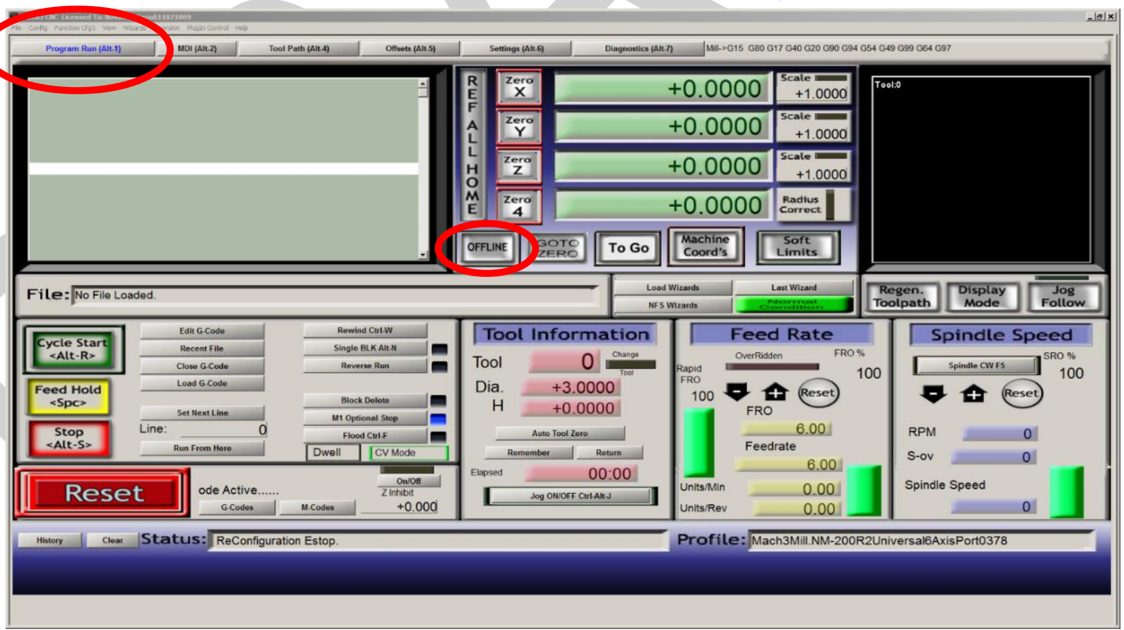


Figure 5-4

Notice the red Reset button. It will be flashing Red/Green (simulation of a light emitting diode). Clicking the red Reset button turns the flashing LED to solid green, and makes Mach3 ready for action!

If you cannot turn the Reset button to solid green, the problem is probably that the Emergency Stop (E-Stop button) has been pressed. Refer to item "2) E-Stop Button", page 4-2, for instructions on restarting the Pulsar CNC Mill after the E-Stop Button has been pushed. You may also try clicking on the *Offline* button which should allow the reset button to function. Most of the tests and demonstrations in this chapter will not work unless Mach3 is reset out of the E-Stop mode.

You should see the Mill Program Run screen. The other major screens, identified on tabs are MDI (manual data input), Tool Path, Offsets, Settings and Diagnostics, (Figure 5-5). Be sure the Program Run screen is selected; its name will be displayed in blue.

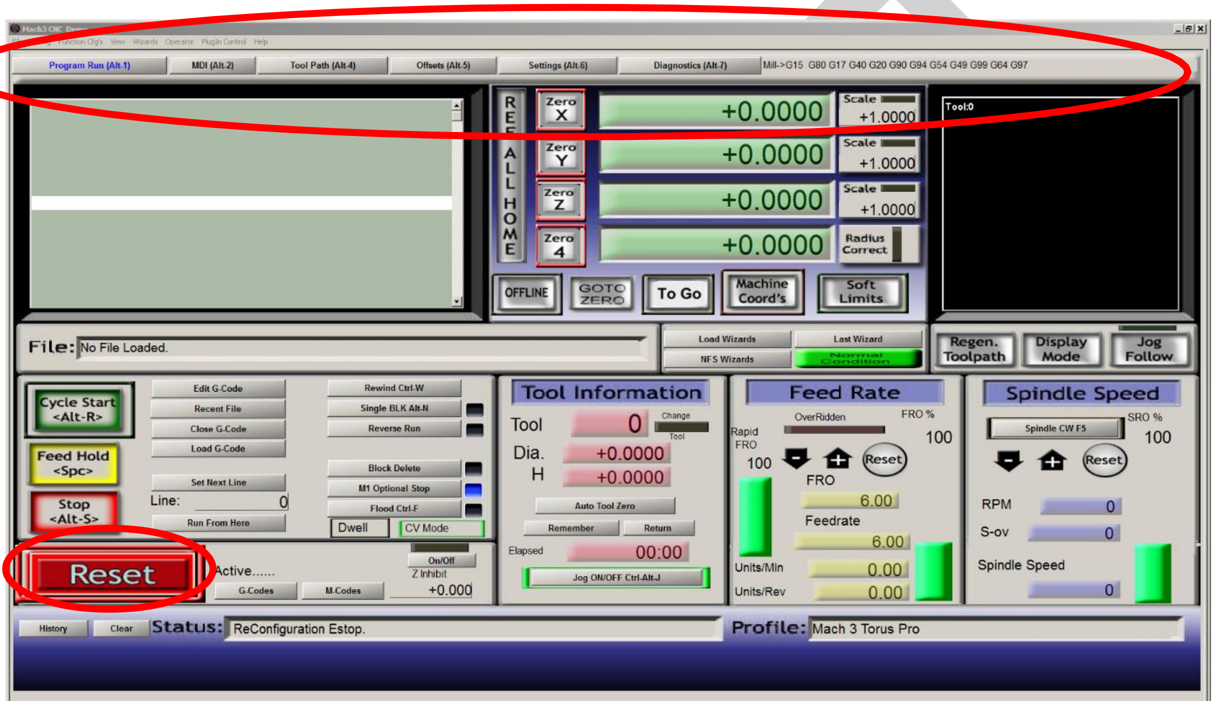


Figure 5-5 Program Run screen

Click the red RESET button (Figure 5-5), and the flashing LED turns to solid green. Mach3 is ready for action!

TYPES OF OBJECTS ON SCREENS

You will see that the Program Run screen is made up of the following types of objects:

- Buttons (ie. Reset, Stop Alt-S, etc.)
- DROs or Digital Read Outs. Anything with a number displayed will be a DRO. This may be a more general use of the term “DRO” than you are accustomed to. The main DROs are the current positions of the X, Y, Z, and A-axes. There are also DROs for feed rate, spindle speed, and other values
- Simulated LEDs (in various sizes and shapes)
- G-Code display window (with its own scroll bars)
- Tool path display (Blank Square on your screen at the moment)

There is one further important type of control that is not on the Program Run screen:

- MDI (Manual Data Input) line on the MDI screen

Buttons, data entry boxes and the MDI line are your inputs to Mach3.

DROs can be displays by Mach3 or can be used as inputs by you. The background color changes when you are inputting ?.

The G-Code window and Tool path displays, provide information from Mach3 to you. You can however, manipulate both of them (ie. scroll the G-Code window and zoom, rotate and pan the Tool path display).

USING BUTTONS AND SHORTCUTS

MOST STANDARD SCREEN BUTTONS HAVE a keyboard hot key which may be a single key or a key combination. This is often shown as part of the name on the button itself or in a label near it. For example, the shortcut to go to the MDI screen is *Alt-2*. Pressing the named key or key combination when the screen is displayed is the same as clicking the button with the mouse. You might like to try using the mouse and keyboard shortcuts to turn on and off jogging, to turn on Flood coolant, and to switch to the MDI screen. Notice that letters are sometimes combined with the *Control* or *Alt* keys. Although letters are shown as uppercase for ease of reading, **do not** use the shift key when typing the shortcuts.

In a workshop, it is often convenient to minimize the times when you need to use a mouse. Physical switches on a control panel can be used to control Mach3 by using a Multiple Pulse Generator (MPG).

If a button does not appear on the current screen, then its keyboard shortcut is not active.

There are certain special keyboard shortcuts which are global across all screens. The

Section entitled Configure System Hotkeys, page 5-40 shows how those are set up.

DATA ENTRY TO DRO

You can enter new data into any DRO by clicking in it with the mouse, by clicking its hot key (where set), or by using the global hot key to select DROs and moving to the one that you want with the arrow keys.

Try entering a feed rate like *45.6* on the Program Run screen (Figure 5-6). Click in the feed rate box and type the numbers. You **must** press the *Enter* key to accept the new value, or press the *Esc* key to revert to the previous one. *Backspace* and *Delete* are not active when inputting to DROs.

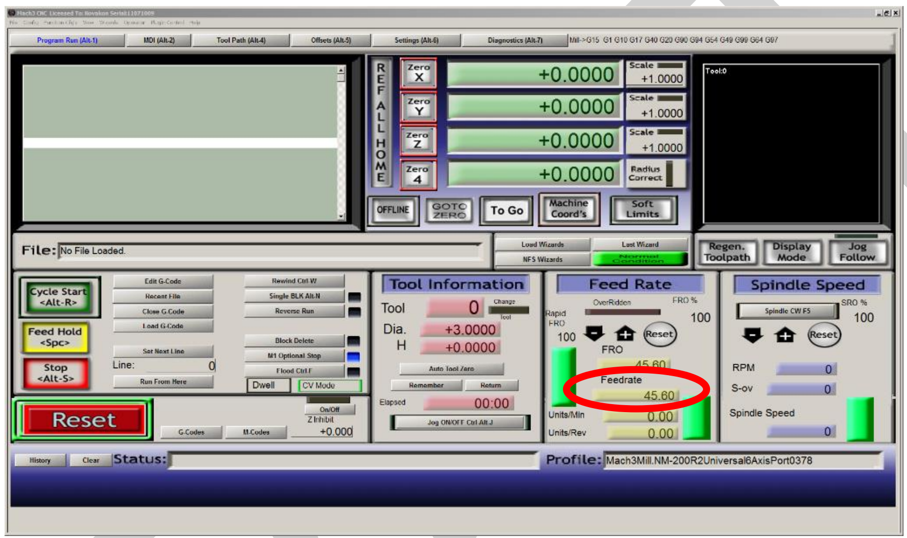


Figure 5-6

Caution: It is not always sensible to put your own data into a DRO. For example, the display of your actual spindle speed is computed by Mach3. Any value you enter will be overwritten. You can put values into the axes DROs, but you should not do it until you have read the *Using Mach3 Mill* manual in detail. It is **not** a way of moving the tool!

JOGGING

You can manually move the tool relative to any place on your work by using various types of jogging. Of course, on the Pulsar Mill, it will be the machine table that moves. We will use the words “move the tool” here for simplicity. The frame of reference used, will assume tool movement. That is, “move the tool to the left” may actually be achieved by moving the table to the right.

CONTROL BUTTONS FLY OUT

The jogging controls are on a special “fly out” screen. This is shown and hidden by using the *Tab* key on the keyboard. Figure 5-7 gives a view of the fly out.

You can also use the keyboard for jogging. The arrow keys are set by default to give you jogging on the X and Y axes and *PgUp/PgDn* jogs the Z-axis. You can re-configure these keys to suit your own preferences. You can use the jogging keys on any screen with the *Jog ON/OFF* button on it.

In Figure 5-7, you will see that the Step LED is shown lit. The *Jog Mode* button toggles between *Continuous, Step,* and *MPG* modes.

In Continuous mode, the chosen axis will jog for as long as you hold the key down. The speed of jogging is set by the *Slow Jog Percentage* DRO. You can enter any value from 0.1% to 100% to get whatever speed you want. The *Up* and *Down* screen buttons beside this DRO will alter its value in 5% steps. If you depress the *Shift* key, then the jogging will occur at 100% speed regardless of the override setting. This allows you to quickly jog near your destination.

In Step mode, each press of a jog key will move the axis by the distance indicated in the *Step* DRO. You can cycle through a list of predefined Step sizes with the *Cycle Jog Step* button. Movement will be at the current Feed Rate.

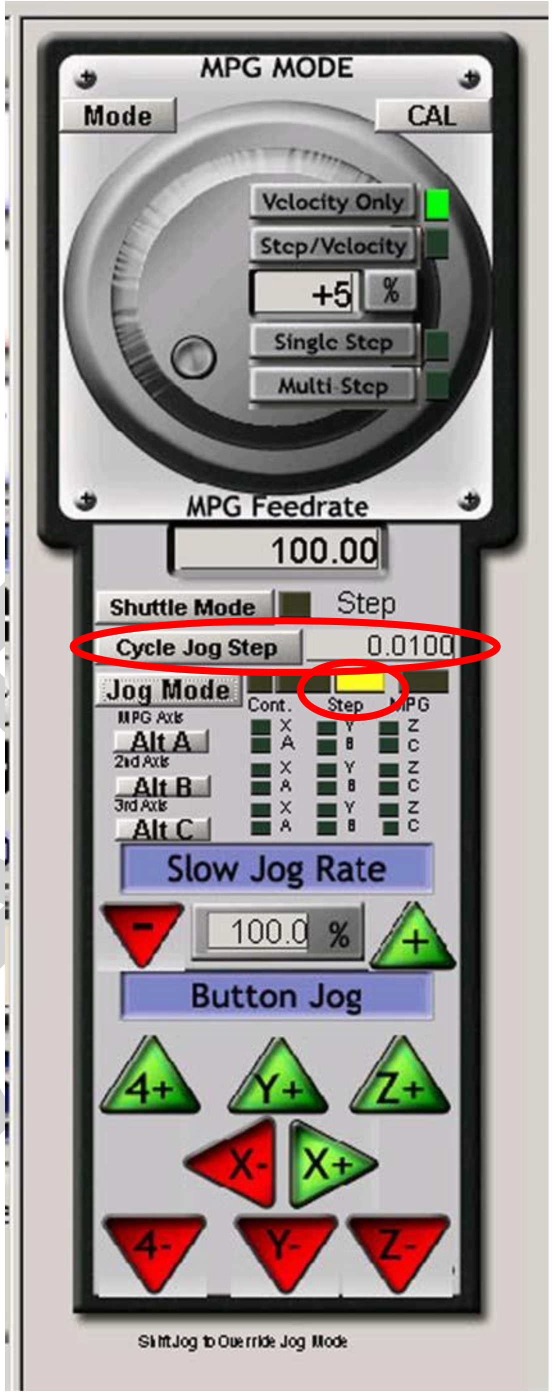


Figure 5-7

Rotary encoders can be interfaced (via the parallel port input pins) to Mach3 as Manual Pulse Generators (MPGs). It is used to perform jogging by turning its knob when in *MPG* mode. The buttons marked *Alt A*, *Alt B,* and *Alt C* cycle through the available axes for each of the three MPGs. The LEDs define which axis is currently selected for jogging.

The other option for jogging is a joystick connected to the PC games port or USB. Mach3 will work with any Windows-compatible “analog joy- stick” (so you could even control your X-axis by a Ferrari steering wheel). The appropriate Windows driver will be needed for the joystick device. The stick is enabled by the Joystick button and for safety reasons, when it is enabled, it must be in the central position.

If you have an actual joystick and it has a throttle control, then this can be configured either to control the jog override speed or control the feed rate override (see Section “CONFIGURING MACH3 FOR YOUR MACHINE AND DRIVES”, page 5-20). Such a joystick is a cheap way of providing very flexible manual control of your machine tool. In addition, you can use multiple joysticks (strictly Axes on Human Interface Devices) by installing manufacturer's profiler software.

Now would be a good time to try all the jogging options on your system. Don't forget that there are keyboard shortcuts for the buttons, so why not identify them and try them. You should soon find a way of working that feels comfortable to you.

MANUAL DATA INPUT (MDI)

Use the mouse or keyboard shortcut to display the MDI (Manual Data Input) screen (Figure 5-8).

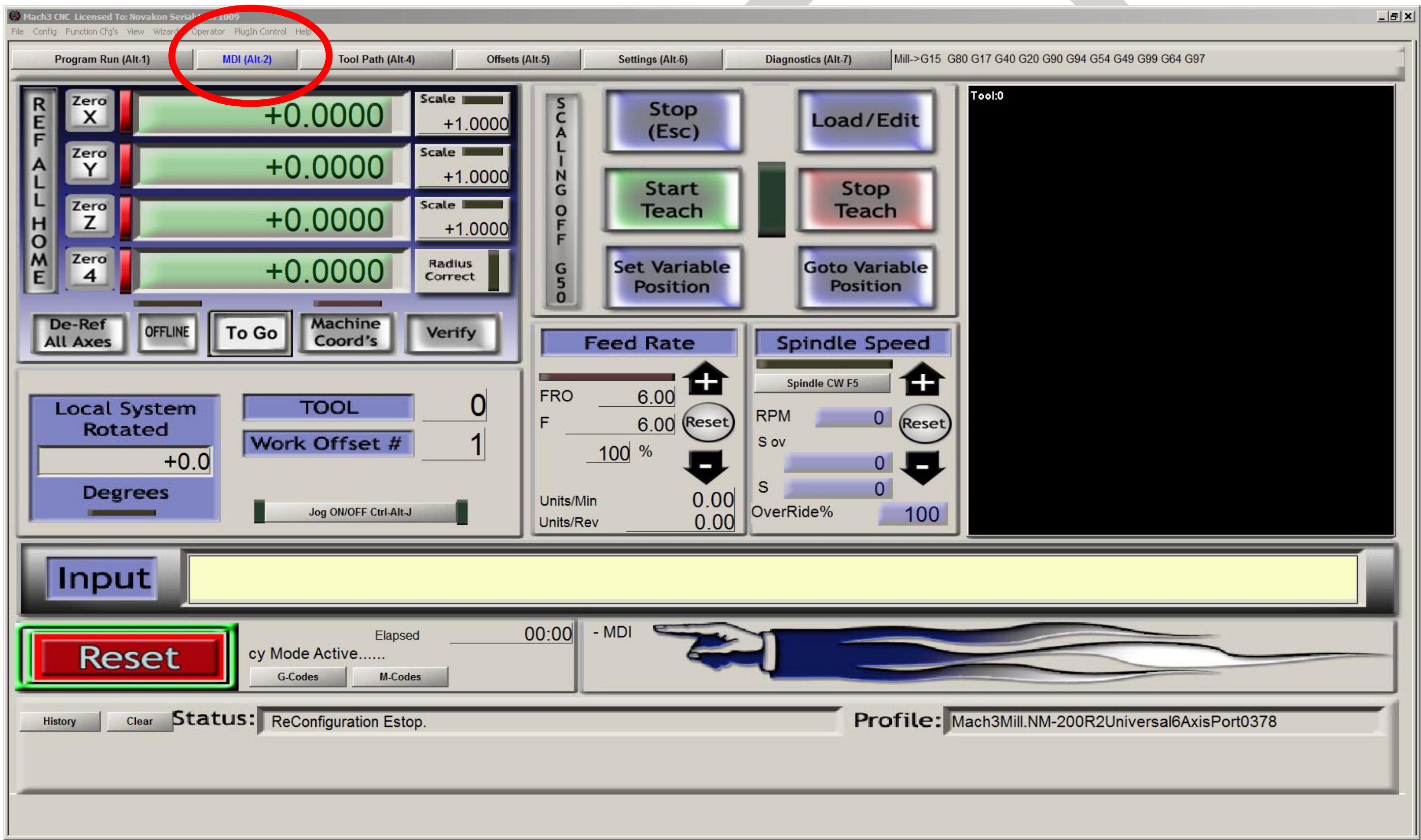


Figure 5-8 MDI (Manual Data Input) screen

This has a single line for data entry. You can click in it to select it or press *Enter*, which will automatically select it. You can type any valid line that could appear in a part program and it will be executed when you press *Enter*. You can discard the line by pressing *Esc*. The Backspace key can be used for correcting mistakes in your typing.

If you know some G-Code commands, you can try them out. Or you can try:

```
G00 X1.6 Y2.3
```

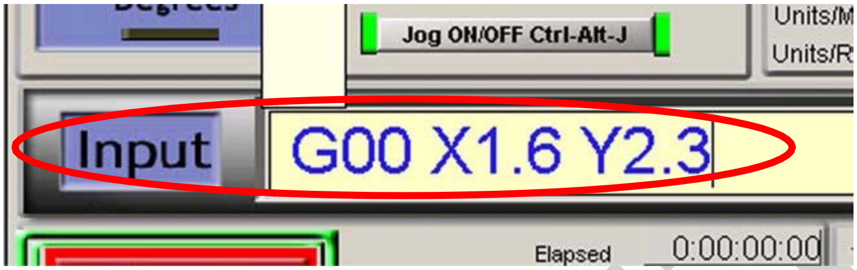


Figure 5-9 MDI EXAMPLE

That command will move the tool to coordinates X = 1.6 units and Y = 2.3 units. (It is G number zero not G letter O.) You will see the axis DROs move to the new coordinates. G0 and G00 are the same command.

Try several different commands (or G0 to different places). If you press the up or down arrow keys while in the MDI line, you will see that Mach3 scrolls you backward and forward through the history of commands you have used. That makes it easy to repeat a command without having to re-type it. When you select the MDI line, you will see a fly out box giving you a preview of this remembered text.

An MDI line or block, as a line of G-Code is sometimes called, can have several commands on it. They will be executed in the "sensible" order as defined in the *Section Order of Execution, page 5-130,* – not necessarily from left to right. For example, setting a feed speed using a command like *F2.5* will take effect before any feed speed movements, even if the *F2.5* appears in the middle or even at the end of the line (block). If in doubt about the order that will be used, enter each MDI command on a separate line.

TEACHING

Mach3 can remember a sequence of lines that you enter using MDI and write them to a file. This file can then be run again and again as a G-Code program.

On the MDI screen, click the *Start Teach* button (Figure 5-10). The LED next to it will light to remind you that you are teaching. Type in a series of MDI lines. Mach3 will execute each command line as you press *Enter*. Mach3 will store the sequence of commands you enter in a named Teach file.

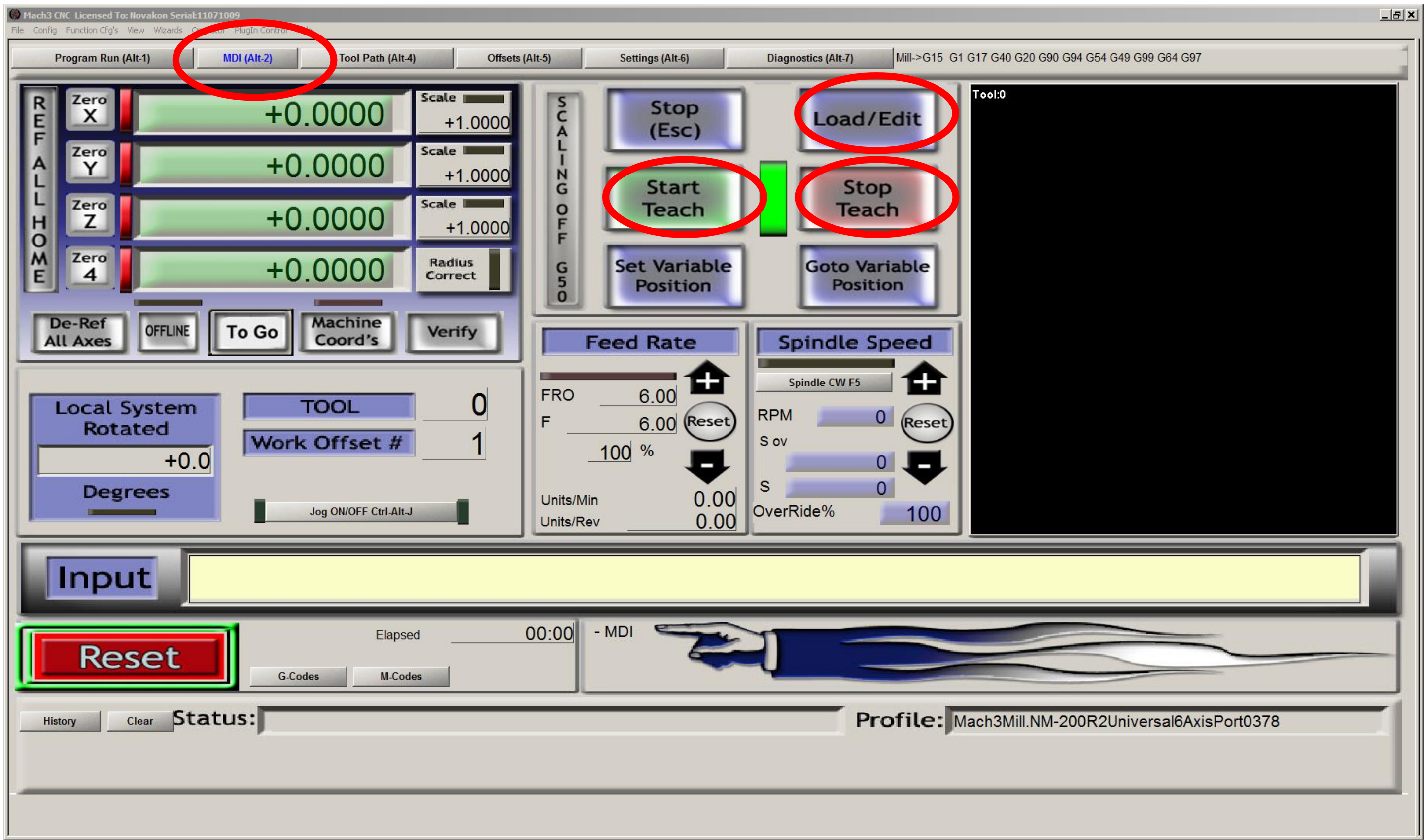


Figure 5-10 TEACHING A RECTANGLE

You can type your own code, or try:

```
g21 f100
g1 x10 y0 g1 x10 y5 x0
y0
```

All the 0 are zeros, not capital O's. When you have finished, click *Stop Teach*. Next, click *Load/Edit (*Figure 5-10*)* and go to the Program Run screen. You will see the lines you typed displayed in the G-Code window (Figure 5-11). If you click Cycle Start, then Mach3 will execute your program.

If you use the editor ("Edit G-Code" button, Figure 5-11), you will be able to correct any mistakes and save the program in a file of your own choosing.

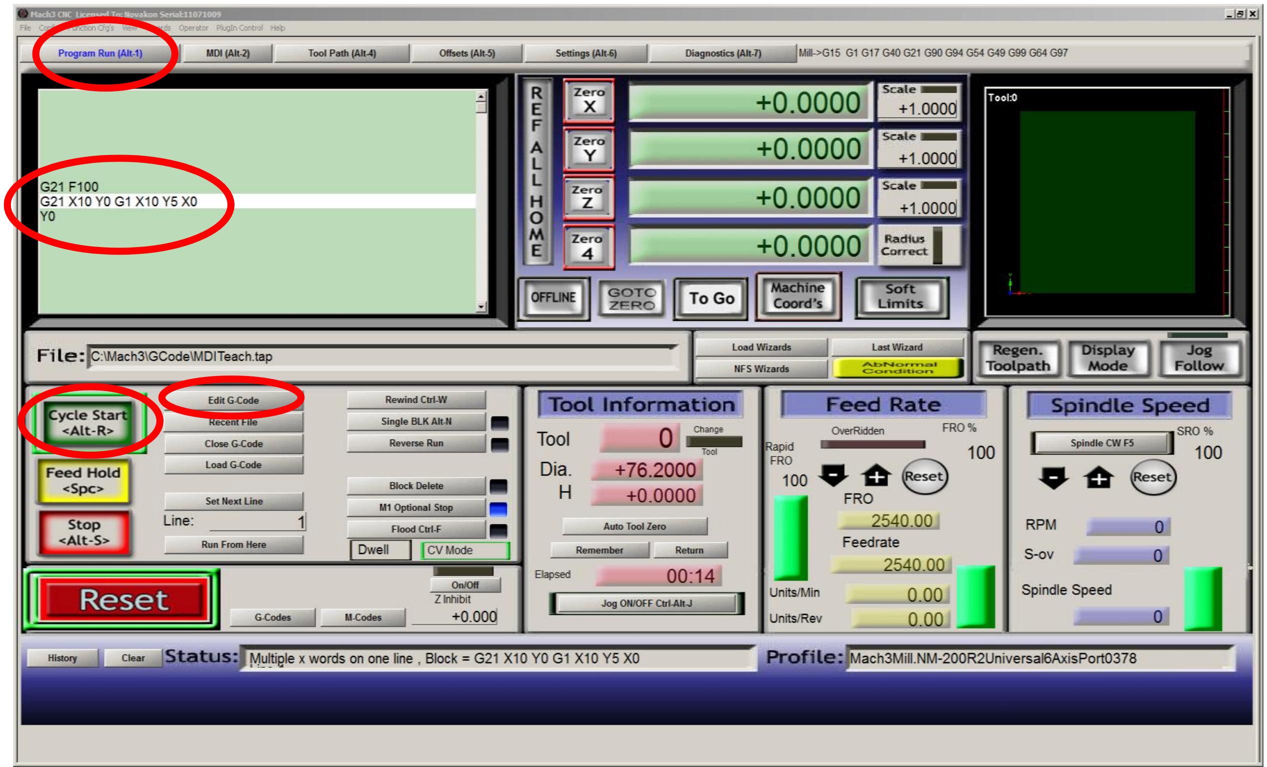


Figure 5-11

WIZARDS – CAM WITHOUT A DEDICATED CAM SOFTWARE

Mach3 allows the use of add-on screens which allow the automation of quite complex tasks by prompting the user to provide the relevant information. In this sense they are rather like the so called Wizards in Windows software that guide you through the information required for a task. The classic Windows Wizard will handle tasks line importing a file to a database or spreadsheet. In Mach3, examples of Wizards include cutting a circular pocket, drilling a grid of holes and digitizing the surface of a model part.

It is easy to try one out. In the Program Run screen, click Load Wizards (Figure 5-12). A table of the Wizards installed on your system will be displayed (Figure 5-13).

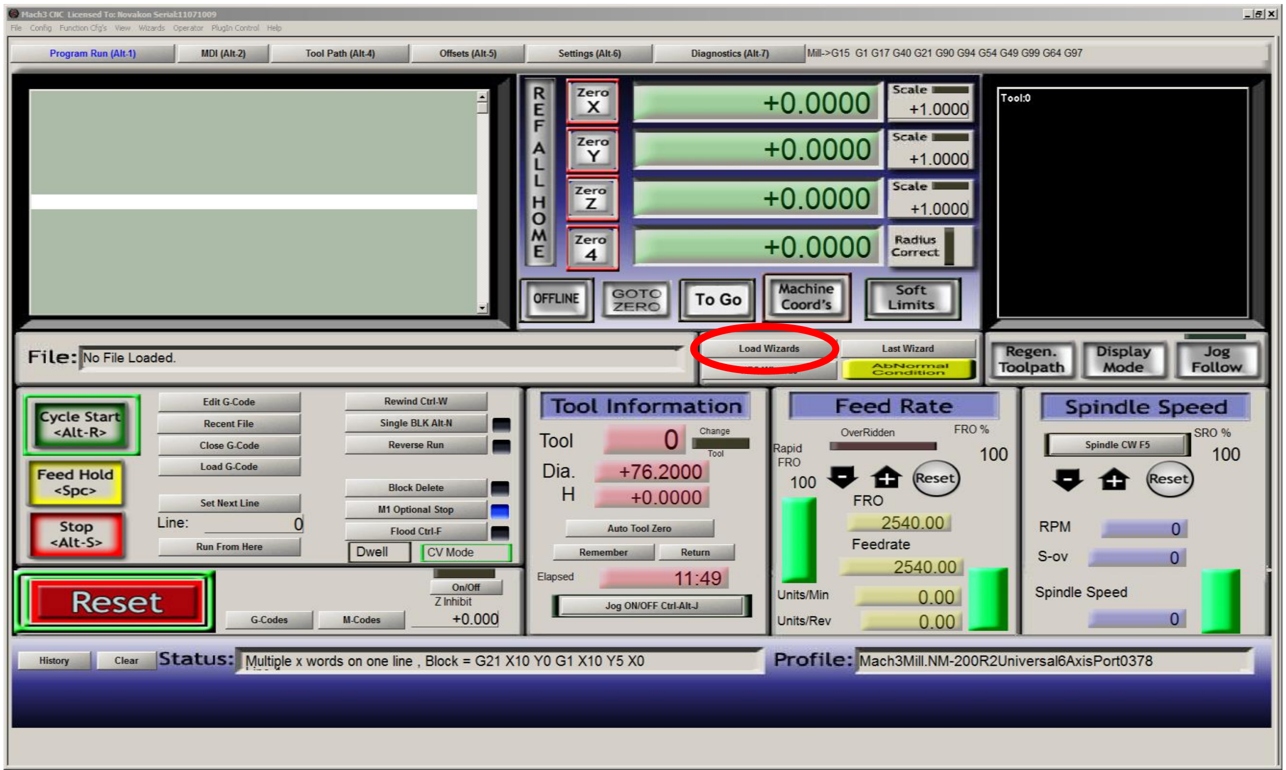


Figure 5-12 Program Run Screen

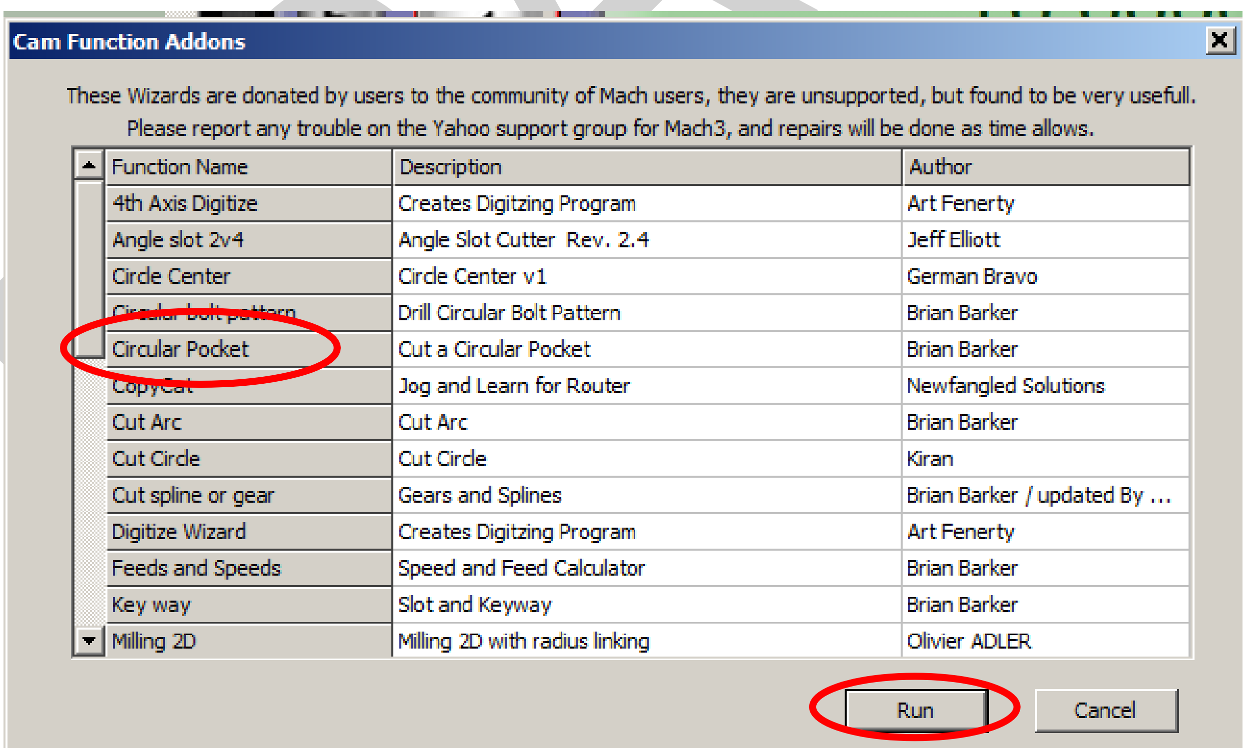


Figure 5-13 Table of Wizards

As an example, click on the line for Circular pocket (Figure 5-13), and click *Run* to display the Mach3 (Figure 5-14). This shows the screen with some default options.

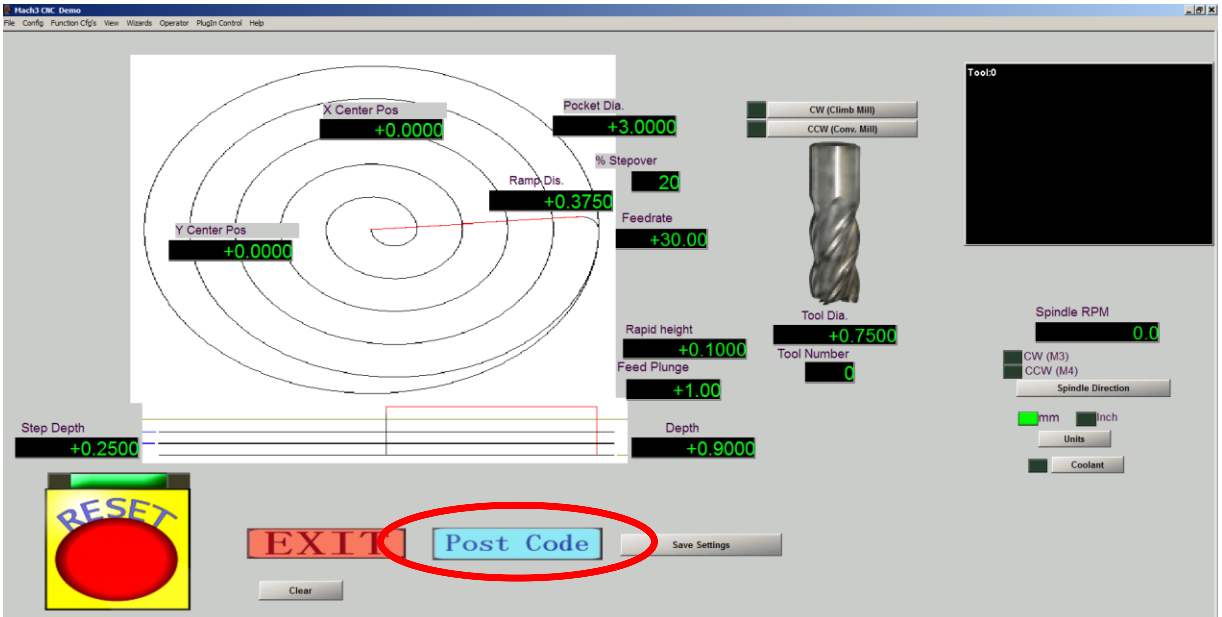


Figure 5-14

Notice that you can choose the units to work in, the position of the center of the pocket, how the tool is to enter the material and so on. You have to set the spindle speed manually using the *Spindle Speed* insert box on the Program Run Screen.

When you are satisfied with the pocket, click the *Post Code* button (Figure 5-14). This writes a G-Code part program and loads it into Mach3. This is just an automation of what you did in the example on Teaching. The tool path display shows the cuts that will be made. You can revise your parameters to take smaller cuts and re-post the code. If you wish, you can save the settings so the next time you run the Wizard, the initial data will be what is currently defined.

When you click Exit you will be returned to the main Mach3 screens (Figure 5-15) and you can run the Wizard generated part program.

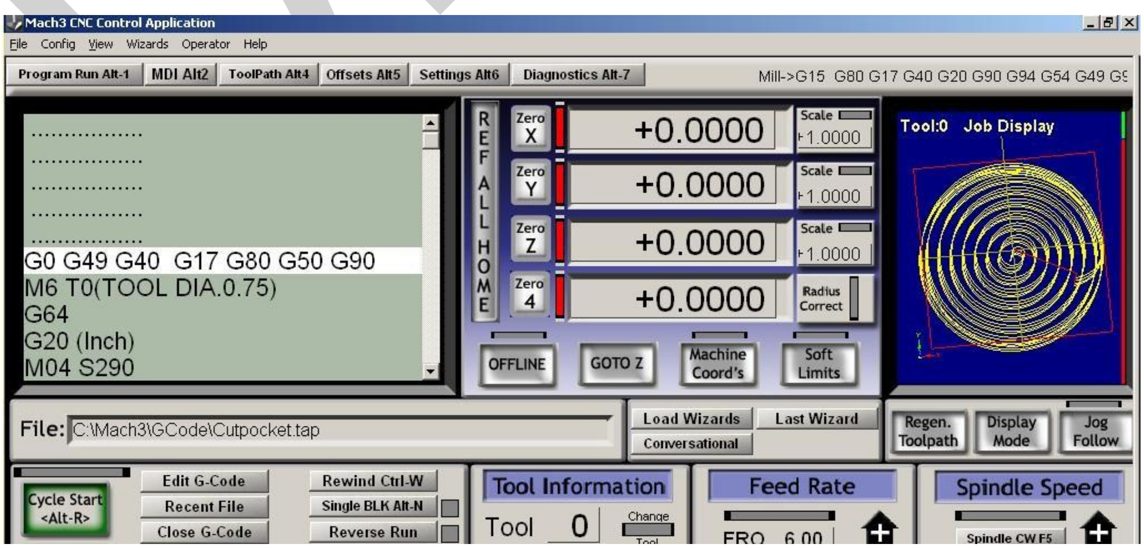


Figure 5-15 THE RESULT OF CIRCULAR POCKET READY TO RUN

RUNNING A G-CODE PROGRAM

Now it is time to input and edit a Part Program. You will be able to edit programs without leaving Mach3.

Use Windows Notepad to enter the following lines into a text file and save it in a convenient folder (ie. My Documents) as spiral. Tap G-Code files should be stored as "TAP" files.

You must choose *All Files* in the *Save As Type* drop-down menu or Notepad editor will append the .TXT to your filename and Mach3 will not be able to identify it.

```
g20 f100 g00 x1 y0 z0 g03 x1
y0 z-0.2 i-1 j0 g03 x1 y0 z-
0.4 i-1 j0 g03 x1 y0 z-0.6 i-
1 j0 g03 x1 y0 z-0.8 i-1 j0
g03 x1 y0 z-1.0 i-1 j0 g03 x1
y0 z-1.2 i-1 j0 m0
```

Again all the "0" are zeros. Don't forget to press the *Enter* key after the *m0*. Use the *File>Load G-Code* menu to load this program. You will notice that it is displayed in the G-Code window.

On the *Program Run* screen you can try the effect of the *Start Cycle*, *Pause, Stop*, and *Rewind* buttons and their shortcuts.

As you run the program you may notice that the highlighted line moves in a peculiar way in the G-Code window. Mach3 reads ahead and plans its moves to avoid the tool path having to slow down more than necessary. This look ahead is reflected in the display and when you pause.

You can go to any line of code scrolling the display so the line is highlighted. You can then use *Run from here*.

Note: You should always run your programs from a hard drive not a floppy drive or USB "key". Mach3 needs high-speed access to the file which ~~it~~ in turn maps into memory. The program file must not be read-only.

TOOL PATH DISPLAY

VIEWING THE TOOL PATH

The Program Run screen has a blank square on it when Mach3 is first loaded. When the Spiral program is loaded, you will see it change to a circle inside a square. You are looking straight down on the tool path for the programmed part, i.e. in Mach3 Mill you are looking perpendicular to the X-Y plane.

The display is like a wire model of the path that the tool will follow placed inside a clear sphere. By dragging the mouse over the window you can rotate the "sphere" and thus see the model from different angles. The set of axes ~~in~~ at the top left hand corner, show you the orientation of the X, Y and Z axes. So, by dragging the mouse from the center in an upward direction, the "sphere" can be manipulated around the axes, allowing viewing of the circle actually representing a spiral cut downward-(in the negative Z direction).

Each of the *G03* lines in the Spiral program above draws a circle while simultaneously lowering the tool 0.2 in the Z direction. You can also see the initial *G00* move which is a straight line.

You can if you wish produce a display like the conventional isometric view of the tool path.

A few minutes of "play" will soon give you confidence in what can be done. Your display may be a different color to that shown in Figure 5-16. The colors can be configured. See Configure ToolPath Section, page 5-42.

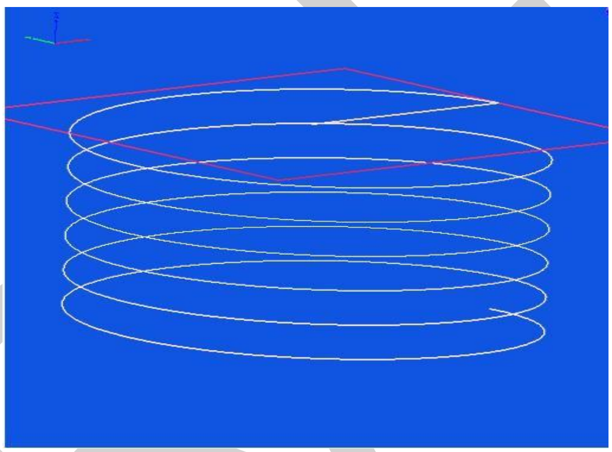


Figure 5-16 TOOL PATH FROM SPIRAL.TAP

PANNING AND ZOOMING THE TOOL PATH DISPLAY

The tool path display can be zoomed by dragging the cursor in its window with the Shift key depressed.

The tool path display can be panned in its window by dragging the cursor in the window with the Right mouse button held.

Double clicking the tool path window restores the display to the original perpendicular view with no zoom applied.

Note: You cannot Pan or Zoom while the machine tool is running.

OTHER SCREEN FEATURES

Finally it is worth browsing through some of the other Wizards and all the screens. As a small challenge you might like to see if you can identify the following useful features:

- A button for estimating the time that a part program will take to run on the actual machine tool
- The controls for overriding the feed rate selected in the part program
- DROs which give the extent of movement of the tool in all axes for the loaded part program
- A screen that lets you set up information like where you want the Z axis to be put to make X and Y moves safe from hitting clamps etc.
- A screen that lets you monitor the logic levels (zero and one) on all Mach3s inputs and outputs

HARDWARE REQUIREMENTS

THE PC PARALLEL PORT AND ITS HISTORY

Mach3 interfaces to the motor drivers, limit switches, and other hardware through your PC's parallel port. This section describes the characteristics of the parallel port.

When IBM designed the original PC (160k floppy disc drive, 64kbytes of RAM!), they provided an interface for connecting printers using a 25 conductor cable. This is the foundation of the parallel port we have on most PCs today. As it is a very simple way of transferring data, it has been used for many things other than connecting printers. You can transfer files between PCs, attach copy protection "dongles," connect peripherals such as scanners and Zip drives, and of course control machine tools using it. The USB interface is taking over many of these functions, and this conveniently leaves the parallel port free for Mach3.

The parallel port connector on the PC is a 25-pin female "D" connector. The connector, as seen from the back of the PC, is shown inFigure 5-17. The arrows give the direction of information flow relative to the PC. Thus, for example, pin 15, the second pin from the right on the bottom row, is an input to the PC.

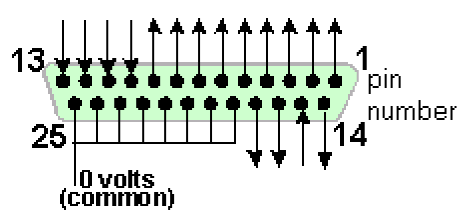


Figure 5-17 PARALLEL PORT FEMALE CONNECTOR (SEEN FROM BACK OF PC)

Note: Convertors which plug into a USB port and have a 25 pin connector will **not** drive a machine tool using Mach3, even though they are perfectly suitable for the simpler task of connecting a printer.

LOGIC SIGNALS

All the signals output by Mach3 and input to it are binary digital (i.e. zeros and ones). These signals are voltages supplied by the output pins or supplied to the input pins of the parallel port. These voltages are measured relative to the computer's 0 volt line, which is connected to pins 18 to 25 of the port connector.

The first successful family (74xx series) of integrated circuits used TTL (transistor-transistor logic). In TTL circuits, any voltage between 0 and 0.8 volts is called “lo” and any voltage between 2.4 and 5 volts is called “hi.” Connecting a negative voltage or anything above 5 volts to a TTL input will produce smoke. The parallel port was originally built using TTL and to this day these voltages define its “lo” and “hi” signals. Notice that in the worst case there is only 1.6 volts difference between them.

It is, of course, arbitrary whether we say that a “lo” represents a logic one or a logic zero. As is explained below, however, letting “lo” = one is usually more desirable in most practical interface circuits.

For an output signal to do anything, some current will have to flow in the circuit connected to it. When it is “hi,” current will flow out of the computer. When it is “lo,” current will flow into the computer. The more current you have flowing in, the harder it is to keep the voltage near zero, so the nearer to the permitted limit of 0.8 volts “lo” will become. Similarly, current flowing out of a “hi” will make the voltage be lower and nearer to the 2.4 volts lower limit. So with too much current the difference between “lo” and “hi” will be even less than 1.6 volts and things will become unreliable. Finally, it's worth noting you are allowed roughly 20 times more current flowing into a “lo” than you are allowed flowing out of a “hi”.

The net result is that it is best to assign logic 1 to be a “lo” signal. Fairly obviously this is called active lo logic. The main practical disadvantage of it is that the device connected to the parallel port has to have a 5 volt supply to it. This is sometimes taken from the PC game port socket or from a power supply in the device that is connected.

Pins 2 to 9 are likely to have similar properties (they are the data pins when printing). Pin 1 is also vital in printing, but the other output pins are little used and may be less powerful in a carefully “optimized” design. A good isolating breakout board will protect you from these electrical compatibility problems.

THE E-STOP CONTROL

The Pulsar Mill has a movable Emergency Stop (E-Stop) button with a big red mushroom head (Figure 5-18). The E-Stop button should be located so that you can easily and safely reach it from wherever you might be when you are operating the machine. This is particularly vital on a CNC machine. Refer to page 4-1 for restarting the Pulsar Mill after the E-Stop button has been pushed.

Figure 5-18

The E-Stop button stops all activity in the Pulsar Mill as quickly as is safely possible. The spindle stops rotating and all axes stop moving. This happens without relying on software – so we are talking about relays and contactors. The circuit tells Mach3 what you have done, and there is a special, mandatory input for this. It is not be good enough to just turn off the AC power for an E-Stop event, because the energy stored in DC smoothing capacitors can allow motors to run on for some considerable time.

HOME REFERENCING IN ACTION

When you request referencing (by button or G-Code) the axis (or axes) that have Home switches defined will travel (at a selectable low speed) in the defined direction until the Home switch operates. The axis will then move back in the other direction so as to be off the switch. During referencing, the limits do not apply.

When you have referenced an axis, then zero or some other value, which is set in the *Home Off(set)* column of the **Config>Homing/Limits** dialog, can be loaded into the axis DRO as its absolute machine coordinate. If you use zero, then the Home switch position is also the machine zero position of the axis. If the reference goes in the negative direction of an axis (usual for X and Y), then you might get referencing to load something like -0.5" into the DRO. This means that the Home is half an inch clear of the limit. This wastes a bit of the axis travel but if you overshoot, when jogging to Home, you will not accidentally trip the limits. See also Software Limits as another way of solving this problem.

If you ask Mach3 to reference **before** you jog off the switch, then it will travel in the opposite direction from its "reference" direction (because the machine already on the Home switch) and stop when the machine gets off the switch. This is fine when you have a separate Home switch or are on the Limit at the **reference end of the axis**. If, however, you are on the other Limit switch (and Mach3 cannot know this, if the switches are shared), then the axis will move away from the actual Home point until it crashes. So the advice is **always jog carefully off the Limit switches, then reference.** If you are concerned about this problem, it is possible to configure Mach3 so it will not automatically jog off the home switch.

SPINDLE AND COOLANT CONTROL

ON/OFF MOTOR CONTROL

M3 and a screen button will request that the spindle starts in a clockwise direction. *M4* will request that the spindle starts in a counterclockwise direction. The Pulsar Mill is not setup to utilize the *M4* command and run in reverse. M5 requests that the spindle stops.

COOLANT

The *M7* command turns on flood coolant for the Pulsar Mill. The *M9* command turns off flood coolant. The *M8* command (mist coolant) is not valid for the Pulsar Mill.

CONFIGURING MACH3 FOR YOUR MACHINE AND DRIVES

This Section is for users that need advanced configuration of Mach3. Mach3, as provided, includes all of the proper configurations required to run the Pulsar Mill as initially shipped from the factory. Any changes made to the default Pulsar Mill settings are at your own risk!

Novakon recommends that you keep a printed copy of how your version of Mach3 is configured. The information will be extremely helpful if you ever need to re-install the software from scratch. Mach3 stores this information in an `.XML` file in the `\Mach3` folder that you can view and print If you created a custom profile for your configuration, there will be an `.XML` file ("`MyMill.xml`," or whatever you named your profile) in the `\Mach3` folder containing that profile information. Although you could edit the `.XML` file, Novakon **strongly** recommends that you do not. Change the Mach3 configuration only through the program dialogs.

A CONFIGURATION STRATEGY

This chapter contains a lot of detail. You should, however, find the configuration process to be straightforward if you take it step by step, testing as you go. A good strategy is to skim the chapter, then work through it on your Pulsar Mill.

Virtually all the work you will do in this chapter is based on dialog boxes reached from the **Config**(ure) menu. These are identified in this manual by, for example, **Config>Ports and Pins,** which means that you choose the **Ports and Pins** entry from the **Config** menu.

INITIAL CONFIGURATION

Begin with the Config>Ports and Pins dialog. Figure 5-19.

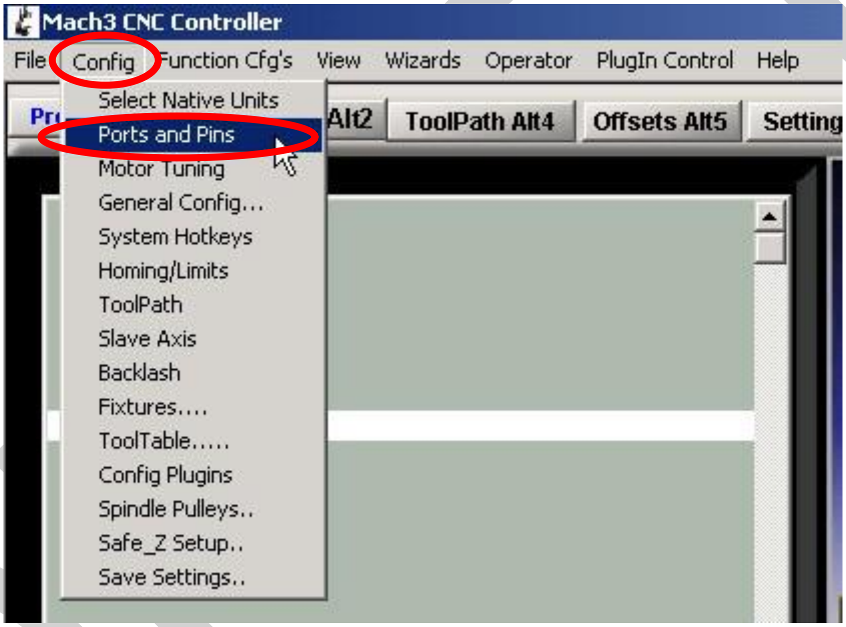


Figure 5-19 SELECTING THE PORTS AND PINS DIALOG FROM THE CONFIG MENU

Select the **Port Setup and Axis Selection** tab on the **Ports and Pins** dialog, as shown in Figure 5-20.

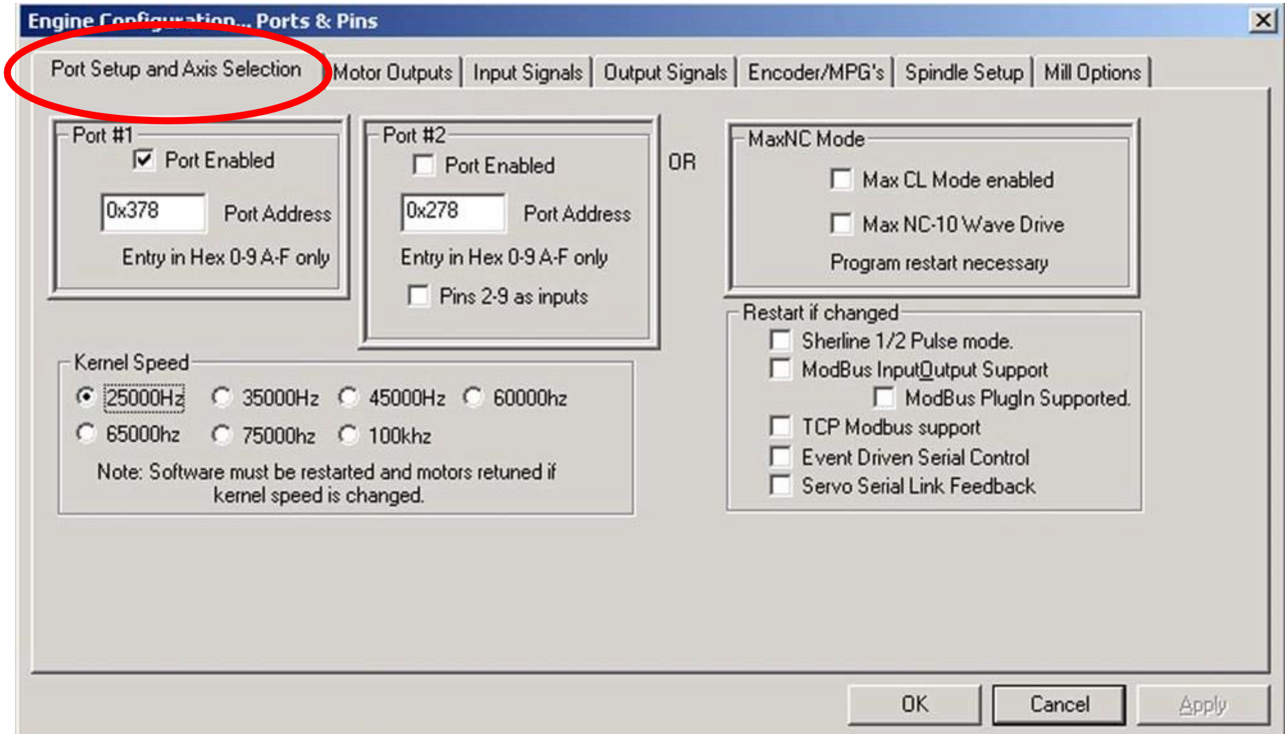


Figure 5-20 PORT SETUP AND AXIS SELECTION TAB ON THE PORTS AND PINS DIALOG

If you are going to use only one parallel port, and it is the one on your computer's motherboard, then the default address of Port 1 of 0x378 (Hexadecimal 378) is almost certainly correct.

If you are using one or more PCI add-on parallel port cards, then you will need to determine the address to which each responds. There are no standards! Open the Windows Control Panel from the Windows *Start* button. Double click on *System* and choose the *Hardware* tab. Click the *Device Manager* button. Expand the tree for the item **Ports (COM & LPT)**.

Double click the first LPT or ECP port. Its properties will be displayed in a new window. Choose the **Resources** tab. The first number in the first IO range line is the address to use. Note the value and close the **Properties** dialog.

Note: Installing or removing any PCI card can change the address of a PCI parallel port card, even if you have not touched it.

If you are going to use a second port, repeat the above steps for it.

Close the Device Manager, System Properties, and Control Panel windows.

Enter the address that you just looked up as the first port's address on the **Port Setup and Axis Selection** dialog. Do not provide the 0x prefix to say it is Hexadecimal, as Mach3 assumes this. If necessary, check *Enabled* for port 2 and enter its address in the same way.

Now click the *Apply* button to save these values. This is most important. Mach3 will not remember values when you change from tab to tab or close the Port & Pins dialog unless you click *Apply*.

DEFINING ENGINE FREQUENCY

The Mach3 driver can run at frequencies from 25,000 Hz (pulses per second) up to 100,000 Hz, depending on the speed of your processor and other loads placed on it when running Mach3.

The frequency you need depends on the maximum pulse rate you need to drive any axis at its top speed. 25,000 Hz will probably be suitable for stepper motor systems. With a 10 micro-step driver, you will get around 750 RPM from a standard 1.8 degree stepper motor with a 25,000 Hz pulse rate. Higher pulse rates are needed to achieve desired motor RPM for servo drives that have high resolution shaft encoders on the motor. Further details are given in the section on Motor Tuning.

Computers with a 1 GHz clock speed will almost certainly be able to run at 35,000 Hz, so you can choose this if you need a high step rate (for example, if you have very fine pitch lead screws).

The demonstration version of Mach3 will run at 25,000 Hz **only**. In addition, if Mach3 is forcibly closed, then on re-start it will automatically revert to 25,000 Hz operation. The actual frequency of the running system is displayed on the standard **Diagnostics** screen.

Click the box next to the desired kernel speed. Don't forget to click the *Apply* button before proceeding.

DEFINING SPECIAL FEATURES

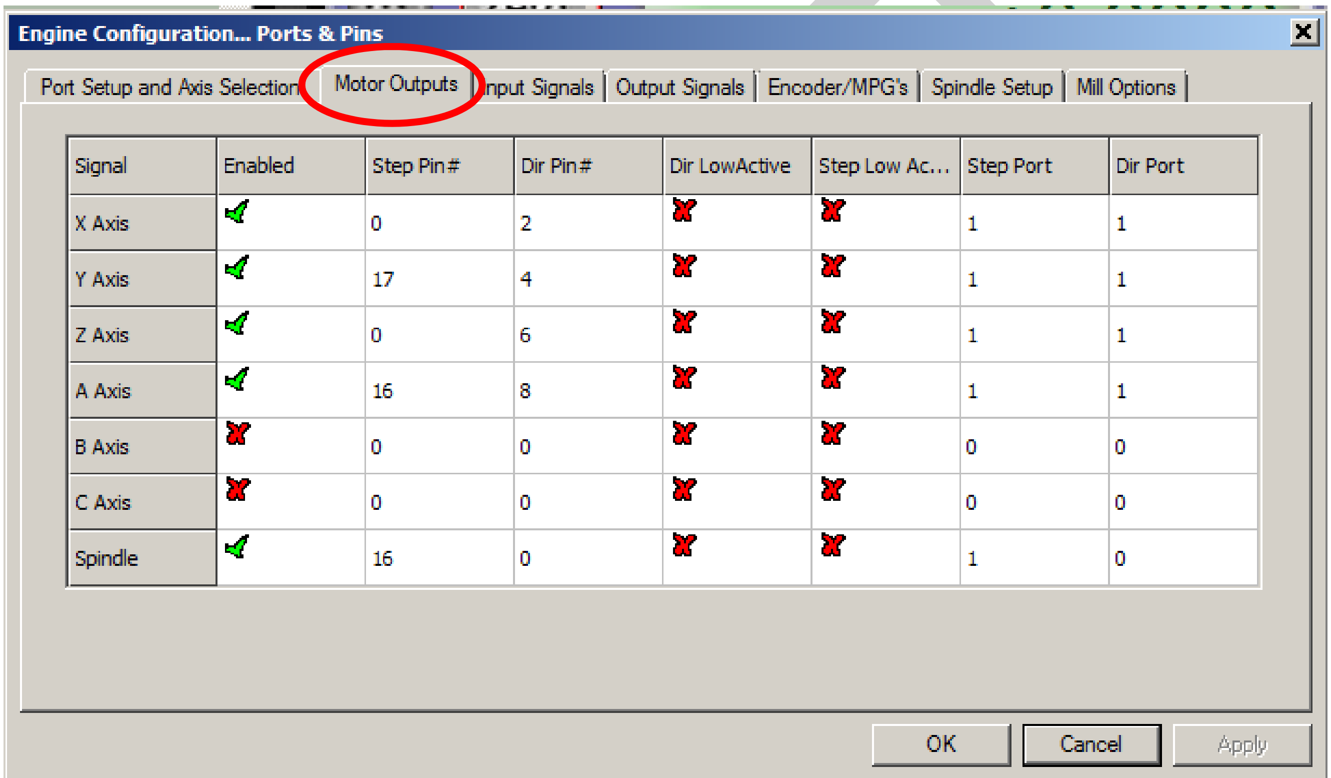
The **Port Setup and Axis Selection** dialog includes check boxes for a variety of special configurations. They should be self-explanatory if you have the relevant hardware in your system. If not, then leave then unchecked. Don't forget to click the *Apply* button before proceeding.

DEFINING INPUT AND OUTPUT SIGNALS TO USE

Now that you have established the basic configuration, it is time to define which input and output signals you will use and which parallel port and pin will be used for each signal. The documentation for your breakout board may give guidance on what outputs to use if it has been designed for use with Mach3, or the board may be supplied with a skeleton Profile (`.XML`) file with these connections already defined.

AXIS AND SPINDLE OUTPUT SIGNALS TO BE USED

View the **Motor Outputs** tab of the **Ports and Pins** dialog (Figure 5-21).



Signal	Enabled	Step Pin#	Dir Pin#	Dir LowActive	Step Low Ac...	Step Port	Dir Port
X Axis	✓	0	2	✗	✗	1	1
Y Axis	✓	17	4	✗	✗	1	1
Z Axis	✓	0	6	✗	✗	1	1
A Axis	✓	16	8	✗	✗	1	1
B Axis	✗	0	0	✗	✗	0	0
C Axis	✗	0	0	✗	✗	0	0
Spindle	✓	16	0	✗	✗	1	0

Figure 5-21 MOTOR OUTPUTS TAB ON THE PORTS AND PINS DIALOG

Define where the drives for your X, Y and Z axes are connected, and click in the **Enabled** column to get a check-mark to Enable these axes. If any axis is enabled that shouldn't be, click in the **Enabled** column to change the green check to a red X.

If you need to edit any boxes in the **Step Pin #**, **Dir Pin #**, **Step Port**, or **Dir Port** columns, double- click the appropriate boxes and edit the entries.

If your interface hardware requires an active-lo signal, ensure that these columns are checked for the Step and Dir signals. Figure 5-21 show the correct settings for a Pulsar with the optional fourth axis (A-axis).

Click the *Apply* button to save the data on this tab.

Select the **Input Signals** tab to display the screen shown in Figure 5-22.

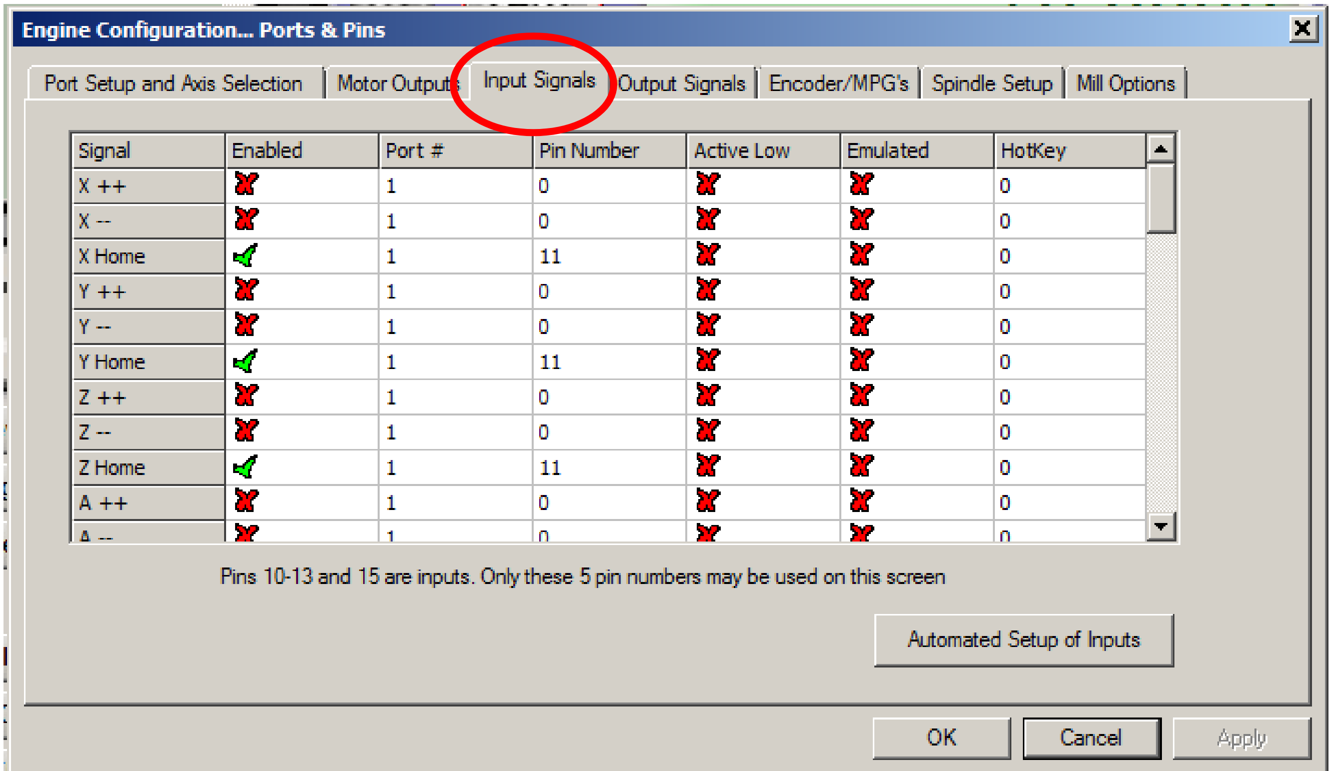


Figure 5-22 INPUT SIGNALS TAB ON THE PORTS AND PINS DIALOG

Table 5-1 LIST OF POSSIBLE INPUT SIGNALS gives the complete list of input signals available in MACH3. Refer to www.machsupport.com for further information on using the Input Signals in Mach3.

Signal	Meaning	Signal	Meaning
X ++	X axis + limit switch hit	OEM Trig #1	User-defined
X --	X axis – limit switch hit	OEM Trig #2	User-defined
X Home	X axis Home switch hit	OEM Trig #3	User-defined
Y ++	Y axis + limit switch hit	OEM Trig #4	User-defined
Y --	Y axis – limit switch hit	OEM Trig #5	User-defined
Y Home	Z axis Home switch hit	OEM Trig #6	User-defined
Z ++	Z axis + limit switch hit	OEM Trig #7	User-defined
Z --	Z axis – limit switch hit	OEM Trig #8	User-defined
Z Home	Z axis Home switch hit	OEM Trig #9	User-defined
A ++	A axis + limit switch hit	OEM Trig #10	User-defined
A --	A axis – limit switch hit	OEM Trig #11	User-defined
A Home	A axis Home switch hit	OEM Trig #12	User-defined
B ++	B axis + limit switch hit	OEM Trig #13	User-defined
B --	B axis – limit switch hit	OEM Trig #14	User-defined
B Home	B axis Home switch hit	OEM Trig #15	User-defined
C ++	C axis + limit switch hit	Timing	Spindle rotation sensor with more than one slot or mark
C --	C axis – limit switch hit	Jog X ++	Move X in + direction
C Home	C axis Home switch hit	Jog X --	Move X in – direction
Input #1	Safety guards not in place, or user-defined	Jog Y ++	Move Y in + direction
Input #2	User-defined	Jog Y --	Move Y in – direction
Input #3	User-defined	Jog Z ++	Move Z in + direction
Input #4	Single step, or user-defined	Jog Z --	Move Z in – direction
Probe	Digitizing probe enable	Jog A ++	Move A in + direction
Index	Spindle rotation sensor with one slot or mark	Jog A --	Move A in – direction
Limit Override	Limit override enable		
E-Stop	Emergency Stop button pressed		
THC On	Plasma torch control		
THC Up	Plasma torch control		
THC Down	Plasma torch control		

Table 5-1 LIST OF POSSIBLE INPUT SIGNALS

OUTPUT SIGNALS

Use the **Output Signals** tab to define the outputs you require. SeeFigure 5-23.

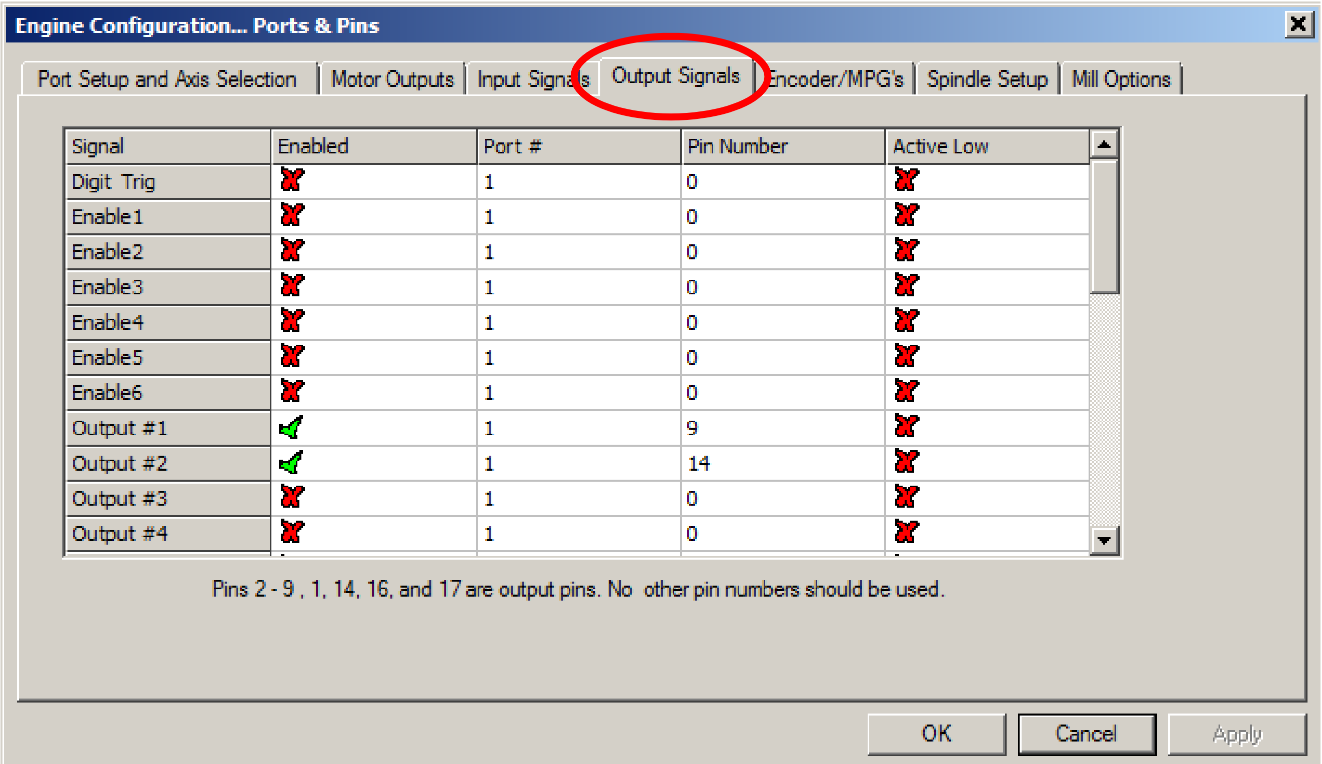


Figure 5-23 OUTPUT SIGNALS TAB ON THE PORTS AND PINS DIALOG

Table 5-2 provides a complete list of output signals.

Signal	**Meaning**	**Signal**	**Meaning**
Digit Trig		Current Hi/Lo	
Enable1		Output #7	
Enable2		Output #8	
Enable3		Output #9	
Enable4		Output #10	
Enable5		Output #11	
Enable6		Output #12	
Output #1		Output #13	
Output #2		Output #14	
Output #3		Output #15	
Output #4		Output #16	
Output #5		Output #17	
Output #6		Output #18	
Charge Pump		Output #19	
Charge Pump2		Output #20	

Table 5-2 Possible Output Signals

You will probably want to use only one *Enable* output (as all the axis drives can be connected to it). If you are using the charge pump/pulse monitor feature, then you can enable your axis drives from its output.

The *Output #* signals are for use to control a stop/start spindle (clockwise and optionally counter clock-wise), the Flood and Mist coolant pumps or valves, and for control by your own customized Mach3 buttons or macros.

The *Charge Pump* line should be enabled and defined if your breakout board accepts this pulse input to continually verify correct operation of Mach3. *Charge Pump2* is used if you have a second breakout board connected to the second port or want to verify the operation of the second port itself.

Click the *Apply* button to save the data on this tab.

DEFINING ENCODER AND MANUAL PULSE GENERATOR (MPG) INPUTS

Use the **Encoder/MPGs** tab to define the connections and the resolution of linear encoders or Manual Pulse Generators (MPGs) used for jogging the axes.

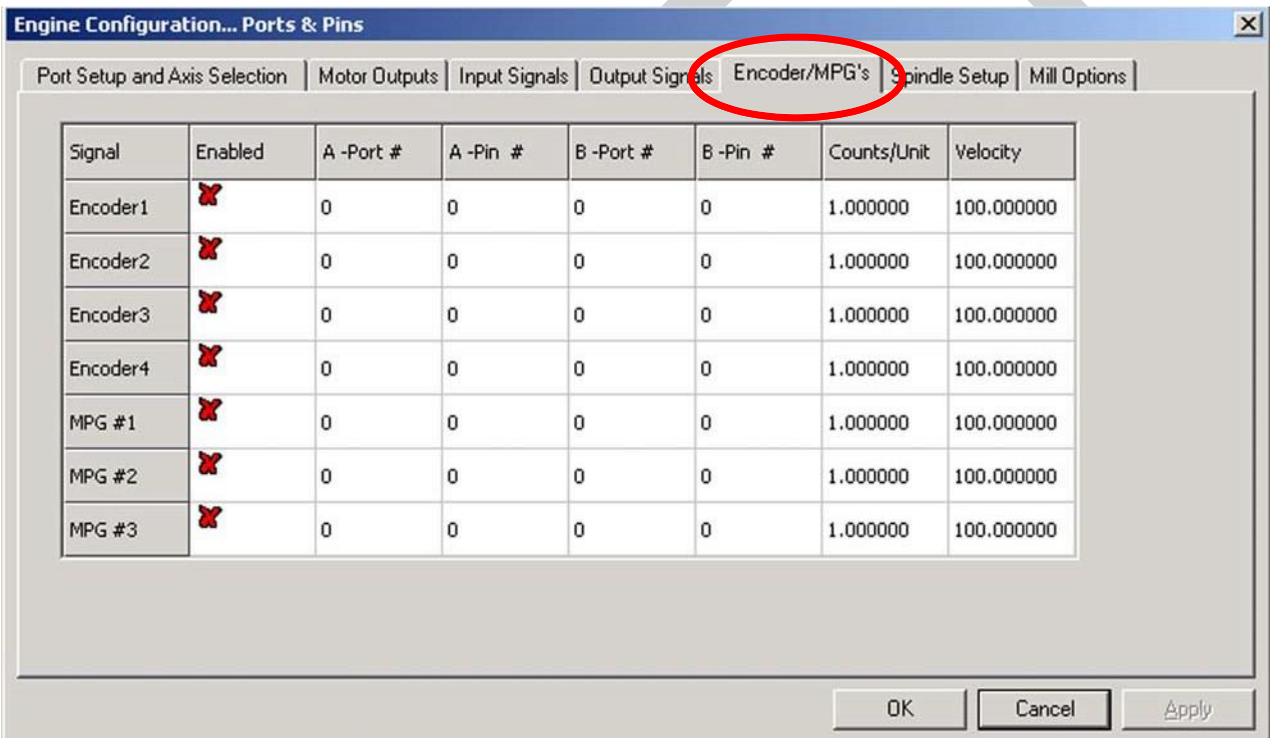


Figure 5-24 Encoder/MPG Tab on the Ports and Pins Dialog

This dialog does not need an active-lo selection. If the encoders count the wrong way, just swap the pins allocated for the A and B inputs.

SETTING UP ENCODERS

The *Counts per unit* value should be set to correspond to the resolution of the encoder and your selection of Native units. For example, a linear scale with rulings at 20 microns produces a count every 5 microns (remember the quadrature signal), or 200 counts per unit (millimeter). If you have Native units set to inches, then it would be 200 x 25.4 = 5080 counts per unit (inch) because there are 25.4 millimeters per inch. The *Velocity* value is not used.

SETTING UP MPGS

The *Counts per unit* value defines the number of quadrature counts that need to be generated for Mach3 to sense movement of the MPG. For a 100 counts-per-revolution (CPR) encoder, a value of 2 should be suitable. For higher resolutions, you will probably want to increase this value to get the mechanical sensitivity you want. A value of 100 works well with 1024 CPR encoders.

The *Velocity* value determines the scaling of pulses sent to the axis being controlled by the MPG. The lower the value given in *Velocity,* the faster the axis will move. Its value is best set by experiment to give a reasonable axis movement speed when spinning the MPG as fast as is comfortable.

CONFIGURING THE SPINDLE

The next tab on **Config>Ports & Pins** is **Spindle Setup**. This is used to define the way in which your spindle and coolant is to be controlled.

The Pulsar Mill uses the Step/Dir Motor Control signal. The correct setup is shown in **Error!**

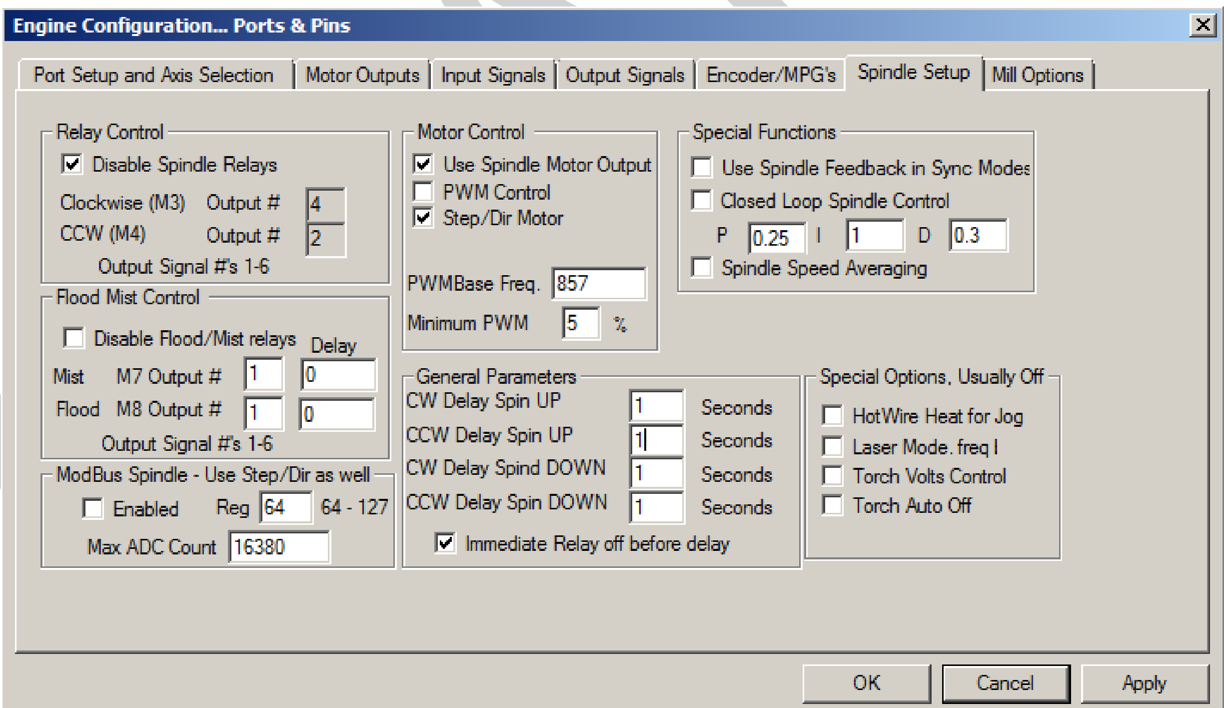


Figure 5-25 Spindle Setup

Reference source not found..

Mach3 can output a pulse width modulated signal whose duty cycle is the percentage of full speed that you require. You could, for example, convert the duty cycle of the signal to a voltage (PWM signal on for 0% of time gives 0 volts, 25% gives 2.5 volts, 50% gives 5 volts, up to 100% gives 10 volts) and use this to control an induction motor with a variable frequency inverter drive

Figure 5-26 and Figure 5-27 show the pulse width at approximately 50% of the cycle and 20% of the cycle.

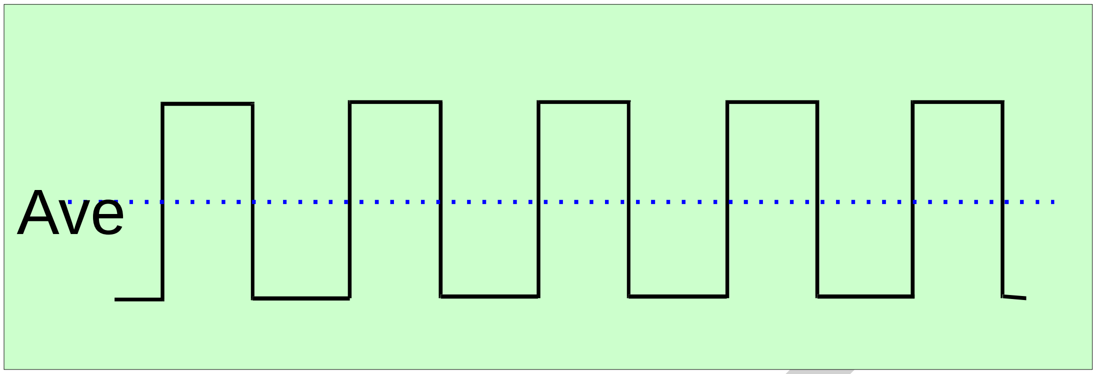


Figure 5-26 A 50% PULSE WIDTH MODULATED SIGNAL

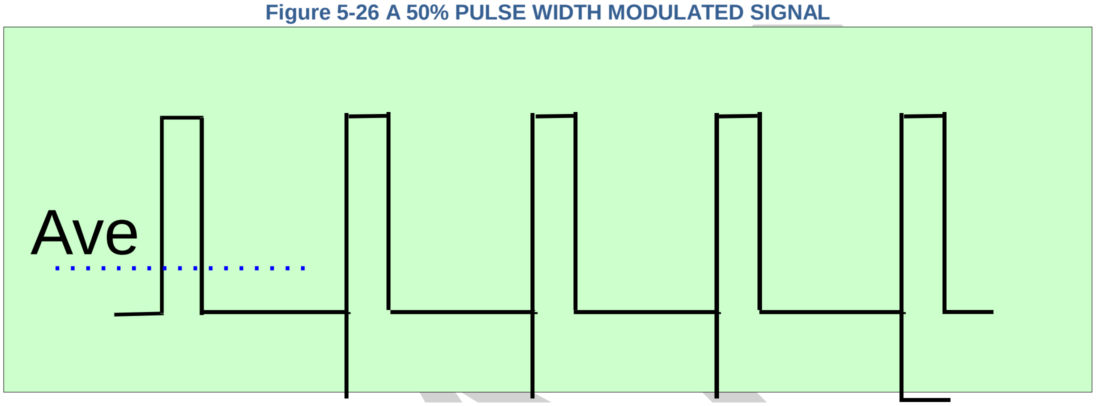


Figure 5-27 A 20% Pulse Width Modulated Signal

To turn the PWM spindle speed signal into a proportional DC voltage, the pulse signal must be transformed. In essence, a circuit must be used to find the average of the pulse width modulated signal. The circuit can be a simple capacitor and resistor, or it can be much more complex depending (a) on how linear you want the relationship between the width and the final output voltage, and (b) on the speed of response you need to the changing pulse width.

The PWM signal is output on the spindle Step pin. You will need to take special precautions to switch off the motor at low speeds using the Motor Clockwise/Counterclockwise outputs.

Refer to the manufacturer's documentation that should come with your controller. Further details may be found by using "PWM converter" or "PWM Digispeed" as a search term to Google or your favorite search engine.

COOLANT CONTROL

Code *M8* turns Flood coolant on, and M9 turns it off. Code *M7*, Mist coolant, is not initially installed on the Pulsar Mill .

GENERAL PARAMETERS

These let you specify a delay after starting or stopping the spindle before Mach3 will execute further commands (that is, a Dwell time). These delays can be used to allow time for acceleration before a cut is made and to provide some software protection from going directly from clockwise to counter clock- wise spindle rotation. Enter the dwell times in seconds.

Immediate Relay off before delay, if checked, will switch the spindle relay off as soon as an *M5* command is executed. If unchecked, the spindle relay will stay on until the spin-down delay period has elapsed.

DEFINING THE SETUP UNITS

With the basic functions working, it's time to configure the axis drives. The first thing to decide is whether you want to define their properties in Metric (millimeters) or Inch units. This is done using the **Config>Select Native Units** dialog. This is shown in Figure 5-28.

Figure 5-28 SELECT NATIVE UNITS DIALOG

The computations for axis configuration will be slightly easier if you choose the same system as your drive train (for example, the ball screw pitch) was made in. A screw with 0.2" lead (5 tpi) is easier to configure in inches than in millimeters. Similarly, a 2mm lead screw will be easier to configure in millimeters. Otherwise, you will need to multiply or divide by 25.4 to convert from one units system to the other when doing the axis configurations. The multiplication or division by 25.4 is not difficult, but it is just something else to think about. You will be able to run part programs using either units, which- ever option you choose here.

There is, on the other hand, a slight advantage in having the setup units be the units in which you usually work. If the units are your usual working unit, you can lock the DROs to display in this system whatever the part program is doing (i.e. switching units by G20 and G21).

So the choice is yours. Use **Config>Select Native Units** to choose Millimeters or Inches (see Figure 5-28). Once you have made a choice, you cannot change it without going back over all the following steps or total confusion will reign! A message box reminds you of this when you use **Config>Select Native Units**.

AUTOMATIC SETTING OF STEPS PER UNIT

Calculating the Steps Per Unit, explains how to calculate the steps per unit, but you may not be able to measure the gearing of your axis drive or know the exact pitch of a screw. Provided you can accurately measure the distance moved by an axis, however, perhaps using a dial test indicator and gage blocks, then Mach3 can calculate the steps per unit that should be configured. For best results, you should get the value approximately correct by calculation, even if you have to guess at some values, before performing the automatic setup.

Select the **Settings Alt6** tab in the main Mach3 CNC Controller window, as shown in Figure 5-29.

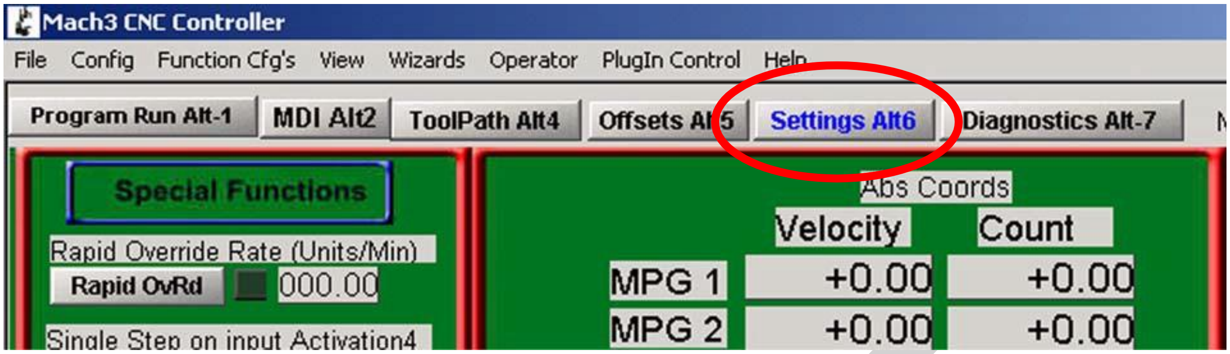


Figure 5-29 Settings Alt6 Tab

5) Figure 5-30 shows the *Set Steps per Unit* button on the **Settings Alt6** screen to initiate the automatic settings process. You will be prompted for the axis that you want to calibrate. Select an axis on the **Pick Axis to Calibrate** pop-up menu and click OK.

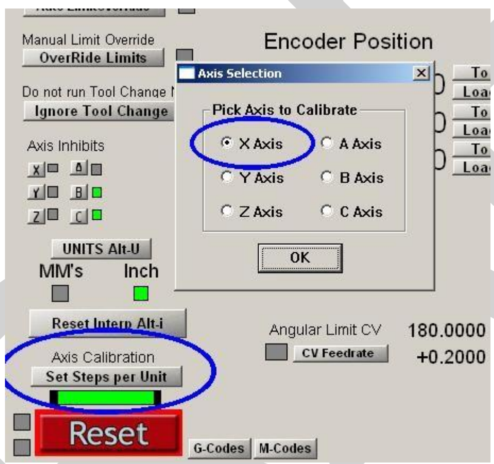


Figure 5-30 Automatic Setting of Steps per Unit

Another pop-up menu, shown in Figure 5-31, asks you to enter a nominal move distance. Mach3 will make that move based on its current settings, which of course may be wildly incorrect. Be ready to press the E-Stop button if the machine seems about to crash because your existing settings are too far off.

Figure 5-31 WINDOW TO ENTER NOMINAL MOVE DISTANCE

Finally, after the move you will be prompted to measure and enter the exact distance that was moved, as shown in Figure 5-32. This will be used to calculate the actual Steps per Unit of your machine axis. The "Saving and Testing Axis" section beginning on page 5-34 describes how to measure axis travel.

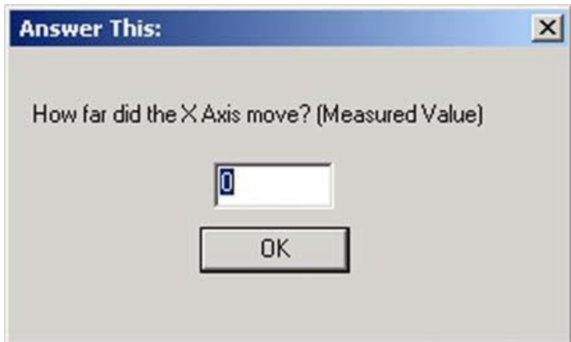


Figure 5-32 Window to Enter Actual Move Distance

CHOOSING AN ACCELERATION VALUE

INERTIA AND FORCES

No motor is able to change the speed of a mechanism instantly. A torque is needed to give angular momentum to the rotating parts (including the motor itself), and torque converted to force by the mechanism (screw and nut etc.) has to accelerate the machine parts and the tool or work piece. Some of the force also goes to overcome friction and, of course, to make the tool cut.

Mach3 will accelerate (and decelerate) the motor at a given rate (i.e. a straight line speed-time curve). If the motor can provide more torque than is needed for the cutting, friction, and inertia forces to be provided at the specified acceleration rate, then all is well. If the torque is insufficient, then the motor will lose steps or stall (if a stepper), or the servo position error will increase (if a servo). If the servo error gets too great, then the drive will probably signal a fault condition, but even if it does not the accuracy of the cut will suffer. This will be explained in more detail shortly.

TESTING DIFFERENT ACCELERATION VALUES

Try starting and stopping your machine with different settings of the Acceleration slider in the **Motor Tuning** dialog. At low acceleration (a gentle slope on the graph), you will be able to hear the speed ramping up and down.

WHY YOU WANT TO AVOID A BIG SERVO ERROR

Most moves made in a part program are coordinated with two, or more, axes moving together. Thus, in a move from (X=0, Y=0) to (X=2, Y=1), Mach3 will move the X axis at twice the speed of the Y axis. It not only co-ordinates the movements at constant speed, but also ensures that the speed-required relationship applies during acceleration and deceleration by accelerating all motions at a speed determined by the "slowest" axis.

If you specify an acceleration for a given axis greater than what the machine can deliver, Mach3 will nevertheless assume it can use that value. If in practice the axis motion lags behind what is commanded (i.e. the servo error is big), then the path cut in the work will be inaccurate.

CHOOSING AN ACCELERATION VALUE

It is possible, knowing all the masses of parts, moments of inertia of the motor and screws, friction forces, and the torque available from the motor, to calculate what acceleration can be achieved with a given error. Ballscrew and linear slide manufacturers' catalogues often include sample calculations.

Unless you must have the ultimate in performance from your machine, however, ArtSoft USA recommends simply setting the acceleration value so that test starts and stops sound “comfortable.” That may not be very scientific, but it usually gives good results. It is also a lot easier than doing all the calculations.

SAVING AND TESTING AXIS

Finally, don't forget to click *Save Axis Settings* to save the acceleration rate before you move on. You should now check your calculations by using manual data input (MDI) to make a defined G0 move and check the results. For a rough check you can use a steel rule. A more accurate test can be made with a Dial Test Indicator (DTI) and a gage block. Strictly, the DTI should be mounted in the tool holder, but for a conventional mill you can use the frame of the machine as the spindle does not move relative to the frame in the X-Y plane.

Suppose you are testing the X axis and have a four inch gage block. Select the MDI screen as shown in Figure 5-33.

Figure 5-33 Selecting MDI

Click in the Input box (Figure 5-34) and enter commands (*G20 G90*) to select inch units and absolute coordinates.

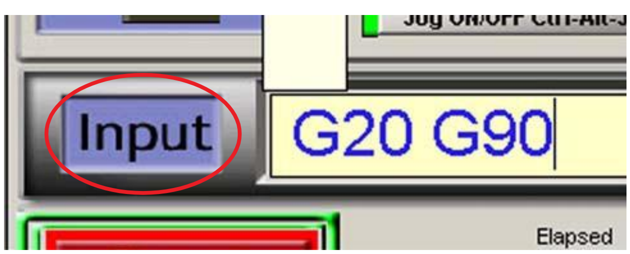


Figure 5-34 MANUALLY ENTERING G20 G90 COMMANDS

Set up a clamp on the table with its surface perpendicular to the table and to the table travel. Jog the axis so the DTI probe touches it. See the Section, *"Jogging" (page 5-7)*, for information about jog control buttons and jog mode settings. Ensure you finish by a move in the minus X direction. Rotate the DTI bezel to zero the reading. This is illustrated in Figure 5-35.

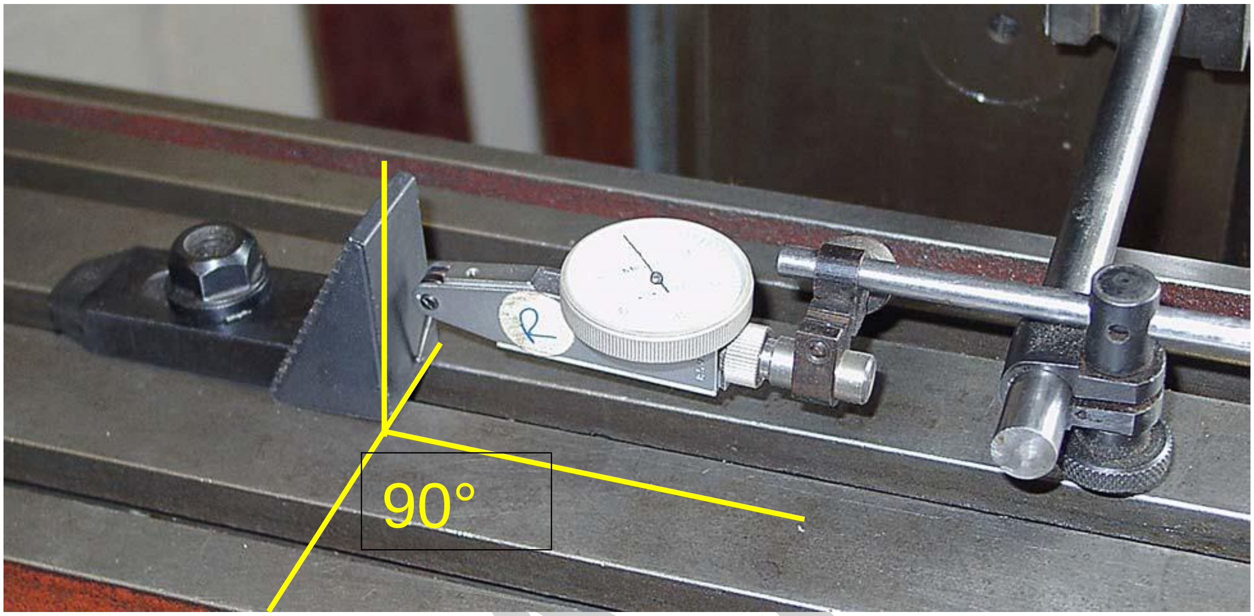


Figure 5-35 Establishing a zero position

Click the **Zero X** button to zero the X axis DRO.

Move the table to X = 4.5 by entering the command *G0 X4.5* in the MDI screen's Input box. The resulting gap between the block and the DTI should be about 4.5". If it is not, then there is something badly wrong with your calculations of the Steps per Unit value. Check and correct this.

Set the gage block against the stop block on the table (recall that in this example we're assuming a four inch block) and move to X = 4.0 by using the command *G0 X4*. This move is in the X minus direction as was the jog to zero the DTI against the block, so the effects of backlash in the mechanism will be eliminated. The reading on the DTI will give your positioning error. It should be only up to a thousandth of an inch or so. Figure 5-36 shows the gage block in position.

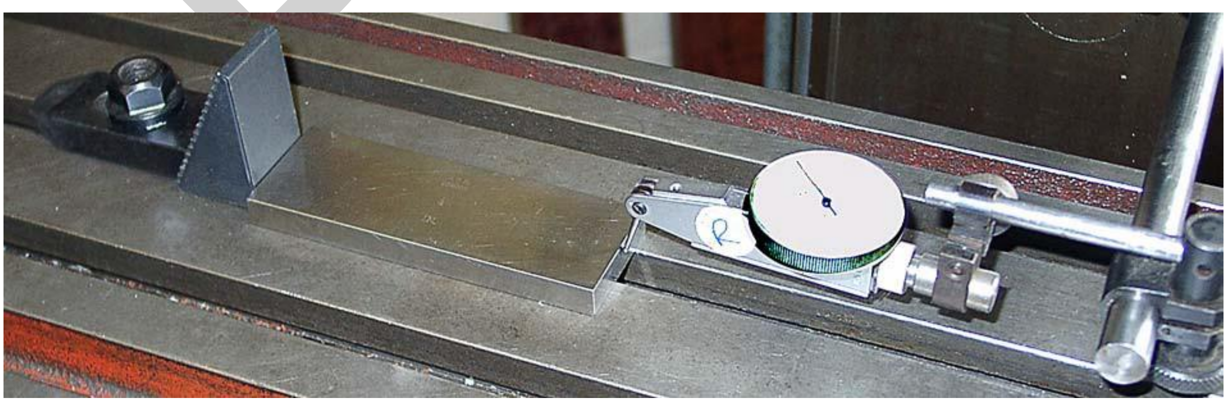


Figure 5-36 GAGE BLOCK IN POSITION

Remove the gage block and issue the command *G0 X0* to move back and check the zero value. Repeat the four inch test to get a set of perhaps 20 test values. See how reproducible the positioning is. If you get big variations, then there is something wrong mechanically. If you get a consistent error, then you can fine tune the Steps per Unit value using the technique described in the Section “Automatic Setting of Steps per Unit” , page 5-31 to achieve maximum accuracy.

Next, you should check that the axis does not lose steps in repeated moves at speed. Remove the gage block. Use the MDI screen to issue the G0 X0 command, and check the zero on the DTI.

Click the **Start Teach** button. Click in the **Input** space, and type in the following program:

```
F1000 (i.e. faster than possible but Mach3 will limit speed)
G20 G90 (Inch and Absolute)
M98 P1234 L50 (run subroutine 50 times) M30 (stop)
O1234
G1 X4
G1 X0 (do a feed rate move and move back) M99 (return)
```

The commands will be executed as you type them, but they will also be saved. When you have typed in all the program commands, click the **Stop Teach** button. Click the **Load/Edit** button. Select the **Program Run** tab. Click the **Cycle Start** button to run the program. Check that the motion sounds smooth.

When it finishes, the DTI should of course read zero. If it does not, adjust the maximum velocity and acceleration of the axis (downward!) and try again. If the program does not run properly, check to be sure you didn’t make a typing error. You can edit the program by clicking the **Edit G-Code** button on the **Program Run** tab.

REPEAT CONFIGURATION OF OTHER AXES

With the experience you will have gained configuring the first axis, you should be able to quickly repeat the process for the other axes.

MOTOR SPEED AND SPINDLE SPEED

Mach3 is used to control the spindle speed of the Pulsar Mill servo motor by Step and Direction pulses. The Step and Direction mode allow you to control the speed of the motor. by using the spindle speed command (the S-word) in the part program.

The machine's pulley ratio for the Pulsar Mill is accessed in Mach3 using the **Config>Spindle Pulleys...** dialog. Figure 5-37 shows the entries for the Pulsar Mill. The Pulsar Mill has only one pulley and therefore does not require a belt change to access its full RPM speed range (1 – 4500 RPM).

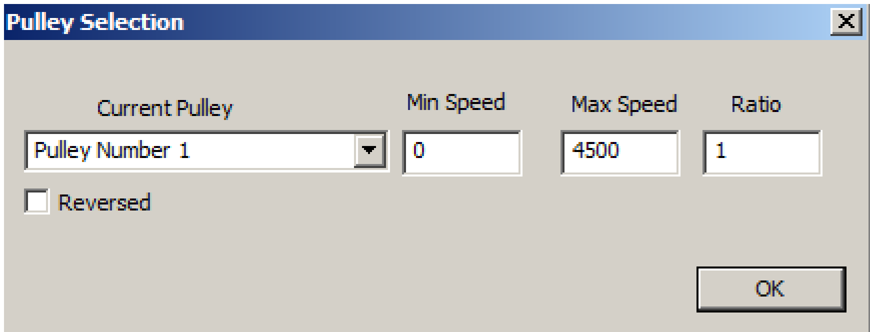


Figure 5-37 SPINDLE PULLEY SELECTION

The maximum speed (**Max Speed**) is the speed at which the **spindle** will rotate when the motor is at full speed. Full speed is achieved by the set *Vel* value on Motor Tuning “Spindle Axis” for Step and Direction. If a speed greater than the Max Speed is requested, Mach3 will display a warning and use the Max Speed value.

The **Min Speed** feature is used to set the minimum spindle speed for the Pulsar Mill mill to 0 RPM. In others words, with the aforementioned settings, Mach3 will run the Pulsar Mill spindle between 1-4500 RPM in one RPM increments.

STEP AND DIRECTION SPINDLE CONTROLLER

To configure the spindle motor for Step and Direction control, check the *Use Spindle Motor Output* and the *Step/Dir Motor* boxes on the **Config>Port and Pins>Spindle Setup** tab (Figure 5-25, page 5-29). Leave *PWM Control* unchecked. Define output pins on the **Config>Ports and Pins>Motor Outputs** tab (Figure 5-21, page 5-24) for the Spindle Step and Spindle Direction. These pins must be connected to your motor drive electronics. *Apply* the changes.

Define External Activation signals on the **Config>Port and Pins>Output Signals** tab to switch the spindle motor controller on/off if you wish to turn off the motor when the spindle is stopped by *M5*. The motor will not be rotating, of course, as Mach3 will not be sending step pulses but, depending on the driver design, the motor may still be dissipating power.

Now move to **Config>Motor Tuning** for the “Spindle Axis.” The units for this will be one revolution. So the Steps per Unit are the number of pulses for one rev (e.g. 2000 for a 10 times micro-stepping drive or 4 x the line count of a servo motor encoder or the equivalent with electronic gearing).

The *Vel* box should be set to the number of revs per second at full speed. For example, a 3600 rpm motor would need to be set to 60. A high line count encoder may limit maximum speed because the maximum pulse rate from Mach3 will be insufficient to drive it (e.g. a 100 line encoder allows 87.5 revs per second on a 35,000 Hz system). The spindle will generally require a powerful motor whose drive electronics is likely to include electronic gearing which overcomes this constraint.

The *Accel* box can be set by experiment to give a smooth start and stop to the spindle. If you want to enter a very small value in the Accel box, you can do this by typing a value rather than using the Accel slider. A spindle run-up time of 30 seconds is quite possible.

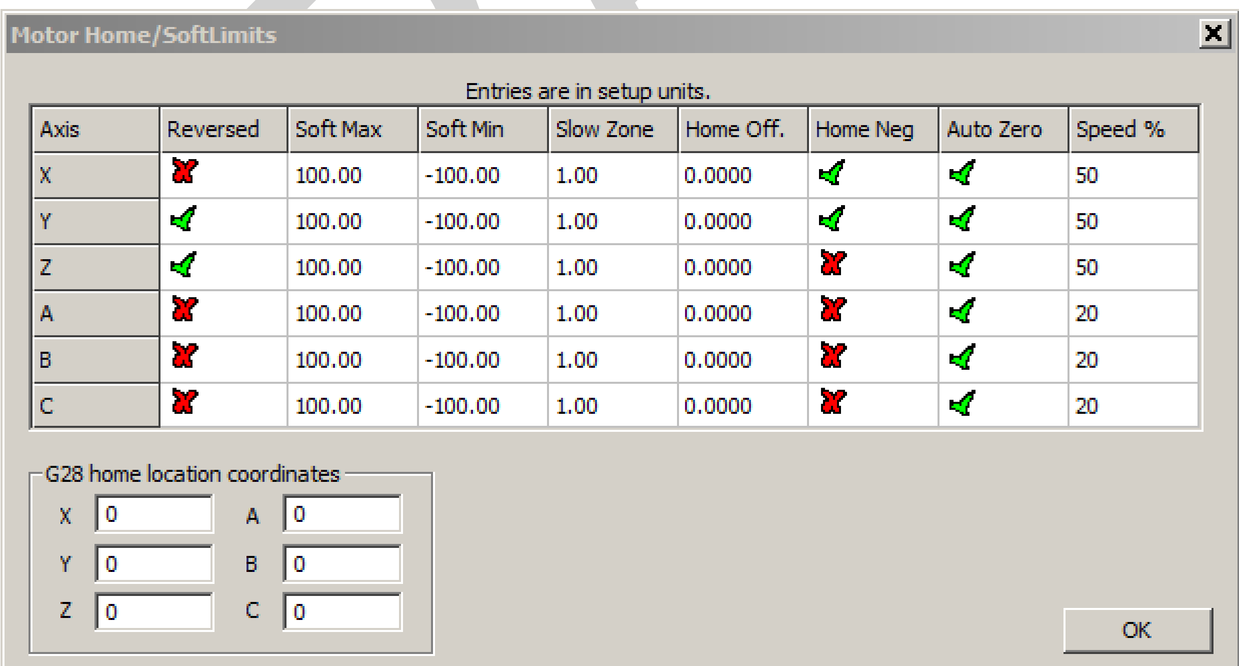
TESTING THE SPINDLE DRIVE

On the Program Run screen, set the spindle speed required to 900 rpm and start it rotating. Measure or estimate the speed. If it is wrong, you will have to revisit your calculations and setup.

OTHER CONFIGURATION

CONFIGURE HOMING AND SOFT LIMITS

The **Config>Homing/Limits** dialog defines what happens when a reference operation (G28.1 or a screen button) is performed. Figure 5-38 shows the dialog.


Motor Home/SoftLimits

Entries are in setup units.

Axis	Reversed	Soft Max	Soft Min	Slow Zone	Home Off.	Home Neg	Auto Zero	Speed %
X	✗	100.00	-100.00	1.00	0.0000	✓	✓	50
Y	✓	100.00	-100.00	1.00	0.0000	✓	✓	50
Z	✓	100.00	-100.00	1.00	0.0000	✗	✓	50
A	✗	100.00	-100.00	1.00	0.0000	✗	✓	20
B	✗	100.00	-100.00	1.00	0.0000	✗	✓	20
C	✗	100.00	-100.00	1.00	0.0000	✗	✓	20

G28 home location coordinates

X	0	A	0
Y	0	B	0
Z	0	C	0

OK

Figure 5-38 CONFIG>HOMING/LIMITS DIALOG

REFERENCING SPEEDS AND DIRECTION

The *Speed %* entry is used to avoid crashing into the stop of an axis at full speed when looking for the reference switch. Mach3 will move the axis at the percentage of full speed that you enter here.

The *Home Neg* entry determines the initial search direction. When you are referencing, Mach3 has no idea of the position of an axis. The direction it moves depends on the *Home Neg* setting. If the relevant box is checked, then the axis will move in the minus direction until the Home input becomes active. If the Home input is already active, then the axis will move in the plus direction. Similarly, if the box is unchecked, then the axis moves in the plus direction until the input is active or the minus direction if it is already active.

POSITION OF HOME SWITCHES

If the *Auto Zero* checkbox is checked, then the axis DROs will be set to the Reference/Home Switch location values defined in the *Home Off.* (offset) column (rather than actual Zero). This can be useful to minimize homing time on a very large and slow axis.

It is, of course, necessary to have separate limit and reference switches if the reference switch is not at the end of an axis.

CONFIGURE SOFT LIMITS

Most implementations of limit switches involve some compromises. Hitting them accidentally will require intervention by the operator and may require the system to be reset and re-referenced. Soft limits can provide a protection against this sort of inconvenient accident.

The software will refuse to allow the axes to move outside the declared range of the soft limits of the X, Y and Z axes. These can be set in the range -999999 to + 999999 units for each axis. When jogging motion gets near to the limit, then its speed will be reduced when inside a *Slow Zone* which is defined for the table.

If the *Slow Zone* is too big, then you will reduce the effective working area of the machine. If they are set too small, then you risk hitting the hardware limits.

The defined limits only apply when switched on using the *Soft Limits* toggle button – see Limits and Miscellaneous control family for details.

If a part program attempts to move beyond a soft limit, then it will raise an error.

The soft limits values are also used to define the cutting envelope if Machine is selected for the tool- path display. You may find them useful for this even if you are not concerned about actual limits.

G28 HOME LOCATION

The *G28 coordinates* define the position in absolute coordinates to which the axes will move when a `G28` is executed. They are interpreted in the current units (G20/G21) and not automatically adjusted if the units system is changed.

CONFIGURE SYSTEM HOTKEYS

Mach3 has a set of global hotkeys that can be used for jogging or to enter values into the MDI line etc. These keys are configured in the **Config>System Hotkeys** dialog shown in Figure 5-39. Click on the button for the required function, then press the key to be used as its hotkey. Its value will be displayed on the dialog. Take care to avoid duplicate use of a code, as that can cause serious confusion.

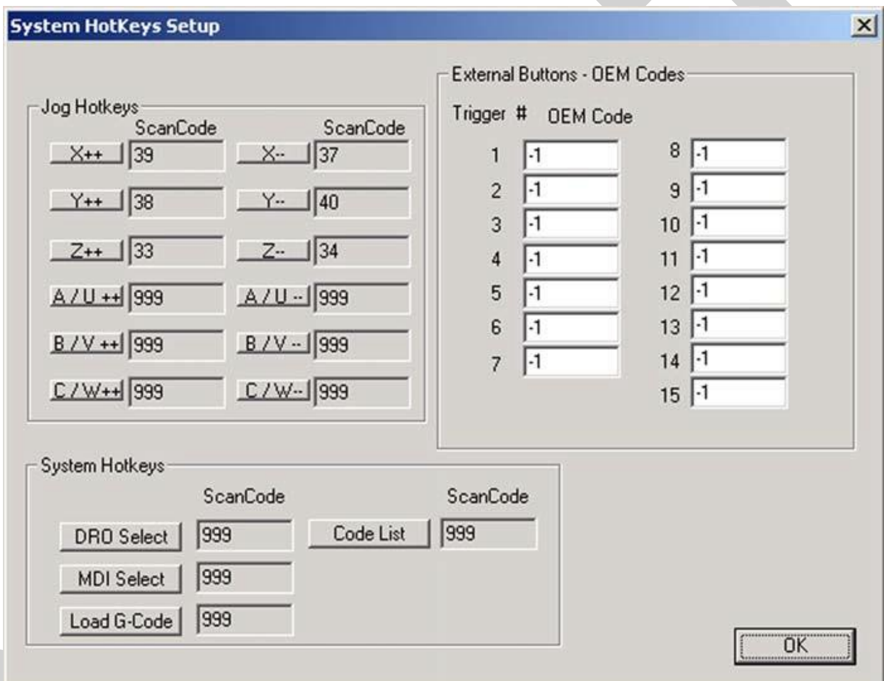


Figure 5-39 Configure System Hot Keys Dialog

This dialog also lets you define the codes for external buttons used as OEM Triggers.

Table 5-3 lists the defaults used for the Jog hotkeys.

Function	Key	Code	Function	Key	Code
X++	Right Arrow	39	X--	Left Arrow	37
Y++	Up Arrow	38	Y--	Right Arrow	40
Z++	Page Up	33	Z--	Page Down	34

Table 5-3 DEFAULTS FOR JOG HOT KEYS

CONFIGURE BACKLASH

The **Config>Backlash** dialog shown in Figure 5-40 lets you provide an estimate of the distance the axis must back up to ensure any backlash is taken up when the final “forward” movement is made. You can also specify the speed at which this movement is to be made.

Mach3 will attempt to compensate for backlash in axis drive mechanisms by attempting to approach each required coordinate from the same direction. While this is useful in applications like drilling or boring, it cannot overcome problems with the machine in continuous cutting where changes in direction occur. Also note:

1. These settings are used only when backlash compensation is enabled by the checkbox.

2. Consider backlash compensation as a “last resort” when the mechanical design of your machine cannot be improved. Using it will generally disable the “constant velocity” features at corners.

3. Mach3 is not able to fully honor the axis acceleration parameters when compensating for backlash, so stepper systems will generally have to be detuned to avoid risk of lost steps.

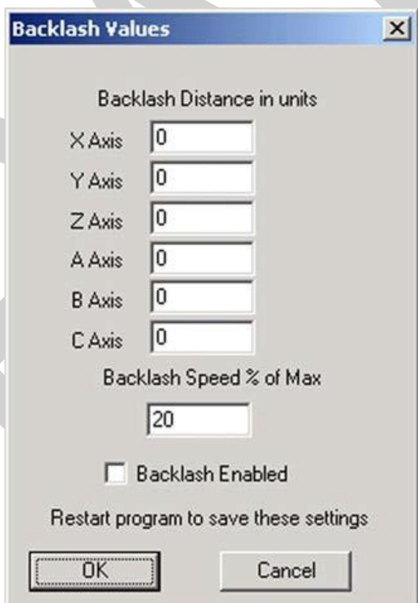


Figure 5-40 Configure Backlash Dialog

The **Config>ToolPath** dialog shown in Figure 5-41 lets you to define how the tool path is displayed.

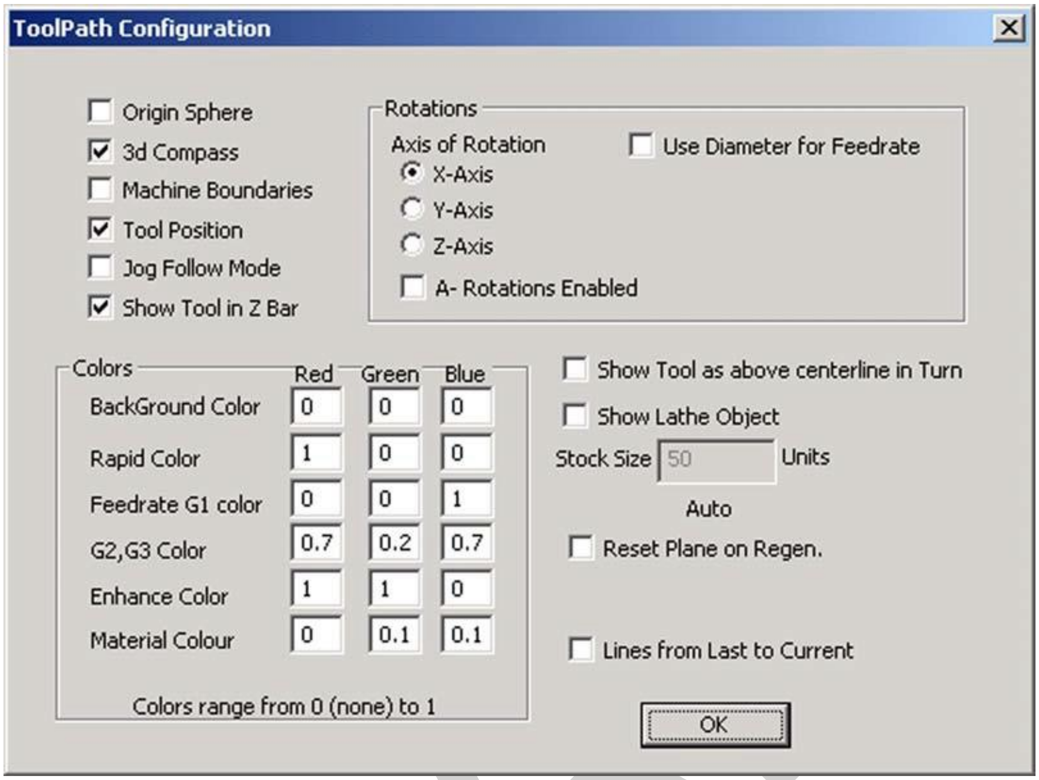


Figure 5-41 CONFIGURE TOOLPATH DIALOG

***Origin sphere**, when checked, displays a filled circle at the point of the tool path display representing X=0, Y=0, Z=0

3D Compass, when checked, shows arrows depicting the directions of positive X, Y, and Z in the tool- path display.

Machine boundaries, when checked, displays a box corresponding to the settings of the soft limits (whether or not they are switched on).

Tool Position, when checked, shows the current position of the tool on the display.

Jog Follow Mode, when checked, causes the lines representing the tool path to move relative to the window as the tool is jogged. In other words, the tool position is fixed in the tool path display window.

***ShowTool as above centerline in** Turn* relates to Mach3Turn (to handle front and rear tool posts).

Show Lathe Object enables the 3D rendering of the object that will be produced by the tool path (Mach3Turn only).

Colors for different elements of the display can be configured. The brightness of each of the additive primary colors Red, Green, and Blue are set on a scale 0 to 1 for each type of line**. Hint:** Use a program such as Photoshop to make a color you like and divide its RGB values by 255 (it uses the scale 0 to 255) to get the values for Mach3.

The *A-axis* values let you specify the position and orientation of the A-axis if it is configured as rotary and the display is enabled by the ***A Rotations*** checkbox.

Reset Plane on Regen reverts the display of the tool path display to the current plane whenever it is regenerated (by double click or button click).

Boxed Graphic displays a box at the boundaries of the tool movement.

GENERAL CONFIGURATION

The **Config>General Config...** dialog shown in **Error! Reference source not found.** lets you define the modes that are active when Mach3 is loaded (i.e. the initial state of the system). The entries are are the initial settings for the Pulsar Mill and are described column

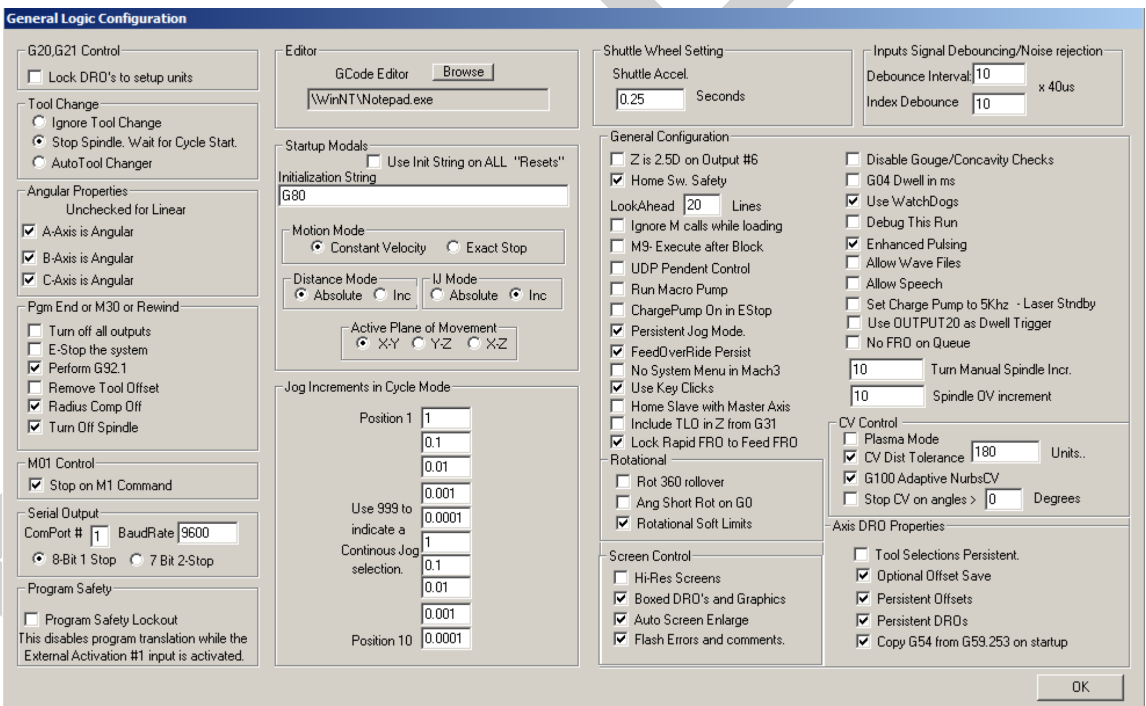


Figure 5-42 GENERAL CONFIGURATION DIALOG

by column.

GENERAL LOGIC CONFIGURATION, COLUMN 1

G20/G21 Control: If Lock DROs to setup units is checked, then even though *G20* and *G21* will alter the way X, Y, Z etc. words are interpreted (inch or millimeter), the DROs will always display in the Setup Unit system.

Tool change**:** If Ignore Tool Change is checked, M6 tool change requests will be ignored. If unchecked, *M6* will call the M6 tool change macro.

If **Stop Spindle, Wait for Cycle Start** is checked, an M6 tool change request will stop the spindle and wait for manual restart.

Auto Tool Changer If checked, then the M6Start/M6End macros will be called.

Angular properties**:** An axis checked as angular is measured in degrees (that is to say *G20/G21* do not alter the interpretation of A, B, C words). If A, B, or C is unchecked, that axis will be measured in linear units similar to X, Y, and Z.

Program end or M30 or Rewind: Defines action(s) to take place at end or a rewind of a part program. Check the required functions. **Caution:** Before checking the items to remove offsets and to perform *G92.1*, you should be absolutely clear how these features work or you may find that the current position has coordinates very different from what you expect at the end of a program.

M01 Control: If Stop on *M1* Command is checked, the optional program stop command M1 will be enabled.

Serial output**:** Defines the COM port number to be used for the serial output channel and the baud rate at which it should output. This port can be written to from VB script in a macro and can be used to control special functions of a machine (e.g. LCD display, tool-changers, axis clamps, swarf conveyor etc.).

Program safety**:** When checked, enables Input #1 as a safety cover interlock.

GENERAL LOGIC CONFIGURATION, COLUMN 2

Editor**:** Specifies the filename of the executable of the editor to be called by the G-Code edit button. The Browse button lets you select a suitable file (e.g. `C:\windows\notepad.exe).`

Startup Modals: Initialization String**:** Specifies a set of valid G-Codes to set the initial state of Mach3 when it is started. These are applied after the values set by Motion Mode, Distance Mode, IJ Mode, and Active Plane of Movement (see below) so may override them. Use the mode selections wherever possible to avoid confusion. If Use Init on ALL “Resets” is checked, then these codes will be applied regardless of the way Mach3 is reset – e.g. after an E-Stop condition.

Motion mode**:** Constant Velocity sets *G64*, Exact Stop sets *G61*.

Distance mode: Absolute sets *G90*, Inc sets *G91*. Absolute is typically the “normal” or conventional mode. Use caution if you change it to incremental. Having the wrong mode selected in relation to what your G-Code program uses can make the tool path look **very** weird.

I/J Mode**:** You can set the interpretation to be placed on I & J in arc moves. This is provided for compatibility with different CAM post-processors and to emulate other machine controllers. In Inc IJ mode, I and J (the center point) are interpreted as relative to the starting point of a center format arc. This is compatible with NIST EMC. In Absolute IJ mode, I and J are the coordinates of the center in the current coordinate system (i.e. after application of work, tool and G92 offsets). If circles always fail to display or to cut properly (especially obvious by them being too big if they are far from the origin), then the IJ mode is not compatible with your part program. An error in this setting is the most frequent cause of questions from users when trying to cut circles.

Active Plane**:** X-Y sets *G17*, Y-Z sets *G19*, X-Z sets *G18*.

Jog Increments in Cycle Mode**:** The Cycle Jog Step button will successively load the values from this list into the Step DRO. This is often more convenient than typing a value into the Step DRO. Enter the special value 999 to switch to Cont Jog Mode. The list is always ten entries long. When stepped to the end, it will cycle to the beginning of the list. A sequence of values can be duplicated to fill out the ten entries required in the list.

GENERAL LOGIC CONFIGURATION, COLUMN 3

Shuttle Wheel Setting**:** Shuttle Accel controls the responsiveness of Mach3 to the MPG when it is being used to control the execution of lines of G-Code. Shuttle Accel is also used by Backlash.

General Configuration: Z is 2.5D on output #6, if checked, controls Output #6 depending on the cur- rent position in the program coordinate system of the Z axis. If Z is greater than 0.0, then Output #6 will be active. You must have a Z axis configured to use this feature, but its Step and Direction outputs can be configured to a non-existent pin, for example Pin 0, Port 0.

Home Sw Safety**,** if checked, will prevent motion of an axis during homing if the home switch is already active. This is useful to prevent mechanical damage on a machine that shares limit switches at both ends of an axis with Home.

LookAhead determines the number of lines of G-Code that the interpreter can buffer for execution. It does not normally require tuning.

Ignore M calls while loading disables M call execution while the G-Code is being interpreted to draw the tool path. Some M calls do things like start spindles, turn on oil pumps, etc., which should execute only when the part program is actually run.

M9 Execute after Block causes a block containing an M9 command to execute the M9 last. *M9* turns off coolant. Some postprocessors write a line like: `M9 G01 X##`, which might perform a cut. If M9 was turned off when it was seen, not at the end of the block, the last cut would be made without coolant.

UDP Pendent Control Special OEM option for Ethernet control pendant. Most users can ignore.

Run Macro Pump, if checked, will on startup look for the file `MacroPump.m1s` in the macro folder for the current profile and will run it every 200 milliseconds.

Charge pump On in E-Stop, if checked, retains the charge pump output (or outputs) even when E-Stop is detected. This is required for the logic of some breakout boards.

***Persistent Jog Mode*,** if checked, remembers the Jog Mode you have chosen between runs of Mach3Mill.

FeedOveride Persist, if checked, retains the selected feed override at the end of a part program run.

No System Menu in Mach3, if checked, eliminates the menu selection bar from the Mach3. This may be useful in a production environment to prevent unauthorized changes to the Mach3 setup parameters. The option will take effect when you restart Mach3. Use CAUTION when selecting this option! It will keep you from making any further changes to the Mach3 setup. Be sure you have a way to re- enable the menu bar if you need it. One way to do so is to replace your configuration `.XML` file with a fresh copy, but that will reset all the other configuration parameters as well.

***Use Key Clicks**,* if checked, enables key click sounds for data entry into DROs and MDI. (Not implemented in current version of Mach3.)

***Home Slave with Master Axis**,* if checked, homes both motors of a master/slave pair at the same time. This will not do axis straightening, however.

Include TLO in Z from G31, if checked, will include the tool offset in the probe data.

***Lock Rapid Feed FRO to Feed FRO**,* if checked, will apply any percentage feed rate override you make to the rapid feed (up to 100% of the motor tuning speed) as well as to the standard feed rate.

***Rotational*: *Rot 360 rollover**,* if checked, will measure a rotary axis modulo 360 (0 to 360 then restart at 0). Otherwise, it will keep counting up (for example, two revolutions would be 720).

Ang Short Rot on G0, if checked, makes any rotary axis treat the position given as an angle modulo 360 degrees. Moves will be by the shortest route to that position. For example, if the axis were at 0 degrees and a request was made to rotate to 359 degrees, it would rotate –1 instead of +359.

***Rotational Soft Limits**,* if checked, applies software limit switches to rotary axes.

Screen Control: *Hi-Res Screens,* if checked, will draw the screen twice to help eliminate pixelization. Use this only if you have a good video card and a fast computer.

Boxed DROs and Graphics, if checked, draws a small border around the G-Code, MDI, and tool path and a small raised edge around the DRO.

Auto Screen Enlarge, if checked, will cause Mach3 to enlarge any screen, and all the objects on it, if it has fewer pixels than the current PC screen mode, ensuring that it fills the entire screen area.

***Flash Errors and comments**,* if checked, will cause any displayed error messages and comments to flash.

GENERAL LOGIC CONFIGURATION, COLUMN 4

Inputs Signal Debouncing/Noise Reduction: *Debounce interval/Index Debounce:* Specifies the number of Mach3 pulses for which a switch must be stable for its signal to be considered valid. For example, if a system runs at 35,000 Hz, 100 would give about a 3 millisecond debounce time (100 ÷ 35000 = 0.0029 sec). The Index pulse and the other inputs have independent settings.

General Configuration (continued from Column 3): ***Disable Gouge/Concavity checks***, if unchecked, Mach3 will check during cutter compensation (*G41* and *G42*) if the tool diameter is too large to cut “insider corners” without gouging the work. Check the box to disable the warning. *G04 Dwell param in Milliseconds*, if checked the command G4 5000 will give a Dwell in running of 5 seconds. If the control is unchecked, the dwell value will be treated as seconds. (*G4 5000* would give a dwell of 1 hour 23 minutes 20 seconds.)

***Use WatchDogs**,* if checked, triggers an E-Stop if Mach3 seems not to be running correctly. You may need to uncheck it if you get spurious E-Stops on slower computers with operations like loading Wizards.

Debug this Run, if checked, gives extra diagnostics to the program designer. ArtSoft USA may ask you to enable this option when you request support.

Enhanced Pulsing, if checked, will ensure the greatest accuracy of timing pulses (and hence smooth- ness of stepper drives) at the expense of additional central processor time. You should generally select this option.

Allow Wave files, if checked, allows Windows `.WAV` sound clips to be played by Mach3. This can be used, for example, to signal errors or attention required by the machine.

Allow Speech, if checked, allows Mach3 to use the Microsoft Speech Agent for system information messages and “right button” Help text. See the Speech option on the Windows Control Panel to configure the voice to be used, speed of speaking, etc.

Set charge pump to 5kHz - Laser Stndby, if checked, charge pump output or output(s) are a 5 kHz signal (for compatibility with some lasers), rather than the standard 12.5kHz signal.

***Use OUTPUT 20 as Dwell Trigger**,* if checked, will turn on Output 20 whenever Dwell is active.

***No FRO on Queue**,* if checked, will delay the application of feed rate override until the queue of commands waiting to be implemented is empty. This is sometimes necessary to avoid exceeding permitted speeds or accelerations when increasing the FRO above 100%.

***Turn Manual Spindle Incr**:* This box lets you enter the increment to raise or lower the spindle RPM using OEM buttons 350 and 351.

***Spindle OV increment**:* This box lets you enter the percentage increment to raise or lower the spindle RPM using OEM buttons 163 and 164.

CV Control: *Plasma Mode*, if checked, controls Mach3's implementation of constant velocity moves to suit the characteristics of plasma cutters. It does anti-dive and tries not to round corners in some circumstances. For the most part, ArtSoft USA suggests you not select this option unless your machine has very poor acceleration and poor step resolution.

CV Dist Tolerance lets you define the permissible tracking error when operating in constant velocity mode. This will affect the amount of rounding on corners. Setting this to a high value will allow faster movement but increase rounding.

G100 Adaptive NumbsCV (Obsolete.)

Stop CV on angles > n CV attempts to maintain constant velocity during all angular or arc moves while obeying the acceleration parameter. This is not possible during some moves, and this option lets you specify the amount of angular movement permissible in CV mode before automatically switching over to Exact Stop mode. Setting to 90 degrees is usually a good compromise.

Axis DRO Properties: *Tool Selections Persistent*, if checked, remembers the selected tool at shut- down of Mach3.

Optional Offset Save, if checked, will prompt to verify that you want to actually do any save requested in *Persistent Offsets*.

Persistent Offsets, if checked, will save the work and tool offsets in the permanent tables you have selected between runs of Mach3Mill. See also *Optional Offset Save.*

Persistent DROs, if checked, the axis DROs will have the same values on startup as when Mach3 is closed down. Note that the positions of the physical axes are unlikely to be preserved if the machine tool is powered down, especially with micro-stepper drives.

Copy G54 from G59.253 on startup, if checked, will re-initialize the G54 offset (i.e. work offset 1) values from the work offset 253 values when Mach3 is started. Check this if you want to start up G54 to always be a fixed coordinate system (e.g. the machine coordinate system) even if a previous user might have altered it and saved a non-standard set of values.

HOW MACH3 CONTROLS ARE EXPLAINED

Although at first sight you may feel daunted by the range of options and data displayed by Mach3, this is actually organized into a few logical groups. We refer to these as Families of Controls. By way of explanation of the term "control", this covers both buttons and their associated keyboard shortcuts used to operate Mach3 and the information displayed by DROs (digital read-outs), labels or LEDs (light emitting diodes).

The elements of each control family are defined for reference in this section. The families are explained in order of importance for most users.

You should, however, note that the actual screens of your Mach3 does not include every control of a family when the family is used. This may be to increase readability of a particular screen or to avoid accidental changes to the part being machined in a production environment

A Screen Designer is provided that allows controls to be removed or added from the screens of a set of screens. You can modify or design screens from scratch so that you can add any controls to a particular screen if your application requires this. For details see the Mach3 Customization wiki available from www.machsupport.com.

SCREEN SWITCHING CONTROLS

These controls appear on each screen. They allow switching between screens and also display *information about the current state of the system.*

***RESET*:** This is a toggle. When the system is Reset the LED glows steadily, the charge pump pulse monitor (if enabled) will output pulses and the Enable outputs chosen will be active.

LABELS: The "intelligent labels" display the last "error" message, the current modes, the file name of the currently loaded part program (if any) and the Profile that is in use.

***SCREEN SELECTION BUTTONS*:** These buttons switch the display from screen to screen. The keyboard shortcuts are given after the names. For clarity in all cases when they are letters they are in upper-case. You should not, however, use the shift key when pressing the shortcut.

***AXIS CONTROL FAMILY*:** This family is concerned with the current position of the tool (or more precisely, the controlled point), (Figure 5-43).

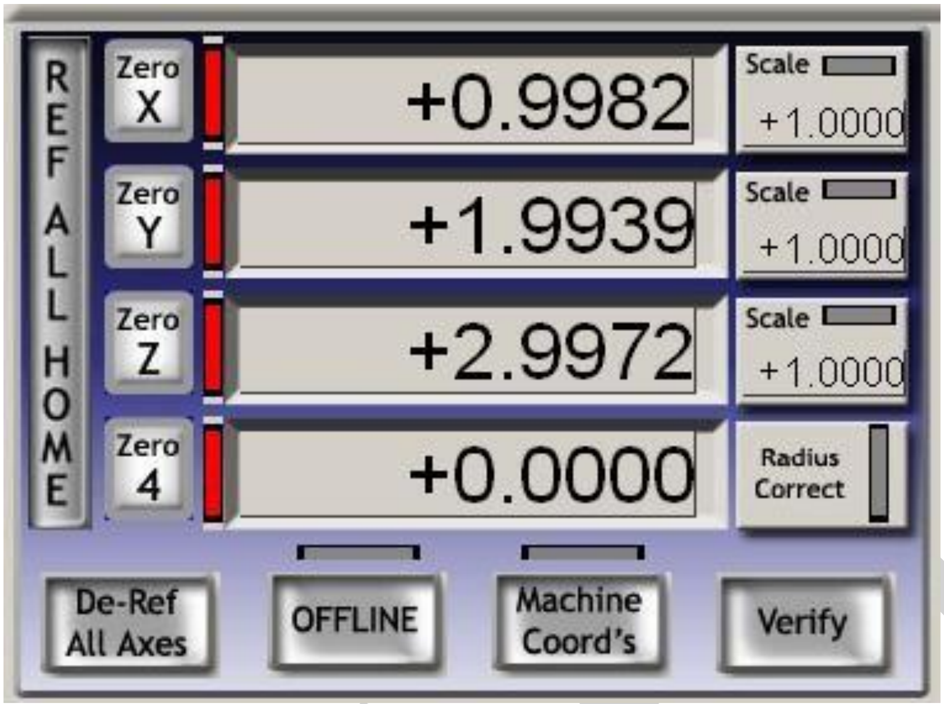


Figure 5-43 AXIS CONTROLL FAMILY

COORDINATE VALUE DRO: These are displayed in the current units (*G20/G21*) unless locked to the setup units on the Config>Logic dialog. The value is the coordinate of the controlled point in the displayed coordinate system. This will generally be the coordinate system of the current Work Offset (initially 1 - i.e. *G54*) together with any G92 offsets applied. It can however be switched to display Absolute Machine Coordinates.

You **can** type a new value into any Axis DRO. This will modify the current Work Offset to make the controlled point in the current coordinate system be the value you have set. You are advised to set up Work Offsets using the Offsets screen until you are fully familiar with working with multiple coordinate systems.

***REFERENCED*:** The LED is green if the axis has been referenced (i.e. is in a known actual position)

Each axis can be referenced using the Ref All button. Individual axes can be referenced on the Diagnostics screen

- If no home/reference switch is defined for the axis, then the axis will not actually be moved but, if Auto Zero DRO when homed is checked in Config>Referencing, then the absolute machine coordinate of the current position of the axis will be set to the value defined for the axis in the Home/Reference switch locations table in the Config>State dialog. This is most often zero.

- If there is a home/reference switch defined for the axis and it is not providing an active input when the Ref is requested, then the axis will be moved in the direction defined in Config>Referencing until the input does become active. It then backs off a short distance so that the input is inactive. If the input is already active then the axis just moves the same short distance into the inactive position. If Auto Zero DRO when homed is checked in Config>Referencing then the absolute machine coordinate of the current position of the axis will be set to the value defined for the axis in the Home/Reference switch locations table in the Config>State dialog.
- The De-Ref All button does not move the axes but stops them being in the referenced state.

MACHINE COORDINATES**:** The MachineCoords button displays absolute machine coordinates. The LED warns that absolute coordinates are being displayed.

SCALE**:** Scale factors for any axes can be set by G51 and can be cleared by G50. If a scale factor (other than 1.0) is set then it is applied to coordinates when they appear in G-code (e.g. as X words, Y words etc.) . The Scale LED will flash as a reminder that a scale is set for an axis. The value defined by G51 will appear, and can be set, in the Scale DRO. Negative values mirror the coordinates about the relevant axis.

SOFTLIMITS**:** The Softlimits button enables the softlimits values defined in Config>Homing/Limits.

VERIFY**:** The Verify button, which is only applicable if you have home switches, will move to them to verify if any steps might have been lost during preceding machining operations.

DIAMETER/RADIUS CORRECTION**:** Rotary axes can have the approximate size of the work piece defined using the Rotational Diameter control family. This size is used when making blended feed rate calculations for coordinated motion including rotational axes. The LED indicates that a non-zero value is defined.

"MOVE TO" CONTROLS: There are many buttons on different screens designed to make it easy to move the tool (controlled point) to a particular location (e.g. for a tool change). These buttons include: Goto Zs to move all axes to zero, Goto Tool Change, Goto Safe Z, Goto Home.

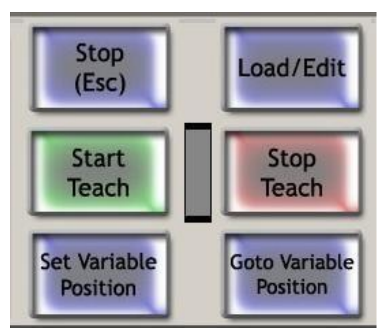


Figure 5-44 Controlled point memories & Teach

In addition Mach3 will remember two different sets of coordinates and go to them on demand. These are controlled *by Set Reference Point* and *Goto Ref Point*, and by *Set Variable Position* and *Goto Variable Position*

G-code lines (blocks) can be entered, for immediate execution, into the MDI (Manual Data Input) line. This is selected by clicking in it or the MDI hotkey (Enter in the default configuration). When the MDI line is active its color changes and a fly out box showing the recently entered commands is displayed. An example is shown in Figure 5-45. The cursor up and down arrow keys can be used to select from the fly out so that you can reuse a line that you have already entered. The *Enter* key causes Mach3 to execute the current MDI line and it remains active for input of another set of commands. The *Esc* key clears the line and de-selects it. You need to remember that when it is selected all keyboard input (and input from a keyboard emulator or custom keyboard) is written in the MDI line rather than controlling Mach3. In particular, jogging keys will not be recognized: you must *Esc* after entering MDI.

Figure 5-45 MDI line

Mach3 can remember all the MDI lines as it executes them and store them in a file by using the Teach facility. Click *Start Teach*, enter the required commands and then click *Stop Teach*. The LED blinks to remind you that you are in Teach Mode. The commands are written in the file with the conventional name

"C:/Mach3/GCode/MDITeach.tap" Clicking Load/Edit will load this file into Mach3 where it can be run or edited in the usual way – you need to go to the Program Run screen to see it. If you wish to keep a given set of taught commands then you should Edit the file and use *Save As* in the editor to give it your own name and put it in a convenient folder.

JOGGING CONTROL FAMILY

Jogging controls are collected on a special screen which flys-out into use when the *Tab* key is pressed on the keyboard. It is hidden by a second press of *Tab*.

This is illustrated in Figure 5-46. Whenever the *Jog ON/OFF* button is displayed on the current screen then the axes of the machine can be jogged using (a) the jog hotkeys – including an MPG connected via a keyboard emulator: the hotkeys are defined in Configure Axis hotkeys; (b) MPG hand wheel (s) connected to an encoder on the parallel port; or a Modbus device (c) joysticks interfaced as USB Human Interface Devices; or (e) as a legacy feature, a Windows compatible analog joystick.

If the *Jog ON/OFF* button is not displayed or it is toggled to OFF then jogging is not allowed for safety reasons.

Figure 5-46 Jogging control

HOTKEY JOGGING

There are three modes. Continuous, Step and MPG which are selected by the *Jog Mode* button and indicated by the LEDs.

Continuous mode moves the axis or axes at the defined slow jog rate while the hotkeys are depressed

The jogging speed used with hotkeys in Continuous mode is set as a percentage of the rapid traverse rate by the *Slow Jog Percentage* DRO. This can be set (in the range 0.1% to 100%) by typing into the DRO. It can be nudged in 5% increments by the buttons or their hotkeys.

This *Slow Jog Percentage* can be overridden by depressing *Shift* with the hotkey(s). An LED beside the Cont. LED indicates this full speed jogging is selected

Step mode moves the axis by one increment (as defined by the *Jog Increment* DRO) for each key press. The current feed rate (as defined by the F word) is used for these moves.

The size of increment can be set by typing it into the *Step* DRO or values can be set in this DRO by cycling through a set of 10 user definable values using the *Cycle Jog Step* button.

Incremental mode is selected by the toggle button or, if in Continuous Mode temporarily selected by holding down Ctrl before performing the jog.

PARALLEL PORT OR MODBUS MPG JOGGING

Up to three quadrature encoders connected to the parallel ports or ModBus can be configured as MPGs for jogging by using the *Jog Mode* button to select *MPG Jog Mode*.

The axis that the MPG will jogs is indicated by the LEDs and the installed axes are cycled through by the Alt-A button for MPG1, Alt-B for MPG2 and Alt-C for MPG3. Over the graphic of the MPG handle are a set of buttons for selecting the MPG mode.

In *MPG Velocity* Mode the velocity of the axis movement is related to the rotational speed of the MPG with Mach3 ensuring that the acceleration of the axis and top speed if honored. This gives a very natural feel to axis movement. *MPG Step/Velocity* mode currently works like velocity mode.

In *Single Step* mode each "click" from the MPG encoder requests one incremental jog step (with the distance set as for hotkey Step jogging). Only one request at a time will be allowed. In other words if the axis is already moving then a “click” will be ignored. In *Multi-step* mode, clicks will be counted and queued for action. Note that this means that for large steps rapid movement of the wheel may mean that the axis moves a considerable distance and for some time after the wheel movement has stopped. The steps are implemented with the feed rate given by the *MPG Feedrate* DRO .

These step modes are of particular use in making very fine controlled movements when setting up work on a machine. You are advised to start using Velocity Mode.

SPINDLE SPEED CONTROL FAMILY

The Pulsar Mill spindle speed can be controlled in three ways: (a) Speed obtain via *S* word used in a part program, (b) Speed fixed/set by typing in to the S DRO (Figure 5-47), and (c) by using the S code via the MDI inputs

The Pulsar Mill spindle can be turned on/off in three ways: (a) Turned on/off via the *M3/M5* word used in a part program, (b) turned on/off by clicking the “*Spindle CWF5*” button (Figure 5-47), and (c) by using the M3/M5 word via the MDI inputs

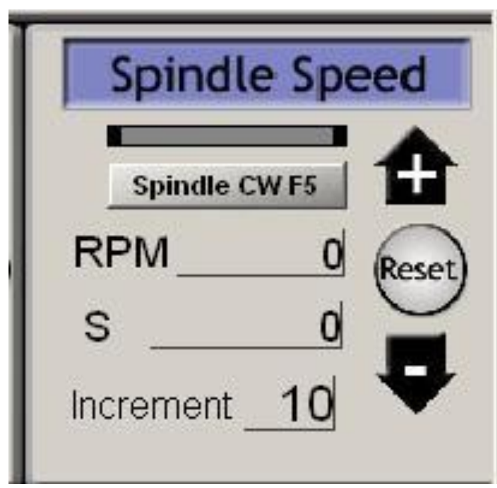


Figure 5-47 SPINDLE SPEED

Mach3 will not allow you to try to set it (in either way) to a speed less than that set in *Min Speed* or greater than that set in *Max Speed* on Config>Port & Pins Spindle, Figure 5-37, page 5-37. If the *Index* input is configured and a sensor which generates pulses as the spindle revolves is connected to its pin, then the current speed will be displayed in the *RPM* DRO. The *RPM* DRO cannot be set by you – use the *S* DRO to command a speed.

FEED CONTROL FAMILY

FEED UNITS PER MINUTE

The Prog Feed DRO (Figure 5-48) gives the feed rate in current units (inches/millimeters per minute). It is set by the F word in a part program or by typing into the F DRO. Mach3 will aim to use this speed as the actual rate of the coordinated movement of the tool through the material. If this rate is not possible because of the maximum permitted speed of any axis then the actual feed rate will be the highest achievable

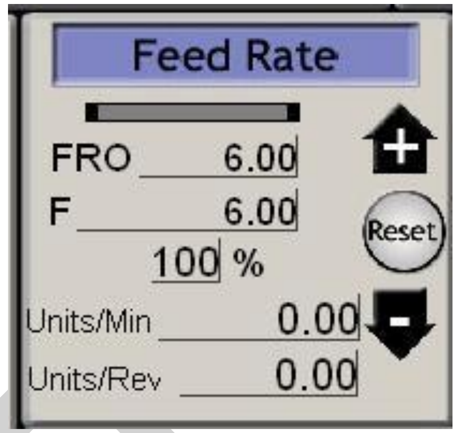


Figure 5-48 FEED CONTROL FAMILY

FEED UNITS PER REV.

As modern cutters are often specified by the permitted cut per "tip" it may be convenient to specify the feed per revolution (i.e. feed per tip x number of tips on tool). The Prog Feed DRO gives the feed rate in current units (inches/millimeters) per rev of the spindle. It is set by the F word in a part program or by typing into the DRO.

A revolution of the spindle can either be determined by the S DRO or from the measured speed by counting index pulses. Config>Logic has a checkbox to define which Mach3 will adopt. To employ Feed units/rev, Mach3 must know the value of the chosen measure of the speed of the spindle (i.e. it must have been (a) defined in an S word or by data entered to S DRO in the Spindle speed control family or (b) the Index must be connected up to measure actual spindle speed).

Notice that the numeric values in the control will be very different unless spindle speed is near to 1 rpm! So using a feed per minute, figure with feed per rev mode will probably produce a disastrous crash.

FEED DISPLAY

The actual feed in operation allowing for the coordinated motion of all axes is displayed in Units/min and Units/rev. If the spindle speed is not set and the actual spindle speed is not measured then the Feed per rev value will be meaningless.

FEED OVERRIDE

Unless *M49* (Disable feed rate override) is in use, the feed rate can be manually overridden, in the range 20% to 299%, by entering a percentage in the DRO. This value can be nudged (in steps of 10%) with the buttons or their keyboard shortcuts and be reset to 100%. The LED warns of an override is in operation.

The FRO DRO displays the calculated result of applying the percentage override to the set feed rate.

PROGRAM RUNNING CONTROL FAMILY

These controls handle the execution of a loaded part program or the commands on an MDI line.

CYCLE START

Safety warning: Note that the Cycle Start button will, in general, start the spindle and axis movement. It should always be configured to require "two hand" operation and if you are assigning your own hotkeys it should not be a single keystroke.

FEEDHOLD

The Feedhold button will stop the execution of the part program as quickly as possible but in a controlled way so it can be restarted by Cycle Start. The spindle and coolant will remain on but can be stopped manually if required.

When in FeedHold (Figure 5-49) you can jog the axes, replace a broken tool etc. If you have stopped the spindle or coolant then you will generally want to turn them on before continuing. Mach3 will however, remember the axis positions at the time of the FeedHold and return to them before continuing the part program

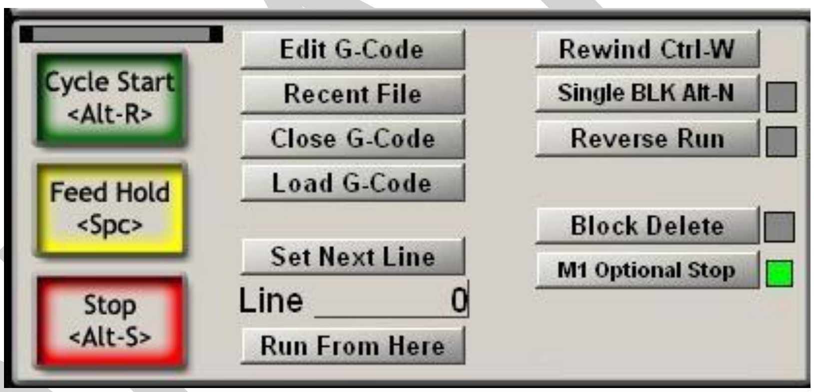


Figure 5-49 Program running family

STOP

Stop halts axis motion as quickly as possible. It may result in lost steps (especially on stepper motor driven axes) and restarting may not be valid.

REWIND

Rewinds the currently loaded part program.

SINGLE BLK

SingleBLK is a toggle (with indicator LED). In Single Block mode a Cycle Start will execute the next single line of the part program and then enter FeedHold.

REVERSE RUN

Reverse Run is a toggle (with indicator LED). It should be used after a Feed Hold or Single Block and the next Cycle Start will cause the part program to run in reverse. This is particularly useful in recovering from a lost arc condition in plasma cutting or a broken tool.

LINE NUMBER

Line DRO is the ordinal number of the current line in the G-code display window (starting from 0). Note that this is not related to the "N word" line number.

You can type into this DRO to set the current line.

RUN FROM HERE

Run from here performs a dummy run of the part program to establish what the modal state (*G20/G21*, *G90/G91*, etc.) should be and then prompts for a move to put the controlled point in the correct position to for the start of the line in Line Number. You should not attempt to Run from here in the middle of a subroutine.

SET NEXT LINE

Like Run from here but without the preparatory mode setting or move.

BLOCK DELETE

The Delete button toggles the Block Delete "switch". If enabled then lines of G-code which start with a slash - i.e. / - will not be executed.

OPTIONAL STOP

The End button toggles the Optional Stop "switch". If enabled then the M01 command will be treated as M00.

FILE CONTROL FAMILY

These controls, are involved with the file of your part program. They should be self-evident in operation.

TOOL DETAILS

In the Tool Details group, controls display the current tool, the offsets for its length and diameter and, on systems with a Digitized input, allow it to be automatically zero to the Z plane.

Unless tool change requests are being ignored (Config>Logic), on encountering an M6 Mach3 will move to Safe Z and stop, flashing the Tool Change LED. You continue (after changing the tool) by clicking Cycle Start.

The elapsed time for the current job is displayed in hours, minutes and seconds.

G-CODE AND TOOLPATH CONTROL FAMILY

The currently loaded part program is displayed in the G-code window. The current line is highlighted and can be moved using the scroll bar on the window.

The Toolpath display, Figure 5-50, shows the path that the controlled point will follow in the X, Y, Z planes. When a part program is executing the path is over painted in the color selected in Config>Toolpath. This over painting is dynamic and is not preserved when you change screens or indeed alter views of the tool path.

On occasions you will find that the display does not exactly follow the planned path. It occurs for the following reason. Mach3 prioritizes the tasks it is doing. Sending accurate step pulses to the machine tool is the first priority. Drawing the tool path is a lower priority. Mach3 will draw points on the tool path display whenever it has spare time and it joins these points by straight lines. So, if time is short, only a few points will be drawn and circles will tend to appear as polygons where the straight sides are very noticeable. This is nothing to worry about.

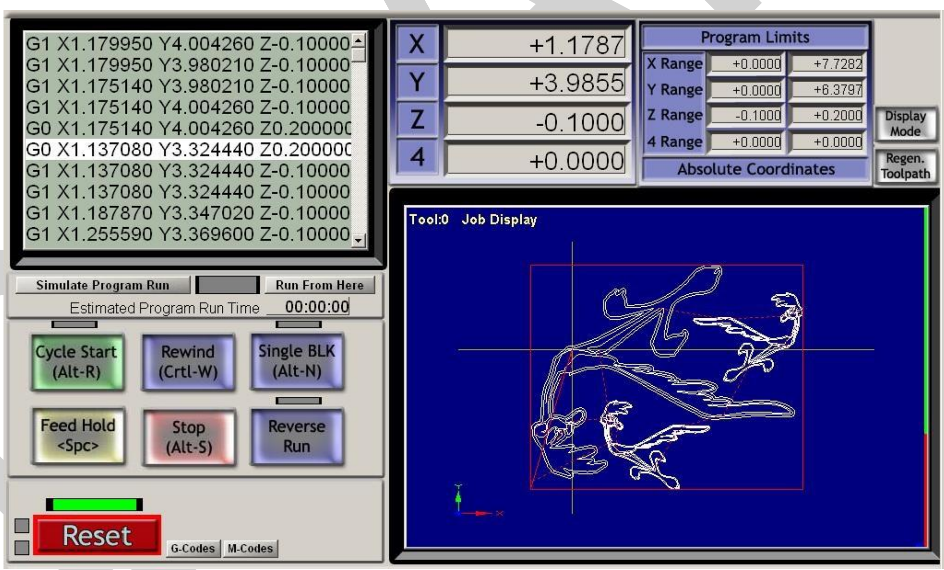


Figure 5-50 Tool path family

The Simulate Program Run button will execute the G-code, but without any tool movement, and allow the time to make the part to be estimated.

The Program Limits data allow you to check the maximum excursion of the controlled point to be reasonable (e.g. not milling the top off the table).

The screenshot also shows axis DROs and some Program Run controls.

If you have defined softlimits which correspond to the size of your machine table then it is often useful to use the Display Mode button to toggle from Job to Table mode to show the tool path in relation to the table. See Figure 5-51.

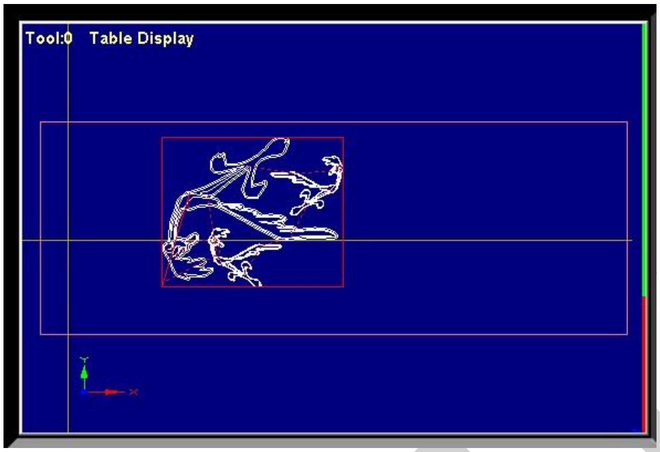


Figure 5-51 Tool path in relation to the table

The tool path display can be rotated by left clicking and dragging the mouse in it. It can be zoomed by shift-left clicking and dragging and can be panned by dragging a right click.

The Regenerate button will regenerate the tool path display from the G-code with the currently enabled fixture and G92 offsets.

Note: It is very important to regenerate the tool path after changing the values of offsets both to get the correct visual effect and because it is used to perform calculations when using G42 and G43 for cutter compensation..

WORK OFFSET AND TOOL TABLE CONTROL FAMILY

Work Offset and Tool tables can be accessed from the Operator menu and, of course, within a part program but it is often most convenient to manipulate them through this family. Refer to Section "*Work Offsets*", page 5-71 for details of the tables and techniques like "Touching". Because of the underlying G-code definitions Work Offset and Tool tables work in slightly different ways.

Warning: Changing the Work and Tool offsets in use will never actually move the tool on the machine although it will of course alter the axis DRO readings. However, a move *G0*, *G1* etc., after setting new offsets will be in the new coordinate system. You must understand what you are doing if you wish to avoid crashes on your machine.

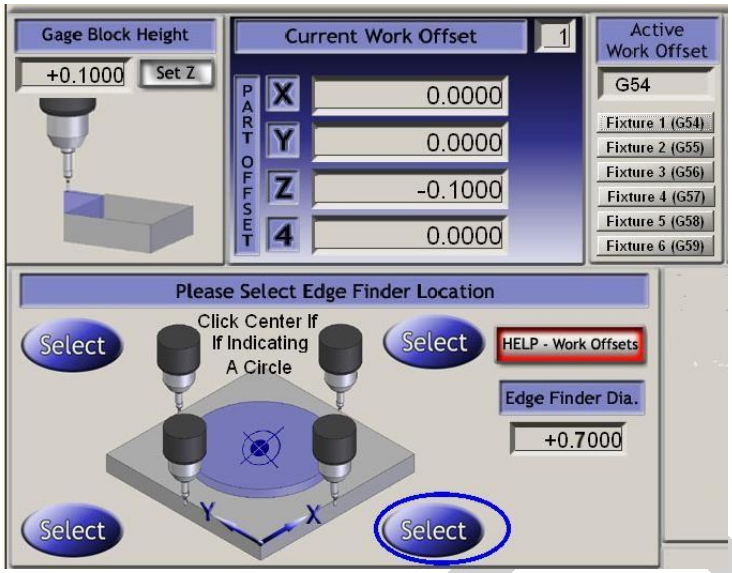


Figure 5-52 WORK OFFSETS FAMILY

WORK OFFSETS

Mach3 by default uses Work Offset number 1. Choosing any value from 1 to 255, and entering it in the Current Work Offset DRO, will make that Work Offset current. Work offsets are sometimes called Fixture Offsets.

Typing into the DRO is equivalent to a part program issuing *G55 to 59* or *G58.1* to *G59.253* (q.v.). You can also set the current offset system using the Fixture buttons. You can change the value of the offset values for the current offset system by typing into the relevant Part Offset DROs. (Part Offset is yet another name for Work and Fixture offsets!)

Values can also be set in these DROs by moving the axes to a desired place and clicking as Set or Select button. The X and Y axes and Z axis are set in slightly different ways. Z is easier to understand so we will describe it first.

The Z offset will usually be set up with a “master tool” in the spindle. The Z for other tools will then be corrected by the tool table. A gage block or sometimes even a piece of foil or paper is slid between the tool and the top of the work (if this is to be Z = 0.0) or the table (if this is to be Z = 0.0). The Z axis is very gently jogged down until the gage is just trapped by the tool. The thickness of the gage is entered into the Gage Block Height DRO and the Set Z button is clicked. This will set up the Z value of the current work offset so that the tool is at the given height.

The process for X and Y is similar except the touching might be done on any of four sides of the part and account has to be taken of the diameter of the tool (or probe) and the thickness of any gage being used to give “feel” to the touching process.

For example to set the bottom edge of a piece of material to be Y = 0.0 with a tool of diameter 0.5" and a 0.1" gage block, you would enter 0.7 in the Edge Finder Dia DRO (i.e. the diameter of the tool plus twice the gage) and click the Select button that is ringed in Figure 5-52, page 5-60.

.Depending on your configuration of Persistent Offsets and Offsets Save in Config>State the new values will be remembered from one run of Mach3 to another.

TOOLS

Tools are numbered from 0 to 255. The tool number is selected by the T word in a part program or entered in the Tool DRO. Its offsets are only applied if they are switched on by the Tool Offset On/Off toggle button (or the equivalent *G43* and *G49* in the part program) In Mach3Mill only the Z offset and Diameter are used for tools. The diameter can be entered in the DRO and the Z-offset (i.e. compensation for tool length) be entered directly or by Touching. The Set Tool Offset feature works exactly as set Z with Work Offsets. Tool Offset data is made persistent between runs in the same way as Work Offset data.

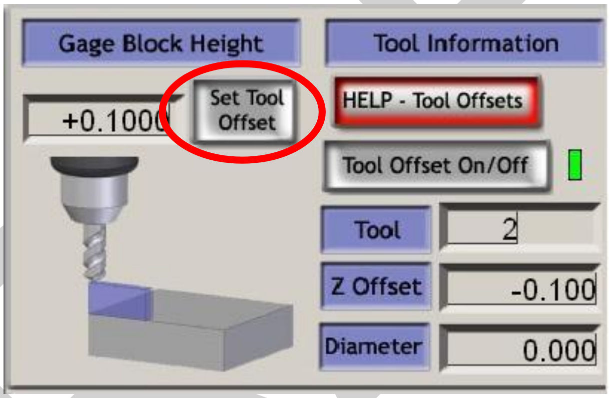


Figure 5-53 TOOL OFFSET

DIRECT ACCESS TO OFFSET TABLES

The tables can be opened and edited directly using the Save Work Offsets and Save Tool Offsets buttons or the Operator>Fixtures (i.e. Work Offsets) and Operator>Tooltable menus.

ROTATIONAL DIAMETER CONTROL FAMILY

As described in the Feed rate control family, it is possible to define the approximate size of a rotated work piece so the rotational axis speed can be correctly included in the blended feed rate. The relevant diameters are entered in the DROs of this family.

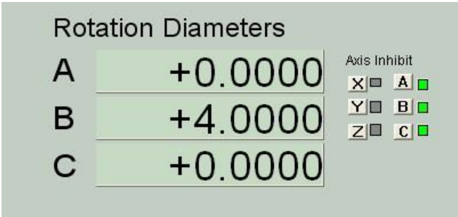


Figure 5-54 Rotational diameters

The Axis control Family has warning LED(s) to indicate the setting of non-zero values here.

Values are not required if rotary movement is not to be coordinated with linear axes. In this case a suitable F word for degrees per minute or degrees per rev should be programmed.

TANGENTIAL CONTROL FAMILY

Figure 5-55 TANGENTIAL CONTROL FAMILY

On a machine to cut vinyl or fabric it is very useful to use a rotary axis to control the direction that the knife points. It will cut best if tangential to the direction in which the X and Y axes are moving at any time.

Mach3 will control the A axis like this for G1 moves. Clearly the point of the knife should be as near to the axis about which a turns and this axis must be parallel to the Z axis of the machine. The feature is enabled by the *Tangential Control* .button. In most applications there is a limit to the angle through which the knife can be turned at a corner while it is in the material. This value is defined in Lift Angle. Any corner where the change in angle required is greater than Lift Angle will cause the Z axis to rise by the value in Lift Z, the knife will turn and then Z will drop so it re-enters the material in the new direction.

LIMITS AND MISCELLANEOUS CONTROL FAMILY

Figure 5-56 LIMITS CONTROL FAMILY

Input Activation 4 Input activation signal 4 can be configured to give a hard wired Single Step function equivalent to the *Single* button in the Program Running control family.

Override Limits Mach3 can use software to override limit switches connected to its inputs.

This can be automatic i.e. the jogging performed immediately after a reset will not be subject to limits until the axis is jogged off the limit switches. The *Toggle* button and warning LED for ***Auto Limit Override*** controls this. As an alternative limits may be locked out using the ***OverRide Limits*** toggle. Its use is indicated by the LED.

Notice that these controls do not apply if limit switches are wired to the drive electronics or to activate E-Stop. In this case an external electrical override switch will be needed to disable the switch circuit while you jog off them.

SYSTEM SETTINGS CONTROL FAMILY

Note: The controls in this family are not in one place on the screens released with Mach3. You will need to hunt for them on Program Run, Settings and Diagnostics screens.

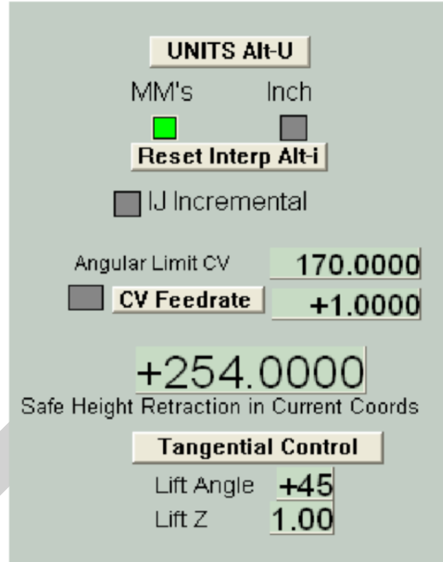


Figure 5-57 System Settings, Safe Z controls etc

Units This toggle implements the G20 and G21 codes to change the current measurement units. You are **strongly** advised **not** to do this except in small fragments of part program on account of the fact that Work Offset and Tool Offset tables are in one fixed set of units

Safe Z This family allows you to define the Z value which is clear of clamps and parts of the work piece. It will be used for homing and changing the tool.

CV MODE/ANGULAR LIMIT

This LED is lit when the system is running in "Constant Velocity" mode. This will give smoother and faster operation than "Exact stop" mode but may cause some rounding at sharp corners depending on the speed of the axis drives. Even when the system is in CV mode a corner with a change of direction more acute than the value given in the *Angular Limit* DRO will be performed as if Exact Stop was selected.

Offline This toggle and warning LED "disconnects" all the output signals of Mach3. This is intended for machine setup and testing. Its use during a part program will cause you all sorts of positioning problems.

ENCODER CONTROL FAMILY

This family displays the values from the axis encoders and allows them to be transferred to and from the main axis DROs

Figure 5-58 Encoder control family

The *Zero* button will reset the corresponding encoder DRO to zero.

The *To DRO* button copies the value into the main axis DRO (i.e. applies this values as a G92 offset).

The *Load DRO* button loads the encoder DRO from the corresponding main axis DRO.

AUTOMATIC Z CONTROL FAMILY

Mach3 has the facility to set a lower limit for moves in the Z axis. See Config>Logic dialog for the static setting of this Inhibit-Z value.

There is also a control family which allows this *Inhibit Z* value to be set while preparing and before running a G-code program. This is shown in Figure 5-59.

Figure 5-59 AUTOMATIC Z CONTROL

Code the program, which might often be a DXF or HPGL import, so that it makes a single cut or set of cuts at the finally desired Z depth (perhaps Z = -0.6 inch assuming top of work piece is Z = 0). The last command should be an M30 (Rewind)

Using the Automatic Z Control controls (a) set the *Z-inhibit* value to the Z for depth for the first roughing cut (perhaps Z= -0.05) (b) the *Lower Z-Inhibit* to the successive cut depths (we might allow 0.1 as the tool has some side support). The whole job will need seven passes to get to Z = -0.6, so (c) enter 7 in *L* (Loop). On pressing Cycle Start the machine will automatically make the series of cuts at increasing Z depth. The DROs track the progress decrementing *L* as they are performed and updating the Z-inhibit value. If the given number of *L* does not reach the part program's requested Z depth then you can update the *L* DRO and restart the program.

LASER TRIGGER OUTPUT FAMILY

Mach3 will output a pulse on the Digitize Trigger Out Pin (if defined) when the X or Y axes pass through trigger points.

The Laser Trigger group of controls allows you to define the grid points in the current units and relative to an arbitrary datum.

Click *Laser Grid Zero* when the controlled point is at the desired grid origin. Define the positions of the grid lines in X and Y axes and click *Toggle* to enable the output of pulses whenever an axis crosses a grid line.

This feature is experimental and subject to change in later releases.

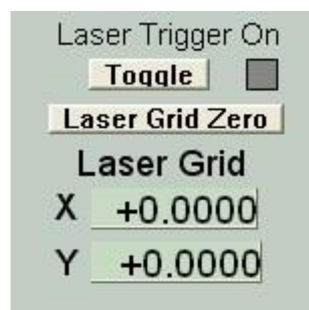


Figure 5-60 Digitize Pulse family

USING WIZARDS

Mach3 Wizards are an extension to the Teach facility which allows you to define some machining operations using one or more special screens. The Wizard will then generate G-Code to make the required cuts.

Examples of Wizards include machining a circular pocket, drilling an array of holes and engraving text.

The *Load Wizards* button displays a table of Wizards (Figure 5-61 – Choosing a Wizard) installed on your system. You choose the one required and click *Run*. The Wizard screen (or sometimes one of several screens) will be displayed.

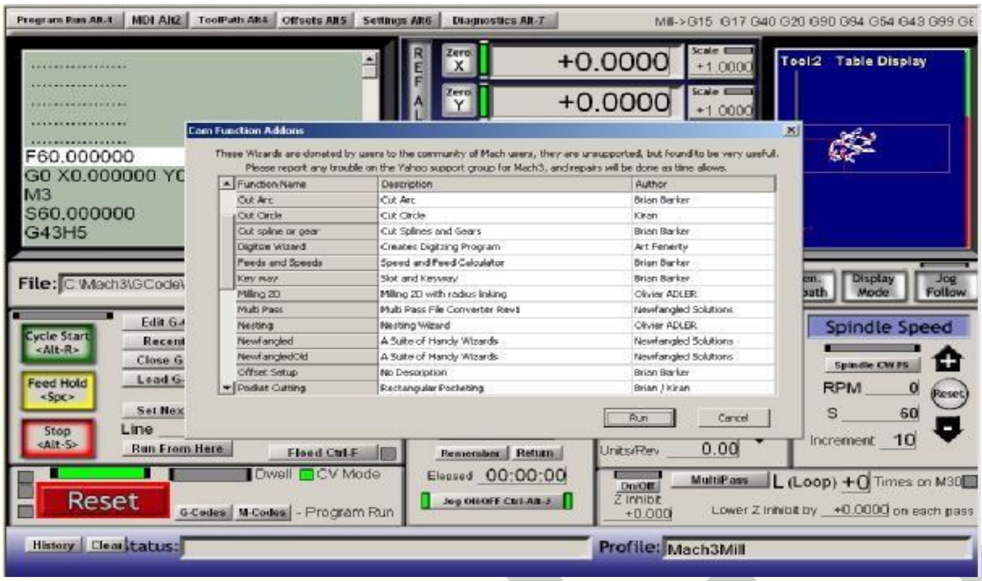


Figure 5-61 – Choosing a Wizard

Figure 5-62 is the Wizard for engraving text.

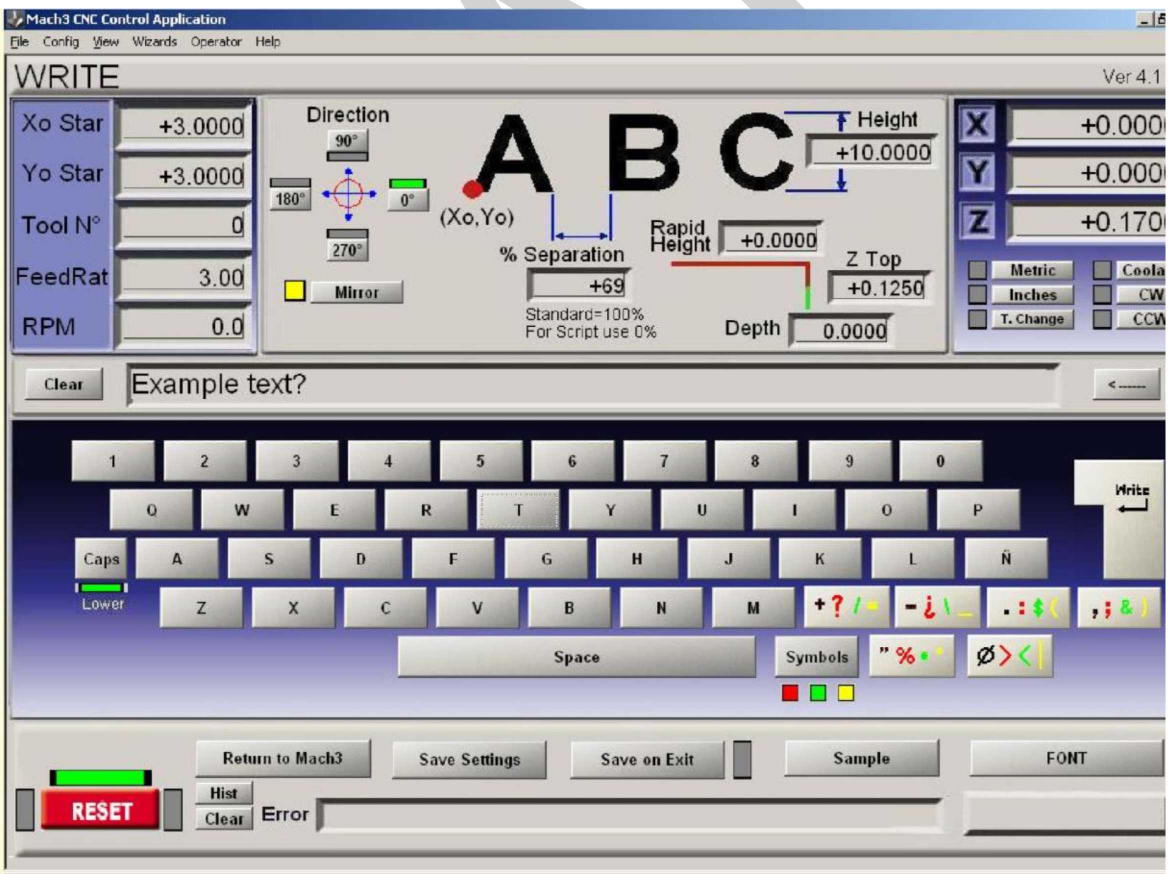


Figure 5-62 – The Write Wizard screen

Wizards have been contributed by several authors and depending on their purpose there are slight differences in the control buttons. Each Wizard will however have a means of posting the G-code to Mach3 (marked Write in Figure 5-62) and a means of returning to the main Mach3 screens. Most Wizards allow you to save your settings so that running the Wizard again gives the same initial values for the DROs etc.

Figure 5-63 shows a section of the tool Path screen after the *Write* button is pressed on Figure 5-62.

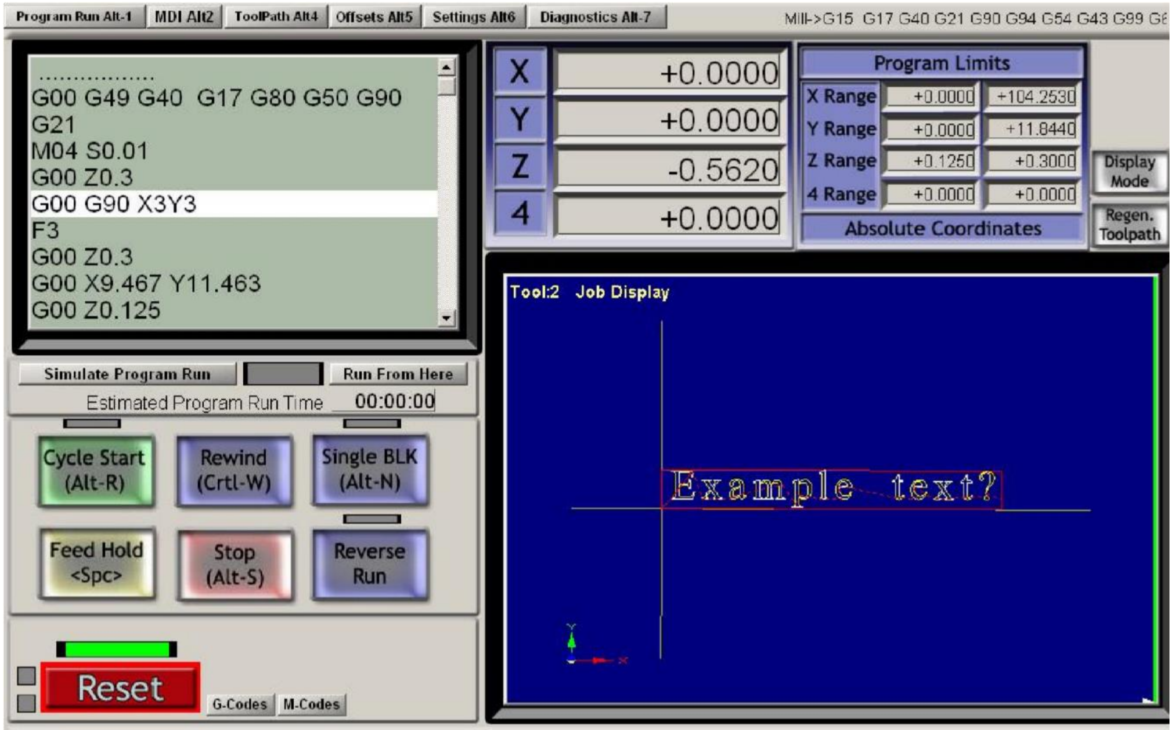


Figure 5-63 After running the Write wizard

The *Last Wizard* buttons runs the wizard you most recently used without the trouble of selecting it from the list.

The *Conversational* button runs a set of wizards designed by Newfangled Solutions. These are supplied with Mach3 but require a separate license for them to be used to generate code.

LOADING A G-CODE PART PROGRAM

If you have an existing part program which was written by hand or a CAD/CAM package then you load it into Mach3 using the *Load G-Code* button. You choose the file from a standard Windows file open dialog. Alternatively you can choose from a list of recently used files which is displayed by the *Recent Files* screen button.

When the file is chosen, Mach3 will load and analyze the code. This will generate a tool path for it, which will be displayed, and will establish the program extreme. The loaded program code will be displayed in the G-code list window. You can scroll through this moving the highlighted current line using the scroll bar.

EDITING A PART PROGRAM

Provided you have defined a program to be used as the G-code editor (in Config>Logic), you can edit the code by clicking the *Edit* button. Your editor will open in a new window with the code loaded into it.

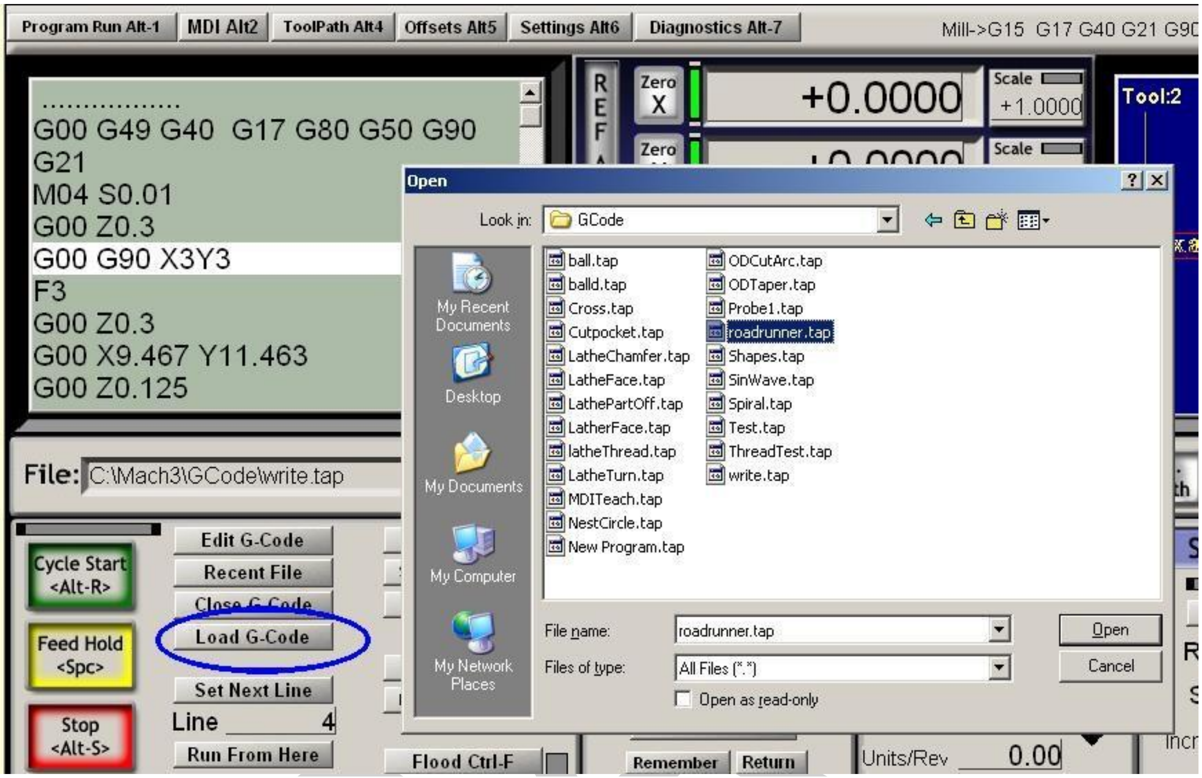


Figure 5-64 – Loading G-Code

When you have finished editing you should save the file and exit the editor. This is probably most easily done by using the close box and replying Yes to the "Do you want to save the changes?" dialog.

While editing, Mach3 is suspended. If you click in its window it will appear to be locked up. You can easily recover by returning to the editor and closing it.

After editing the revised code will again be analyzed and used to regenerate the tool path and extreme. You can regenerate the tool path at any time using the *Regenerate* button.

MANUAL PREPARATION AND RUNNING A PART PROGRAM

INPUTTING A HAND-WRITTEN PROGRAM

If you want to write a program "from scratch" then you can either do so by running the editor outside Mach3 and saving the file or you can use the *Edit* button to display the notepad screen shown in Figure 5-65. In this case you will have to Save As the completed file and exit the editor. In both cases you will have to use File>Load G-code to load your new program into Mach3.

Warning: Errors in lines of code are generally ignored. You should not rely on being given a detailed syntax check.

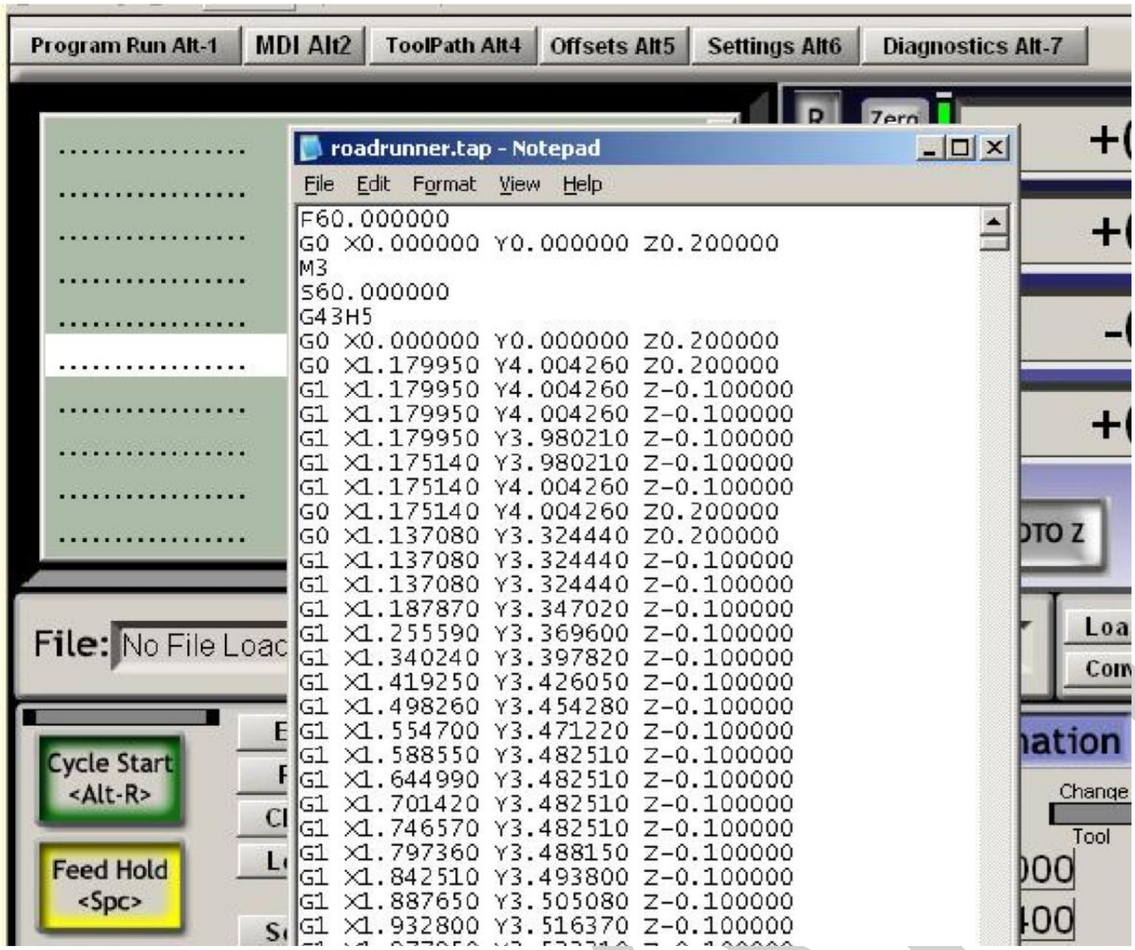


Figure 5-65 Editing a Parts Program

BEFORE YOU RUN A PART PROGRAM

It is good practice for a part program to make no assumptions about the state of the machine when it starts. It should therefore include *G17/G18/G19, G20/G21, G40, G49, G61/G62, G90/G91,* and *G93/G94*.

You should ensure that the axes are in a known reference position - probably by using the *Ref All* button.

You need to decide whether the program starts with an S word or if you need to set the spindle speed by hand or by entering a value in the *S* DRO.

You will need to ensure that a suitable feed rate is set before any *G01/G02/G03* commands are executed. This may be done by an F word or entering data into the *F* DRO.

Next you may need to select a Tool and/or Work Offset.

Finally, unless the program has been proved to be valid you should attempt a dry run, cutting "air" to see that nothing terrible happens.

RUNNING YOUR PROGRAM

You should monitor the first run of any program with great care. You may find that you need to override the feed rate or, perhaps, spindle speed to minimize chattering or to optimize production. When you want to make changes you should either do this on the "fly" or use the *Pause* button, make your changes and the click *Cycle Start*.

COORDINATE SYSTEMS, TOOL TABLE AND FIXTURES

This section explains how Mach3 works out where exactly you mean when you ask the tool to move to a given position. It describes the idea of a coordinate system, defines the Machine Coordinate System and shows how you can specify the lengths of each Tool, the position of a work piece in a Fixture and, if you need to, to add your own variable Offsets.

Mach3 can be used without a detailed understanding of this chapter but you will find that using its concepts makes setting up jobs on the Pulsar Mill much quicker and more reliable.

MACHINE COORDINATE SYSTEM

You have seen that most Mach3 screens have DROs labelled "X Axis", "Y Axis" etc. If you are going to make parts accurately and minimize the chance of your tool crashing into anything you need to understand exactly what these values mean at all times when you are setting up a job or running a part program.

This is easiest to explain looking at a machine. We have chosen an imaginary machine that makes it easier to visualize how the coordinate system works. Figure 5-66 shows what it is like.

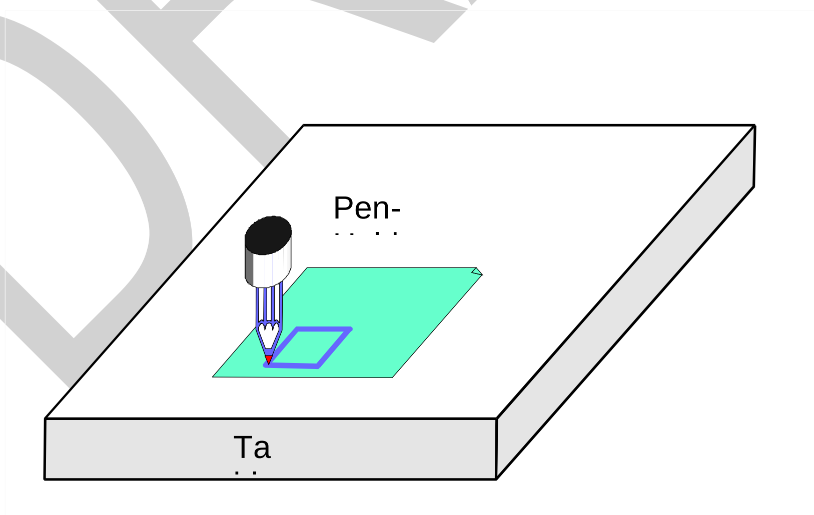


Figure 5-66 Basic Drawing Machine

It is a machine for producing drawings with a ballpoint or felt tipped pen on paper or cardboard. It consists of a fixed table and a cylindrical pen-holder which can move left and right (X direction), front and back (Y direction) and up and down (Z-direction). The figure shows a square which has just been drawn on the paper.

Figure 5-67 shows the Machine Coordinate System which measures (lets say in inches) from the surface of the table at its bottom left hand corner. As you will see the bottom left corner of the paper is at X=2, Y=1 and Z=0 (neglecting paper thickness). The point of the pen is at X=3, Y=2 and it looks as though Z=1.3.

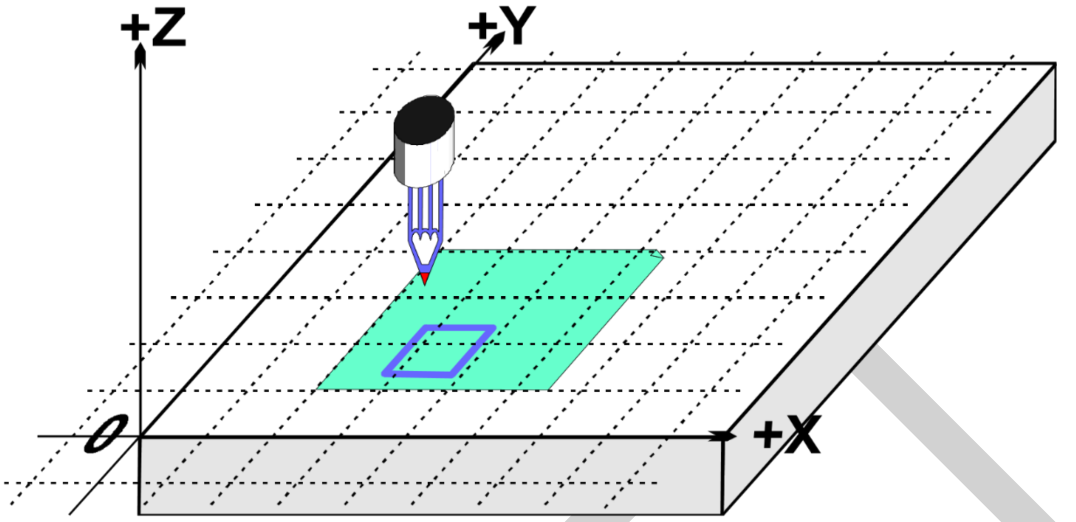


Figure 5-67 Machine Coordinate System

If the point of the pen was at the corner of the table then, on this machine, it would be in its Home or referenced position. This position is often defined by the position of Home switches which the machine moves to when it is switched on. At any event there will be a zero position

for each axis called the absolute machine zero. We will come back to where Home might actually be put on a real machine.

The point of the pen, like the end of a cutting tool, is where things happen and is called the Controlled Point. The Axis DROs in Mach3 always display the coordinates of the Controlled Point relative to some coordinate system. The reason you are having to read this chapter is that it is not always convenient to have the zeros of the measuring coordinate system at a fixed place of the machine (like the corner of the table in our example).

A simple example will show why this is so. The following part program looks, at first sight, suitable for drawing the 1" square in Figure 5-67:

```
N10 G20 F10 G90 (set up imperial units, a slow feed rate etc.)
N20 G0 Z2.0 (lift pen)
N30 G0 X0.8 Y0.3 (rapid to bottom left of square)
N40 G1 Z0.0  (pen down)
N50 Y1.3 (we can leave out the G1 as we have just done one)
N60 X1.8
N70 Y0.3 (going clockwise round shape)
N80 X0.8
N90 G0 X0.0 Y0.0 Z2.0 (move pen out of the way and lift it)
N100 M30 (end program)
```

Even if you cannot yet follow all the code it is easy to see what is happening.

For example on line N30 the machine is told to move the Controlled Point to X=0.8, Y=0.3.

By line N60 the Controlled Point will be at X=1.8, Y=1.3 and so the DROs will read:

X Axis 1.8000 Y Axis 1.3000 Z Axis 0.0000

The problem, of course, is that the square has not been drawn on the paper like in Figure 5-66 but on the table near the corner. The part program writer has measured from the corner of the paper but the machine is measuring from its machine zero position.

WORK OFFSETS

Mach3, like all machine controllers, allows you to move the origin of the coordinate system or, in other words where it measures from (i.e. where on the machine is to considered to be zero for moves of X, Y Z etc.)

This is called offsetting the coordinate system.

Figure 5-68 shows what would happen if we could offset the Current Coordinate system to the corner of the paper. **Remember** the G-code always moves the Controlled Point to the numbers given in the Current Coordinate system.

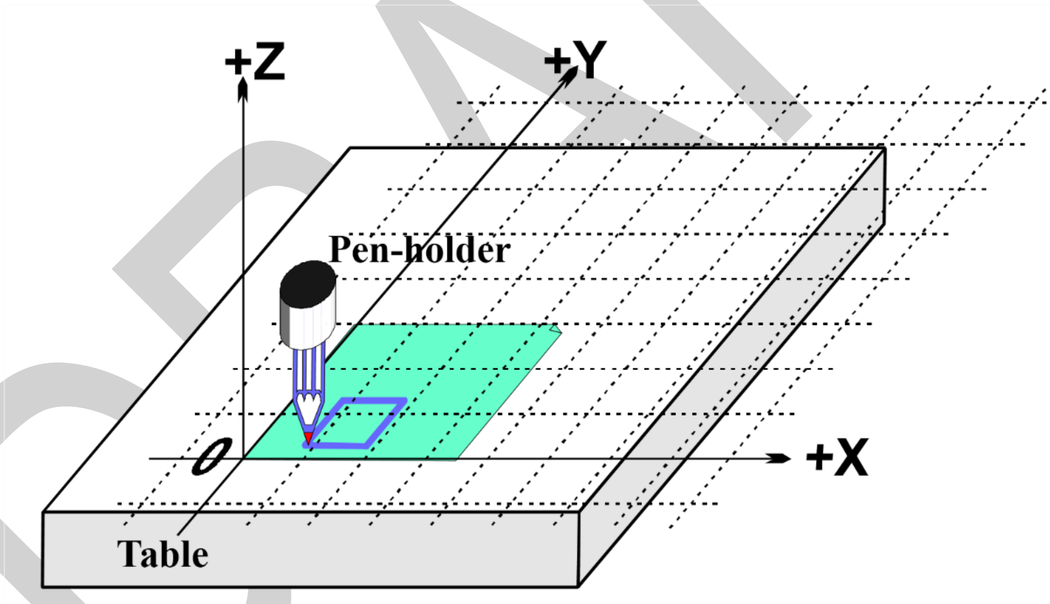


Figure 5-68 Coordinate System Origin Offset to Corner of Paper

As there will usually be some way of fixing sheets of paper, one by one, in the position shown, this offset is called a Work **offset** and the 0, 0, 0 point is the origin of this coordinate system**.**

This offsetting is so useful that there are several ways of doing it using Mach3 but they are all organized using the Offsets screen (see Appendix 1 for a screenshot)

SETTING WORK ORIGIN TO A GIVEN POINT

The most obvious way consists of two steps:

1) Display the Offsets screen. Move the Controlled Point (pen) to where you want the new origin to be. This can be done by jogging or, if you can calculate how far it is from the current position you can use G0s with manual data input

2) Click the *Touch* button next to each of the axes in the Current Work Offset part of the screen. On the first Touch you will see that the existing coordinate of the Touched axis is put into the Part Offset DRO and the axis DRO reads zero. Subsequent Touches on other axes copy the Current Coordinate to the offset and zero that axis DRO.

If you wonder what has happened then the following may help. The work offset values are always added the numbers in the axis DROs (i.e. the current coordinates of the controlled point) to give the absolute machine coordinates of the controlled point. Mach3 will display the absolute coordinates of the controlled point if you click the *Machine Coords* button. The LED flashes to warn you that the coordinates shown are absolute ones.

There is another way of setting the offsets which can be used if you know the position of where you want the new origin to be.

The corner of the paper is, by eye, about 2.6" right and 1.4" above the Home/Reference point at the corner of the table. Let's suppose that these figures are accurate enough to be used.

1) Type 2.6 and 1.4 into the X and Y Offset DROs. The Axis DROs will change (by having the offsets subtracted from them). Remember you have not moved the actual position of the Controlled point so its coordinates must change when you move the origin.

2) If you want to you could check all is well by using the MDI line to G00 X0 Y0 Z0. The pen would be touching the table at the corner of the paper.

We have described using work offset number 1. You can use any numbers from 1 to 255. Only one is in use at any time and this can be chosen by the DRO on the Offsets screen or by using G-codes (G54 to G59 P253) in your part program.

The final way of setting a work offset is by typing a new value into an axis DRO. The current work offset will be updated so the controlled point is referred to by the value now in the axis DRO. Notice that the machine does not move; it is merely that the origin of coordinate system has been changed. The Zero-X, Zero-Y etc. buttons are equivalent to typing 0 into the corresponding axis DRO.

You are advised not to use this final method until you are confident using work offsets that have been set up using the Offsets screen.

So, to recap the example, by offsetting the Current Coordinate system by a work offset we can draw the square at the right place on the paper wherever we have taped it down to the table.

HOME IN A PRACTICAL MACHINE

As mentioned above, although it looks tidy at first sight, it is often not a good idea to have the Home Z position at the surface of the table. Mach3 has a button to *Reference all* the axes (or you can Reference them individually). For an actual machine which has home switches installed, this will move each linear axes (or chosen axis) until its switch is operated then move slightly off it. The absolute machine coordinate system origin (i.e. machine zero) is then set to given X, Y, Z etc. values - frequently 0.0. You can actually define a non-zero value for the home switches if you want but ignore this for now!

The Z home switch is generally set at the highest Z position above the table. Of course if the reference position is machine coordinate Z=0.0 then all the working positions are lower and will be negative Z values in machine coordinates.

Again if this is not totally clear at present do not worry. Having the Controlled Point (tool) out of the way when homed is obviously practically convenient and it is easy to use the work offset(s) to set a convenient coordinate system for the material on the table.

WHAT ABOUT DIFFERENT LENGTHS OF TOOL?

If you are feeling confident so far then it is time to see how to solve another practical problem.

Suppose we now want to add a red rectangle to the drawing.

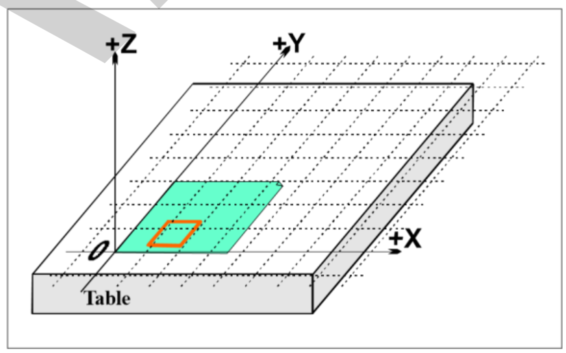


Figure 5-69 Now we want another color

We jog the Z axis up and put the red pen in the holder in place of the blue one. Sadly the red pen is longer than the blue one so when we go to the Current Coordinate System origin the tip smashes into the table. (Figure 5-70)

Mach3, like other CNC controllers, has a way for storing information about the tools (pens in our system). This **Tool Table** allows you to tell the system about up to 256 different tools.

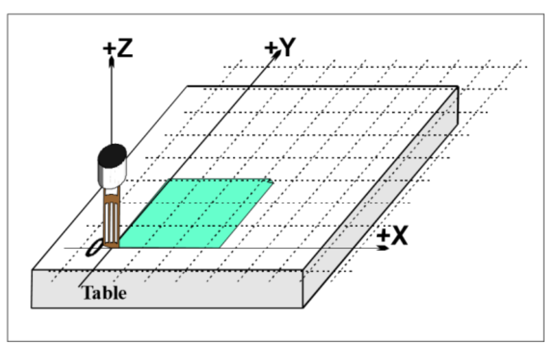


Figure 5-70 Disaster at 0,0,0!

On the Offsets screen you will see space for a Tool number and information about the tool. The DROs are labelled *Z-offset*, *Diameter* and *T*. Ignore the DRO Touch Correction and its associated button marked On/Off for now. By default you will have Tool #0 selected but its offsets will be switched OFF. Information about the tool diameter is also used for Cutter Compensation (q.v.)

PRE-SETTABLE TOOLS

We will assume your machine has a tool holder system which lets you put a tool in at exactly the same position each time. This might be a mill with lots of chucks or something like the Novakon Quick Change Tooling. If your tool position is different each time then you will have to set up the offsets each time you change it. This will be described later.

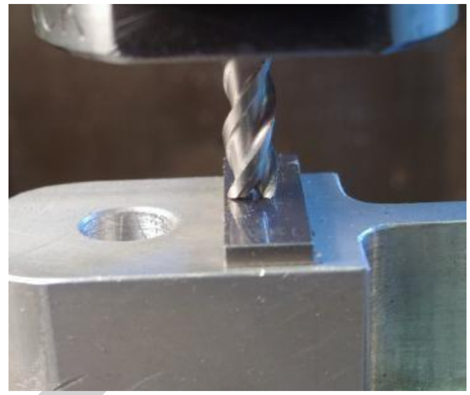


Figure 5-71 Endmill in a preset table holder

In our drawing machine, suppose the pens register in a blind hole that is 1" deep in the pen holder. The red pen is **4.2**" long and the blue one 3.7" long.

Suppose the machine has just been referenced/homed and a work offset defined for the corner of the paper with Z = 0.0 being the table using the bottom face of the empty pen holder. You would jog the Z axis up say to 5" and fit the blue pen. Enter "1" (which will be the blue pen) in the Tool number DRO but do not click *Offset On/Off* to ON yet. Jog the Z down to touch the paper. The Z axis DRO would read 2.7 as the pen sticks 2.7" out of the holder. Then you click the Touch button by the Z offset. This would load the (2.7") into the Z offset of Tool #1. Clicking the *Offset On/Off* toggle would light the LED and apply the tool offset and so the Z axis DRO will read 0.0. You could draw the square by running the example part program as before.

Next to use the red pen you would jog the Z axis up (say to Z = 5.0 again) to take out the blue pen and put in the red. Physically swapping the pens obviously does not alter the axis DROs. Now you would, switch Off the tool offset LED, select Tool #2 , jog and *Touch* at the corner of the paper. This would set up tool 2's Z offset to 3.2". Switching On the offset for Tool #2 again will display Z = 0.0 on the axis DRO so the part program would draw the red square (over the blue one).

Now that tools 1 and 2 are set up you can change them as often as you wish and get the correct Current Coordinate system by selecting the appropriate tool number and switching its offsets on. This tool selection and switching on and off of the offsets can be done in the part program (T word, M6, G43 and G49) and there are DROs on the standard Program Run screen.

NON-PRE-SETTABLE TOOLS

Some tool holders do not have a way of refitting a given tool in exactly the same place each time. For example the collet of a router is usually bored too deep to bottom the tool. In this case it may still be worth setting up the tool offset (say with tool #1) each time it is changed. If you do it this way you can still make use of more than one work offset (see 2 and 3 pin fixtures illustrated below in Figure 5-72 and Figure 5-73. If you do not have a physical fixture it may be just as easy to redefine the Z of the work offsets offsets each time you change the tool.

HOW THE OFFSET VALUES ARE STORED

The 254 work offsets are stored in one table in Mach3. The 255 tool offsets and diameters are stored in another table. You can view these tables using the Work Offsets Table and Tool Offsets Table buttons on the offsets screen. These tables have space for additional information which is not at present used by Mach3

Mach3 will generally try to remember the values for all work and tool offsets from one run of the program to another but will prompt you on closing down the program to check that you do want to save any altered values. Check boxes on the Config>State dialog (q.v.) allow you to change this behavior so that Mach3 will either automatically save the values without bothering to ask you or will never save them automatically.

However the automatic saving options are configured, you can use the Save button on the dialogs which display the tables to force a save to occur.

DRAWING LOTS OF COPIES - FIXTURES

Now imagine we want to draw on many sheets of paper. It will be difficult to tape each one in the same place on the table and so will be necessary to set the work offsets each time. Much better would be to have a plate with pins sticking out of it and to use pre-punched paper to register on the pins. You will probably recognize this as an example of a typical fixture which has long been used in machine shops. Figure 5-72 shows the machine so equipped. It would be common for the fixture to have dowels or something similar so that it always mounts in the same place on the table.

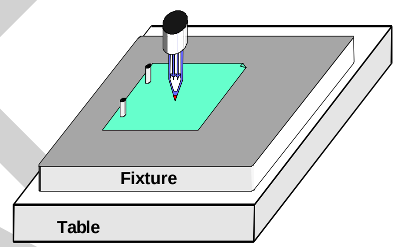


Figure 5-72 Machine with Two Pin Fixture

We could now move Current Coordinate system by setting the work offsets #1 to the corner of the paper on the actual fixture. Running the example program would draw the square exactly as before. This will of course take care of the difference in Z coordinates caused by the thickness of the fixture. We can put new pieces of paper on the pins and get the square in exactly the right place on each with no further setting up.

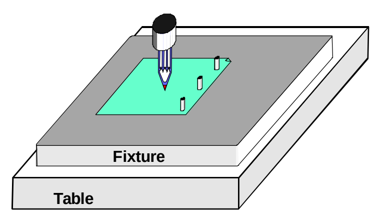


Figure 5-73 Three Pin Fixture

We might also have another fixture for three-hole paper (Figure 5-74) and might want to swap between the two and three pin fixtures for different jobs so work offset #2 could be defined for the corner of the paper on the three pin fixture.

You can of course define any point on the fixture as the origin of its offset coordinate system. For the drawing machine we would want to make the bottom left corner of the paper be X=0 & Y=0 and the top surface of the fixture be Z=0.

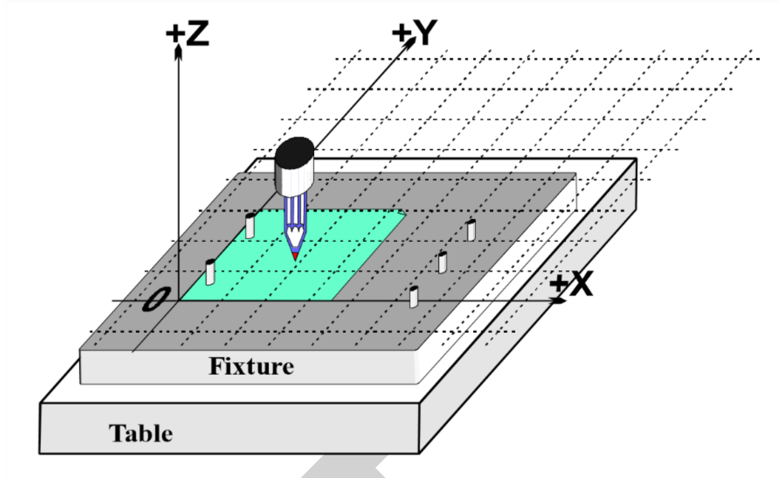


Figure 5-74 A Double Fixture

It is common for one physical fixture to be able to be used for more than one job. Figure 5-75 shows the two and three hole fixtures combined. You would of course have two entries in the work offset corresponding to the offsets to be used for each. In Figure 5-73, the Current Coordinate system is shown set for using the two-hole paper option.

PRACTICALITIES OF "TOUCHING"

END MILLS

On a manual machine tool it is quite easy to feel on the handles when a tool is touching the work but for accurate work it is better to have a feeler (perhaps a piece of paper or plastic from a candy bar) or slip gage so you can tell when it is being pinched. This is illustrated on a mill in Figure 5-75.

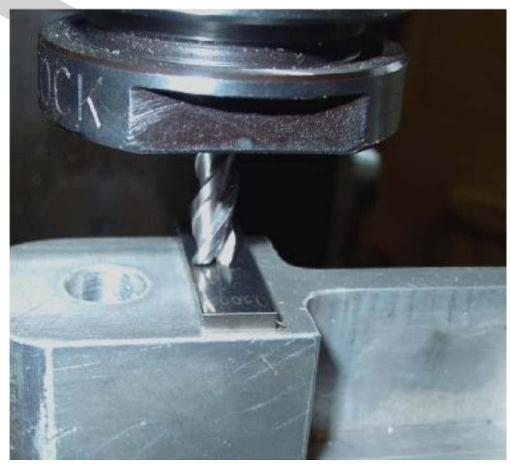


Figure 5-75 Using a Slip Gauge whe Touching Z Offset on a Mill

On the Offset screen you can enter the thickness of this feeler or slip gage into the DRO beside the Set Tool Offset button. When you use Sret Tool Offset to set an offset DRO for a too, then the thickness of the gage will be allowed for.

For example suppose you had the axis DRO Z = -3.518 with the 0.1002" slip lightly held. Choose Tool #3 by typing 3 in the Tool DRO. Enter 0.1002 in the DRO in Gage Block Height and click Set Tool Offset. After the touch the axis DRO reads Z = 0.1002 (i.e the Controlled Point is 0.1002) and tool 3 will have has Z offset -0.1002.Figure 5-76 shows this process just before clicking Set Tool Offset.

If you have an accurate cylindrical gage and a reasonable sized flat surface on the top of the work piece, then using it can be even better than jogging down to a feeler or slip gage. Jog down so that the roller will not pass under the tool. Now very slowly jog up until you can just roll it under the tool. Then you can click the Touch button. There is an obvious safety advantage in that jogging a bit too high does no harm; you just have to start again. Jogging down to a feeler or gage risks damage to the cutting edges of the tool.

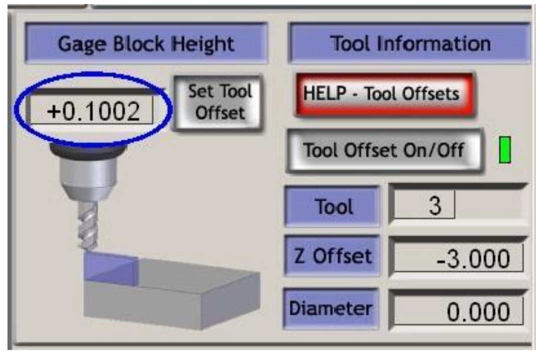


Figure 5-76 – Entering Z offset data

EDGE FINDING

It is very difficult to accurately set a mill to an edge in X or Y due to the flutes of the tool. A special edge-finder tool helps here,Figure 5-77 shows the minus X edge of a part being found.

The Touch Correction can be used here as well. You will need the radius of the probe tip and the thickness of any feeler or slip gage.

Figure 5-77 - Edge-finder in use on a mill

G52 & G92 OFFSETS

There are two further ways of offsetting the Controlled Point using G-codes *G52* and *G92*.

When you issue a *G52* you tell Mach3 that for any value of the controlled point (e.g. X=0, Y= 0) you want the actual machine position offset by adding the given values of X, Y and/or Z.

When you use *G92* you tell Mach3 what you want the coordinates of the current Controlled Point to be values given by X, Y and/or Z.

Neither *G52* nor *G92* move the tool they just add another set of offsets to the origin of the Current Coordinate system.

A simple example of using *G52* is where you might wish to produce two identical shapes ate different places on the work piece. The code we looked at before draws a 1" square with a corner at X = 0.8, Y = 0.3:

```
G20 F10 G90 (set up imperial units, a slow feed rate etc.)
G0 Z2.0 (lift pen)
G0 X0.8 Y0.3 (rapid to bottom left of square)
G1 Z0.0 (pen down)
Y1.3 (we can leave out the G1 as we have just done one)
X1.8
Y0.3 (going clockwise round shape)
X0.8
G0 X0.0 Y0.0 Z2.0 (move pen out of the way and lift it)
```

If we want another square but the second one with its corner at X= 3.0 and Y = 2.3 then the above code can be used twice but using G52 to apply and offset before the second copy.

```
G20 F10 G90 (set up imperial units, a slow feed rate etc.)
G0 Z2.0 (lift pen)
G0 X0.8 Y0.3 (rapid to bottom left of square)
G1 Z0.0 (pen down)
Y1.3 (we can leave out the G1 as we have just done one)
X1.8
Y0.3 (going clockwise round shape)
X0.8
G0 Z2.0 (lift pen)
G52 X2.2 Y2 (temporary offset for second square)
G0 X0.8 Y0.3 (rapid to bottom left of square)
G1 Z0.0 (pen down)
Y1.3 (we can leave out the G1 as we have just done one)
X1.8
Y0.3 (going clockwise round shape)
X0.8
G52 X0 Y0 (Get rid of temporary offsets)
G0 X0.0 Y0.0 Z2.0 (move pen out of the way and lift it)
```

Copying the code is not very elegant but as it is possible to have a G-code subroutine (See *M98* and *M99*) the common code can be written once and called as many times as you need – twice in this example.

The subroutine version is shown below. The pen up/down commands have been tidied up and the subroutine actually draws at 0,0 with a *G52* being used for setting the corner of both squares:

G20 F10 G90 (set up imperial units, a slow feed rate etc.)
G52 X0.8 Y0.3 (start of first square)
G52 X0.8 Y0.3
M98 P1234 (call subroutine for square in first position)
G52 X3 Y2.3 (start of second square)
M98 P1234 (call subroutine for square in second position)
G52 X0 Y0 {IMPORTANT – get rid of G52 offsets)
M30 (rewind at end of program)
O1234 (Start of subroutine 1234)
G0 X0 Y0 (rapid to bottom left of square)
G1 Z0.0 (pen down)
Y1 (we can leave out the G1 as we have just done one)
X1
Y0 (going clockwise round shape)
X0
G0 Z2.0 (lift pen)
M99 (return from subroutine)

Notice that each *G52* applies a new set of offsets which take no account of any previously issued *G52*.

USING G92

The simplest example with *G92* is, at a given point, to set X & Y to zero but you can set any values. The easiest way to cancel *G92* offsets is to enter *G92.1* on the MDI line.

TAKE CARE WITH G52 AND G92

You can specify offsets on as many axes as you like by including a value for their axis letter. If an axis name is not given then its offset remains unaltered.

Mach3 uses the same internal mechanisms for *G52* and *G92* offsets; it just does different calculations with your X, Y and Z words. If you use *G52* and *G92* together you (and even Mach3) will become so confused that disaster will inevitably occur. If you really want to prove you have understood how they work, set up some offsets and move the controlled point to a set of coordinates, say X=2.3 and Y=4.5. Predict the absolute machine coordinates you should have and check them by making Mach3 display machine coordinates with the "Mach" button.

Do not forget to clear the offsets when you have used them.

Warning**!** Almost everything that can be done with G92 offsets can be done better using work offsets or perhaps *G52* offsets. Because *G92* relies on where the controlled point is as well as the axis words at the time *G92* is issued, changes to programs can easily introduce serious bugs leading to crashes.

Many operators find it hard to keep track of three sets of offsets (Work, Tool and *G52/G92*) and if you get confused you will soon break either your tool or worse your machine!

TOOL DIAMETER

Suppose the blue square drawn using our machine is the outline for a hole in the lid of a child's shape-sorter box into which a blue cube will fit. Remember G-codes move the Controlled Point. The example part program drew a 1" square. If the tool is a thick felt pen then the hole will be significantly smaller than 1" square (Figure 5-78).

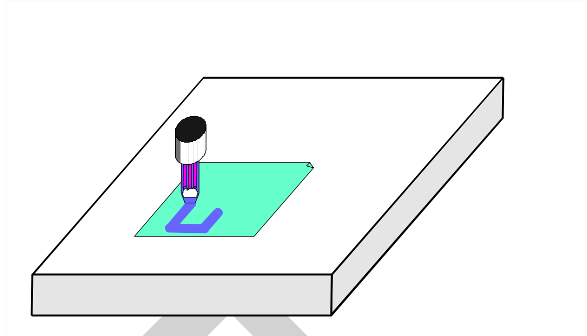


Figure 5-78 Using a large diameter tool (felt pen)

The same problem obviously occurs with an end mill/slot drill. You may want to cut a pocket or be leaving an island. These need different compensation.

This sounds easy to do but in practice there are many "devils in the detail" concerned with the beginning and end of the cutting. It is usual for a Wizard or your CAD/CAM software to deal with these issues. Mach3, however, allows a part program to compensate for the diameter of the chosen tool with the actual cutting moves being specified as, say, the 1" square. This feature is important if the author of the part program does not know the exact diameter of the cutter that will be used (e.g. it may be smaller than nominal due to repeated sharpening). The tool table lets you define the diameter of the tool or, is some applications, the difference from the nominal tool diameter of the actual tool being used – perhaps after multiple sharpening. See Cutter Compensation chapter for full details.

CUTTER COMPENSATION

Cutter compensation is a feature of Mach3 which you many never have to use. Most CAD/CAM programs can be told the nominal diameter of your mill and will output part programs which cut the part outline or pocket which you have drawn by themselves allowing for the tool diameter. Because the CAD/CAM software has a better overall view of the shapes being cut it may be able to do a better job than Mach3 can when avoiding gouges at sharp internal corners. Having compensation in Mach3 allows you to: (a) use a tool different in diameter from that programmed (e.g. because it has be reground) or (b) to use a part program that describes the desired outline rather than the path of the center of the tool (perhaps one written by hand).

However, as compensation is not trivial, it is described in this section should you need to use it.

INTRODUCTION TO COMPENSATION

As we have seen Mach1 controls the movement of the Controlled Point. In practice no tool (except perhaps a Vengraver) is a point so cuts will be made at a different place to the Controlled Point depending on the radius of the cutter.

It is generally easiest to allow your CAD/CAM software to take account of this when cutting out pockets or the outline of shapes.

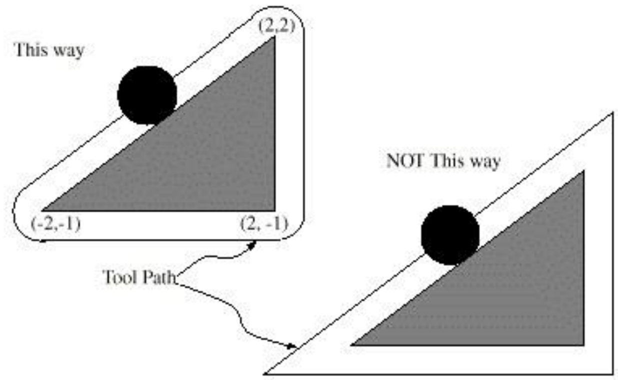


Figure 5-79 Two possible tool paths to cut triangle

Mach3 does, however, support calculations to compensate for the diameter (radius) of the cutter. In industrial applications this is aimed at allowing for a cutter which, through regrinding, is not exactly the diameter of the tool assumed when the part program was written. The compensation can be enabled by the machine operator rather than requiring the production of another part program.

Of the face of it, the problem should be easy to solve. All you need to do is to offset the controlled point by an appropriate X and Y to allow for the tool radius. Simple trigonometry gives the distances depending on the angle the direction of cut makes to the axes.

In practice it is not quite so easy. There are several issues but the main one is that the machine has to set a Z position before it starts cutting and at that time it does not know the direction in which the tool is going to be moving. This problem is solved by providing "pre-entry moves" which take place in waste material of the part. These ensure that the compensation calculations can be done before the actual part outline is being cut. Choice of a path which runs smoothly into the part's outline also optimizes the surface finish. An exit move is sometimes used to maintain the finish at the end of a cut.

TWO KINDS OF CONTOUR

Mach3 handles compensation for two types of contour:

♦ The contour given in the part program code is the edge of material that is not to be machined away. We will call this type a "material edge contour". This is the sort of code that might be "hand written"

♦ The contour given in the NC code is the tool path that would be followed by a tool of exactly the correct radius. We will call this type a "tool path contour". This is the sort of code that a CAD/CAM program might produce if it is aware of the intended cutter diameter

The interpreter does not have any setting that determines which type of contour is used, but the numerical description of the contour will, of course, differ (for the same part geometry) between the two types and the values for diameters in the tool table will be different for the two types.

MATERIAL EDGE CONTOUR

When the contour is the edge of the material, the outline of the edge is described in the part program. For a material edge contour, the value for the diameter in the tool table is the actual value of the diameter of the tool. The value in the table must be positive. The NC code for a material edge contour is the same regardless of the (actual or intended) diameter of the tool.

Example1:

Here is an NC program which cuts material away from the outside of the triangle in Figure 5-79, page 5-81. In this example, the cutter compensation radius is the actual radius of the tool in use, which is 0.5. The value for the diameter in the tool table is twice the radius, which is 1.0.

> *N0010 G41 G1 X2 Y2* (turn compensation on and make entry move)
> *N0020 Y-1* (follow right side of triangle)
> *N0030 X-2* (follow bottom side of triangle)
> *N0040 X2 Y2* (follow hypotenuse of triangle)
> *N0050 G40* (turn compensation off)

This will result in the tool following a path consisting of an entry move and the path shown on the left going clockwise around the triangle. Notice that the coordinates of the triangle of material appear in the NC code. Notice also that the tool path includes three arcs which are not explicitly programmed; they are generated automatically.

TOOL PATH CONTOUR

When the contour is a tool path contour, the path is described in the part program. It is expected that (except for during the entry moves) the path is intended to create some part geometry. The path may be generated manually or by a CAD/CAM program, considering the part geometry which is intended to be made. For Mach3 to work, the tool path must be such that the tool stays in contact with the edge of the part geometry, as shown on the left side of Figure 5-79, page 5-81. If a path of the sort shown on the right of Figure 5-79 is used, in which the tool does not stay in contact with the part geometry all the time, the interpreter will not be able to compensate properly when undersized tools are used.

For a tool path contour, the value for the cutter diameter in the tool table will be a small positive number if the selected tool is slightly oversized and will be a small negative number if the tool is slightly undersized. As implemented, if a cutter diameter value is negative, the interpreter compensates on the other side of the contour from the one programmed and uses the absolute value of the given diameter. If the actual tool is the correct size, the value in the table should be zero.

Tool Path Contour example:

Suppose the diameter of the cutter currently in the spindle is 0.97, and the diameter assumed in generating the tool path was 1.0. Then the value in the tool table for the diameter for this tool should be -0.03. Here is an NC program which cuts material away from the outside of the triangle in the figure.

N0010 G1 X1 Y4.5 (make alignment move)
N0020 G41 G1 Y3.5 (turn compensation on and make first entry move)
N0030 G3 X2 Y2.5 I1 (make second entry move)
N0040 G2 X2.5 Y2 J-0.5 (cut along arc at top of tool path)
N0050 G1 Y-1 (cut along right side of tool path)
N0060 G2 X2 Y-1.5 I-0.5 (cut along arc at bottom right of tool path)
N0070 G1 X-2 (cut along bottom side of tool path)
N0080 G2 X-2.3 Y-0.6 J0.5 (cut along arc at bottom left of tool path)
N0090 G1 X1.7 Y2.4 (cut along hypotenuse of tool path)
N0100 G2 X2 Y2.5 I0.3 J-0.4 (cut along arc at top of tool path)
N0110 G40 (turn compensation off)

This will result in the tool making an alignment move and two entry moves, and then following a path slightly inside the path shown on the left in Figure 5-79 Two possible tool paths to cut triangle, going clockwise around the triangle. This path is to the right of the programmed path even though G41 was programmed, because the diameter value is negative.

PROGRAMMING ENTRY MOVES

In general, an alignment move and an entry moves are needed to begin compensation correctly. The tool should be at least a diameter away from the finished cut before the entry move is made.

MACH 2 G AND M-CODE LANGUAGE REFERENCE

LINEAR AXES

The X, Y, and Z axes form a standard right-handed coordinate system of orthogonal linear axes. Positions of the three linear motion mechanisms are expressed using coordinates on these axes.

ROTATIONAL AXES

The rotational axes are measured in degrees as wrapped linear axes in which the direction of positive rotation is counterclockwise when viewed from the positive end of the corresponding X, Y, or Z-axis. By "wrapped linear axis," we mean one on which the angular position increases without limit (goes towards plus infinity) as the axis turns counterclockwise and decreases without limit (goes towards minus infinity) as the axis turns clockwise. Wrapped linear axes are used regardless of whether or not there is a mechanical limit on rotation.

Clockwise or counterclockwise is from the point of view of the work piece. If the work piece is fastened to a turntable which turns on a rotational axis, a counterclockwise turn from the point of view of the work piece is accomplished by turning the turntable in a direction that (for most common machine configurations) looks clockwise from the point of view of someone standing next to the machine.

SCALING INPUT

It is possible to set up scaling factors for each axis. These will be applied to the values of X, Y, Z, A, B, C, I, J and R words whenever these are entered. This allows the size of features machined to be altered and mirror images to be created - by use of negative scale factors.

The scaling is the first thing done with the values and things like feed rate are always based on the scaled values.

The offsets stored in tool and fixture tables are not scaled before use. Scaling may, of course, have been applied at the time the values were entered (say using G10).

CONTROLLED POINT

The controlled point is the point whose position and rate of motion are controlled. When the tool length offset is zero (the default value), this is a point on the spindle axis (often called the gauge point) that is some fixed distance beyond the end of the spindle, usually near the end of a tool holder that fits into the spindle. The location of the controlled point can be moved out along the spindle axis by specifying some positive amount for the tool length offset. This amount is normally the length of the cutting tool in use, so that the controlled point is at the end of the cutting tool.

CO-ORDINATED LINEAR MOTION

To drive a tool along a specified path, a machining system must often co-ordinate the motion of several axes. We use the term "coordinated linear motion" to describe the situation in which, nominally, each axis moves at constant speed and all axes move from their starting positions to their end positions at the same time. If only the X, Y, and Z axes (or any one or two of them) move, this produces motion in a straight line, hence the word "linear" in the term. In actual motions, it is often not possible to maintain constant speed because acceleration or deceleration is required at the beginning and/or end of the motion. It is feasible, however, to control the axes so that, at all times, each axis has completed the same fraction of its required motion as the other axes. This moves the tool along the same path, and we also call this kind of motion coordinated linear motion.

Co-ordinated linear motion can be performed either at the prevailing feed rate, or at rapid traverse rate. If physical limits on axis speed make the desired rate unobtainable, all axes are slowed to maintain the desired path.

FEED RATE

The rate at which the controlled point or the axes move is nominally a steady rate which may be set by the user. In the Interpreter, the interpretation of the feed rate is as follows unless inverse time feed rate (G93) mode is being used:

- ♦ For motion involving one or more of the linear axes (X, Y, Z and optionally A, B, C), without simultaneous rotational axis motion, the feed rate means length units per minute along the programmed linear XYZ(ABC) path

- ♦ For motion involving one or more of the linear axes (X, Y, Z and optionally A, B, C), with simultaneous rotational axis motion, the feed rate means length units per minute along the programmed linear XYZ(ABC) path combined with the angular velocity of the rotary axes multiplied by the appropriate axis Correction Diameter multiplied by pi ($\pi = 3.14152\ldots$); i.e. the declared "circumference" of the part

- ♦ For motion of one rotational axis with X, Y, and Z axes not moving, the feed rate means degrees per minute rotation of the rotational axis.

- ♦ For motion of two or three rotational axes with X, Y, and Z axes not moving, the rate is applied as follows. Let dA, dB, and dC be the angles in degrees through which the A, B, and C axes, respectively, must move. Let $D = \text{sqrt}(dA^2 + dB^2 + dC^2)$. Conceptually, D is a measure of total angular motion, using the usual Euclidean metric. Let T be the amount of time required to move through D degrees at the current feed rate in degrees per minute. The rotational axes should be moved in coordinated linear motion so that the elapsed time from the start to the end of the motion is T plus any time required for acceleration or deceleration.

ARC MOTION

Any pair of the linear axes (XY, YZ, XZ) can be controlled to move in a circular arc in the plane of that pair of axes. While this is occurring, the third linear axis and the rotational axes can be controlled to move simultaneously at effectively a constant rate. As in coordinated linear motion, the motions can be coordinated so that acceleration and deceleration do not affect the path.

If the rotational axes do not move, but the third linear axis does move, the trajectory of the controlled point is a helix.

The feed rate during arc motion is as described in Feed Rate above. In the case of helical motion, the rate is applied along the helix. Beware as other interpretations are used on other systems.

COOLANT

Flood coolant and mist coolant may each be turned on independently. They are turned off together.

DWELL

A machining system may be commanded to dwell (i.e., keep all axes unmoving) for a specific amount of time. The most common use of dwell is to break and clear chips or for a spindle to get up to speed. The units in which you specify Dwell are either seconds or Milliseconds depending on the setting on Configure>Logic

UNITS

Units used for distances along the X, Y, and Z axes may be measured in millimeters or inches. Units for all other quantities involved in machine control cannot be changed. Different quantities use different specific units. Spindle speed is measured in revolutions per minute. The positions of rotational axes are measured in degrees. Feed rates are expressed in current length units per minute or in degrees per minute, as described above.

Warning: We advise you to check very carefully the system's response to changing units while tool and fixture offsets are loaded into the tables, while these offsets are active and/or while a part program is executing

CURRENT POSITION

The controlled point is always at some location called the "current position" and Mach3 always knows where that is. The numbers representing the current position are adjusted in the absence of any axis motion if any of several events take place:

- ♦ Length units are changed (but see Warning above)
- ♦ Tool length offset is changed
- ♦ Coordinate system offsets are changed.

SELECTED PLANE

There is always a "selected plane", which must be the XY-plane, the YZ-plane, or the XZ-plane of the machining system. The Z-axis is, of course, perpendicular to the XY-plane, the X-axis to the YZ-plane, and the Y-axis to the XZ-plane.

TOOL TABLE

Zero or one tool is assigned to each slot in the tool table.

TOOL CHANGE

Mach3 allows you to implement a procedure for implementing automatic tool changes using macros or to change the tools by hand when required.

PALLET SHUTTLE

Mach3 allows you to implement a procedure for implementing pallet shuttle using macros.

PATH CONTROL MODES

The machining system may be put into any one of two path control modes: (1) exact stop mode, (2) constant velocity mode. In exact stop mode, the machine stops briefly at the end of each programmed move. In constant velocity mode, sharp corners of the path may be rounded slightly so that the feed rate may be kept up. These modes are to allow the user to control the compromise involved in turning corners because a real machine has a finite acceleration due to the inertia of its mechanism.

Exact stop does what it says. The machine will come to rest at each change of direction and the tool will therefore precisely follow the commanded path.

Constant velocity will overlap acceleration in the new direction with deceleration in the current one in order to keep the commanded feed rate. This implies a rounding of any corner but faster and smoother cutting. This is particularly important in routing and plasma cutting.

The lower the acceleration of the machine axes, the greater will be the radius of the rounded corner.

In Plasma mode (set on Configure Logic dialog) the system attempts to optimize corner motion for plasma cutting by a proprietary algorithm.

It is also possible to define an limiting angle so that changes in direction of more than this angle will always be treated as Exact Stop even though Constant Velocity is selected. This allows gentle corners to be smoother but avoids excessive rounding of sharp corners even on machines with low acceleration on one or more axes. This feature is enabled in the Configure Logic dialog and the limiting angle is set by a DRO. This setting will probably need to be chosen experimentally depending on the characteristics of the machine tool and, perhaps, the tool path of an individual job.

INTERPRETER INTERACTION WITH CONTROLS

FEED AND SPEED OVERRIDE CONTROLS

Mach3 commands which enable (M48) or disable (M49) the feed and speed override switches. It is useful to be able to override these switches for some machining operations. The idea is that optimal settings have been included in the program, and the operator should not change them.

BLOCK DELETE CONTROL

If the block delete control is ON, lines of code which start with a slash (the block delete character) are not executed. If the switch is off, such lines are executed.

OPTIONAL PROGRAM STOP CONTROL

The optional program stop control (see Configure>Logic) works as follows. If this control is ON and an input line contains an M1 code, program execution is stopped at the end on the commands on that line until the *Cycle Start* button is pushed.

TOOL FILE

Mach3 maintains a tool file for each of the 254 tools which can be used. Each data line of the file contains the data for one tool. This allows the definition of the tool length (Z axis), tool diameter (for milling) and tool tip radius (for turning)

THE LANGUAGE OF PART PROGRAMS

OVERVIEW

The language is based on lines of code. Each line (also called a "block") may include commands to the machining system to do several different things. Lines of code may be collected in a file to make a program.

A typical line of code consists of an optional line number at the beginning followed by one or more "words." A word consists of a letter followed by a number (or something that evaluates to a number). A word may either give a command or provide an argument to a command. For example, `G1 X3` is a valid line of code with two words. "G1" is a command meaning "move in a straight line at the programmed feed rate," and "X3" provides an argument value (the value of X should be 3 at the end of the move). Most commands start with either G or M (for General and Miscellaneous). The words for these commands are called "G codes" and "M codes."

The language has two commands (M2 or M30), either of which ends a program. A program may end before the end of a file. Lines of a file that occur after the end of a program are not to be executed in the normal flow so will generally be parts of subroutines.

PARAMETERS

A Mach3 machining system maintains an array of 10,320 numerical parameters. Many of them have specific uses. The parameter which are associated with fixtures are persistent over time. Other parameters will be undefined when Mach3 is loaded. The parameters are preserved when the interpreter is reset. The parameters with meanings defined by Mach3 are given in Table 5-4 System Defined Parameters

Parameter number	Meaning	Parameter number	Meaning
5161	G28 home X	5261	Work offset 3 X
5162	G28 home Y	5262	Work offset 3 Y
5163	G28 home Z	5263	Work offset 3 Z
5164	G28 home A	5264	Work offset 3 A
5165	G28 home B	5265	Work offset 3 B
5166	G28 home C	5266	Work offset 3 C
5181	G30 home X	5281	Work offset 4 X
5182	G30 home Y	5282	Work offset 4 Y
5183	G30 home Z	5283	Work offset 4 Z
5184	G30 home A	5284	Work offset 4 A
5185	G30 home B	5285	Work offset 4 B
5186	G30 home C	5286	Work offset 4 C
5191	Scale X Scale Y	5301	Work offset 5 X
5192	Scale Z	5302	Work offset 5 Y
5193	Scale A	5303	Work offset 5 Z
5194	Scale B	5304	Work offset 5 A
5195	Scale C	5305	Work offset 5 B
5196	G92 offset X	5306	Work offset 5 C
5211	G92 offset Y	5321	Work offset 6 X
5212	G92 offset Z	5322	Work offset 6 Y
5213	G92 offset A	5323	Work offset 6 Z
5214	G92 offset B	5324	Work offset 6 A
5215	G92 offset C	5325	Work offset 6 B
5216	Current Work	5326	Work offset 6 C
5220	offset number		
5221	Work offset 1 X		And so on every
5222	Work offset 1 Y		20
5223	Work offset 1 Z		values until
5224	Work offset 1 A	10281	
5225	Work offset 1 B	10282	Work offset 254 X
5226	Work offset 1 C	10283	Work offset 254 Y
5241	Work offset 2 X	10284	Work offset 254 Z
5242	Work offset 2 Y	10285	Work offset 254 A
5243	Work offset 2 Z	10286	Work offset 254 B
5244	Work offset 2 A	10301	Work offset 254 C
5245	Work offset 2 B	10302	Work offset 255 X
5246	Work offset 2 C	10303	Work offset 255 Y
		10304	Work offset 255 Z
		10305	Work offset 255 A
		10306	Work offset 255 B
			Work offset 255 C

Table 5-4 System Defined Parameters

COORDINATE SYSTEMS

The machining system has an absolute coordinate system and 254 work offset (fixture) systems.

You can set the offsets of tools by `G10 L1 P~ X~ Z~`. The P word defines the tool offset number to be set.

You can set the offsets of the fixture systems using `G10 L2 P~ X~ Y~ Z~ A~ B~ C~` The P word defines the fixture to be set. The `X, Y, Z` etc. words are the coordinates for the origin of for the axes in terms of the absolute coordinate system.

You can select one of the first seven work offsets by using `G54, G55, G56, G57, G58, G59`. Any of the 255 work offsets can be selected by `G59 P~` (e.g. `G59 P23` would select fixture 23). The absolute coordinate system can be selected by `G59 P0`.

You can offset the current coordinate system using `G92 or G92.3`. This offset will then applied on top of work offset coordinate systems. This offset may be cancelled with `G92.1 or G92.2`. You can make straight moves in the absolute machine coordinate system by using `G53` with either `G0 or G1`.

FORMAT OF A LINE

A permissible line of input code consists of the following, in order, with the restriction that there is a maximum (currently 256) to the number of characters allowed on a line.

- ♦ an optional block delete character, which is a slash "/”.
- ♦ an optional line number.
- ♦ any number of words, parameter settings, and comments.
- ♦ an end of line marker (carriage return or line feed or both).

Any input not explicitly allowed is illegal and will cause the Interpreter to signal an error or to ignore the line.

Spaces and tabs are allowed anywhere on a line of code and do not change the meaning of the line, except inside comments. This makes some strange-looking input legal. For example, the line `g0x +0. 12 34y 7` is equivalent to `g0 x+0.1234 y7` Blank lines are allowed in the input. They will be ignored.

Input is case insensitive, except in comments, i.e., any letter outside a comment may be in upper or lower case without changing the meaning of a line.

LINE NUMBER

A line number is the letter N followed by an integer (with no sign) between 0 and 99999 written with no more than five digits (000009 is not OK, for example). Line numbers may be repeated or used out of order, although normal practice is to avoid such usage. A line number is not required to be used (and this omission is common) but it must be in the proper place if it is used.

SUBROUTINE LABELS

A subroutine label is the letter O followed by an integer (with no sign) between 0 and 99999 written with no more than five digits (000009 is not permitted, for example). Subroutine labels may be used in any order but must be unique in a program although violation of this rule may not be flagged as an error. Nothing else except a comment should appear on the same line after a subroutine label.

WORD

A word is a letter other than N or O followed by a real value.

Words may begin with any of the letters shown in Table 5-5.

Letter	Meaning
A	A-axis of machine
B	B-axis of machine
C	C-axis of machine
D	tool radius compensation number
F	feed rate
G	general function (see Table 5)
H	tool length offset index
I	X-axis offset for arcs X offset in G87 canned cycle
J	Y-axis offset for arcs Y offset in G87 canned cycle
K	Z-axis offset for arcs Z offset in G87 canned cycle
L	number of repetitions in canned cycles/subroutines key used with G10
M	miscellaneous function (see Table 7)
N	line number
O	Subroutine label number
P	dwell time in canned cycles dwell time with G4 key used with G10
Q	feed increment in G83 canned cycle repetitions of subroutine call
R	arc radius canned cycle retract level
S	spindle speed
T	tool selection
U	Synonymous with A
V	Synonymous with B
W	Synonymous with C
X	X-axis of machine
Y	Y-axis of machine
Z	Z-axis of machine

Table 5-5 Word Initial Letters

The table includes N and O for completeness, even though, as defined above, line numbers are not words. Several letters (I, J, K, L, P, R) may have different meanings in different contexts.

A real value is some collection of characters that can be processed to come up with a number. A real value may be an explicit number (such as 341 or -0.8807), a parameter value, an expression, or a unary operation value. Definitions of these follow immediately. Processing characters to come up with a number is called "evaluating". An explicit number evaluates to itself.

NUMBER

The following rules are used for (explicit) numbers. In these rules a digit is a single character between 0 and 9.

- ♦ A number consists of (1) an optional plus or minus sign, followed by (2) zero to many digits, followed, possibly, by (3) one decimal point, followed by (4) zero to many digits provided that there is at least one digit somewhere in the number.
- ♦ There are two kinds of numbers: integers and decimals. An integer does not have a decimal point in it; a decimal does.
- ♦ Numbers may have any number of digits, subject to the limitation on line length. Only about seventeen significant figures will be retained, however (enough for all known applications).
- ♦ A non-zero number with no sign as the first character is assumed to be positive.

Notice that initial (before the decimal point and the first non-zero digit) and trailing (after the decimal point and the last non-zero digit) zeros are allowed but not required. A number written with initial or trailing zeros will have the same value when it is read as if the extra zeros were not there.

Numbers used for specific purposes by Mach3 are often restricted to some finite set of values or some to some range of values. In many uses, decimal numbers must be close to integers; this includes the values of indexes (for parameters and carousel slot numbers, for example), M codes, and G codes multiplied by ten. A decimal number which is supposed be close to an integer is considered close enough if it is within 0.0001 of an integer.

PARAMETER VALUE

A parameter value is the hash character # followed by a real value. The real value must evaluate to an integer between 1 and 10320. The integer is a parameter number, and the value of the parameter value is whatever number is stored in the numbered parameter.

The # character takes precedence over other operations, so that, for example, `#1+2` means the number found by adding 2 to the value of parameter 1, not the value found in parameter 3. Of course, `#[1+2]` does mean the value found in parameter 3. The `#` character may be repeated; for example `##2` means the value of the parameter whose index is the (integer) value of parameter 2.

EXPRESSIONS AND BINARY OPERATIONS

6) An expression is a set of characters starting with a left bracket [and ending with a balancing right bracket]. In between the brackets are numbers, parameter values, mathematical operations, and other expressions. An expression may be evaluated to produce a number. The expressions on a line are evaluated when the line is read, before anything on the line is executed. An example of an expression is:

```
[1+acos[0]-[#3**[4.0/2]]]
```

7) Binary operations appear only inside expressions. Nine binary operations are defined. There are four basic mathematical operations: addition (+), subtraction (-), multiplication (*), and division (/). There are three logical operations: non-exclusive or (OR), exclusive or (XOR), and logical and (AND). The eighth operation is the modulus operation (MOD). The ninth operation is the "power" operation (**) of raising the number on the left of the operation to the power on the right.

8) The binary operations are divided into three groups. The first group is: power. The second group is: multiplication, division, and modulus. The third group is: addition, subtraction, logical non-exclusive or, logical exclusive or, and logical and. If operations are strung together (for example in the expression `[2.0/3*1.5-5.5/11.0]`), operations in the first group are to be performed before operations in the second group and operations in the second group before operations in the third group. If an expression contains more than one operation from the same group (such as the first / and * in the example), the operation on the left is performed first. Thus, the example is equivalent to: `[((2.0/3)*1.5)(5.5/11.0)]` which simplifies to: `[1.0-0.5]` which is 0.5.

9) The logical operations and modulus are to be performed on any real numbers, not just on integers. The number zero is equivalent to logical false, and any non-zero number is equivalent to logical true.

UNARY OPERATION VALUE

A unary operation value is either "ATAN" followed by one expression divided by another expression (for example `ATAN[2]/[1+3]`) or any other unary operation name followed by an expression (for example `SIN[90]`). The unary operations are: ABS (absolute value), ACOS (arc cosine), ASIN (arc sine), ATAN (arc tangent), COS (cosine), EXP (e raised to the given power), FIX (round down), FUP (round up), LN (natural logarithm), ROUND (round to the nearest whole number), SIN (sine), SQRT (square root), and TAN (tangent). Arguments to unary operations which take angle measures (COS, SIN, and TAN) are in degrees. Values returned by unary operations which return angle measures (ACOS, ASIN, and ATAN) are also in degrees.

The FIX operation rounds towards the left (less positive or more negative) on a number line, so that `FIX[2.8]=2` and `FIX[-2.8]=-3`, for example. The FUP operation rounds towards the right (more positive or less negative) on a number line; `FUP[2.8]=3` and `FUP[-2.8]=-2`, for example.

PARAMETER SETTING

A parameter setting is the following four items one after the other:

- ♦ a pound character #
- ♦ a real value which evaluates to an integer between 1 and 10320,
- ♦ an equal sign = , and
- ♦ a real value. For example "#3 = 15" is a parameter setting meaning "set parameter 3 to 15."

A parameter setting does not take effect until after all parameter values on the same line have been found. For example, if parameter 3 has been previously set to 15 and the line `#3=6 G1 x#3` is interpreted, a straight move to a point where x equals 15 will occur and the value of parameter 3 will be 6.

COMMENTS AND MESSAGES

A line that starts with the percent character, `%`, is treated as a comment and not interpreted in any way.

Printable characters and white space inside parentheses is a comment. A left parenthesis always starts a comment. The comment ends at the first right parenthesis found thereafter. Once a left parenthesis is placed on a line, a matching right parenthesis must appear before the end of the line. Comments may not be nested; it is an error if a left parenthesis is found after the start of a comment and before the end of the comment. Here is an example of a line containing a comment: `G80 M5 (stop motion)`

An alternative form of comment is to use the two characters `//`. The remainder of the line is treated as a comment. Comments do not cause the machining system to do anything.

A comment that is included in parentheses, contains a message if `MSG,` appears after the left parenthesis and before any other printing characters. Variants of `MSG,` which include white space and lower case characters are allowed. Note that the comma which is required. The rest of the characters before the right parenthesis are considered to be a message to the operator. Messages are displayed on screen in the "Error" intelligent label.

ITEM REPEATS

A line may have any number of G words, but two G words from the same modal group may not appear on the same line.

A line may have zero to four M words. Two M words from the same modal group may not appear on the same line.

For all other legal letters, a line may have only one word beginning with that letter.

If a parameter setting of the same parameter is repeated on a line, `#3=15 #3=6`, for example, only the last setting will take effect. It is silly, but not illegal, to set the same parameter twice on the same line.

If more than one comment appears on a line, only the last one will be used; each of the other comments will be read and its format will be checked, but it will be ignored thereafter. It is expected that putting more than one comment on a line will be very rare.

ITEM ORDER

The three types of item whose order may vary on a line (as given at the beginning of this section) are word, parameter setting, and comment. Imagine that these three types of item are divided into three groups by type.

The first group (the words) may be reordered in any way without changing the meaning of the line.

If the second group (the parameter settings) is reordered, there will be no change in the meaning of the line unless the same parameter is set more than once. In this case, only the last setting of the parameter will take effect. For example, after the line `#3=15 #3=6` has been interpreted, the value of parameter 3 will be 6. If the order is reversed to `#3=6 #3=15` and the line is interpreted, the value of parameter 3 will be 15.

If the third group (the comments) contains more than one comment and is reordered, only the last comment will be used.

If each group is kept in order or reordered without changing the meaning of the line, then the three groups may be interleaved in any way without changing the meaning of the line. For example, the line `g40 g1 #3=15 (so there!) #4=-7.0` has five items and means exactly the same thing in any of the 120 possible orders such as `#4=-7.0 g1 #3=15 g40 (so there!)` for the five items.

COMMANDS AND MACHINE MODES

Mach3 has many commands which cause a machining system to change from one mode to another, and the mode stays active until some other command changes it implicitly or explicitly. Such commands are called "modal". For example, if coolant is turned on, it stays on until it is explicitly turned off. The G codes for motion are also modal. If a G1 (straight move) command is given on one line, for example, it will be executed again on the next line if one or more axis words is available on the line, unless an explicit command is given on that next line using the axis words or cancelling motion. "Non-modal" codes have effect only on the lines on which they occur. For example, G4 (dwell) is non-modal.

MODAL GROUPS

Modal commands are arranged in sets called "modal groups", and only one member of a modal group may be in force at any given time. In general, a modal group contains commands for which it is logically impossible for two members to be in effect at the same time (like measure in inches vs. measure in millimeters). A machining system may be in many modes at the same time, with one mode from each modal group being in effect. The modal groups are shown in Table 5-6.

The modal Groups for G codes are :

- group 1 = {G00, G01, G02, G03, G38.2, G80, G81, G82, G84, G85, G86, G87, G88, G89} motion
- group 2 = {G17, G18, G19} plane selection
- group 3 = {G90, G91} distance mode
- group 5 = {G93, G94} feed rate mode
- group 6 = {G20, G21} units
- group 7 = {G40, G41, G42} cutter radius compensation
- group 8 = {G43, G49} tool length offset
- group 10 = {G98, G99} return mode in canned cycles
- group 12 = {G54, G55, G56, G57, G58, G59, G59.xxx} coordinate system selection
- group 13 = {G61, G61.1, G64} path control mode

The modal groups for M codes are:

- group 4 = {M0, M1, M2, M30} stopping
- group 6 = {M6} tool change
- group 7 = {M3, M4, M5} spindle turning
- group 8 = {M7, M8, M9} coolant (special case: M7 and M8 may be active at the same time)
- group 9 = {M48, M49} enable/disable feed and speed override controls

In addition to the above modal groups, there is a group for non-modal G codes:

- group 0 = {G4, G10, G28, G30, G53, G92, G92.1, G92.2, G92.3}

Table 5-6 Modal Groups

For several modal groups, when a machining system is ready to accept commands, one member of the group must be in effect. There are default settings for these modal groups. When the machining system is turned on or otherwise re-initialized, the default values are automatically in effect.

Group 1, the first group on the table, is a group of G codes for motion. One of these is always in effect. That one is called the current motion mode. It is an error to put a G-code from group 1 and a G-code from group 0 on the same line if both of them use axis words. If an axis word-using G-code from group 1 is implicitly in effect on a line (by having been activated on an earlier line), and a group 0 G-code that uses axis words appears on the line, the activity of the group 1 G-code is suspended for that line. The axis word-using G-codes from group 0 are G10, G28, G30, and G92. Mach3 displays the current mode at the top of each screen.

G CODES

G codes of the Mach3 input language are shown in Table 5-7and are the described in detail.

The descriptions contain command prototypes, set in `courier` type.

In the command prototypes, the tilde (~) stand for a real value. As described earlier, a real value may be (1) an explicit number, 4.4, for example, (2) an expression, [2+2.4], for example, (3) a parameter value, #88, for example, or (4) a unary function value, acos[0], for example.

In most cases, if axis words (any or all of `X~, Y~, Z~, A~, B~, C~, U~, V~, W~`) are given, they specify a destination point. Axis numbers relate to the currently active coordinate system, unless explicitly described as being in the absolute coordinate system. Where axis words are optional, any omitted axes will have their current value. Any items in the command prototypes not explicitly described as optional are required. It is an error if a required item is omitted.

U, V and W are synonyms for A, B and C. Use of A with U, B with V etc. is erroneous (like using A twice on a line). In the detailed descriptions of codes U, V and W are not explicitly mentioned each time but are implied by A, B or C.

In the prototypes, the values following letters are often given as explicit numbers. Unless stated otherwise, the explicit numbers can be real values. For example, `G10 L2` could equally well be written `G[2*5] L[1+1]`. If the value of parameter 100 were 2, `G10 L#100` would also mean the same. Using real values which are not explicit numbers as just shown in the examples is rarely useful.

If `L~` is written in a prototype the "~" will often be referred to as the "L number". Similarly the "~" in `H~` may be called the "H number", and so on for any other letter.

If a scale factor is applied to any axis then it will be applied to the value of the corresponding X, Y, Z, A/U, B/V, C/W word and to the relevant I, J, K or R words when they are used.

Summary of G-codes	
G0	Rapid positioning
G1	Linear interpolation
G2	Clockwise circular/helical interpolation
G3	Counterclockwise circular/Helical interpolation
G4	Dwell
G10	Coordinate system origin setting
G12	Clockwise circular pocket
G13	Counterclockwise circular pocket
G15/G16	Polar Coordinate moves in G0 and G1
G17	XY Plane select
G18	XZ plane select
G19	YZ plane select
G20/G21	Inch/Millimeter unit
G28	Return home
G28.1	Reference axes
G30	Return home
G31	Straight probe
G40	Cancel cutter radius compensation
G41/G42	Start cutter radius compensation left/right
G43	Apply tool length offset (plus)
G49	Cancel tool length offset
G50	Reset all scale factors to 1.0
G51	Set axis data input scale factors
G52	Temporary coordinate system offsets
G53	Move in absolute machine coordinate system
G54	Use fixture offset 1
G55	Use fixture offset 2
G56	Use fixture offset 3
G57	Use fixture offset 4
G58	Use fixture offset 5
G59	Use fixture offset 6 / use general fixture number
G61/G64	Exact stop/Constant Velocity mode
G68/G69	Rotate program coordinate system
G70/G71	Inch/Millimeter unit
G73	Canned cycle - peck drilling
G80	Cancel motion mode (including canned cycles)
G81	Canned cycle - drilling
G82	Canned cycle - drilling with dwell
G83	Canned cycle - peck drilling
G84	Canned cycle - right hand rigid tapping
G85/G86/G 88/G89	Canned cycle - boring
G90	Absolute distance mode
G91	Incremental distance mode
G92	Offset coordinates and set parameters
G92.x	Cancel G92 etc.
G93	Inverse time feed mode
G94	Feed per minute mode
G95	Feed per rev mode
G98	Initial level return after canned cycles
G99	R-point level return after canned cycles

Table 5-7 G Codes

RAPID LINEAR MOTION - G0

For rapid linear motion, program `G0 X~ Y~ Z~ A~ B~ C~`, where all the axis words are optional, except that at least one must be used. The G0 is optional if the current motion mode is G0. This will produce coordinated linear motion to the destination point at the current traverse rate (or slower if the machine will not go that fast). It is expected that cutting will not take place when a G0 command is executing.

If G16 has been executed to set a Polar Origin then for rapid linear motion to a point described by a radius and angle `G0 X~ Y~` can be used. `X~` is the radius of the line from the G16 polar origin and `Y~` is the angle in degrees measured with increasing values counterclockwise from the 3 o'clock direction (i.e. the conventional four quadrant conventions).

Coordinates of the current point at the time of executing the *G16* are the polar origin.

It is an error if:

- ♦ All axis words are omitted.

If cutter radius compensation is active, the motion will differ from the above; see Cutter Compensation. If G53 is programmed on the same line, the motion will also differ; see Absolute Coordinates.

LINEAR MOTION AT FEED RATE - G1

For linear motion at feed rate (for cutting or not), program `G1 X~ Y~ Z~ A~ B~ C~`, where all the axis words are optional, except that at least one must be used. The G1 is optional if the current motion mode is G1. This will produce coordinated linear motion to the destination point at the current feed rate (or slower if the machine will not go that fast).

If G16 has been executed to set a polar origin then linear motion at feed rate to a point described by a radius and angle `G0 X~ Y~` can be used. `X~` is the radius of the line from the G16 polar origin and `Y~` is the angle in degrees measured with increasing values counterclockwise from the 3 o'clock direction (i.e. the conventional four quadrant conventions).

Coordinates of the current point at the time of executing the *G16* are the polar origin.

It is an error if:

- ♦ all axis words are omitted.

If cutter radius compensation is active, the motion will differ from the above; see Cutter Compensation. If G53 is programmed on the same line, the motion will also differ; see Absolute Coordinates.

ARC AT FEED RATE - G2 AND G3

A circular or helical arc is specified using either *G2* (clockwise arc) or *G3* (counterclockwise arc). The axis of the circle or helix must be parallel to the X, Y, or Z-axis of the machine coordinate system. The axis (or, equivalently, the plane perpendicular to the axis) is selected with *G17* (Z-axis, XY-plane), *G18* (Y-axis, XZ-plane), or *G19* (X-axis, YZ-plane). If the arc is circular, it lies in a plane parallel to the selected plane.

If a line of code makes an arc and includes rotational axis motion, the rotational axes turn at a constant rate so that the rotational motion starts and finishes when the XYZ motion starts and finishes. Lines of this sort are hardly ever programmed.

If cutter radius compensation is active, the motion will differ from the above; see Cutter Compensation.

Two formats are allowed for specifying an arc. We will call these the center format and the radius format. In both formats the *G2* or *G3* is optional if it is the current motion mode.

RADIUS FORMAT ARC

In the radius format, the coordinates of the end point of the arc in the selected plane are specified along with the radius of the arc. Program `G2 X~ Y~ Z~ A~ B~ C~ R~` (or use *G3* instead of *G2*). R is the radius. The axis words are all optional except that at least one of the two words for the axes in the selected plane must be used. The R number is the radius. A positive radius indicates that the arc turns through 180 degrees or less, while a negative radius indicates a turn of 180 degrees to 359.999 degrees. If the arc is helical, the value of the end point of the arc on the coordinate axis parallel to the axis of the helix is also specified. It is an error if:

- ♦ both of the axis words for the axes of the selected plane are omitted,
- ♦ the end point of the arc is the same as the current point.

It is not good practice to program radius format arcs that are nearly full circles or are semicircles (or nearly semicircles) because a small change in the location of the end point will produce a much larger change in the location of the center of the circle (and, hence, the middle of the arc). The magnification effect is large enough that rounding error in a number can produce out-of-tolerance cuts. Nearly full circles are outrageously bad, semicircles (and nearly so) are only very bad. Other size arcs (in the range tiny to 165 degrees or 195 to 345 degrees) are OK.

Here is an example of a radius format command to mill an arc:

```
G17 G2 x 10 y 15 r 20 z 5.
```

That means to make a clockwise (as viewed from the positive Z-axis) circular or helical arc whose axis is parallel to the Z-axis, ending where X=10, Y=15, and Z=5, with a radius of 20. If the starting value of Z is 5, this is an arc of a circle parallel to the XY-plane; otherwise it is a helical arc.

In the center format, the coordinates of the end point of the arc in the selected plane are specified along with the offsets of the center of the arc from the current location. In this format, it is OK if the end point of the arc is the same as the current point. It is an error if:

♦ When the arc is projected on the selected plane, the distance from the current point to the center differs from the distance from the end point to the center by more than 0.0002 inch (if inches are being used) or 0.002 millimeter (if millimeter's are being used).

The center is specified using the I and J words. There are two ways of interpreting them. The usual way is that I and J are the center relative to the current point at the start of the arc. This is sometimes called *Incremental IJ mode.* The second way is that I and J specify the center as actual coordinates in the current system. This is rather misleadingly called *Absolute IJ mode.* The IJ mode is set using the Configure>State menu when Mach3 is set up. The choice of modes are to provide compatibility with commercial controllers. You will probably find Incremental to be best. In Absolute it will, of course usually be necessary to use both I and J words unless by chance the arc's center is at the origin.

When the XY-plane is selected, program `G2 X~ Y~ Z~ A~ B~ C~ I~ J~` (or use *G3* instead of *G2*). The axis words are all optional except that at least one of X and Y must be used. I and J are the offsets from the current location or coordinates depending on IJ mode (X and Y directions, respectively) of the center of the circle. I and J are optional except that at least one of the two must be used. It is an error if:

♦ X and Y are both omitted,

♦ I and J are both omitted.

When the XZ-plane is selected, program `G2 X~ Y~ Z~ A~ B~ C~ I~ K~` (or use *G3* instead of *G2*). The axis words are all optional except that at least one of X and Z must be used. I and K are the offsets from the current location or coordinates depending on IJ mode (X and Z directions, respectively) of the center of the circle. I and K are optional except that at least one of the two must be used. It is an error if:

♦ X and Z are both omitted,

♦ I and K are both omitted.

When the YZ-plane is selected, program `G2 X~ Y~ Z~ A~ B~ C~ J~ K~` (or use *G3* instead of *G2*). The axis words are all optional except that at least one of Y and Z must be used. J and K are the offsets from the current location or coordinates depending on IJ mode (Y and Z directions, respectively) of the center of the circle. J and K are optional except that at least one of the two must be used. It is an error if:

- ♦ Y and Z are both omitted,
- ♦ J and K are both omitted.

Here is an example of a center format command to mill an arc in Incremental IJ mode:

```
G17 G2 x10 y16 i3 j4 z9
```

That means to make a clockwise (as viewed from the positive z-axis) circular or helical arc whose axis is parallel to the Z-axis, ending where X=10, Y=16, and Z=9, with its center offset in the X direction by 3 units from the current X location and offset in the Y direction by 4 units from the current Y location. If the current location has X=7, Y=7 at the outset, the center will be at X=10, Y=11. If the starting value of Z is 9, this is a circular arc; otherwise it is a helical arc. The radius of this arc would be 5.

The above arc in Absolute IJ mode would be:

```
G17 G2 x10 y16 i10 j11 z9
```

In the center format, the radius of the arc is not specified, but it may be found easily as the distance from the center of the circle to either the current point or the end point of the arc.

DWELL - G4

For a dwell, program `G4 P~`. This will keep the axes unmoving for the period of time in seconds or milliseconds specified by the P number. The time unit to be used is set up on the Config>Logic dialog. For example, with units set to Seconds, *`G4 P0.5`* will dwell for half a second. It is an error if:

- ♦ the P number is negative.

SET COORDINATE SYSTEM DATA TOOL AND WORK OFFSET TABLES - G10

See details of tool and work offsets for further information on coordinate systems

To set the offset values of a tool, program, `G10 L1 P~ X~ Z~` A~, where the P number must evaluate to an integer in the range 0 to 255, the tool number and Offsets of the tool specified by the P number are reset to the given. The A number will reset the tool tip radius. Only those values for which an axis word is included on the line will be reset. The Tool diameter cannot be set in this way.

To set the coordinate values for the origin of a fixture coordinate system, program, `G10 L2 P~ X~ Y~ Z~ A~ B~ C~`, where the P number must evaluate to an integer in the range 1 to 255, the fixture number (Values 1 to 6 corresponding to G54 to G59) and all axis words are optional. The coordinates of the origin of the coordinate system specified by the P number are reset to the coordinate values given (in terms of the absolute coordinate system). Only those coordinates for which an axis word is included on the line will be reset.

It is an error if:

- The *P* number does not evaluate to an integer in the range 0 to 255.

If origin offsets (made by G92 or G92.3) were in effect before G10 is used, they will continue to be in effect afterwards.

The coordinate system whose origin is set by a G10 command may be active or inactive at the time the G10 is executed.

The values set will not be persistent unless the tool or fixture tables are saved using the buttons on Tables screen.

Example: *`G10 L2 P1 x3.5 y17.2`* sets the origin of the first coordinate system (the one selected by G54) to a point where X is 3.5 and Y is 17.2 (in absolute coordinates). The *Z* coordinate of the origin (and the coordinates for any rotational axes) are whatever those coordinates of the origin were before the line was executed.

CLOCKWISE/COUNTERCLOCKWISE CIRCULAR POCKET - G12 AND G13

These circular pocket commands are a sort of canned cycle which can be used to produce a circular hole larger than the tool in use or with a suitable tool (like a woodruff key cutter) to cut internal grooves for "O" rings etc.

Program *`G12 I~`* for a clockwise move and *`G13 I~`* for a counterclockwise move.

The tool is moved in the X direction by the value if the *I* word and a circle cut in the direction specified with the original X and Y coordinates as the center. The tool is returned to the center.

Its effect is undefined if the current plane is not XY.

EXIT AND ENTER POLAR MODE - G15 AND G16

It is possible for G0 and G1 moves in the X/Y plane only to specify coordinates as a radius and angle relative to a temporary center point. Program `G16` to enter this mode. The current coordinates of the controlled point are the temporary center.

Program `G15` to revert to normal Cartesian coordinates.

```
G0 X10 Y10      // normal G0 move to 10,10
G16 //start of polar mode.
G10X10Y45 (this will move to X 17.xxx, Y 17.xxx which is
    a spot on a circle) (of radius 10 at 45 degrees from
    the initial coordinates of 10, 10.)
```

This can be very useful, for example, for drilling a circle of holes. The code below moves to a circle of holes every 10 degrees on a circle of radius 50 mm center X = 10, Y = 5.5 and peck drills to Z = -0.6

```
G21             // metric
G0 X10Y5.5
G16
G1 X50 Y0       //polar move to a radius of 50 angle 0deg
G83 Z-0.6       // peck drill
G1 Y10          // ten degrees from original center
G83 Z-0.6
G1 Y20          // 20 degrees....etc...
G1 Y30
G1 Y40
> ...etc....
G15             //back to normal Cartesian
```

Notes:

1) You must not make X or Y moves other than by using *G0* or *G1* when *G16* is active.

2) This *G16* is different to a Fanuc implementation in that it uses the current point as the polar center. The Fanuc version requires a lot of origin shifting to get the desired result for any circle not centered on 0,0

PLANE SELECTION - G17, G18, AND G19

Program *G17* to select the XY-plane, *G18* to select the XZ-plane, or *G19* to select the YZ plane. The effects of having a plane selected are discussed in under *G2/3* and Canned cycles

LENGTH UNITS - G20 AND G21

Program *G20* to use inches for length units. Program *G21* to use millimeters.

It is usually a good idea to program either *G20* or *G21* near the beginning of a program before any motion occurs, and not to use either one anywhere else in the program. It is the responsibility of the user to be sure all numbers are appropriate for use with the current length units. See also *G70/G71* which are synonymous.

RETURN TO HOME - G28 AND G30

A home position is defined (by parameters 5161-5166). The parameter values are in terms of the absolute coordinate system, but are in unspecified length units.

To return to home position by way of the programmed position, program `G28 X~ Y~ Z~ A~ B~ C~` (or use *G30*). All axis words are optional. The path is made by a traverse move from the current position to the programmed position, followed by a traverse move to the home position. If no axis words are programmed, the intermediate point is the current point, so only one move is made.

REFERENCE AXES G28.1

Program `G28.1 X~ Y~ Z~ A~ B~ C~` to reference the given axes. The axes will move at the current feed rate towards the home switch(es), as defined by the Configuration. When the absolute machine coordinate reaches the value given by an axis word then the feed rate is set to that defined by Configure>Config Referencing. Provided the current absolute position is approximately correct, then this will give a soft stop onto the reference switch(es).

STRAIGHT PROBE – G31

The Straight Probe Command Program `G31 X~ Y~ Z~ A~ B~ C~` to perform a straight probe operation. The rotational axis words are allowed, but it is better to omit them. If rotational axis words are used, the numbers must be the same as the current position numbers so that the rotational axes do not move. The linear axis words are optional, except that at least one of them must be used. The tool in the spindle must be a probe.

It is an error if:

- ♦ the current point is less than 0.254 millimeter or 0.01 inch from the programmed point.
- ♦ *G31* is used in inverse time feed rate mode,
- ♦ any rotational axis is commanded to move,
- ♦ no X, Y, or Z-axis word is used.

In response to this command, the machine moves the controlled point (which should be at the end of the probe tip) in a straight line at the current feed rate toward the programmed point. If the probe trips, the probe is retracted slightly from the trip point at the end of command execution. If the probe does not trip even after overshooting the programmed point slightly, an error is signaled.

After successful probing, parameters 2000 to 2005 will be set to the coordinates of the location of the controlled point at the time the probe tripped and a triplet giving X, Y and Z at the trip will be written to the triplet file if it has been opened by the `M40` macro/`OpenDigFile()` function (q.v.)

Using the Straight Probe Command Using the straight probe command, if the probe shank is kept nominally parallel to the Zaxis (i.e., any rotational axes are at zero) and the tool length offset for the probe is used, so that the controlled point is at the end of the tip of the probe:

- ♦ without additional knowledge about the probe, the parallelism of a face of a part to the XY-plane may, for example, be found.
- ♦ if the probe tip radius is known approximately, the parallelism of a face of a part to the YZ or XZ-plane may, for example, be found.
- ♦ if the shank of the probe is known to be well-aligned with the Z-axis and the probe tip radius is known approximately, the center of a circular hole, may, for example, be found.
- ♦ if the shank of the probe is known to be well-aligned with the Z-axis and the probe tip radius is known precisely, more uses may be made of the straight probe command, such as finding the diameter of a circular hole.

If the straightness of the probe shank cannot be adjusted to high accuracy, it is desirable to know the effective radii of the probe tip in at least the +X, -X, +Y, and -Y directions. These quantities can be stored in parameters either by being included in the parameter file or by being set in a Mach3 program.

sing the probe with rotational axes not set to zero is also feasible. Doing so is more complex than when rotational axes are at zero, and we do not deal with it here.

As a usable example, the code for finding the center and diameter of a circular hole is shown in below. For this code to yield accurate results, the probe shank must be well aligned with the Z-axis, the cross section of the probe tip at its widest point must be very circular, and the probe tip radius (i.e., the radius of the circular cross section) must be known precisely. If the probe tip radius is known only approximately (but the other conditions hold), the location of the hole center will still be accurate, but the hole diameter will not.

```
N010 (probe to find center and diameter of circular hole)
N020 (This program will not run as given here. You have to)
N030 (insert numbers in place of <description of number>.)
N040 (Delete lines N020, N030, and N040 when you do that.)
N050 G0 Z<Z-value of retracted position> F <feed rate>
N060 #1001=<nominal X-value of hole center>
N070 #1002=<nominal Y-value of hole center>
N080 #1003=<some Z-value inside the hole>
N090 #1004=<probe tip radius>
N100 #1005=[<nominal hole diameter>/2.0 - #1004]
N110 G0 X#1001 Y#1002 (move above nominal hole center)
N120 G0 Z#1003 (move into hole - to be cautious, substitute G1 for G0 here)
N130 G31 X[#1001 + #1005] (probe +X side of hole)
N140 #1011=#2000 (save results)
N150 G0 X#1001 Y#1002 (back to center of hole)
N160 G31 X[#1001 - #1005] (probe -X side of hole)
N170 #1021=[[#1011 + #2000] / 2.0] (find pretty good X-value of hole center)
N180 G0 X#1021 Y#1002 (back to center of hole)
N190 G31 Y[#1002 + #1005] (probe +Y side of hole)
N200 #1012=#2001 (save results)
N210 G0 X#1021 Y#1002 (back to center of hole)
N220 G31 Y[#1002 - #1005] (probe -Y side of hole)
N230 #1022=[[#1012 + #2001] / 2.0] (find very good Y-value of hole center)
N240 #1014=[#1012 - #2001 + [2 * #1004]] (find hole diameter in Y-direction)
N250 G0 X#1021 Y#1022 (back to center of hole)
N260 G31 X[#1021 + #1005] (probe +X side of hole)
N270 #1031=#2000 (save results)
N280 G0 X#1021 Y#1022 (back to center of hole)
N290 G31 X[#1021 - #1005] (probe -X side of hole)
N300 #1041=[[#1031 + #2000] / 2.0] (find very good X-value of hole center)
N310 #1024=[#1031 - #2000 + [2 * #1004]] (find hole diameter in X-direction)
N320 #1034=[[#1014 + #1024] / 2.0] (find average hole diameter)
N330 #1035=[#1024 - #1014] (find difference in hole diameters)
N340 G0 X#1041 Y#1022 (back to center of hole)
N350 M2 (that's all, folks)
```

In code above, an entry of the form <description of number> is meant to be replaced by an actual number that matches the description of number. After this section of code has executed, the X-value of the center will be in parameter 1041, the Y-value of the center in parameter 1022, and the diameter in parameter 1034. In addition, the diameter parallel to the X-axis will be in parameter 1024, the diameter parallel to the Y-axis in parameter 1014, and the difference (an indicator of circularity) in parameter 1035. The probe tip will be in the hole at the XY center of the hole.

The example does not include a tool change to put a probe in the spindle. Add the tool change code at the beginning, if needed.

CUTTER RADIUS COMPENSATION - G40, G41, AND G42

To turn cutter radius compensation off, program `G40`. It is OK to turn compensation off when it is already off.

Cutter radius compensation may be performed only if the XY-plane is active.

To turn cutter radius compensation on left (i.e., the cutter stays to the left of the programmed path when the tool radius is positive), program `G41 D~` To turn cutter radius compensation on right (i.e., the cutter stays to the right of the programmed path when the tool radius is positive), program `G42 D~` The D word is optional; if there is no D word, the radius of the tool currently in the spindle will be used. If used, the D number should normally be the slot number of the tool in the spindle, although this is not required. It is OK for the D number to be zero; a radius value of zero will be used.

G41 and *G42* can be qualified by a P-word. This will override the value of the diameter of the tool (if any) given in the current tool table entry.

It is an error if:

- ♦ the D number is not an integer, is negative or is larger than the number of carousel slots,
- ♦ the XY-plane is not active,
- ♦ cutter radius compensation is commanded to turn on when it is already on.

The behavior of the machining system when cutter radius compensation is ON is described in the chapter of Cutter Compensation. Notice the importance of programming valid entry and exit moves.

TOOL LENGTH OFFSETS - G43, G44 AND G49

To use a tool length offset, program `G43 H~`, where the H number is the desired index in the tool table. It is expected that all entries in this table will be positive. The H number should be, but does not have to be, the same as the slot number of the tool currently in the spindle. It is OK for the H number to be zero; an offset value of zero will be used. Omitting H has the same effect as a zero value.

G44 is provided for compatibility and is used if entries in the table give negative offsets.

It is an error if:

- ♦ The H number is not an integer, is negative, or is larger than the number of carousel slots.

To use no tool length offset, program `G49`

It is OK to program using the same offset already in use. It is also OK to program using no tool length offset if none is currently being used.

SCALE FACTORS G50 AND G51

To define a scale factor which will be applied to an X, Y, Z, A, B, C, I & J word before it is used program `G51 X~ Y~ Z~ A~ B~ C~` where the X, Y, Z etc. words are the scale factors for the given axes. These values are, of course, never themselves scaled.

It is not permitted to use unequal scale factors to produce elliptical arcs with *G2* or *G3*.

To reset the scale factors of all axes to 1.0 program `G50`

TEMPORARY COORDINATE SYSTEM OFFSET – G52

To offset the current point by a given positive or negative distance (without motion), program `G52 X~ Y~ Z~ A~ B~ C~`, where the axis words contain the offsets you want to provide. All axis words are optional, except that at least one must be used. If an axis word is not used for a given axis, the coordinate on that axis of the current point is not changed. It is an error if:

- ♦ all axis words are omitted.

G52 and *G92* use common internal mechanisms in Mach3 and may not be used together.

When *G52* is executed, the origin of the currently active coordinate system moves by the values given.

The effect of *G52* is cancelled by programming `G52 X0 Y0` etc.

Here is an example. Suppose the current point is at X=4 in the currently specified coordinate system, then `G52 X7` sets the X-axis offset to 7, and so causes the X-coordinate of the current point to be -3.

The axis offsets are always used when motion is specified in absolute distance mode using any of the fixture coordinate systems. Thus all fixture coordinate systems are affected by *G52*.

MOVE IN ABSOLUTE COORDINATES - G53

For linear motion to a point expressed in absolute coordinates, program `G1 G53 X~ Y~ Z~ A~ B~ C~` (or similarly with *G0* instead of *G1*), where all the axis words are optional, except that at least one must be used. The *G0* or *G1* is optional if it is in the current motion mode. *G53* is not modal and must be programmed on each line on which it is intended to be active. This will produce coordinated linear motion to the programmed point. If *G1* is active, the speed of motion is the current feed rate (or slower if the machine will not go that fast). If *G0* is active, the speed of motion is the current traverse rate (or slower if the machine will not go that fast).

It is an error if:

- ♦ *G53* is used without *G0* or *G1* being active,
- ♦ *G53* is used while cutter radius compensation is on.

See page 5-91 for an overview of coordinate systems.

SELECT WORK OFFSET COORDINATE SYSTEM - G54 TO G59 & G59 P~

To select work offset #1, program `G54`, and similarly for the first six offsets. The system number-G-code pairs are: (*1-G54*), (*2-G55*), (*3-G56*), (*4-G57*), (*5-G58*), (*6-G59*).

To access any of the 254 work offsets (1 - 254) program `G59 P~` where the P word gives the required offset number. Thus `G59 P5` is identical in effect to `G58`.

It is an error if:

- ♦ one of these G-codes is used while cutter radius compensation is on.

See page 5-91 for an overview of coordinate systems.

SET PATH CONTROL MODE - G61, AND G64

Program `G61` to put the machining system into exact stop mode, or `G64` for constant velocity mode. It is OK to program for the mode that is already active. These modes are described in detail above.

ROTATE COORDINATE SYSTEM – G68 AND G69

Program *G68 A~ B~ I~ R~* to rotate the program coordinate system.

A~ is the X coordinate and *B~* the Y coordinate of the center of rotation in the current coordinate system (i.e. including all work and tool offsets and *G52/G92* offsets.).

R~ is the rotation angle in degrees (positive is CCW viewed from the positive Z direction).

I~ is optional and the value is not used. If I~ is present it causes the given R value to be added to any existing rotation set by *G68*.

e.g. *G68 A12 B25 R45* causes the coordinate system to be rotated by 45 degrees about the point Z=12, Y=25.

Subsequently: *G68 A12 B35 I1 R40* leaves the coordinate system rotated by 85 degrees about X = 12, Y=25 Program G69 to cancel rotation.

Notes:

- This code only allows rotation when the current plane is X-Y.
- The I word can be used even if the center point is different from that used before although, in this case, the results need careful planning. It could be useful when simulating engine turning.

LENGTH UNITS – G70 AND G71

10) Program *G70* to use inches for length units. Program *G71* to use millimeters.

11) It is usually a good idea to program either G70 or G71 near the beginning of a program before any motion occurs, and not to use either one anywhere else in the program. It is the responsibility of the user to be sure all numbers are appropriate for use with the current length units. See also *G20/G21* which are synonymous and preferred.

CANNED CYCLE – HIGH SPEED PECK DRILL G73

The *G73* cycle is intended for deep drilling or milling with chip breaking. See also G83. The retracts in this cycle break the chip but do not totally retract the drill from the hole. It is suitable for tools with long flutes which will clear the broken chips from the hole. This cycle takes a Q number which represents a "delta" increment along the Z-axis. Program.

G73 X~ Y~ Z~ A~ B~ C~ R~ L~ Q~

- Preliminary motion, as described in *G81* to *G89* canned cycles.
- Move the Z-axis only at the current feed rate downward by delta or to the Z position, whichever is less deep.
- Rapid back out by the distance defined in the *G73 Pullback* DRO on the Settings screen.
- Rapid back down to the current hole bottom, backed off a bit.
- Repeat steps 1, 2, and 3 until the Z position is reached at step 1.
- Retract the Z-axis at traverse rate to clear Z.

It is an error if:

- the Q number is negative or zero.

CANCEL MODAL MOTION - G80

Program `G80` to ensure no axis motion will occur. It is an error if:

- Axis words are programmed when G80 is active, unless a modal group 0 G code is programmed which uses axis words.

CANNED CYCLES - G81 TO G89

The canned cycles *G81* through *G89* have been implemented as described in this section. Two examples are given with the description of *G81* below.

All canned cycles are performed with respect to the currently selected plane. Any of the three planes (XY, YZ, ZX) may be selected. Throughout this section, most of the descriptions assume the XY-plane has been selected. The behavior is always analogous if the YZ or XZ-plane is selected.

Rotational axis words are allowed in canned cycles, but it is better to omit them. If rotational axis words are used, the numbers must be the same as the current position numbers so that the rotational axes do not move.

All canned cycles use X, Y, R, and Z numbers in the NC code. These numbers are used to determine X, Y, R, and Z positions. The R (usually meaning retract) position is along the axis perpendicular to the currently selected plane (Z-axis for XY plane, X-axis for YZ plane, Y-axis for XZ-plane). Some canned cycles use additional arguments.

For canned cycles, we will call a number "sticky" if, when the same cycle is used on several lines of code in a row, the number must be used the first time, but is optional on the rest of the lines. Sticky numbers keep their value on the rest of the lines if they are not explicitly programmed to be different. The R number is always sticky.

In incremental distance mode: when the XY-plane is selected, X, Y, and R numbers are treated as increments to the current position and Z as an increment from the Z-axis position before the move involving Z takes place; when the YZ or XZ-plane is selected, treatment of the axis words is analogous. In absolute distance mode, the X, Y, R, and Z numbers are absolute positions in the current coordinate system.

The L number is optional and represents the number of repeats. L=0 is not allowed. If the repeat feature is used, it is normally used in incremental distance mode, so that the same sequence of motions is repeated in several equally spaced places along a straight line. In absolute distance mode, L > 1 means "do the same cycle in the same place several times," Omitting the L word is equivalent to specifying L=1. The L number is not sticky.

When L>1 in incremental mode with the XY-plane selected, the X and Y positions are determined by adding the given X and Y numbers either to the current X and Y positions (on the first go-around) or to the X and Y positions at the end of the previous go-around (on the repetitions). The R and Z positions do not change during the repeats.

The height of the retract move at the end of each repeat (called "clear Z" in the descriptions below) is determined by the setting of the retract mode: either to the original Z position (if that is above the R position and the retract mode is G98), or otherwise to the R position.

It is an error if:

- ♦ X, Y, and Z words are all missing during a canned cycle,
- ♦ A P number is required and a negative P number is used,
- ♦ An L number is used that does not evaluate to a positive integer,
- ♦ Rotational axis motion is used during a canned cycle,
- ♦ Inverse time feed rate is active during a canned cycle,
- ♦ Cutter radius compensation is active during a canned cycle.

When the XY plane is active, the Z number is sticky, and it is an error if:

- ♦ The Z number is missing and the same canned cycle was not already active,
- ♦ The R number is less than the Z number.

When the XZ plane is active, the Y number is sticky, and it is an error if:

- ♦ The Y number is missing and the same canned cycle was not already active,
- ♦ The R number is less than the Y number.

When the YZ plane is active, the X number is sticky, and it is an error if:

- ♦ The X number is missing and the same canned cycle was not already active,
- ♦ The R number is less than the X number.

PRELIMINARY AND IN-BETWEEN MOTION

At the very beginning of the execution of any of the canned cycles, with the XY-plane selected, if the current Z position is below the R position, the Z-axis is traversed to the R position. This happens only once, regardless of the value of L.

In addition, at the beginning of the first cycle and each repeat, the following one or two moves are made:

- ♦ Straight traverse parallel to the XY-plane to the given XY-position,
- ♦ Straight traverse of the Z-axis only to the R position, if it is not already at the R position.

If the XZ or YZ plane is active, the preliminary and in-between motions are analogous.

G81 CYCLE

The G81 cycle is intended for drilling. Program `G81 X~ Y~ Z~ A~ B~ C~ R~ L~`

- ♦ Preliminary motion, as described above.
- ♦ Move the Z-axis only at the current feed rate to the Z position.
- ♦ Retract the Z-axis at traverse rate to clear Z.

Example 1. Suppose the current position is (1, 2, 3) and the XY-plane has been selected, and the following NC code is interpreted.*G90 G81 G98 X4 Y5 Z1.5 R2.8*

This calls for absolute distance mode (*G90*), old "Z" retract mode (*G98*) and calls for the *G81* drilling cycle to be performed once. The X number and X position are 4. The Y number and Y position are 5. The Z number and Z position are 1.5. The R number and clear Z are 2.8. The following moves take place.

- ♦ Traverse parallel to the XY-plane to (4,5,3)
- ♦ Traverse parallel to the Z-axis to (4,5,2.8)
- ♦ Feed parallel to the Z-axis to (4,5,1.5)
- ♦ Traverse parallel to the Z-axis to (4,5,3)

Example 2. Suppose the current position is (1, 2, 3) and the XY-plane has been selected, and the following line of NC code is interpreted.

```
G91 G81 G98 X4 Y5 Z-0.6 R1.8 L3
```

This calls for incremental distance mode (G91), old "Z" retract mode and calls for the G81 drilling cycle to be repeated three times. The X number is 4, the Y number is 5, the Z number is -0.6 and the R number is 1.8. The initial X position is 5 (=1+4), the initial Y position is 7 (=2+5), the clear Z position is 4.8 (=1.8+3), and the Z position is 4.2 (=4.8-0.6). Old Z is 3.0

The first move is a traverse along the Z-axis to (1,2,4.8), since old Z < clear Z.

The first repeat consists of 3 moves.

- ♦ Traverse parallel to the XY-plane to (5,7,4.8)
- ♦ Feed parallel to the Z-axis to (5,7, 4.2)
- ♦ Traverse parallel to the Z-axis to (5,7,4.8)

The second repeat consists of 3 moves. The X position is reset to 9 (=5+4) and the Y position to 12 (=7+5).

- ♦ Traverse parallel to the XY-plane to (9,12,4.8)
- ♦ Feed parallel to the Z-axis to (9,12, 4.2)
- ♦ Traverse parallel to the Z-axis to (9,12,4.8)

The third repeat consists of 3 moves. The X position is reset to 13 (=9+4) and the Y position to 17 (=12+5).

- ♦ Traverse parallel to the XY-plane to (13,17,4.8)
- ♦ Feed parallel to the Z-axis to (13,17, 4.2)
- ♦ Traverse parallel to the Z-axis to (13,17,4.8)

G82 CYCLE

The G82 cycle is intended for drilling. Program `G82 X~ Y~ Z~ A~ B~ C~ R~ L~ P~`

- ♦ Preliminary motion, as described above.
- ♦ Move the Z-axis only at the current feed rate to the Z position.
- ♦ Dwell for the P number of seconds.
- ♦ Retract the Z-axis at traverse rate to clear Z.

G83 CYCLE

The *G83* cycle (often called peck drilling) is intended for deep drilling or milling with chip breaking. See also *G73*. The retracts in this cycle clear the hole of chips and cut off any long stringers (which are common when drilling in aluminum). This cycle takes a Q number which represents a "delta" increment along the Z-axis. Program `G83 X~ Y~ Z~ A~ B~ C~ R~ L~ Q~`

- ♦ Preliminary motion, as described above.

Move the Z-axis only at the current feed rate downward by delta or to the Z position, whichever is less deep.

- ♦ Rapid back out to the clear Z.
- ♦ Rapid back down to the current hole bottom, backed off a bit.
- ♦ Repeat steps 1, 2, and 3 until the Z position is reached at step 1.
- ♦ Retract the Z-axis at traverse rate to clear Z.

It is an error if:

- ♦ the Q number is negative or zero.

G84 CYCLE (MAY NOT BE ACTIVE IN MACH3)

The *G84* cycle is intended for right-hand tapping with a tap tool. Program `G84 X~ Y~ Z~ A~ B~ C~ R~ L~`

- ♦ Preliminary motion, as described above.
- ♦ Start speed-feed synchronization.
- ♦ Move the Z-axis only at the current feed rate to the Z position.
- ♦ Stop the spindle.
- ♦ Start the spindle counterclockwise.
- ♦ Retract the Z-axis at the current feed rate to clear Z.
- ♦ If speed-feed synch was not on before the cycle started, stop it.
- ♦ Stop the spindle.
- ♦ Start the spindle clockwise.

The spindle must be turning clockwise before this cycle is used.

It is an error if:

- ♦ the spindle is not turning clockwise before this cycle is executed.

With this cycle, the programmer must be sure to program the speed and feed in the correct proportion to match the pitch of threads being made. The relationship is that the spindle speed equals the feed rate times the pitch (in threads per length unit). For example, if the pitch is 2 threads per millimeter, the active length units are millimeters, and the feed rate has been set with the command F150, then the speed should be set with the command S300, since 150 x 2 = 300.

If the feed and speed override switches are enabled and not set at 100%, the one set at the lower setting will take effect. The speed and feed rates will still be synchronized.

G85 CYCLE

The *G85* cycle is intended for boring or reaming, but could be used for drilling or milling.

Program `G85 X~ Y~ Z~ A~ B~ C~ R~ L~`

- ♦ Preliminary motion, as described above.
- ♦ Move the Z-axis only at the current feed rate to the Z position.
- ♦ Retract the Z-axis at the current feed rate to clear Z.

G86 CYCLE

The *G86* cycle is intended for boring. This cycle uses a P number for the number of seconds to dwell. Program `G86 X~ Y~ Z~ A~ B~ C~ R~ L~ P~`

- ♦ Preliminary motion, as described above.
- ♦ Move the Z-axis only at the current feed rate to the Z position.
- ♦ Dwell for the P number of seconds.
- ♦ Stop the spindle turning.
- ♦ Retract the Z-axis at traverse rate to clear Z.
- ♦ Restart the spindle in the direction it was going.

The spindle must be turning before this cycle is used. It is an error if:

- ♦ the spindle is not turning before this cycle is executed.

G87 CYCLE

The *G87* cycle is intended for back boring. Program `G87 X~ Y~ Z~ A~ B~ C~ R~ L~ I~ J~ K~`

The situation, as shown in Figure 5-80 G87 back boring sequence, is that you have a through hole and you want to counter bore the bottom of hole. To do this you put an L-shaped tool in the spindle with a cutting surface on the UPPER side of its base. You stick it carefully through the hole when it is not spinning and is oriented so it fits through the hole, then you move it so the stem of the L is on the axis of the hole, start the spindle, and feed the tool upward to make the counter bore. Then you stop the tool, get it out of the hole, and restart it.

This cycle uses I and J numbers to indicate the position for inserting and removing the tool. I and J will always be increments from the X position and the Y position, regardless of the distance mode setting. This cycle also uses a K number to specify the position along the Z axis of the controlled point top of the counter bore. The K number is a Z-value in the current coordinate system in absolute distance mode, and an increment (from the Z position) in incremental distance mode.

- ♦ Preliminary motion, as described above.
- ♦ Move at traverse rate parallel to the XY-plane to the point indicated by I and J.
- ♦ Stop the spindle in a specific orientation.
- ♦ Move the Z-axis only at traverse rate downward to the Z position.
- ♦ Move at traverse rate parallel to the XY-plane to the X,Y location.
- ♦ Start the spindle in the direction it was going before.
- ♦ Move the Z axis only at the given feed rate upward to the position indicated by K.
- ♦ Move the Z-axis only at the given feed rate back down to the Z position.
- ♦ Stop the spindle in the same orientation as before.

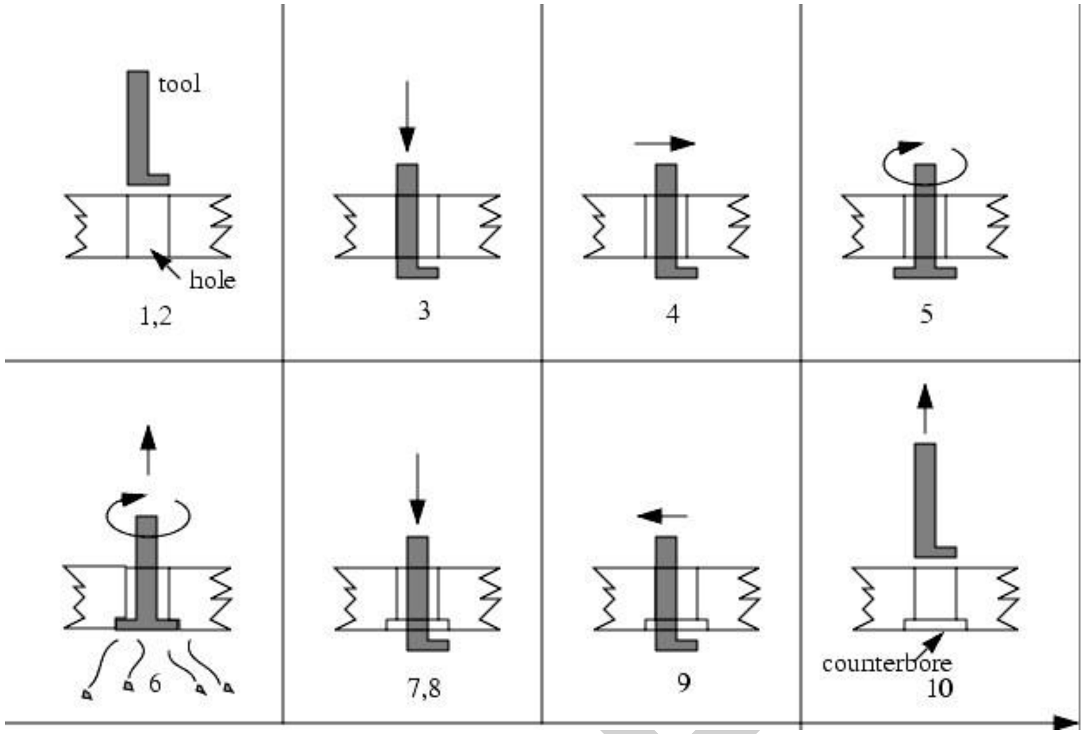


Figure 5-80 G87 back boring sequence

Move at traverse rate parallel to the XY-plane to the point indicated by I and J.

- ♦ Move the Z-axis only at traverse rate to the clear Z.
- ♦ Move at traverse rate parallel to the XY-plane to the specified X,Y location.
- ♦ Restart the spindle in the direction it was going before.

When programming this cycle, the I and J numbers must be chosen so that when the tool is stopped in an oriented position, it will fit through the hole. Because different cutters are made differently, it may take some analysis and/or experimentation to determine appropriate values for I and J.

G88 CYCLE

he *G88* cycle is intended for boring. This cycle uses a P word, where P specifies the number of seconds to dwell. Program `G88 X~ Y~ Z~ A- B~ C~ R~~ L~ P~`

- ♦ Preliminary motion, as described above.
- ♦ Move the Z-axis only at the current feed rate to the Z position.
- ♦ Dwell for the P number of seconds.
- ♦ Stop the spindle turning.
- ♦ Stop the program so the operator can retract the spindle manually.
- ♦ Restart the spindle in the direction it was going.

G89 CYCLE

The *G89* cycle is intended for boring. This cycle uses a P number, where P specifies the number of seconds to dwell. program `G89 X~ Y~ Z~ A~ B~ C~ R~ L~ P~`

- ♦ Preliminary motion, as described above.
- ♦ Move the Z-axis only at the current feed rate to the Z position.
- ♦ Dwell for the P number of seconds.
- ♦ Retract the Z-axis at the current feed rate to clear Z.

SET DISTANCE MODE - G90 AND G91

Interpretation of Mach3 code can be in one of two distance modes: absolute or incremental.

To go into absolute distance mode, program *G90*. In absolute distance mode, axis numbers (X, Y, Z, A, B, C) usually represent positions in terms of the currently active coordinate system. Any exceptions to that rule are described explicitly in this section describing G-Codes.

To go into incremental distance mode, program *G91*. In incremental distance mode, axis numbers (X, Y, Z, A, B, C) usually represent increments from the current values of the numbers.

When the Rigid Tapping switch located on the BOB is in the ON position, the `G91` mode is used to implement the Novakon Pulsar Rigid Tapping cycle where `A` is used to specify the number of spindle rotations nececessary to tap a hole to a certain `Z` depth. See Chapter 6, page 6-1 for more information on rigid tapping.

I and J numbers always represent increments, regardless of the distance mode setting. K numbers represent increments in all but one usage (the *G87* boring cycle), where the meaning changes with distance mode.

SET IJ MODE - G90.1 AND G91.1

Interpretation of the IJK values in *G02* and *G03* codes can be in one of two distance modes: absolute or incremental.

To go into absolute IJ mode, program `G90.1`. In absolute distance mode, IJK numbers represent absolute positions in terms of the currently active coordinate system.

To go into incremental IJ mode, program `G91.1`. In incremental distance mode, IJK numbers usually represent increments from the current controlled point.

Incorrect settings of this mode will generally result in large incorrectly oriented arcs in the tool path display.

G92 OFFSETS - G92, G92.1, G92.2, G92.3

See the Section on Machine Coordinate Systems, page 5-69 for full details. You are strongly advised not to use this legacy feature on any axis where there is another offset applied.

To make the current point have the coordinates you want (without motion), program `G92 X~ Y~ Z~ A~ B~ C~`, where the axis words contain the axis numbers you want. All axis words are optional, except that at least one must be used. If an axis word is not used for a given axis, the coordinate on that axis of the current point is not changed. It is an error if:

- ♦ All axis words are omitted.

G52 and *G92* use common internal mechanisms in Mach3 and may not be used together.

When G92 is executed, the origin of the currently active coordinate system moves. To do this, origin offsets are calculated so that the coordinates of the current point with respect to the moved origin are as specified on the line containing the G92. In addition, parameters 5211 to 5216 are set to the X, Y, Z, A, B, and C-axis offsets. The offset for an axis is the amount the origin must be moved so that the coordinate of the controlled point on the axis has the specified value.

Here is an example. Suppose the current point is at X=4 in the currently specified coordinate system and the current X-axis offset is zero, then `G92 X7` sets the X-axis offset to -3, sets parameter 5211 to -3, and causes the X-coordinate of the current point to be 7.

The axis offsets are always used when motion is specified in absolute distance mode using any of the fixture coordinate systems. Thus all fixture coordinate systems are affected by *G92*.

Being in incremental distance mode has no effect on the action of *G92*.

Non-zero offsets may be already be in effect when the *G92* is called. They are in effect discarded before the new value is applied. Mathematically the new value of each offset is A+B, where A is what the offset would be if the old offset were zero, and B is the old offset. For example, after the previous example, the X-value of the current point is 7. If `G92 X9` is then programmed, the new X-axis offset is -5, which is calculated by [[7-9] + 3]. Put another way the `G92 X9` produces the same offset whatever *G92* offset was already in place.

To reset axis offsets to zero, program `G92.1 or G92.2` *G92.1* sets parameters 5211 to 5216 to zero, whereas *G92.2* leaves their current values alone.

To set the axis offset values to the values given in parameters 5211 to 5216, program `G92.3`

You can set axis offsets in one program and use the same offsets in another program. Program *G92* in the first program. This will set parameters 5211 to 5216. Do not use *G92.1* in the remainder of the first program. The parameter values will be saved when the first program exits and restored when the second one starts up. Use *G92.3* near the beginning of the second program. That will restore the offsets saved in the first program.

SET FEED RATE MODE - G93, G94 AND G95

Three feed rate modes are recognized: inverse time, units per minute and units per revolution of spindle. Program *G93* to start the inverse time mode (this is very infrequently employed). Program *G94* to start the units per minute mode. Program *G95* to start the units per rev mode.

In inverse time feed rate mode, an *F* word means the move should be completed in [one divided by the *F* number] minutes. For example, if the *F* number is 2.0, the move should be completed in half a minute.

In units per minute feed rate mode, an *F* word on the line is interpreted to mean the controlled point should move at a certain number of inches per minute, millimeters per minute, or degrees per minute, depending upon what length units are being used and which axis or axes are moving.

In units per rev feed rate mode, an *F* word on the line is interpreted to mean the controlled point should move at a certain number of inches per spindle revolution, millimeters per spindle revolution, or degrees per spindle revolution, depending upon what length units are being used and which axis or axes are moving.

When the inverse time feed rate mode is active, an *F* word must appear on every line which has a *G1, G2*, or *G3* motion, and an *F* word on a line that does not have *G1, G2*, or *G3* is ignored. Being in inverse time feed rate mode does not affect *G0* (rapid traverse) motions. It is an error if:

- ♦ Inverse time feed rate mode is active and a line with *G1, G2*, or *G3* (explicitly or implicitly) does not have an *F* word.

SET CANNED CYCLE RETURN LEVEL - G98 AND G99

12) When the spindle retracts during canned cycles, there is a choice of how far it retracts:

    1) Retract perpendicular to the selected plane to the position indicated by the *R* word, or

    2) Retract perpendicular to the selected plane to the position that axis was in just before the canned cycle started (unless that position is lower than the position indicated by the *R* word, in which case use the *R* word position).

To use option (1), program *G99* To use option (2), program *G98* Remember that the R word has different meanings in absolute distance mode and incremental distance mode.

BUILT-IN M CODES

M codes interpreted directly by Mach3 are shown in Table 5-8.

M-code	Meaning
M0	Program stop
M1	Optional program stop
M2	Program end
M3/4	Rotate spindle clockwise/counterclockwise
M5	Stop spindle rotation
M6	Tool change (by two macros)
M7	Mist coolant on
M8	Flood coolant on
M9	All coolant off
M30	Program end and Rewind
M47	Repeat program from first line
M48	Enable speed and feed override
M49	Disable speed and feed override
M98	Call subroutine
M99	Return from subroutine/repeat

Table 5-8 Built in M-codes

PROGRAM STOPPING AND ENDING - M0, M1, M2, M30

To stop a running program temporarily (regardless of the setting of the optional stop switch), program *M0*.

To stop a running program temporarily (but only if the optional stop switch is on), program *M1*.

It is OK to program *M0* and *M1* in MDI mode, but the effect will probably not be noticeable, because normal behavior in MDI mode is to stop after each line of input, anyway.

If a program is stopped by an *M0, M1*, pressing the cycle start button will restart the program at the following line.

To end a program, program *M2 or M30*. M2 leaves the next line to be executed as the *M2* line. *M30* "rewinds" the G-code file. These commands can have the following effects depending on the options chosen on the Configure>Logic dialog:

- ♦ Axis offsets are set to zero (like *G92.2*) and origin offsets are set to the default (like *G54*).
- ♦ Selected plane is set to XY (like *G17*).
- ♦ Distance mode is set to absolute (like *G90*).
- ♦ Feed rate mode is set to Units per minute mode (like *G94*).
- ♦ Feed and speed overrides are set to ON (like *M48*).
- ♦ Cutter compensation is turned off (like *G40*).
- ♦ The spindle is stopped (like *M5*).
- ♦ The current motion mode is set to *G1* (like *G1*).
- ♦ Coolant is turned off (like *M9*).

o more lines of code in the file will be executed after the *M2* or *M30* command is executed. Pressing cycle start will resume the program (*M2*) or start the program back at the beginning of the file (*M30*).

SPINDLE CONTROL - M3, M4, M5

To start the spindle turning clockwise at the currently programmed speed, program `M3`.

To start the spindle turning counterclockwise at the currently programmed speed, program `M4`.

For a PWM or Step/Dir spindle the speed is programmed by the *S* word. For an on/off spindle control it will be set by the gearing/pulleys on the machine.

To stop the spindle from turning, program `M5`.

It is OK to use *M3* or *M4* if the spindle speed is set to zero. If this is done (or if the speed override switch is enabled and set to zero), the spindle will not start turning. If, later, the spindle speed is set above zero (or the override switch is turned up), the spindle will start turning. It is permitted to use *M3* or *M4* when the spindle is already turning or to use *M5* when the spindle is already stopped but see the discussion on safety interlocks in configuration for the implications of a sequence which would reverse an already running spindle.

TOOL CHANGE - M6

Provided tool change requests are not to be ignored (as defined in Configure>Logic), Mach3 will call a macro (q.v) *M6Start* when the command is encountered. It will then wait for Cycle Start to be pressed, execute the macro *M6End* and continue running the part program. You can provide Visual Basic code in the macros to operate your own mechanical tool changer and to move the axes to a convenient location to tool changing if you wish.

If tool change requests are set to be ignored (in Configure>Logic) then *M6* has no effect.

COOLANT CONTROL - M7, M8, M9

To turn flood coolant on, program `M8`.

To turn mist coolant on, program `M7` (Not currently active with the Pulsar Mill).

To turn all coolant off, program `M9`.

It is always OK to use any of these commands, regardless of what coolant is on or off.

RE-RUN FROM FIRST LINE - M47

On encountering an *M47* the part program will continue running from its first line.

It is an error if:

- ♦ *M47* is executed in a subroutine

The run can be stopped by the Pause or Stop buttons

See also the use of *M99* outside a subroutine to achieve the same effect.

OVERRIDE CONTROL - M48 AND M49

13) To enable the speed and feed override, program `M48`. To disable both overrides, program `M49`. It is OK to enable or disable the switches when they are already enabled or disabled.

CALL SUBROUTINE - M98

This has two formats:

1) To call a subroutine program within the current part program file code `M98 P~ L~` or `M98 ~P ~Q` The program must contain an *O* line with the number given by the *P* word of the Call . This *O* line is a sort of "label" which indicates the start of the subroutine. The *O* line may not have a line number (*N* word) on it. It, and the following code, will normally be written with other subroutines and follow either an *M2, M30* or *M99* so it is not reached directly by the flow of the program.

2) To call a subroutine which is in a separate file code `M98(`*filename*`)L~` for example `M98 (test.tap)`

For both formats:

The L word (or optionally the *Q* word) gives the number of times that the subroutine is to be called before continuing with the line following the *M98*. If *the L (Q*) word is omitted then its value defaults to 1.

By using parameters values or incremental moves a repeated subroutine can make several roughing cuts around a complex path or cut several identical objects from one piece of material.

Subroutine calls may be nested. That is to say a subroutine may contain a *M98* call to another subroutine. As no conditional branching is permitted it is not meaningful for subroutines to call themselves recursively.

RETURN FROM SUBROUTINE

To return from a subroutine program `M99` Execution will continue after the *M98* which called the subroutine.

If *M99* is written in the main program, i.e. not in a subroutine, then the program will start execution from the first line again. See also *M47* to achieve the same effect.

MACRO M-CODES

MACRO OVERVIEW

If any M-code is used which is not in the above list of built-in codes then Mach3 will attempt to find a file named "`Mxx.M1S`" in the Macros folder. If it finds the file then it will execute the VB script program it finds within it.

The Operator>Macros menu item displays a dialog which allows you to see the currently installed macros, to Load, Edit and Save or Save As the text. The dialog also has a Help button which will display the VB functions which can be called to control Mach3. For example you can interrogate the position of axes, move axes, interrogate input signals and control output signals.

New macros can be written using an external editor program like Notepad and saved in the Macros folder or you can load an existing macro within Mach3, totally rewrite it and save it with a different file name.

OTHER INPUT CODES

SET FEED RATE - F

To set the feed rate, program `F~` Depending on the setting of the Feed Mode toggle the rate may be in units-per-minute or units-per-rev of the spindle.

The units are those defined by the *G20/G21* mode. Depending on the setting in *Configure>Logic* a revolution of the spindle may be defined as a pulse appearing on the Index input or be derived from the speed requested by the *S* word or *Set Spindle speed* DRO.

The feed rate may sometimes be overridden as described in *M48* and *M49* above.

SET SPINDLE SPEED - S

To set the speed in revolutions per minute (rpm) of the spindle, program `S~` The spindle will turn at that speed when it has been programmed to start turning. It is OK to program an *S* word whether the spindle is turning or not. If the speed override switch is enabled and not set at 100%, the speed will be different from what is programmed. It is OK to program `S0`; the spindle will not turn if that is done. It is an error if:

- ♦ the *S* number is negative.

If a *G84* (tapping) canned cycle is active and the feed and speed override switches are enabled, the one set at the lower setting will take effect. The speed and feed rates will still be synchronized. In this case, the speed may differ from what is programmed, even if the speed override switch is set at 100%.

SELECT TOOL – T

To select a tool, program `T~` where the T number is the slot number in the tool changer (of course a rack for manual changing) for the tool.

Even if you have an automatic tool changer, the tool is not changed automatically by the *T* word. To do this use *M06*. The *T* word just allows the changer to get the tool ready.

M06 (depending on the settings in *Config>Logic*) will operate the tool changer or stop execution of the part-program so you can change the tool by hand. The detailed execution of these changes is set in the *M6Start* and *M6End* macros. If you require anything special you will have to customize these

The *T* word, itself, does not actually apply any offsets. Use *G43* or *G44*, q.v., to do this. The *H* word in G43/G44 specifies which tool table entry to use to get the tool offset. Notice that this is different to the action when you type a tool slot number into the T DRO. In this case an implied G43 is performed so the length offset for the tool will be applied assuming that the slot number and the tool table entry number are the same.

It is OK, but not normally useful, if *T* words appear on two or more lines with no tool change. It is OK to program T0; no tool will be selected. This is useful if you want the spindle to be empty after a tool change. It is an error if:

- ♦ A negative *T* number is used, or a T number larger than 255 is used.

ERROR HANDLING

This section describes error handling in Mach3. If a command does not work as expected or does not do anything check that you have typed it correctly. Common mistakes are GO, instead of G0 i.e. letter O instead of zero) and too many decimal points in numbers. Mach3 does not check for axis over travel (unless software limits are in use) or excessively high feeds or speeds. Nor does it does not detect situations where a legal command does something unfortunate, such as machining a fixture.

ORDER OF EXECUTION

The order of execution of items on the same line is critical to safe and effective machine operation. Items are executed in the order shown in Table 5-9 when they occur on the same line.

Order	Item
1	Comment (including message)
2	Set feed rate mode (G93, G94, G95)
3	Set feed rate (F)
4	Set spindle speed (S)
5	Select tool
6	Tool change (M6) and Execute M-code macros
7	Spindle On/Off (M3, M4, M5)
8	Coolant On/Off (M7, M8, M9)
9	Enable/disable overrides (M48, M49)
10	Dwell (G4)
11	Set active plane (G17, G18, G18)
12	Set length units (G20, G21)
13	Cutter radius compensation On/Off (G40, G41, G42)
14	Tool table offset On/Off (G43, G49)
15	Fixture table select (G54 - G58 & G59 P~)
16	Set path control mode (G61, G61.1, G64)
17	Set distance mode (G90, G91)
18	Set canned cycle return level mode (G98, G99)
19	Home, or change coordinate system data (G10), or set offsets (G92, G94)
20	Perform motion (G0 to G3, G12, G13, G80 to G89 as modified by G53
21	Stop or repeat (M0, M1, M2, M30, M47, M99)

Table 5-9 Order of execution on a line

6 RIGID TAPPING

Rigid Tapping with the Novakon Pulsar CNC Mill uses simple G-code. The proprietary BOB accurately controls the spindle revolutions in coordination with the Z feed rate to tap threaded holes.

The basic G-code for rigid tapping includes setting the Incremental Distance Mode (G91), the tapping feed rate (F), the depth of the hole to be tapped (Z), and the number of treads (A). The number of threads is calculated by multiplying the depth of the hole to be tapped by the treads per inch (tpi) of the tap. Novakon has teamed with BobCAD and created a post processor to automatically generate the correct code required to accomplish rigid tapping. Contact Novakon International for more information.

The following is an intuitive example of G-code used to center drill, drill and tap two ½-13 holes.

(NOVAKON PULSAR MILL)

(CENTER DRILL, DRILL AND TAP TWO ½-13 HOLES)

%

G00 G17 G20 G40.1 G49 G50 G54 G80 G90 (Initialization string.)

(CENTER DRILL TWO HOLES)

M6 T6 (Call the Tool Change for the Center Drill.)

G43 H6 (Set the machine offset for the Center Drill.)

S1500 M03 (Turn ON the spindle clockwise at 1500 RPM.)

M08 (Turn ON the coolant.)

G81 G98 X0.0000 Y0.0000 Z-0.2631 R0.1000 F6.5000 (Implement the Canned Drill Cycle and center drill first hole at X = 0.0 and Y = 0.0.)

X1.0000 (Center drill the second hole at X = 1.0 and Y = 0.0.)

G80 (Cancel the Canned Drill Cycle.)

G00 Z0.2000 (Rapid raise the center drill 0.2 inches above hole.)

M09 M05 (Turn OFF the coolant and the spindle.)

(DRILL TWO HOLES)

M6 T227	(Call the tool Change for the drill bit.)
G43 H227	(Set the machine offset for the drill bit.)
S1750 M03	(Turn ON the spindle clockwise at 1750 RPM.)
M08	(Turn ON the coolant.)
G81 G98 X0.0000 Y0.0000 Z-1.200 R0.1000 F6.7000	(Implement the Canned Drill Cycle and drill first hole at X = 0.0 and Y = 0.0.)
X1.0000	(Drill the second hole at X = 1.0 and Y = 0.0.)
G80	(Cancel the Canned Drill Cycle.)
G00 Z0.2000	(Rapid raise the center drill 0.2 inches above hole.)
M09 M05	(Turn OFF the coolant and the spindle.)

(Novakon Rigid Tapping)

M6 T130	(Call the Tool Change for the ½-13 tap.)
G43 H130	(Set the machine offset for the ½-13 tap.)
M08	(Turn on the coolant – **DO NOT** turn ON the spindle.)
G84 G98 X0.0000 Y0.0000 F150.0000	(Implement the Canned Rigid Tap Cycle, move to first hole at X = 0.0, Y = 0.0 and set the tapping Feed Rate to 150 inches per minute.)
Z0.0000	(Lower the tap to the top of the hole.)
G91	(Implement the Novakon Rigid Tapping Cycle.)
G01	(Use the specified Feed Rate to tap the hole.)

Z-1.0000 A13.0000	(Rotate the spindle clockwise 13.0000 revolutions as the tap is lowered one inch at the specified Feed Rate. If you are using BobCAD, it will automatically calculate the correct tpi.)
Z1.0000 A-13.0000	(Rotate the spindle counterclockwise 13.0000 revolutions as tap is raised one inch at the specified Feed Rate.)
G90 Z0.2000	(Implement the Absolute Distance Mode and move the tap 0.2 inches above hole.)
X1.0000	(Move to the second hole at X = 1.0 and Y = 0.0.)
Z0.0000	(Lower the tap to the top of the hole.)
Z-1.0000 A13.0000	(Rotate the spindle clockwise 13.0000 revolutions as the tap is lowered one inch at the specified Feed Rate.)
Z1.0000 A-13.0000	(Rotate the spindle counterclockwise 13.0000 revolutions as tap is raised one inch at the specified Feed Rate.)
G90 Z0.2000	(Implement the Absolute Distance Mode and move to the tap 0.2 inches above hole.)
G80	(Cancel the Novakon Rigid Tapping Cycle.)
M09 M05	(Turn OFF the coolant and the spindle.)
M30	(End of program.)

%

7 GLOSSARY

Bed & Column - The bed and the column are a box type construction made of high quality grade 25 cast iron. They are designed for high rigidity with suitable cross section and ribs for reinforcement. Proper casting treatment is given to relieve the castings of any undue stress before assembly and machining is done. The X-axis, Y-axis and Z-axis slide components including the headstock assembly and table are mounted on the same base and column structure.

Bellows - The Y-Axis is protected from chips and coolant by industrial metal bellows. The Z-Axis ball screw is protected from chips and coolant by industrial accordion bellows.

Chip Containment Drawer - The PULSAR Mill is equipped with a dedicated drawer to collect chips from the milling operation. The drawer is easily removable to allow disposal of chips.

Cog Drive Belt - The drive belt for the motor is accessible by removing the Lever Draw Bar and the servo motor.

Coolant Containment Drawer - The PULSAR Mill is equipped with a large dedicated drawer and two 90 watt pumps for circulating coolant. One pump provides coolant for use during the milling operation. The other dedicated pump provides fluid for wash down and cleaning debris from the mill and tray. The drawer is easily removable to allow changing the coolant.

X-axis Gib Adjustment Screw - The PULSAR Mill have a gib adjustment screw located on the front left side of the table. This screw presses a metal plate (the gib) to the ways of the saddle, increasing or decreasing the tension when moving the table along the X-axis. Before adjusting the gibs, make sure they are well lubricated to achieve the desired result. (Reference chapter for details.)

Lever Draw Bar (LDB) - The LDB allows easy insertion and removal of quick change tooling.

Lubrication Distribution Blocks - The shielded metal lines of the lubrication system intersect at these blocks to deliver lubrication to the various parts of your Pulsar Mill.

Mill Table - The mill table supports your work piece and work holding devices. It travels along the X & Y axes via Ball screws, the PULSAR Mill has 4 5/8” T-Slots.

Mounting Points - There are four mounting points on the Pulsar CNC Mill. The mounting points are M10 holes located at each corner of the base of the mill. Make sure that your machine is properly mounted before operating it.

One Shot Lubrication System - The one shot or single stroke lubrication system is located on the left side of the machine stand. The dovetail ways and ball nuts are lubricated when the lubrication system is activated. To be sure that enough lubrication is provided, a minimum of two pumps is recommended and it is also advisable to visually check the lubrication delivery on the lubricated surfaces. Lubrication is very important for machine tools since it greatly affects machine life. Use only recommended lubrication oils which are clean and free from foreign matter. Follow the maintenance scheduled in this manual.

Optional X-Axis and Y-Axis Servo Motors - The PULSAR Mill can also be equipped with X and Y servo motors. The X/Y-Axis servo motors provides power for table movement along the X/Y-Axis.

Proximity Switch Covers - The proximity switches on the Pulsar Mill are protected from coolant and chips with removable sheet metal covers.

Rigid Tapping - A rigid tapping cycle synchronizes the machine spindle rotation and feed to match a specific thread pitch. Higher tapping speeds are possible when using a rigid tapping cycle.

Saddle - The mill table is supported by the saddle along the Y-Axis.

Spindle Drive Motor - The spindle drive servo motor allows control of the spindle from 1 to 4500 rpm in one rpm increments.

Spindle with R-8 Nose Taper - The Pulsar Mill utilizes a spindle with an R-8 taper nose for supporting and locating work holding tools. Any standard tool holding equivalent to the R-8 specifications can be mounted on the spindle with suitable adapters.

Proximity Switches - A proximity switch is located on each axis of travel. These switches are the mechanical end position limits and they also are used to home the slides.

X-Axis Stepper Motor - The PULSAR Mill is equipped with X axis stepper motor. The X-Axis stepper motor provides power for table movement along the X-Axis and is located under the left side of the mill table.

Z-Axis Servo Motor - The PULSAR Mill can also be equipped with X, Y and Z servo motors. The Z-Axis servo motor provides power for head/spindle movement along the Z-Axis. See the machines specifications for motor sizes.

Modular Electrical Cabinet - The control system is connected to the machine via the modular electrical cabinet.

Y-Axis Stepper Motor - This motor drives the table along the dovetail ways of the Y-Axis. (Located at the rear base of the machine).

8 MAINTENANCE

Maintaining your machine as per the schedule documented in this manual will help prolong the life of the machine and aid in the production of precision work pieces. It should be noted that the maintenance schedules listed herein, are intended to be used as a guide. The environment and working conditions of your shop should be taken into consideration.

LUBRICATION AND DAILY MAINTENANCE SCHEDULE

DAILY MAINTENANCE

AREA OF MAINTENANCE	MAINTENANCE TASK	OCCURRENCE
Lubrication System	Check Oil Level Pressure build up during hand pumping Check for distribution film of oil on all Sliding surfaces Check for leaks	Before Machining
Cutting Tools & Tool Holders	Check for proper operation of the LDB Tighten work holding devices	Before Machining
Coolant Level	Check that coolant level is above pump intake	Before Machining
Machine Work Area	Check for leakage and cleanliness	Before & After Machining
Cleaning	Clean the work holding devices Clean Guards Clean Machine Clean Trays	Before & After Machining
External Wiring & Cables	Check all cable connections Check for crimped or frayed cables	Before Machining
Machine General Inspection	Check entire machine for lose or missing fasteners	Before Machining
Spindle	Clean Spindle Taper	Before Machining

Table 8-1

AREA OF MAINTENANCE	MAINTENANCE TASK	OCCURRENCE
Axis Backlash	Check and compensate if necessary	Every 6 Months
Ball Screw Guards	Check condition	Every 6 Months or as Required
Spindle Drive Belt	Check condition Check tension	Every 6 Months or as Required
Gib Adjustments	Check table motion for fish tailing movement & adjust (Follow Instructions)	Every 6 Months
Machine Base / Table	Check if level Check mounting bolts for loosening	Every 6 Months
Tool Holder / LDB	Check for breakage Check for thread damage	Every Month
Electrical Cabinets	Check for cleanliness Check and secure any loose connections	Every 3 Months
Electrical Elements	Check for proper working of push button switches	Every 3 Months
Proximity Switches	Check for proper operations	Every 3 Months
Motors	Check for condition and testing	Every Month

Table 8-2

LUBRICATION SYSTEM

This section covers the lubrication of your machine. There are two primary areas of lubrication: areas lubricated by the single stroke lube system and the life greased bearings.

CAUTION

Remember, proper lubrication is very important. It greatly affects the performance and longevity of your machine. If the machine is operated without supplying the lubricating oil, it will cause seizure of the sliding surfaces. Before operating, visually inspect the oil on the actual machine surfaces.

CAUTION

Use only recommended lubrication oils which are clean and free from foreign matter. Periodically clean the tank and strainers/filters, if provided, inspect the equipment's functioning or lube supply pipes for damaged to ensure optimum machine operation.

.

LUBRICATING POINTS	LUBRICATING SYSTEM & RECOMMENDED OIL GRADE	QUANTITY
Headstock Bearings	**Grease** Kluber Isoflex Bu-15	Life Grease
Ball screws & Guide ways	**Manual One Shot Lubrication** Mobil Vactra Oil No.2 Way Oil Medium	One Litre
Quill & Ball screw Bearings	**Grease** Kluber Isoflex Bu-15	Life Grease

Table 8-3

BALLSCREWS AND GUIDEWAYS

The ball screws and dovetail slides are lubricated by the single stoke lubrication system. Follow the lubrication schedule in Table 8-3. When lubricating the dovetail slides, pull the lever of the one shot lube system away and down from the machine. The number of pumps required will depend on your shop environment. Enough pumps should be administered until a thin layer of lubrication is present on the slides.

HEADSTOCK & SPINDLE BEARINGS

The bearings are of a precision class of accuracy. The spindle bearings are lubricated for life with high grade grease, such as Kluber Isoflex NBU-15. This grease maintains its lubrication properties at both low and high temperatures (-70º F to 120º F). There is no necessary lubrication for these bearings.

If the spindle starts making abnormal noises or gets very warm (above 120º F), check for damage to the spindle bearings after stopping the machine completely. The spindle should be warmed up for approximately 15-20 minutes when the machine has been stopped for a long period (ie. 5 or more days).

PERIODICAL MAINTENANCE TASKS

AXIS BEARING ADJUSTMENT / AXIS BACKLASH

Backlash is the amount of movement the screw makes before the table engages. There are a number of signs that may indicate that there is excessive backlash with your system:

- Rough/uneven surface finishes
- Dimension inaccuracies
- Table shakes under machining force

BACKLASH COMPENSATION

Even if none of these signs are present, it is recommended to check for backlash compensation every 3-6 months depending on your usage of the machine. This section of the manual is addressing mechanical backlash compensation.

There are four main mechanical reasons for backlash:

1) Pre-loaded ball nut is damaged and is causing axial play between the nut and the screw. If the ball nut is damaged, the only solution to this is to replace the ball screw and ball nut assembly.

2) The end support bearings are damaged. If you find that the end bearings are damaged, follow the procedure below for replacing the bearings.

3) The hexagon socket cap ball nut mounting screws are loose. In this case move the table to the end of the stroke so that the nut mounting screws are accessible. Tighten the bolts and make sure the bolts or housing threads are not damaged before tightening.

4) The end bearing tightening nut is loose. Follow steps 6 and 7 in the following procedure “Y-Axis Ball Screw End Bearing Adjustment” to tighten the bearings whenever required.

Y-AXIS BALL SCREW END BEARING ADJUSTMENT

For this procedure, you will need to refer to **Error! Reference source not found.**, page **Error! Bookmark not defined.**.

1) Remove axis motor.

2) Detach bellows from saddle.

3) Loosen the part #28 and 29, (**Error! Reference source not found.**), page **Error! Bookmark not defined.**, to remove the bearing housing #20. Loosen the end bearing nuts first. Remove the damaged bearing from the housing and ball screw.

4) Check the opposed pair of angular contact bearings for damage. Make sure the bearings are not damaged or the housing of the bearing ID seating is not heavily scoured or damaged. If the bearings are damaged, acquire the new set of bearings for replacement. Also ensure the new sets of bearings are packed with proper amount and type of grease before tightening. (See Section 7 "Maintenance", page 8-1 for lubrication specifications).

5) Install the inner bearing #30, (**Error! Reference source not found.**), page **Error! Bookmark not defined.**, first on the ball screw, making sure the bearing orientation is the same as the original mounting direction.

6) Install bearing housing #20, (**Error! Reference source not found.**), page 11-9, square with the ball screw axis, making sure the bearing housing #20 is securely positioned with part # 28 and #29.

7) Install bearing #21, (**Error! Reference source not found.**), page **Error! Bookmark not defined.**, from the outside of the housing, making sure the bearing orientation is the same as the original mounting direction. The bearing inner chase contact angle must be mounted to oppose each other in an outward direction.

8) Tighten the first nut #25 closer to the housing/bearing, (**Error! Reference source not found.**), page **Error! Bookmark not defined.**, to tighten the bearings against the housing. Loosen the nut and re-tighten again the bearing properly to position. Install the second jam nut #25 to secure the first nut as well.

9) Rotate the screw manually to ensure the motion of the screw is smooth and uniform and the bearings are not creating any stop and go motion or are completely jammed.

Z-AXIS BALL SCREW END BEARING ADJUSTMENT

For this procedure, you will need to refer to **Error! Reference source not found.**, page **Error! Bookmark not defined.**.

1) Remove axis motor.

2) Detach bellows from saddle.

3) Loosen the part #28 and 29, (**Error! Reference source not found.**), page **Error! Bookmark not defined.**, to remove the bearing housing #20. Loosen the end bearing nuts first. Remove the damaged bearing from the housing and ball screw.

4) Check the opposed pair of angular contact bearings for damage. Make sure the bearings are not damaged or the housing of the bearing ID seating is not heavily scoured or damaged. If the bearings are damaged, acquire the new set of bearings for replacement. Also ensure the new sets of bearings are packed with proper amount and type of grease before tightening. (See Section 7 “Maintenance”, page 8-1 for lubrication specifications).

5) Install the inner bearing #30, (**Error! Reference source not found.**), page **Error! Bookmark not defined.**, first on the ball screw, making sure the bearing orientation is the same as the original mounting direction.

6) Install bearing housing #20, (**Error! Reference source not found.**), page 11-9, square with the ball screw axis, making sure the bearing housing #20 is securely positioned with part # 28 and #29.

7) Install bearing #21, (**Error! Reference source not found.**), page **Error! Bookmark not defined.**, from the outside of the housing, making sure the bearing orientation is the same as the original mounting direction. The bearing inner chase contact angle must be mounted to oppose each other in an outward direction.

8) Tighten the first nut #25 closer to the housing/bearing, (**Error! Reference source not found.**), page **Error! Bookmark not defined.**, to tighten the bearings against the housing. Loosen the nut and re-tighten again the bearing properly to position. Install the second jam nut #25 to secure the first nut as well.

9) Rotate the screw manually to ensure the motion of the screw is smooth and uniform and the bearings are not creating any stop and go motion or are completely jammed.

ADJUSTMENT OF GIBS ON THE X, Y & Z AXIS SLIDES

The X, Y and Z axis slides have straight gibs. If the slide motion proves out to be sloppy or uneven, adjust the gibs evenly using the set screw provided on the front and side of the casting.

1) Make sure that the slide is properly lubricated before you start adjusting the screws.

2) Loosen the gib hex head bolts.

3) Tighten the gib set screws all the way so there is no table movement.

4) Loosen all set screws one at a time, by a quarter of a turn and then check the motion for “fish tailing” and uneven motions to and fro.

5) Adjust all the set screws until the “fish tailing” movement is eliminated.

6) Tighten the gib hex head bolts. Do NOT exceed 100 in*lb torque.

The tighter the gib, the more accurate it will be in motion. The power required to move the slide may also go up due to the increased sliding friction. Removing and cleaning the gibs before installation is also helpful.

With the gibs property adjusted and the slides lubricated, the torque required to move the slide slowly, should not exceed 15%-20% of the maximum motor torque available.

9 LEVER DRAW BAR (LDB)

This chapter will help you to familiarize yourself with the major components and adjustments to the Lever Draw Bar (LDB).

There are four separate points that affect the overall slack adjustment for the LDB pull handle.

1) The first is the draw bar itself with the Belvedere spring washers. Ideally, you want to tighten the draw bar as far it can go and back off enough for the draw bar stroke to open the R8 collet enough to insert the tool. The stroke needs about 1-2 thread turns of the draw bar which leaves about .070-.100” of available stroke. This will give you the maximum spring pressure for the tool allowing some linear travel to open the jaws.

Figure 9-1

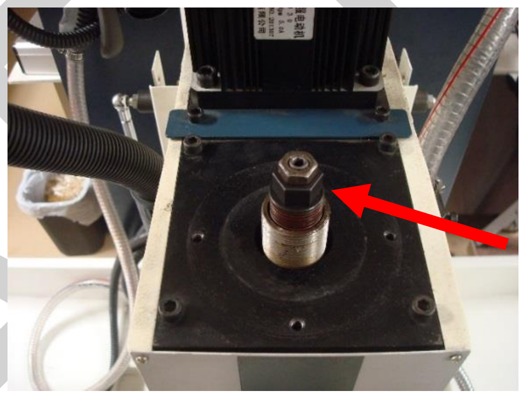


Figure 9-2

2) Once this is done, the second adjustable parameter is the LDB base. The four screws hold it down to the head plate. The spindle collar nut is then tightened and backed off so it just clears the spindle from rubbing while rotating. There are three set screws that lock the nut into place. (Apply Lock Tight to the three set screws to prevent the screws from loosening when spindle is running. Do not over tighten these set screws.) This adjustment limits the upward movement before it contacts the spindle collar nut, starting the clamping (pinching) action.

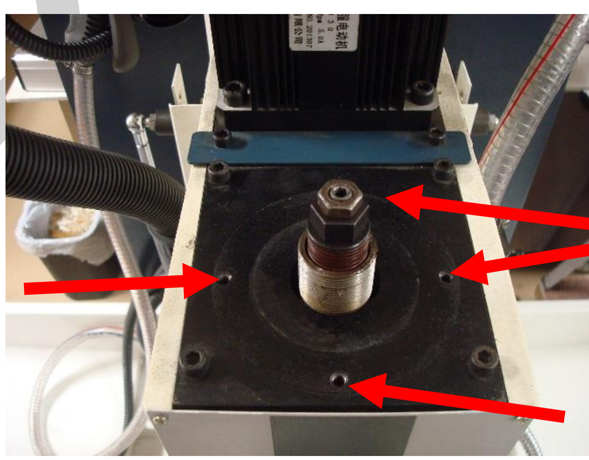


Figure 9-3

3) The third adjustment is the set screw with nut on the draw bar head itself. The set screw and nut that can be lengthened to help adjust the total length. This is adjusted only if the fourth adjustment does not allow the draw bar to be fine-tuned for lever engagement.

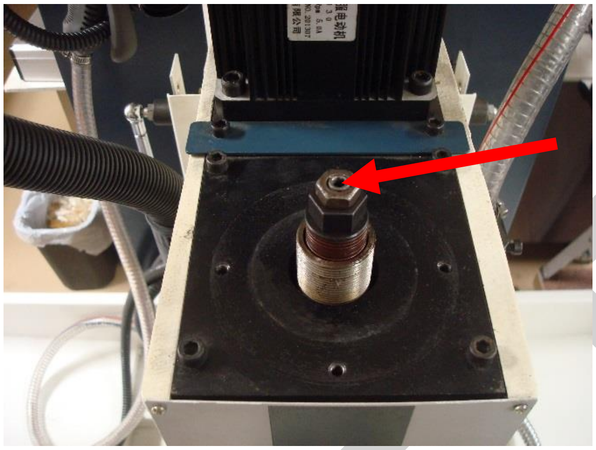


Figure 9-4

4) The fourth method for adjustment of LDB is the raising or lowering of the LDB head block itself (Figure 9-6 and Figure 9-5). When the LDB block is backed off from rubbing on the spindle, there should be about 20-30 degrees of lever movement before it contacts the spindle. If there is not enough stroke available in the handle, loosen the set screw on the draw bar head and rotate the LDB head block by +/- one turn to fine tune it. Recheck the degrees of lever movement and continue to adjust the LDB head block until the desired results are achieved. Lock the set screw and recheck the alignment.

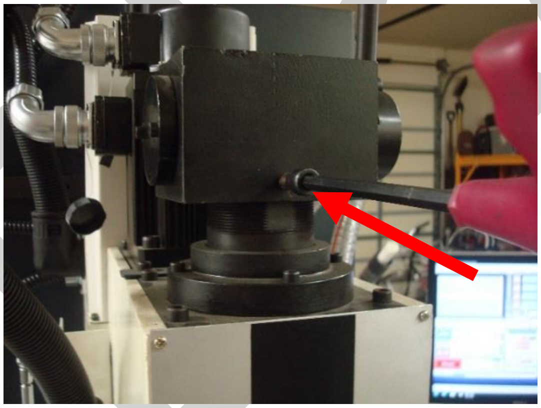


Figure 9-6

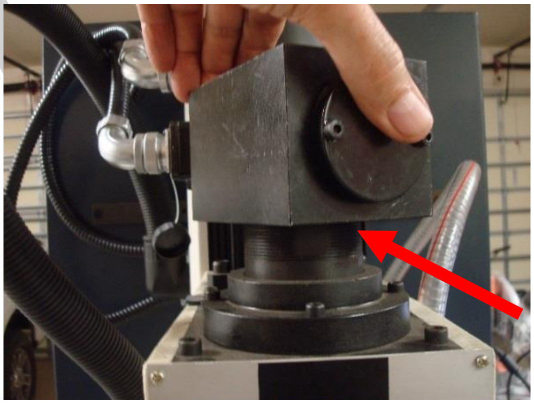


Figure 9-5

10 TOOL HOLDING

OVERVIEW

In milling operations cutters are mounted in a tool holder which is drawn into the spindle by a drawbar system. The Pulsar Mill utilizes the readily available Bridgeport® R-8 standard tooling. The standard drawbar for R-8 tapered tooling is 7/16” diameter with 20 threads per inch (TPI).

USING THE LDB

The Pulsar Mill with the LDB is optimize to be used with a ¾” R-8 flat faced collect to accept ¾” quick change tooling readily available in the industry.

The LDB is simple to use.

1) Pull down on the LDB handle.

2) Insert or remove quick change tooling.

3) Release the LDB handle slowly as the LDB handle is spring loaded. In other words don’t let LDB slip our of your hand and snap back.

TOOL HOLDERS

There are several tool holders that can be used in the spindle of the Pulsar Mill.

Arbors come in different sizes and lengths with one end tapered to suit the inside spindle taper. To accommodate quick change tooling, a special flat faced R-8 taper is required and included with the mill. The term arbor, is used for a general work holding device. Arbors can mount drill chucks, slitting saws, gear cutters and shell end mills. An arbor is secured in place with the drawbar system.

Collets (Spring Collets) are the most commonly used work holding device. They come in a variety of styles and sizes. They are bored to hold a specific diameter. As the collet is drawn into the spindle of the machine, the spring collet “closes” or “clamps” down on the tool securing it in place.

Collets (Set-Screw) are made specifically to hold the most common diameter tools up to 1/2” in size, commonly in 1/16” increments. (Larger collets are available with sizes up to ¾”) Set-Screw collets are bored to a specific size to hold the common diameter tools with a corresponding shank size. The tools are secured in the collet by a set screw that clamps down on the the side of the tool.

11 WARRANTY

NOVAKON WARRANTY

Novakon warrants its machines for a period of one (1) year from the date of invoice. If within one (1) year from the date of invoice, the Novakon Pulsar CNC Mill fails due to a defect in material or workmanship, then Novakon will, at its discretion, repair and/or replace the components with a new manufactured part(s) free of charge. This warranty does not cover labor, repairing costs or replacing parts. Customers have 14 days from date of delivery to check machine functionality and report any damages. Novakon will pay for return freight of damaged or defective part(s) from carrier's mishandling at its discretion. Claims must be made within this 14-day time frame or the customer will be responsible for the return freight.

This warranty does not apply to defects due directly or indirectly to misuse, abuse, negligence, accidents, repairs, lack of routine maintenance or an act of God. This warranty is also void if the serial number of the machine has been removed (where applicable), altered or modified.

An investigation will be made by Novakon to determine whether the warranty applies. To qualify, listed below are some of the causes of machine failure that this warranty does not cover.

Normal Wear – All mechanical devices need periodical parts service and replacement. This warranty will not cover repair when normal use has exhausted the life of a part(s) or component(s).

Improper Maintenance – The life of the machine and control system depends upon the conditions under which it operates and the care it receives. Applications of this machine may be in dusty and dirty environments, which can cause what appears to be premature wear. Such wear when caused by dirt, dust, cleaning grit or any abrasive material is not covered under warranty.

Non Original Novakon Parts – Problems caused by part(s) that are not original Novakon CNC machine parts.

Machine Installations – Improper installation of the machine or control system can prevent starting, may cause unsatisfactory machine performance and can shorten machine life.

Faulty Lubrication -- Part(s) which are broken due to operation with insufficient or contaminated lubricating oil, or incorrect grade of lubricating oil.

Repair or adjustment of associated part or assemblies which are not manufactured by Novakon including control systems, installed on Novakon CNC machines.

Part(s) damaged by excessive speed or overheating. Refer to the instruction manual for the recommended working environment and maintenance schedules.

Part(s) broken by excessive vibration caused by improper mounting of the machine or tools, installation, unbalanced set-up, improper attachment of work pieces or other abuse in operation.

Part(s) which are determined to have failed due to improper use or excessive wear caused by continuous use in a production environment. In cases such as this, Novakon will inspect the machine or part(s) and will be the sole judge of the merit of the claim.

Mishandling, improper operation, using the tool or control system for operations other than what was the intended use.

Warranty will be voided if modification to the original equipment has been made.

Transportation charges of part(s) submitted for repair and/or replacement under this warranty are the responsibility of the purchaser. Prior to the return of the machine or component, a Return Merchandise Authorization (RMA) number must be assigned by Novakon in order for Novakon to accept the return shipment(s).

Transportation charges for part(s)/machine submitted for repair and/or replacement under this warranty will be the customer's responsibility. If the part(s) or machine returned is found to be functional, an inspection fee and the return freight charges will be charged to customer. (A $100.00 US hourly fee will be applicable for such an inspection, and a minimum charge of $100.00 will be billed). If the part(s) is determined to be non functional upon an in-house inspection, Novakon will repair and/or replace the part(s)/machine and pay for the return freight and insurance to customer. No warranty registration is required. Please provide your invoice as proof of purchase. In the event that the invoice is not provided, Novakon will establish the purchase date and this date will be used to determine the warranty period.

12 PULSAR Mill SPECIFICATIONS

General

Table	23.6” x 7.8”
Work Cube	15.75” x 9.5” x 12.25”
Maximum Weight on Table	450 lbs.
Table Slots	Four 7/16” T-Slots
Table Travel X Axis	15.75” 400 mm
Saddle Travel Y Axis	9.5” 241 mm
Head Travel Z Axis	12.25” 311 mm
Castings	Cast Iron
Table Surface	Precision Ground
Ways (X, Y & Z-Axis)	Dovetail
Rigid Tapping Capability	Yes

Servo Spindle

Motor Power Rating	1.2KW AC Servo
Maximum Torque	8 N-m
Spindle Control	Infinite Variable
Drive System	Geared Belt
Minimum Speed	1 RPM
Maximum Speed	4,500 RPM
Ball Screw Incremental Error	+/- 0.0005”
Ball Screw Cumulative Error	0.00025”
Quill Diameter	3.15”
Spindle Bearings	Angular Contact Bearings
Spindle to Column	11.50”
Spindle to Table	2” – 14.25”
Spindle Taper	R-8

Axis

Stepper Motors (X, Y)	130SM0430
Optional Servo Motors (X, Y, Z)	90ST-M04025
Ball Screws	Pre-loaded Hardened & precision Ground
Ball Screw Pitch	5mm
Ball Screw Diameter	20mm
Ball Screw Precision	P4
Motor Mount (Stepper)	NEMA 34 (X, Y, & Z Axis)
Stepper Angle	1,8 Degrees
Stepper Holding Torque	? N-m
Stepper Peak Torque	? N-m
Optional Servo Holding Torque	4 N-m
Optional Servo Peak Torque	12 N-m
Phase	2
Current	10.0 amps
Voltage	30-300 V
Torque	106 in*lbs.

Electrical

Power Requirement	220 VAC @ 15 A (50-60 Hertz)

CNC Specifications

CNC Machine Dimensions	37” W x 37” D x 44” H
Machine Weight	?? lbs
Machine Footprint	13” L x 23 3/4” W
Shipping Dimensions	46” W x 49”D x 56” H
lbs.	440 kg
Maximum Work Capacities	
Drilling mild steel	??
Tapping	??”
Machine & Coolant	66" W x 44" D x 76" H

Stand Specifications

Stand Dimensions	20” W x 24” D x 36” H
Stand Weight	?? lbs.
Shipping Weight	Included with Machine Shipping Weight.

NOTE: Due to continual design improvements, specifications may change. Visit our website for up to date specifications.

PULSAR MILL DIMENSIONS

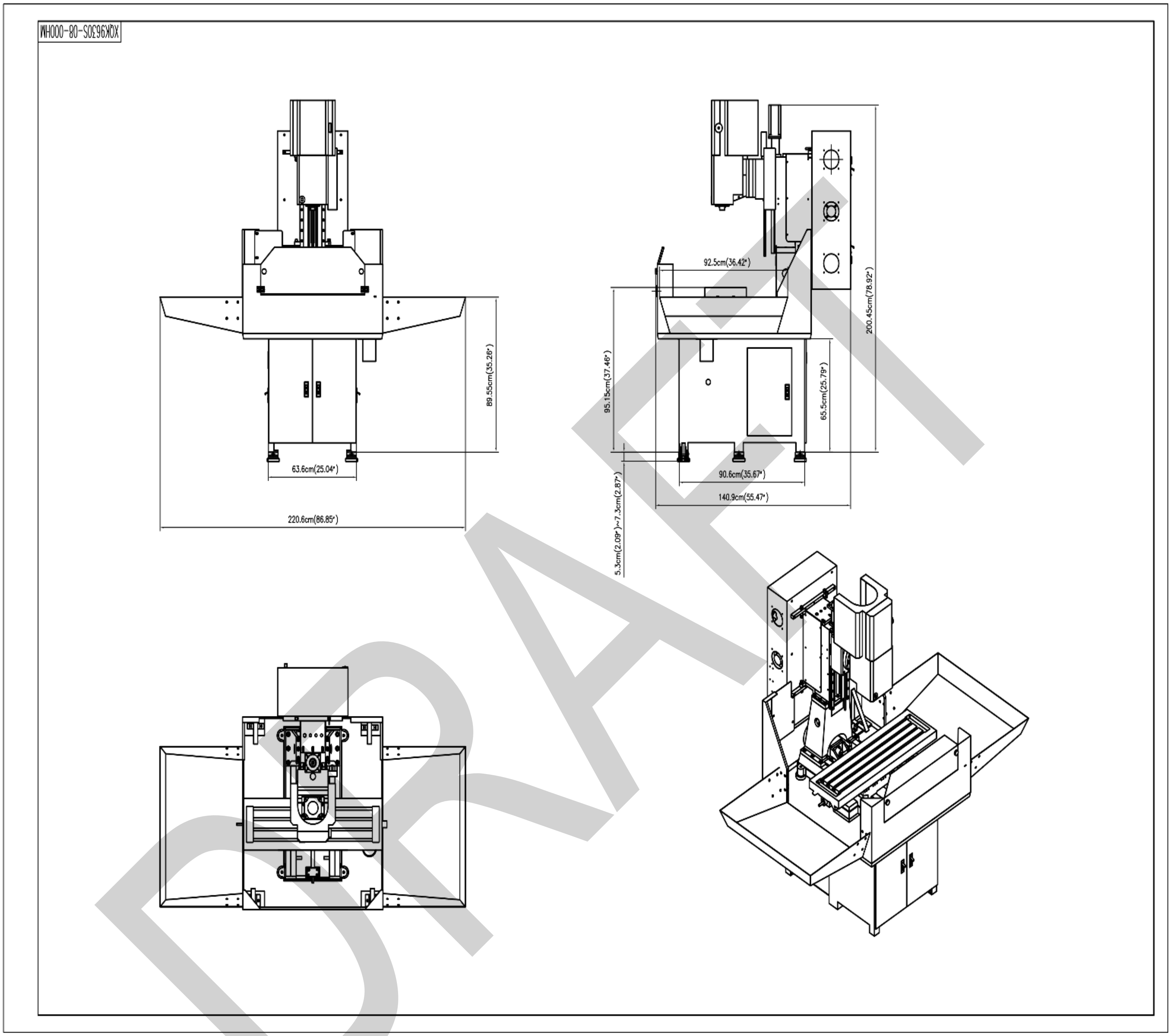
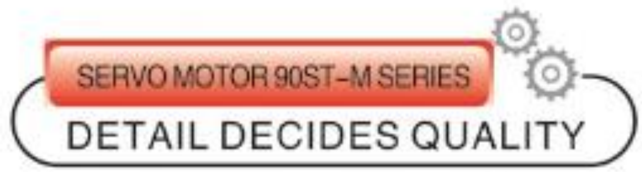


Figure 12-1

OPTIONAL X, Y & Z-AXIS AC SERVO MOTORS (MODEL 90ST-M04025)



Motor model	90ST-M02430	90ST-M03520	90ST-M04025
Rated power (Kw)	0.75	0.73	1.0
Rated voltage (V)	220	220	220
Rated cutter (A)	2.4	3.5	4
Rated speed (rpm)	300	2000	2500
Holding torque (N • m)	2.4	3.5	4
Peak torque (N • m)	7.2	10.5	12
Peak cutter (A)	7.2	10.5	12
Voltage constant (V/1000r/min)	62	65	61
Torque coefficient (N • m/A)	1.0	1.0	1.0
Rotor inertia (Kg • m^2)	0.25×10^{-3}	0.34×10^{-3}	0.37×10^{-3}
Phase Resistance (Ω)	4.4	2.6	2.0
Phase Inductance (mH)	11.6	7.6	6.4
Mechanical time constant (ms)	2.2	2.39	2.3
Weight (Kg)	3.0	3.8	4.2
Encoder line number (PPR)	2500		
Insulation class	Class B (130℃)		
Safety class	IP65		
The operating of Environmental Conditions	-20℃-+50℃ Temperature: -20℃-+50℃ Humidity90%RH No dewing		

Motor winding plug					
Motor winding plug	Winding lead wire	U	V	W	PE
	Plug serial number	2	3	4	1

Encoder plug																
Encoder plug	Signal lead wire	5V	0V	A+	B+	Z+	A-	B-	Z-	U+	V+	W+	U-	V-	W-	PE
	Plug serial number	2	3	4	5	6	7	8	9	10	11	12	13	14	15	1

Figure 12-2 Optional X, Y & Z Axis Servo Motors Specifications (90ST-M0405)

Figure 12-3 Optional X, Y & Z Axis Servo Motors Dimensions (90ST-M0405)

AC SPINDLE SERVO MOTOR PARAMETER TABLE

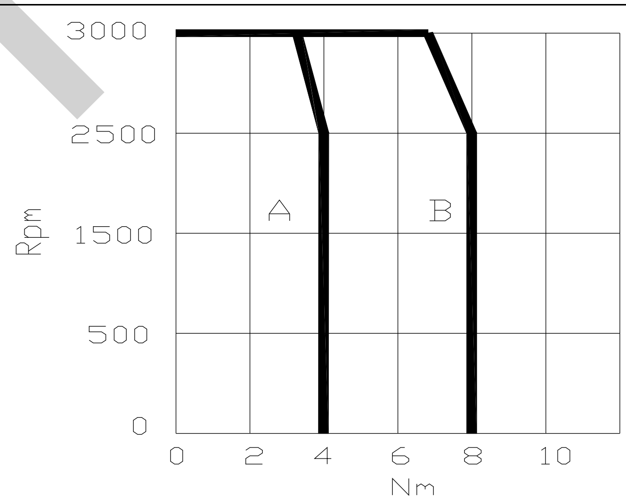
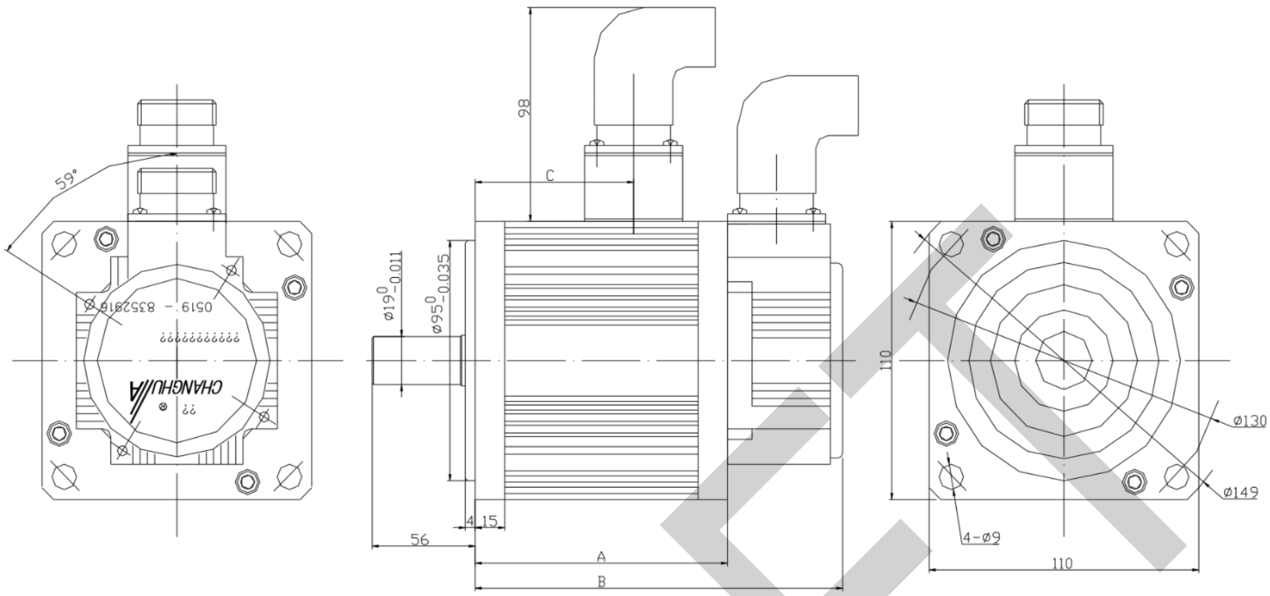
Motor Model	110SM04030
Power (KW)	1.2
Zero-Speed Torque （Nm）	4
Rated Speed (RPM）	3000
Rated Current (A）	5.0
Rotor Inertia（Kgm^2）	0.65×10^{-3}
Electrical Time Constant F (ms)	2.99
Encoder Resolution P/R）	2500
Motor Insulation Class	B
Adapter Driver Model	Panasonic and other domestic or import the appropriate drive
Operating Voltage (AC)	3ΦAC220V -15% ~ +10% 50/60Hz
Work Duty Cycle	Continuous
Environmental Conditions	Operating Temperature:(0 ~ 55) ℃Storage Temperature:(-20 ~ 80) ℃ Humidity: Less than 90% (No condensation) Vibration: Less than 0.5g (4.9m/s^2), 10 ~ 60Hz (Non-Continuous Operation)
Control	Sine Wave PWM Control
Torque Speed Curve	

Table 12-1 Torque-speed graph (T-M) (A: continual work area; B: short time work area)

Figure 12-4

110



zero-speed torque (N.m)	2	4	5	6
A (mm)	106(171)	136(201)	151(216)	166(231)
B (mm)	154(219)	184(249)	199(264)	214(279)
C (mm)	71	101	116	131

A, B is the size of safe brake

Figure 12-5 SERIES 110 AC SERVO MOTOR DIMENSIONS

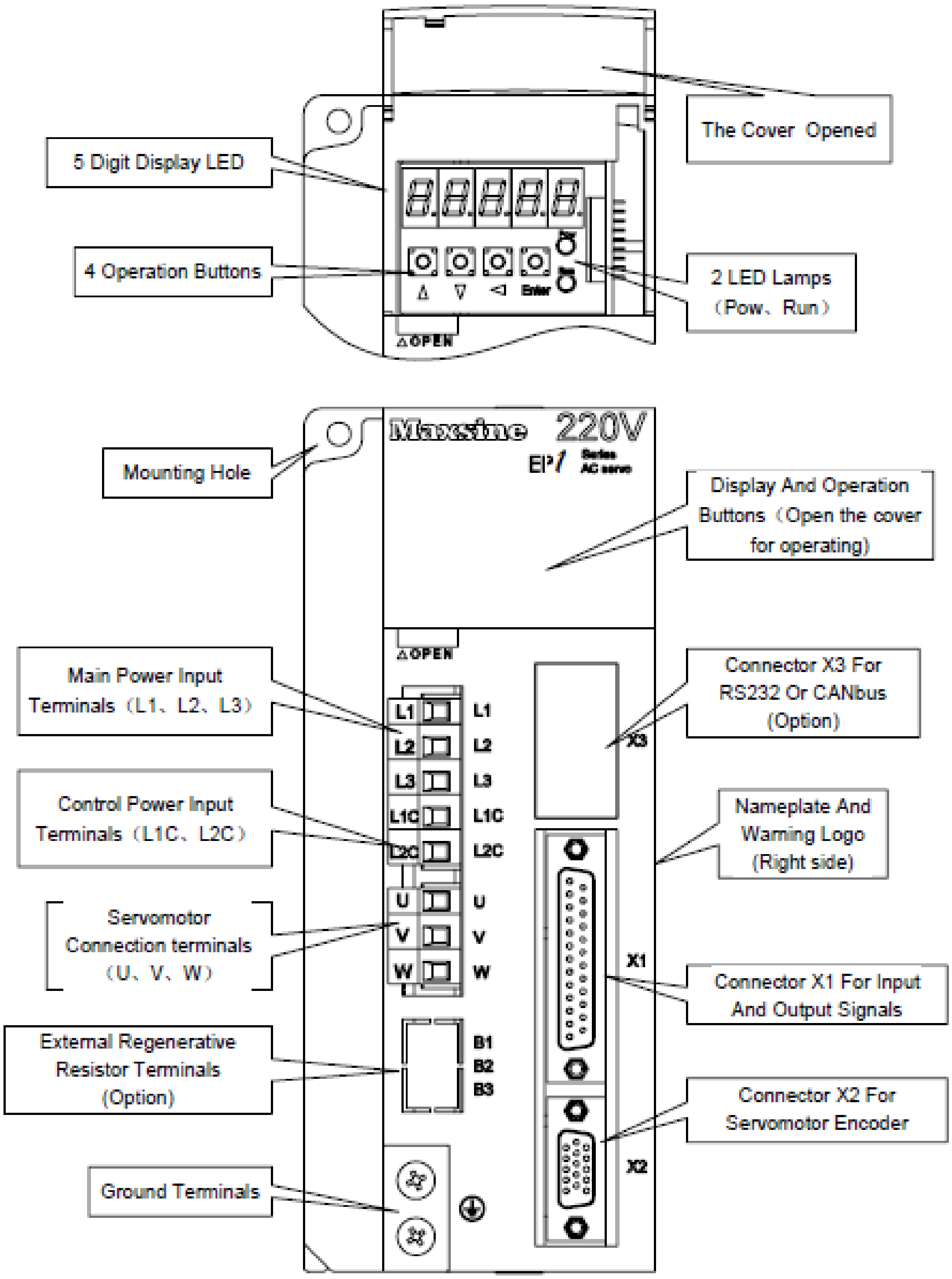


Figure 12-6 Maxine EP1 220 VAC Servo Front Panel

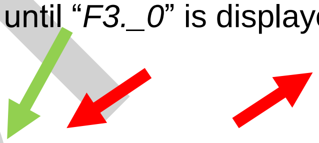
SPINDLE SERVO CONTROLLER PARAMETER SETTINGS

Certain Maxsine EP-1 spindle servo controller parameter settings are programmed different than the default factory settings. Figure 12-7 shows the affected parameter settings.

Parameter	Setting
P4	0
P17	1.5
P29	1000
P30	750
P31	29984
P35	0
P36	0
P40	50
P80	12
P97	3
P98	1
P100	18
P101	19
P131	-3

Figure 12-7 Maxsine EP-1 Spindle Servo Parameters

1) Pressing and releasing the "<" button will highlight one of the displayed digits by making it blink, indicating that the digit can be modified using the "Up Arrow" and "Down Arrow" buttons. Press and release the "<" button one or more times, as needed, until the left-most zero begins blinking. Continue pressing and releasing the "Up Arrow" button until "*F3._0*" is displayed on the LED panel.



FOURTH AXIS DIMENSIONS (SIX INCH)

TSL160-1C

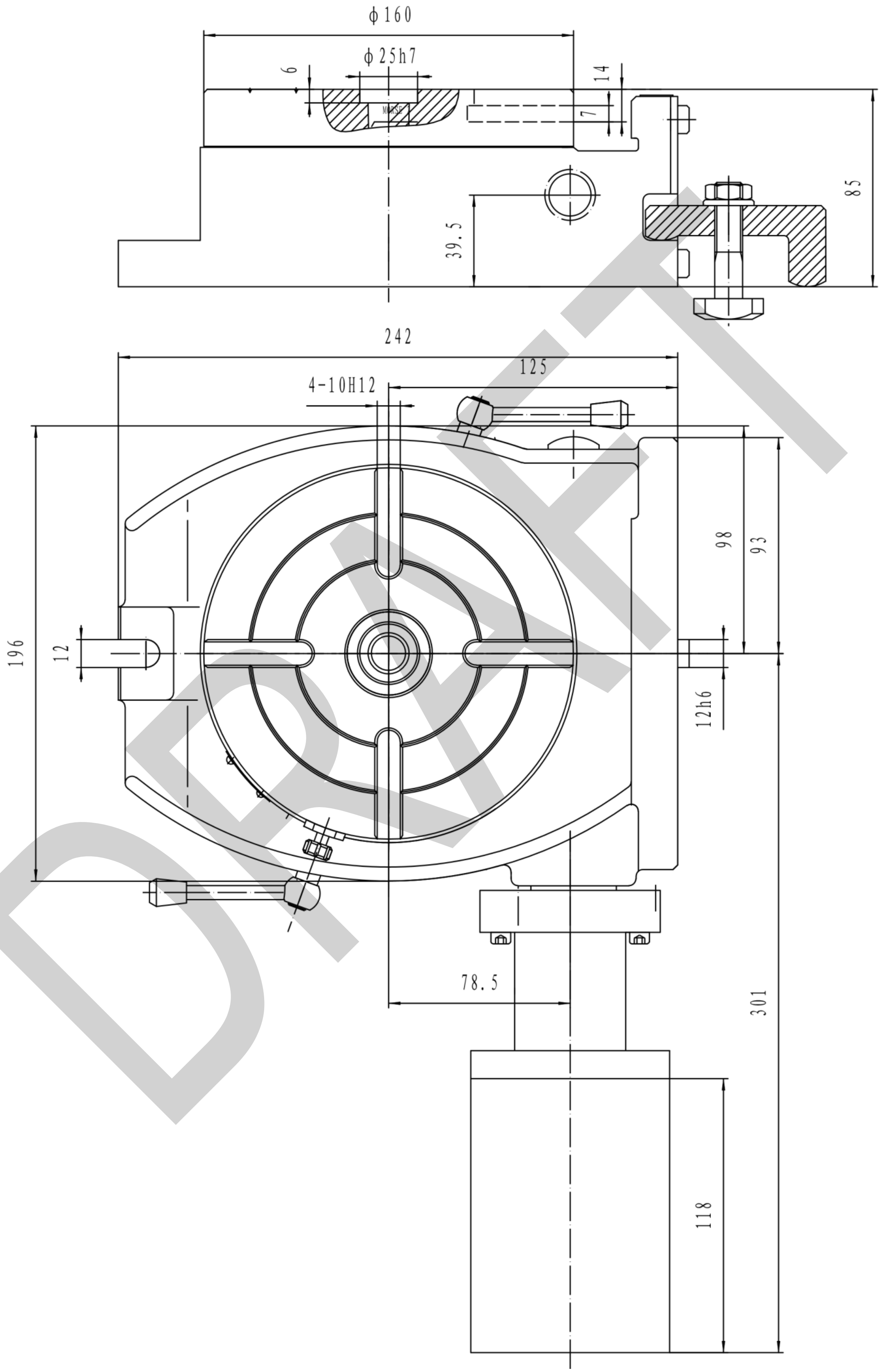
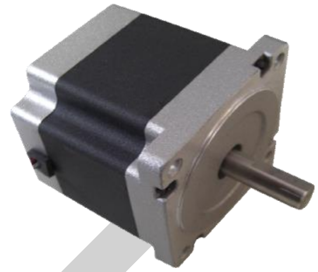


Figure 12-8 Six Inch Fourth Axis

FOURTH AXIS STEPPER MOTOR

HYBRID STEPPING MOTOR 85HS80-4208-001

(Optional Fourth Axis Rotary Table)



Tolerance...± 5%

Maximum Operating Temperature.............................80℃ Max

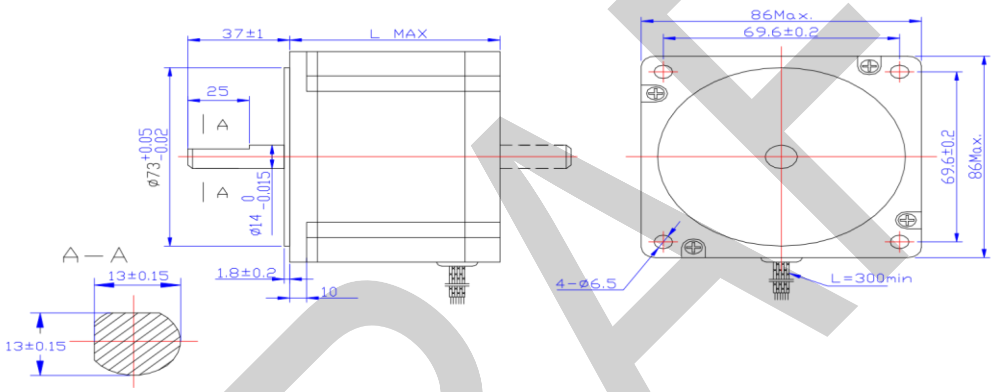
Temperature Operating Range................................-20℃ - +50℃

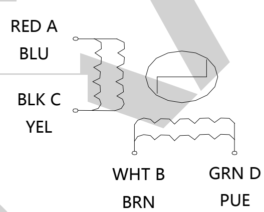
Resistance..100MΩMin.500 VDC

Resistance Limit..500VAC/1 minute

Model	Step Angle (°)	Length L(mm)	Nominal Voltage (V)	Current (A)	Resistance (Ω)	Inductance (mH)	Torque (N.m)	Poles (NO.)	Inertia (kg.cmsq)	Weight (kg)
85HS80-4208-001	1.8	80	4.6	2.97	1.55	6.76	3.2	8	1.4	2.3

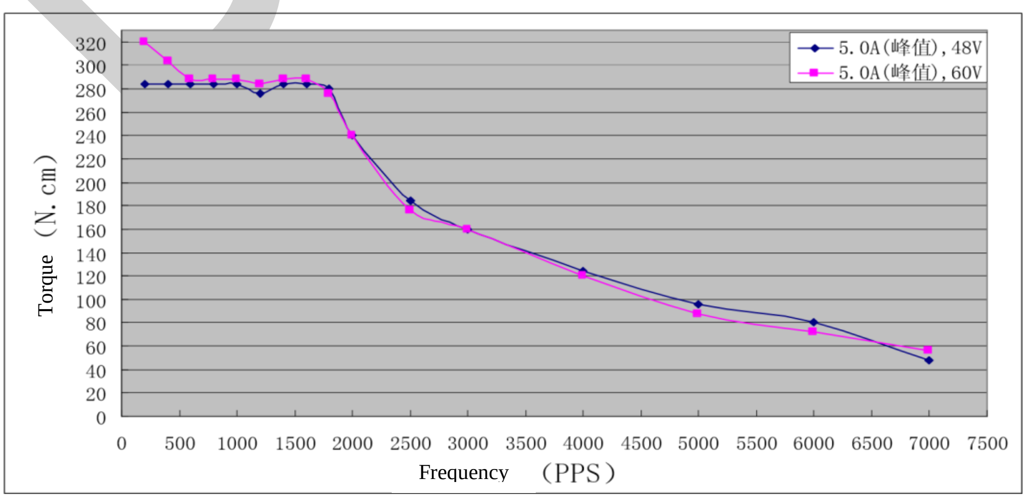
Dimensions





4 Leads Parallel

Model 85HS80 Driver SD-2H086MB：Four Wire Bi-Polar 4 Poles



13 WIRING SCHEMATICS AND MACHINE PARTS

OVERVIEW

This section shows all the necessary parts of the machine. All the diagrams are labeled and numbered. The reference codes and numbers are found on the Reference Chart after each diagram.

Please have this section ready whenever you have questions, inquiries or you are in need of technical help regarding the machine.

If replacement parts are required, please e-mail Novakon at: support@Novakon.net

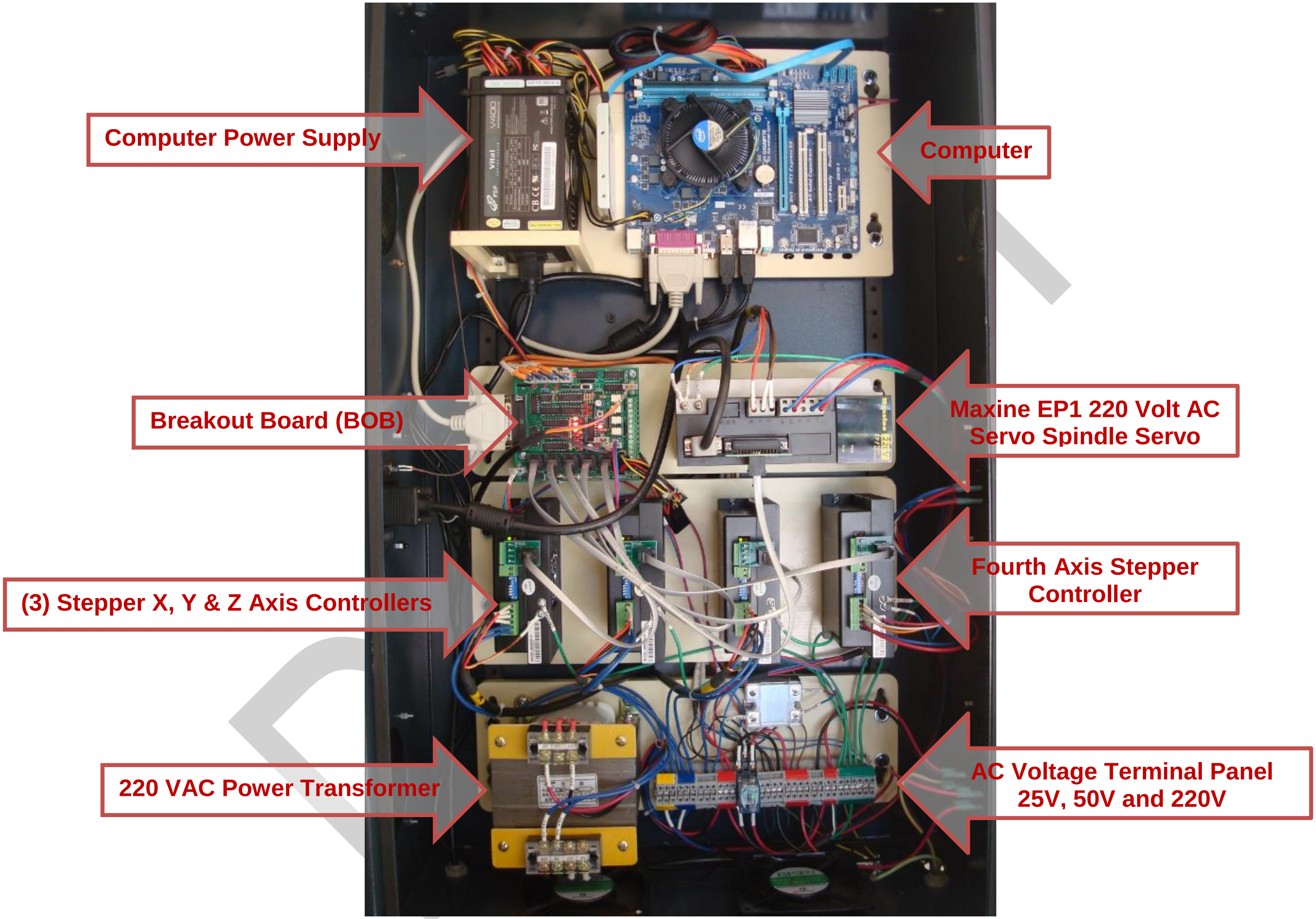


Figure 13-1 Electrical Control Panel

NOVAKON BOB REV 0

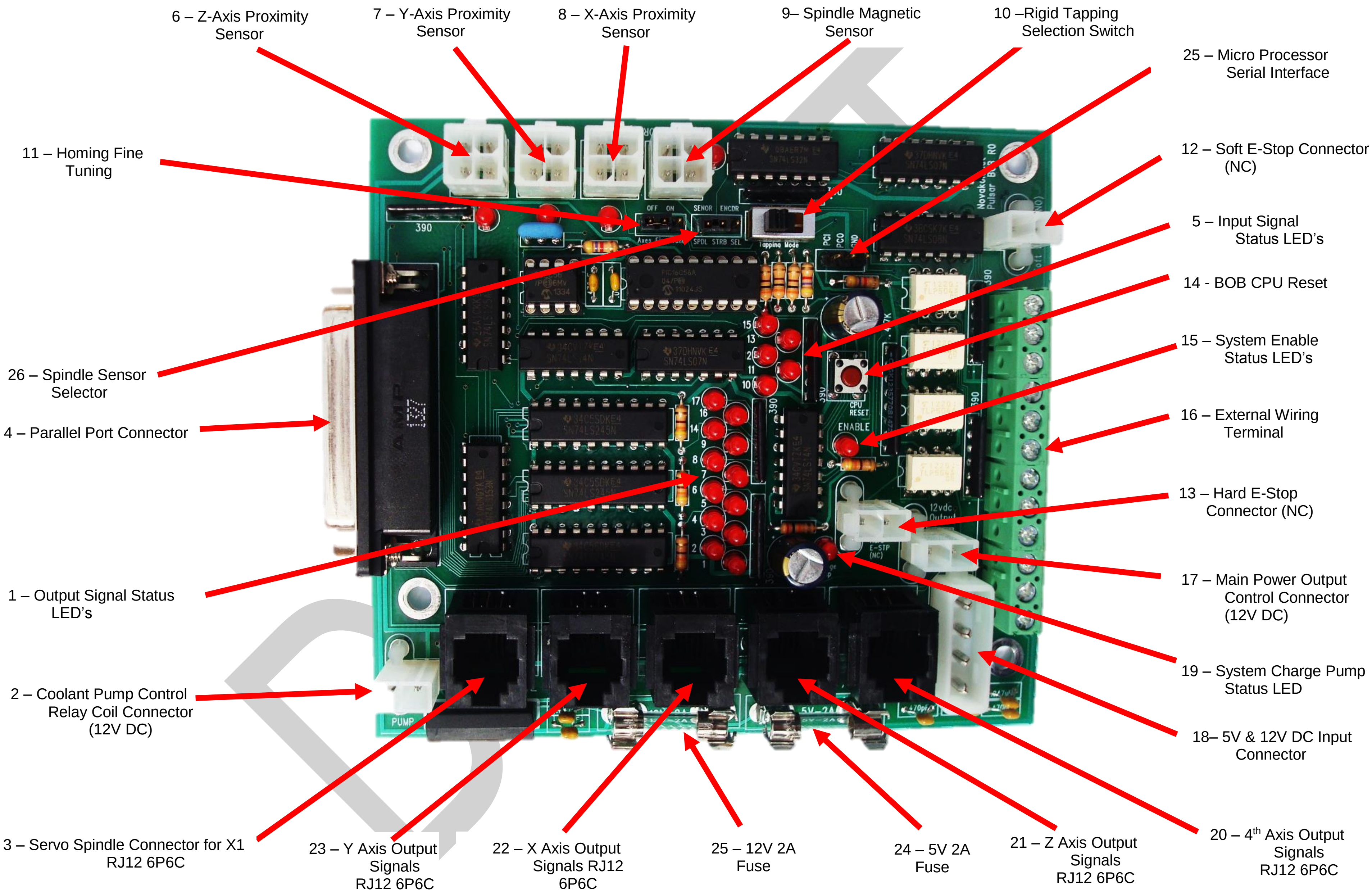


Figure 13-2 Novakon Pulsar BOB Revision 0

Novakon Pulsar BOB Rev 0 Item Description			
NO.	**Item**	**Description**	
Connectors			
2	Coolant Pump Control Connector	Output 9 controls 12 VDC output for relay control to turn on/off coolant pump 220 VAC power	2 Wires
4	Parallel Port Connector	Provides I/O from BOB to PC	DB25
6	Spindle Magnetic Home Sensor Connector	The sensors share Input 12 to monitor HOME position and RPM input	3 Wires (GND, 12 VDC, Signal)
7	Y –Axis Proximity Home Sensor Connector	The sensors share Input 11 to monitor HOME position. Any one of 3 axes reaches their home position will trigger the input	3 Wires (GND, 12 VDC, Signal)
8	X-Axis Proximity Home Sensor Connector	The sensors share Input 11 to monitor HOME position. Any one of 3 axes reaches their home position will trigger the input	3 Wires (GND, 12 VDC, Signal)
9	Z-Axis Proximity Home Sensor Connector	The sensors share Input 11 to monitor HOME position. Any one of 3 axes reaches their home position will trigger the input	3 Wires (GND, 12 VDC, Signal)
12	Soft E-Stop Connector (N.O.)	Input 10 monitors E-stop status to enable/disable Mach3 Reset	2 Wires
13	Hard E-Stop Connector (N.C.)	Connector is used to cut off the machine's main driver power when the manual E-stop button is pressed through the main power relay (PC remains on)	2 Wires
17	Main Power Output Control Connector	Provides 12 VDC for main power relay coil when BOB becomes activated	2 Wires
18	5 &12 VDC Input Connector	This power comes from the PC power supply	4 Wires

MOD JACK 6P6C			
3	Servo Spindle Connector	Tapping Mode ON: The Jack Module Outputs Signals to the servo driver (Puls+, Puls-, Dir+, Dir-) from Outputs 7 & 8 Tapping Mode OFF: The Jack Module Outputs Signals to the Servo driver (Puls+, Puls-, Dir+, Dir-) from Pins 14 & 16	Pin 7: Puls+; Pin 8: Dir+; GND: Puls-, Dir-
20	4th Axis Output Signals Connector	The Jack Module Outputs Signals to Stepper Driver (Puls+, Puls-, Dir+, Dir-) through Outputs 7 & 8	Pin 7: Puls+; Pin 8: Dir+; GND: Puls-, Dir-
21	Z-Axis Output Signals Connector	The Jack Module Outputs signals to stepper driver (Puls+, Puls-, Dir+, Dir-) through Outputs 5 & 6	Pin 5: Puls+; Pin 6: Dir+; GND: Puls-, Dir-
22	X-Axis Output Signals Connector	The Jack module outputs signals to stepper driver (Puls+, Puls-, Dir+, Dir-) through Outputs 1 & 2	Pin 1: Puls+; Pin 2: Dir+; GND: Puls-, Dir-
23	Y-Axis Output Signals Connector	The Jack module outputs signals to stepper driver (Puls+, Puls-, Dir+, Dir-) through Outputs 3 & 4	Pin 3: Puls+; Pin 4: Dir+; GND: Puls-, Dir-
Button & Switch			
10	Tap Mode Selector	When selected Rigid Tapping is ON	
14	BOB CPU Reset	Press the reset button; the BOB will reset the CPU program	
Terminals			
16	External Wiring Terminals	The terminal Strip can accept Active Low Input Signals and Source +12DC, +5VDC and GND Outputs	
D-sub Connectors			
4	Parallel Port Connector	The parallel port connector is used for communicating between the computer and Mach3	DB25 Female

Status LEDs			
1	Outputs Signal Status LEDs	Indicates the status of all the outputs (High or Low)	
5	Inputs Signal Status LEDs	Indicates the status of all the inputs (High or Low)	
15	System Enable Status LED	Indicates Mach3 is in control of the system; ON: Enabled; OFF: Disable	
19	System Charge Pump Status LED	Slow Pulse Rate: LED Flashes Slow at 1 Hz; Fast Pulse Rate: LED Flashes at a faster Rate of 3 Hz or more; Steady ON or OFF: Processor is Locked Up	Depressing the CPU Reset button will reboot the BOB

Shunts			
11	Homing Fine Tuning	Select Encoder Feed Back Strobe for More Precise Home Tuning	ON/OFF
26	Spindle Sensor Selector	Select Source for Spindle RPM Clock (Magnetic Spindle Sensor or Spindle Server Encoder). This Input is applied to Input Pin 12	Sensor/Encoder
Fuses			
24	5V 2 Amp	5V Fused Output Protection	
25	12V 2 Amp	12V Fused Output Protection	

Note: Pulsar BOB VER 0 input voltage:

Computer power supply 5 VDC & 12 VDC

5 VDC load current; 400-420mA

12 VDC load current; 80-90mA without running coolant pump relay; 160mA when coolant pump is on

PULSAR STEPPER WIRING DIAGRAM

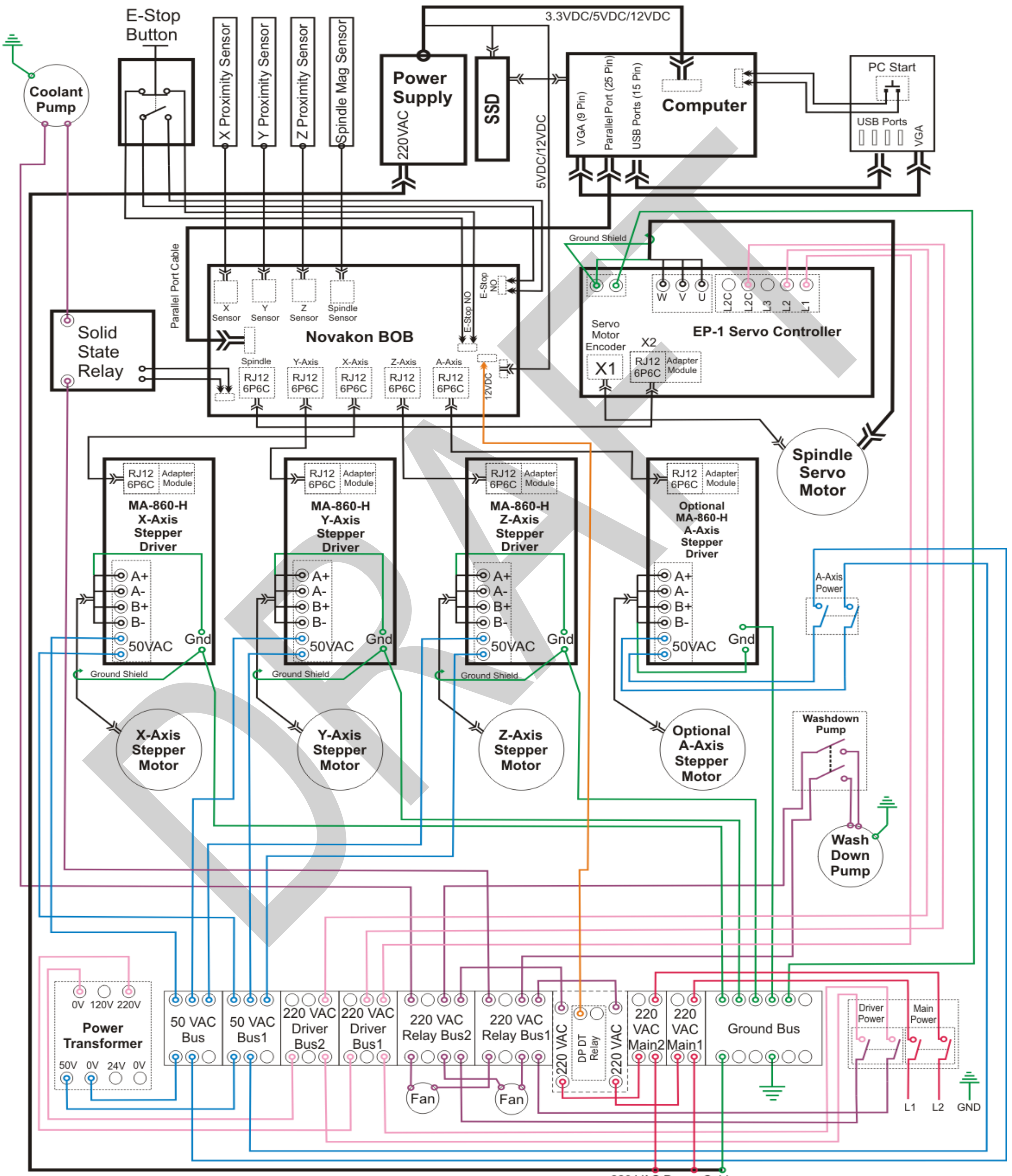


Figure 13-3 Pulsar Stepper Wiring Diagram

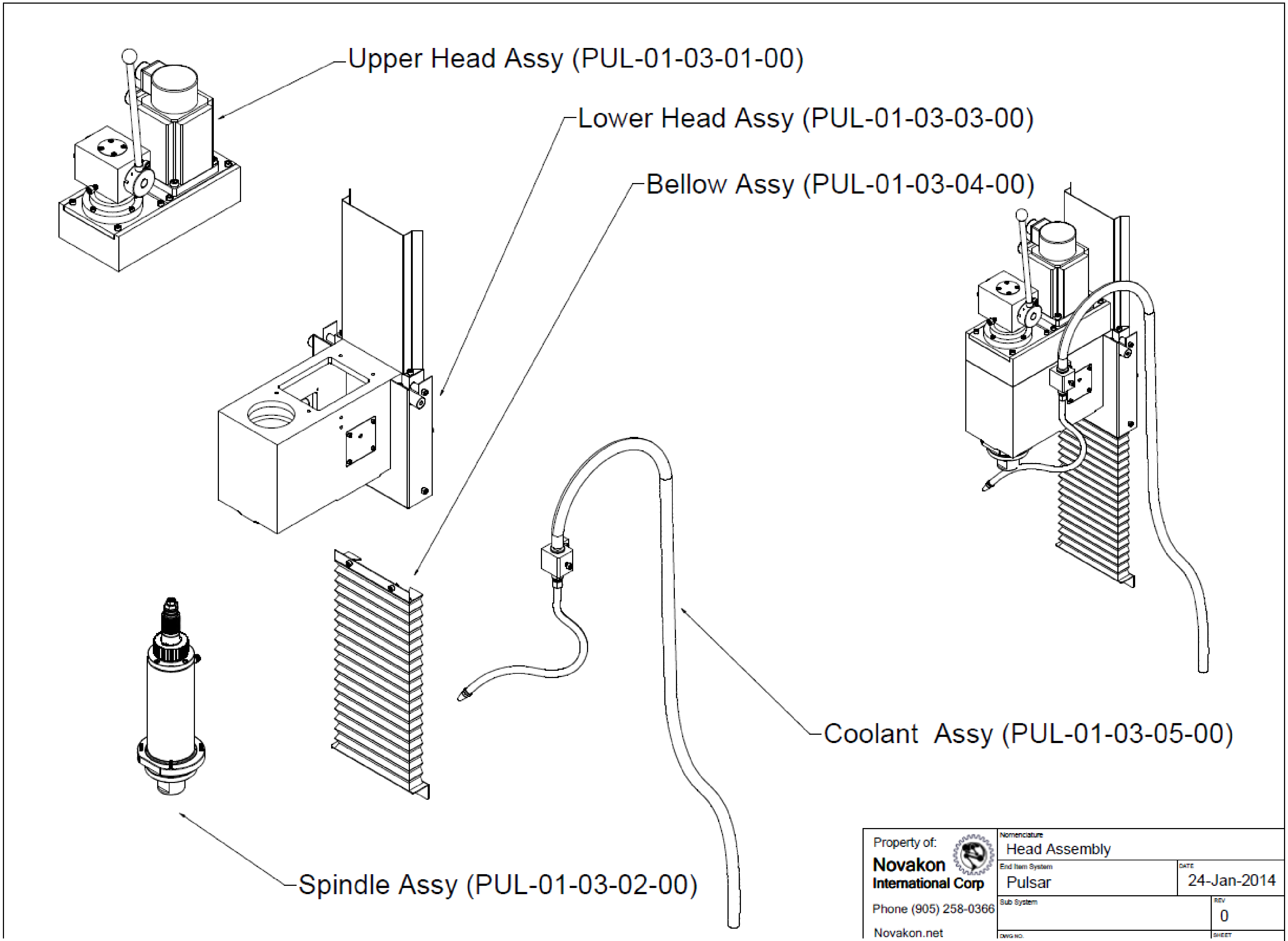
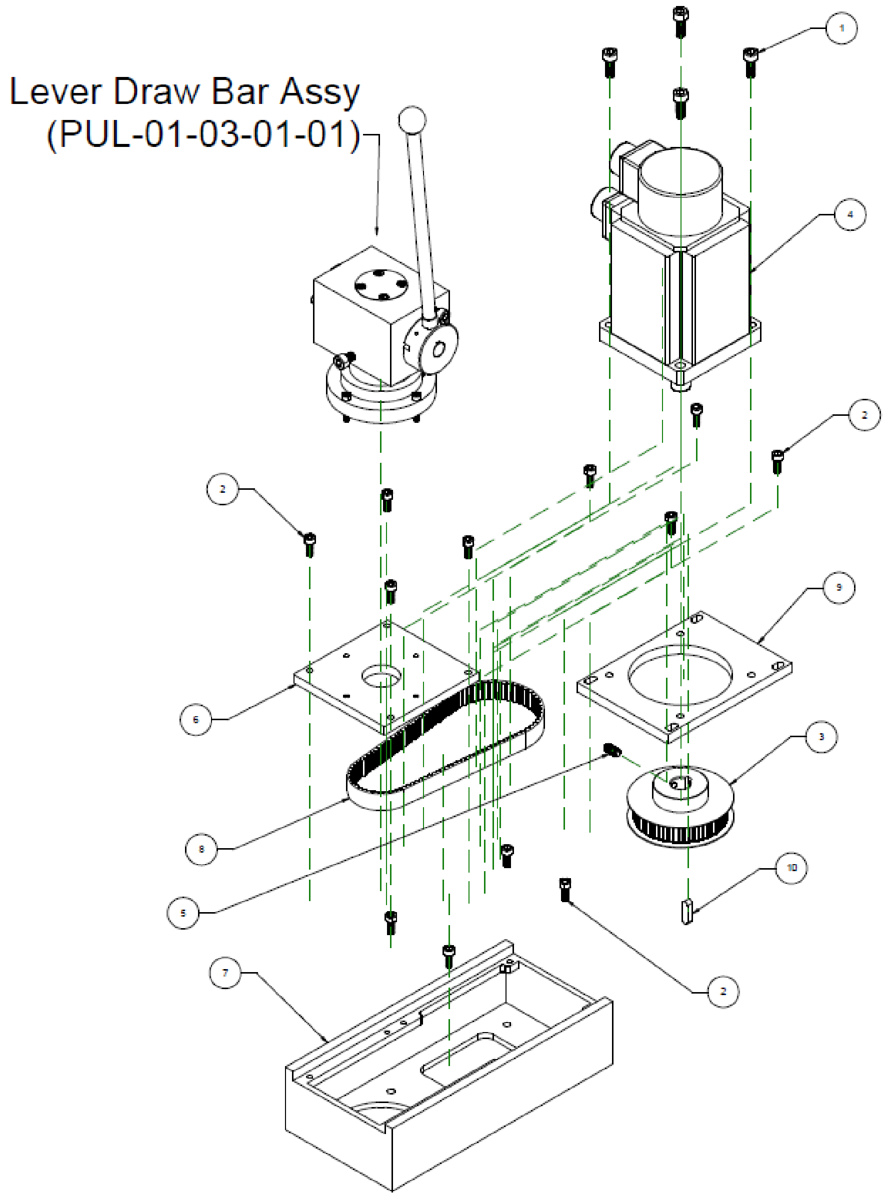
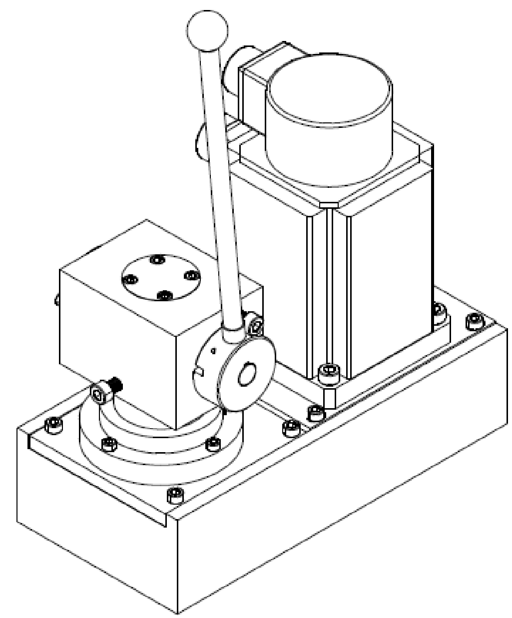


Figure 13-4 Pulsar Exploded Parts Oview



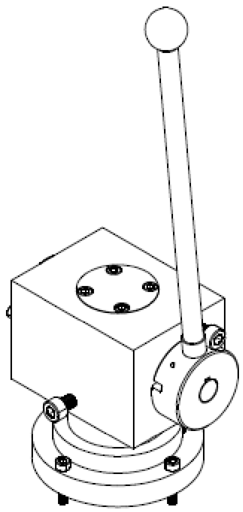
Item Number	Quantity	Part Number	Part Name	Revision	Comment
1	6		M8x20		
2	12		M6x15		
3	1	PUL-01-03-01-02	Servo Motor Pulley		
4	1	110SM04030	Servo Motor		1.2KW 3PH 3000RPM
5	1		Set Screw, M8x15		
6	1	PUL-01-03-01-03	Spindle cover Plate		
7	1	PUL-01-03-01-04	Servo Belt Guard		
8	1		Belt		
9	1	PUL-01-03-01-05	Servo Motor Slide		
10	1		Key,6x6x25		

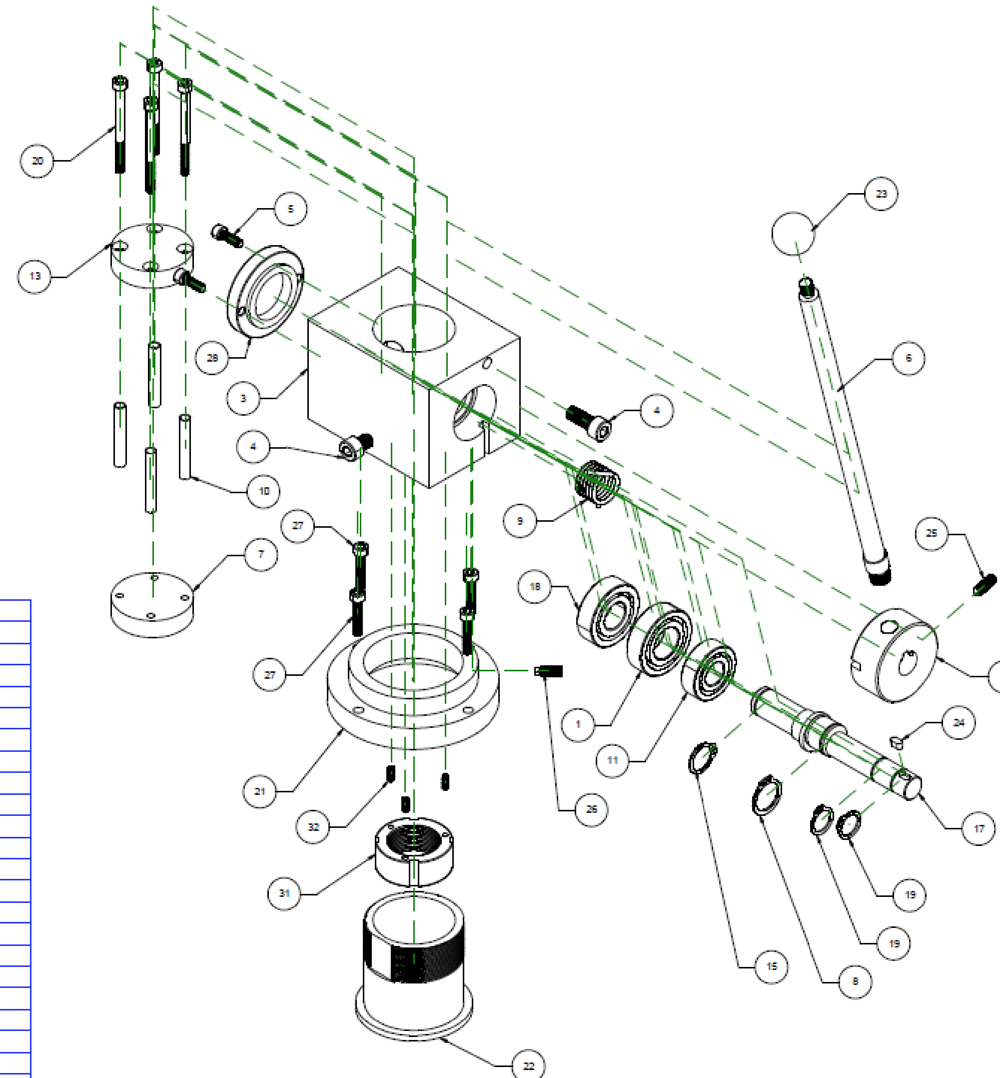


Property of: Novakon International Corp	Nomenclature: Upper Head Assy	
Phone (905) 258-0366	End Item System: Pulsar	DATE: 18-Jan-2014
Novakon.net	Sub System:	REV: 0
	DWG NO.:	SHEET: 1 of 1

Figure 13-5 Head Assembly

LEVER DRAW BAR ASSEMBLY



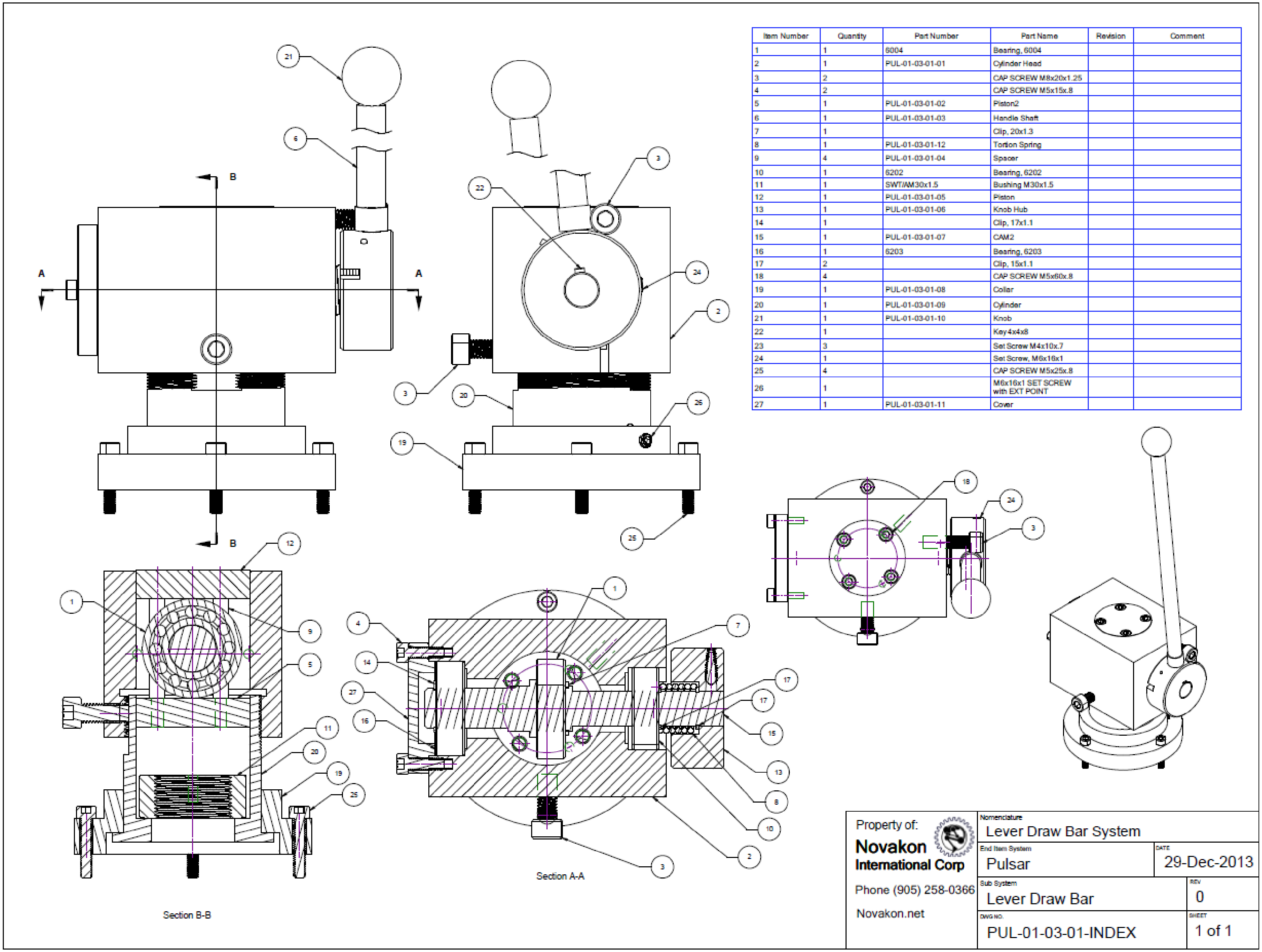


Item Number	Quantity	Part Number	Part Name	Revision	Comment
1	1		Bearing, 6004		
3	1	PUL-01-03--01-01-01	Cylinder Head		
4	2		M8x20		
5	2		M5x15		
6	1	PUL-01-03--01-01-02	Handle Shaft		
7	1	PUL-01-03--01-01-03	Lower Piston		
8	1		Clip, 20x1.3		
9	1		Tortion Spring		
10	4	PUL-01-03--01-01-04	Spacer		
11	1		Bearing, 6202		
13	1	PUL-01-03--01-01-05	Upper Piston		
14	1	PUL-01-03--01-01-06	Knob Hub		
15	1		Clip, 17x1.1		
17	1	PUL-01-03--01-01-07	CAM Shaft		
18	1		Bearing, 6203		
19	2		Clip, 15x1.1		
20	4		M5x60		
21	1	PUL-01-03--01-01-08	Collar		
22	1	PUL-01-03--01-01-09	Cylinder		
23	1	PUL-01-03--01-01-10	Knob		
24	1		Key 4x4x8		
25	1		Set Screw, 6mm x 16		
26	1		M16x16 SET SCREW with EXT POINT		
27	4		M5x25		
28	1	PUL-01-03--01-01-11	Cover		
31	1		Threaded Bushing M30x1.5x18		
32	3		Set Screw M4x10		

Novakon International Corp
Phone (905) 258-0366
Novakon.net

Nomenclature: Lever Draw Bar Assy
End Item System: Pulsar
Sub System:
DWG NO. PUL-01-03-01-01
REV
DATE 26-Jan-2014
SHEET 1 of 1

Figure 13-6 Lever Draw Bar Assembly



Item Number	Quantity	Part Number	Part Name	Revision	Comment
1	1	6004	Bearing, 6004		
2	1	PUL-01-03-01-01	Cylinder Head		
3	2		CAP SCREW M8x20x1.25		
4	2		CAP SCREW M5x15x.8		
5	1	PUL-01-03-01-02	Piston2		
6	1	PUL-01-03-01-03	Handle Shaft		
7	1		Clip, 20x1.3		
8	1	PUL-01-03-01-12	Tortion Spring		
9	4	PUL-01-03-01-04	Spacer		
10	1	6202	Bearing, 6202		
11	1	SWT/AM30x1.5	Bushing M30x1.5		
12	1	PUL-01-03-01-05	Piston		
13	1	PUL-01-03-01-06	Knob Hub		
14	1		Clip, 17x1.1		
15	1	PUL-01-03-01-07	CAM2		
16	1	6203	Bearing, 6203		
17	2		Clip, 15x1.1		
18	4		CAP SCREW M5x60x.8		
19	1	PUL-01-03-01-08	Collar		
20	1	PUL-01-03-01-09	Cylinder		
21	1	PUL-01-03-01-10	Knob		
22	1		Key 4x4x8		
23	3		Set Screw M4x10x.7		
24	1		Set Screw, M6x16x1		
25	4		CAP SCREW M5x25x.8		
26	1		M6x16x1 SET SCREW with EXT POINT		
27	1	PUL-01-03-01-11	Cover		

Figure 13-8 Lever Draw Bar System

HEAD AND SPINDLE ASSEMBLY

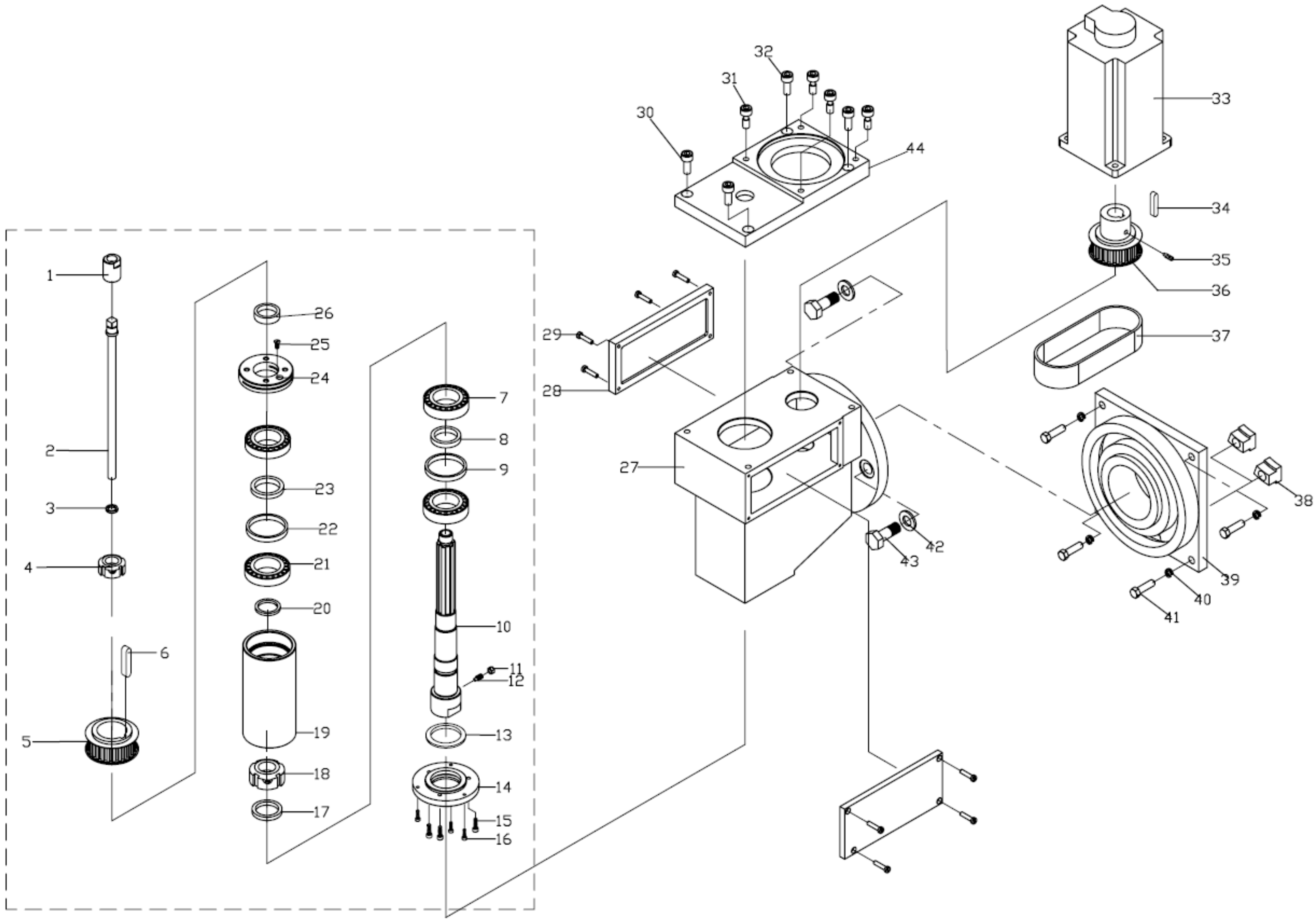
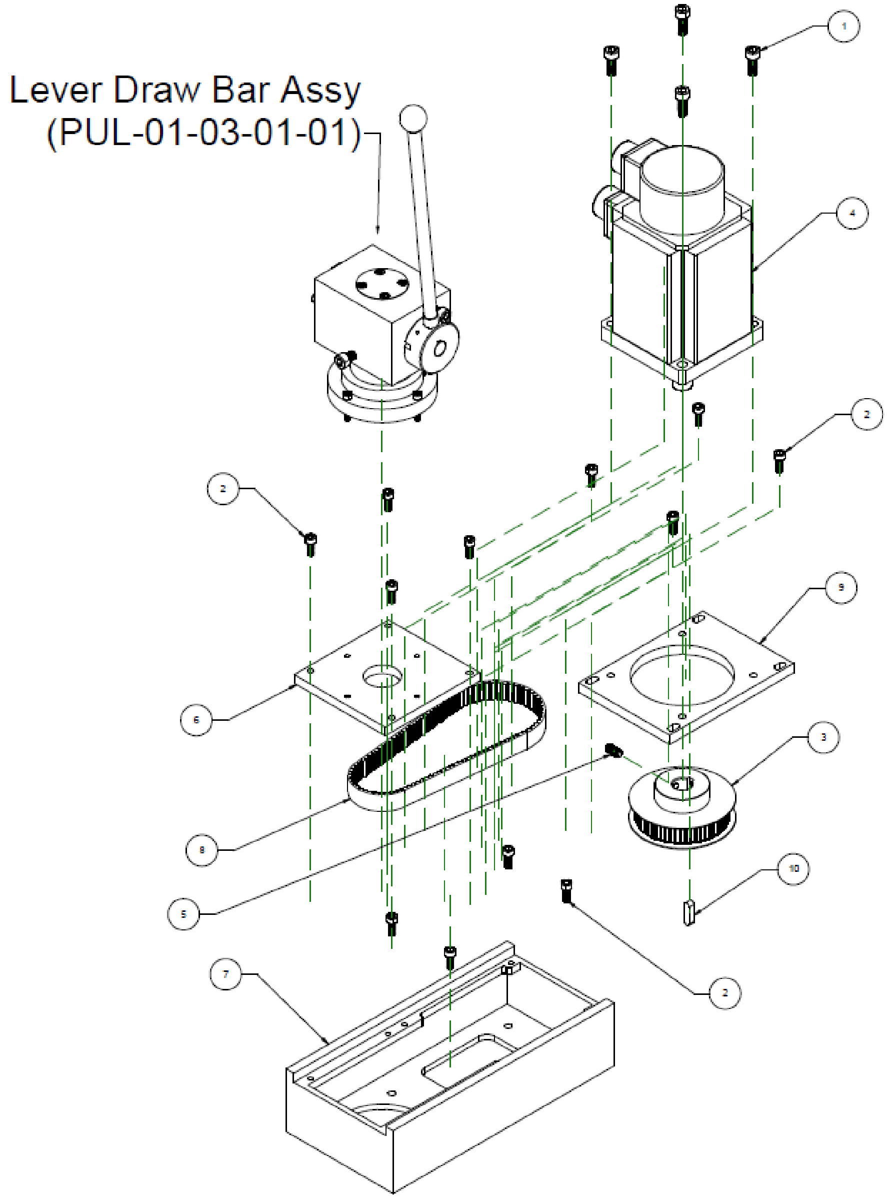
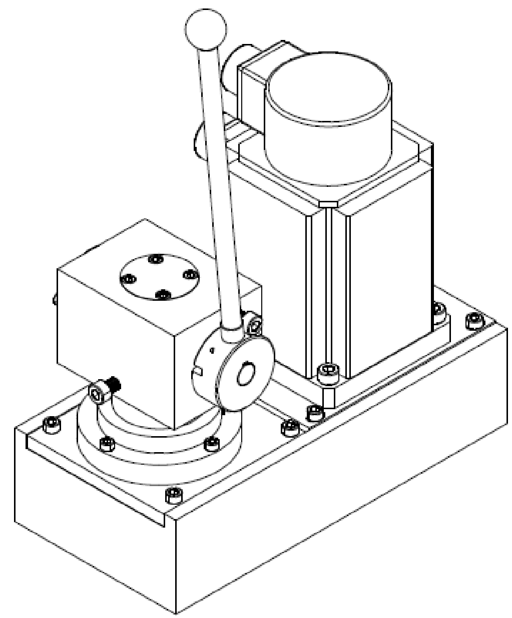


Figure 13-9 Head and Spindle Assembly

UPPER HEAD ASSEMBLY



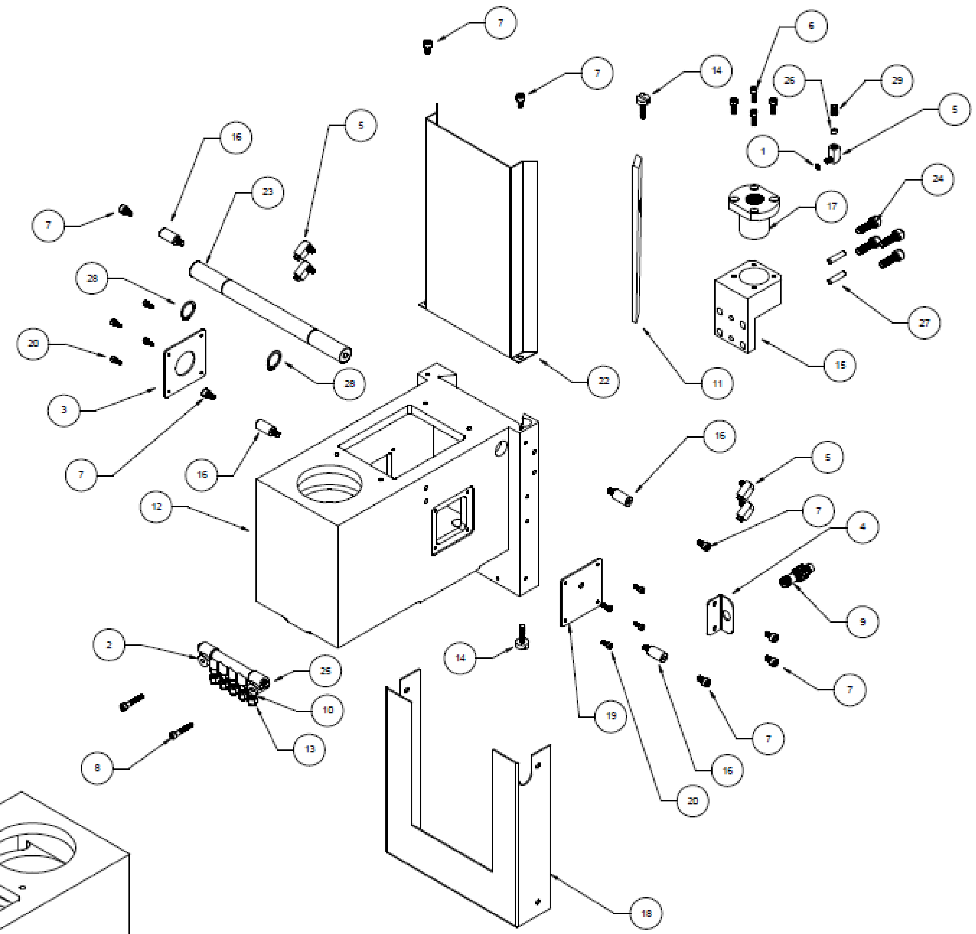
Item Number	Quantity	Part Number	Part Name	Revision	Comment
1	6		M8x20		
2	12		M6x15		
3	1	PUL-01-03-01-02	Servo Motor Pulley		
4	1	110SM04030	Servo Motor		1.2KW 3PH 3000RPM
5	1		Set Screw, M8x15		
6	1	PUL-01-03-01-03	Spindle cover Plate		
7	1	PUL-01-03-01-04	Servo Belt Guard		
8	1		Belt		
9	1	PUL-01-03-01-05	Servo Motor Slide		
10	1		Key,6x6x25		

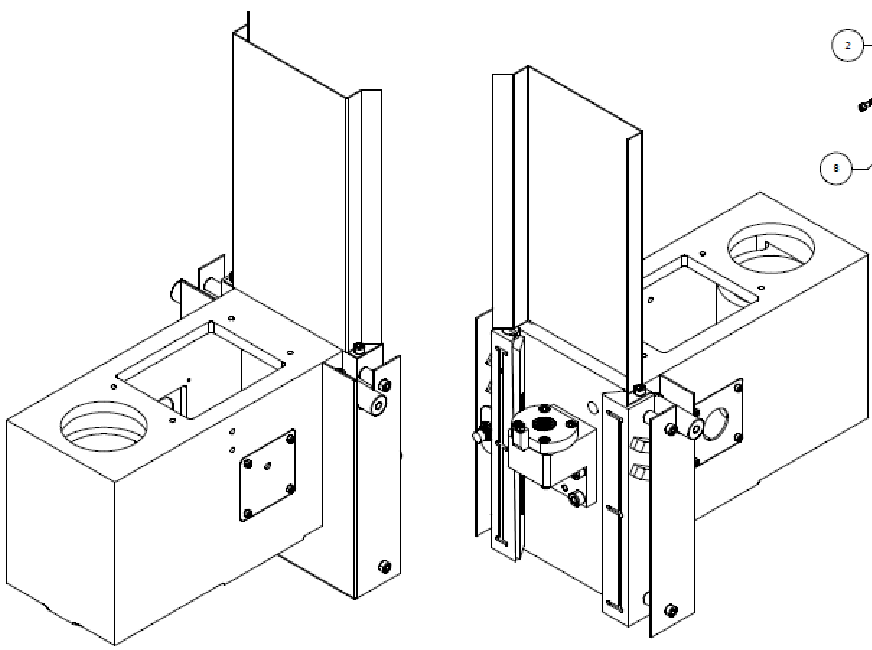


Property of: **Novakon International Corp** Phone (905) 258-0366 Novakon.net	Nomenclature: Upper Head Assy	
	End Item System: Pulsar	DATE: 18-Jan-2014
	Sub System:	REV: 0
	DWG NO.:	SHEET: 1 of 1

Figure 13-10 Upper Head Assembly

Item Number	Quantity	Part Number	Part Name	Revision	Comment
1	10		O-Ring		
2	1		Oil Manifold x7		
3	1	PUL-01-03-03-01	Head Side Plate		
4	1	PUL-01-03-03-02	Sensor Bracket		
5	5		Adaptor-Elbow		
6	4		M5x15		
7	8		M5x10		
8	2		M5x30		
9	1		Sensor NPN		
10	5		Relief Valve		
11	1	PUL-01-03-03-03	Z GIB		
12	1	PUL-01-03-03-04	Head		
13	5		Line Connector		
14	2		GIB Adjusting Screw		
15	1	PUL-01-03-03-05	Z-axis Ball Nut Mount		
16	4	PUL-01-03-03-06	Standoff		
17	1		Ball Nut		
18	1	PUL-01-03-03-07	Lower Bellow Trim		
19	1	PUL-01-03-03-08	Head Side Plate-Right Side		
20	8		M4x10		
22	1	PUL-01-03-03-10	Upper Z-axis Cover Plate		
23	1	PUL-01-03-03-11	Gas Spring Support		
24	4		M8x25		
25	1		Plug		
26	12		Furrel		
27	2		Pin, 6 x 25mm		
28	2		Snap Ring, 20 x 1.2mm		
29	6		Adaptor		

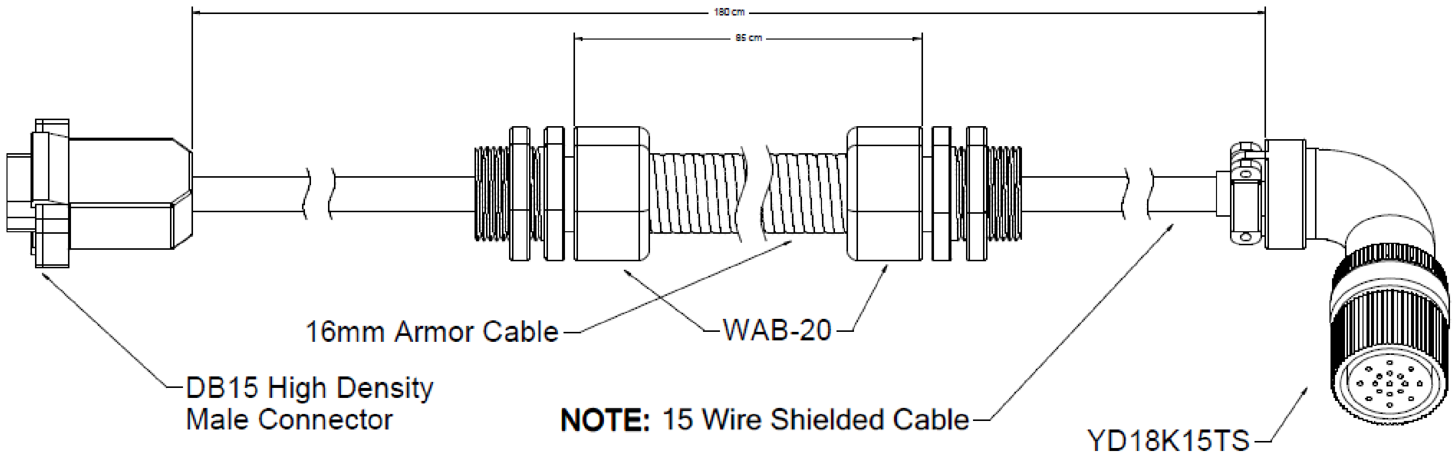




Property of:
Novakon International Corp
Phone (905) 258-0366
Novakon.net

Nomenclature		
Lower Head Assy		
End Item System: Pulsar	DATE: 18-Jan-2014	
Sub System	REV: 0	
DWG NO.	SHEET: 1 of 1	

Figure 13-11 Lower Head Assembly

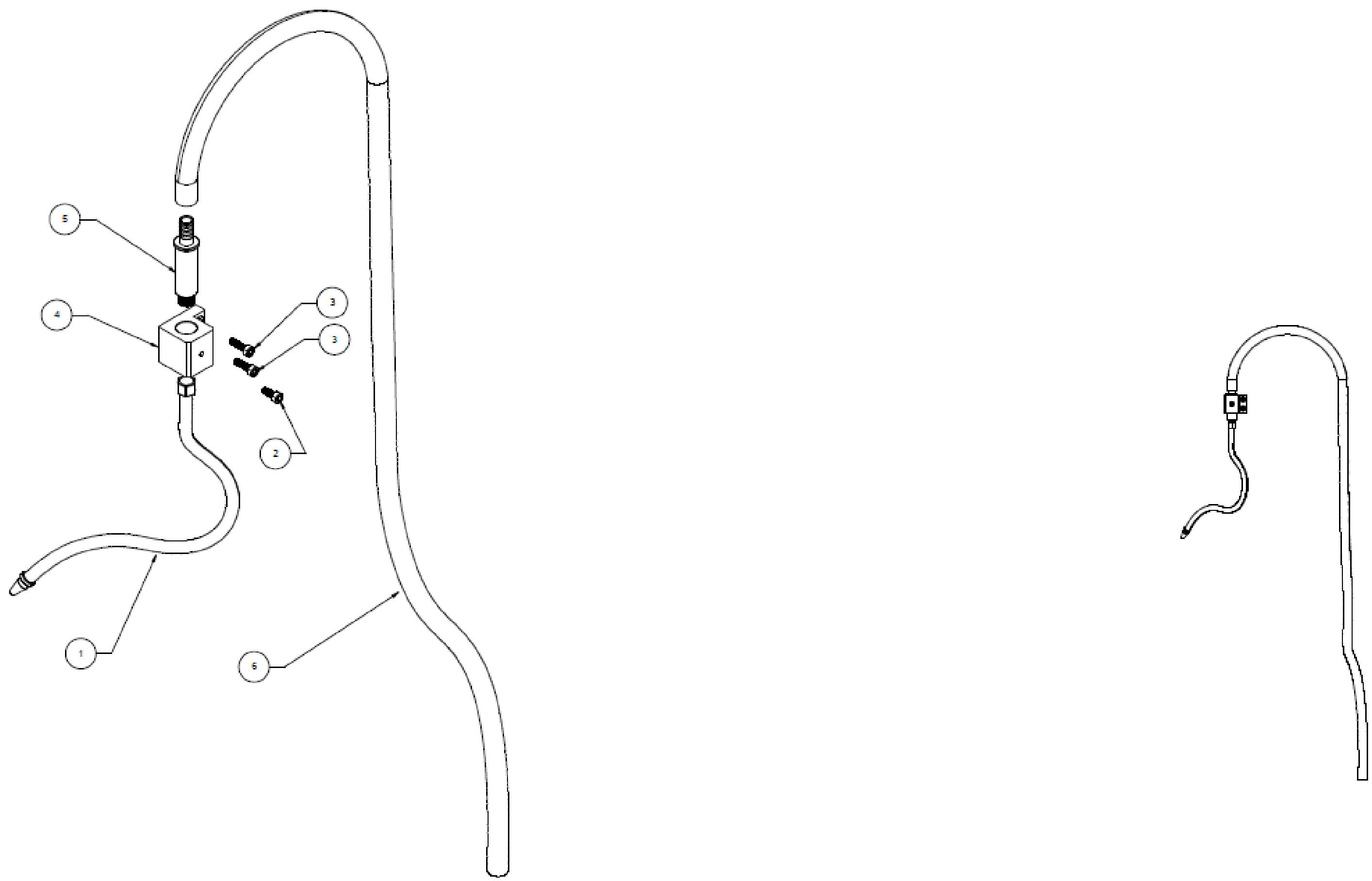


DB15	Wire Color	Motor Connector	Function
1	Orange	11	V+
2	Yellow	10	U+
3	Green	6	Z+
4	Blue	5	B+
5	Purple	4	A+
6	Orange/White	14	V-
7	Yellow/White	13	U-
8	Green/White	9	Z-
9	Blue/White	8	B-
10	Purple/White	7	A-
11	Brown/White	15	W-
12	Brown	12	W+
13	Red Red/White	2	5VDC
14	Black Black/White	3	Ground
15	Bare Wire	1	FG

Property of:
Novakon
International Corp
Phone (905) 258-0366
Novakon.net

Nomenclature		
Servo X-Axis Encoder Cable Assy		
End Item System: Pulsar	DATE: 6-Jan-2014	
Sub System:	REV: 1	
DWG NO.:	SHEET: 1 of 1	

Figure 13-12 Servo X-Axis Encoder Cable Assembly



Item Number	Quantity	Part Number	Part Name	Revision	Comment
1	1		Nozzle		
2	1		M6x15		
3	2		M6x20		
4	1	PUL-01-03-03-01	Coolant Adapter Base		
5	1	PUL-01-03-03-02	Coolant Adapter		
6	1		Hose		

Property of: **Novakon International Corp**	Nomenclature: Coolant Nozzle Assy	
Phone (905) 258-0366	End Item System: Pulsar	DATE: 29-Dec-2013
Novakon.net	Sub System: Coolant Nozzle	REV: 0
	DWG NO: PUL-01-03-03-INDEX	SHEET: 1 of 1

Figure 13-13 Coolant Nozzle Assembly