CENTRAL MACHINERY®

Mach3
Configured

Item:66052

CNC Mill User Manual

Mach3
Configured

KX1 CNC Mill
User Manual

Serial Number:		Register Number:	

Make a note of your Serial number and Register number here for future reference.

Disclaimer

1. This whole document with all it's sections is to be used as guidance only.
2. All comments, suggestions, operations, and instructions, stated herein are subject to change without notice.
3. Customers should make their own risk assessments and take appropriate measures to comply with health and safety regulations prevailing in their country.
4. We or our agents take no responsibility for loss, damage or injury caused:
   - by the use of this machine,
   - by inappropriate safety measures adopted by the user,
   - as a result of work carried out by unqualified person(s),
   - by issues generally considered to be Force Majeure (e.g. acts of God).

E. & O.E.

Contents

Part A: Quick Start Guide

Part C: Technical Guide

Mach3
Configured

CNC Milling Machines

Quick Start Guide

Congratulations on purchasing a Sieg KX Series CNC Mill!

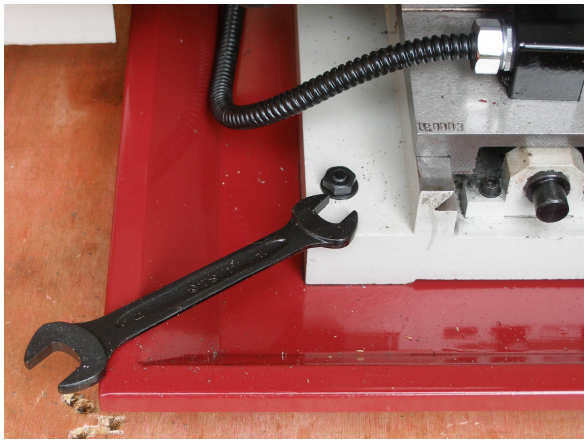
This quick start guide is designed to get you cutting as quickly as possible so please take some time at this stage to read through it before going any further.

We have tried to make the guide as "Beginner Friendly" as possible and even if you are already a Mach3 expert, we highly recommend you read through it at least once.

First, we'll deal briefly with the machine itself. Next we will move on to the Mach3 software installation on your computer. Finally, we will work through setting the machine up ready to make your first cut.

1. Unpacking and Setting Up Your Sieg CNC Milling Machine

1. The KX Series mill is bolted onto the base of the crate. There are four bolts, two at the front and two at the back. You can see the two front bolts either side of the main machine base in the photo above.

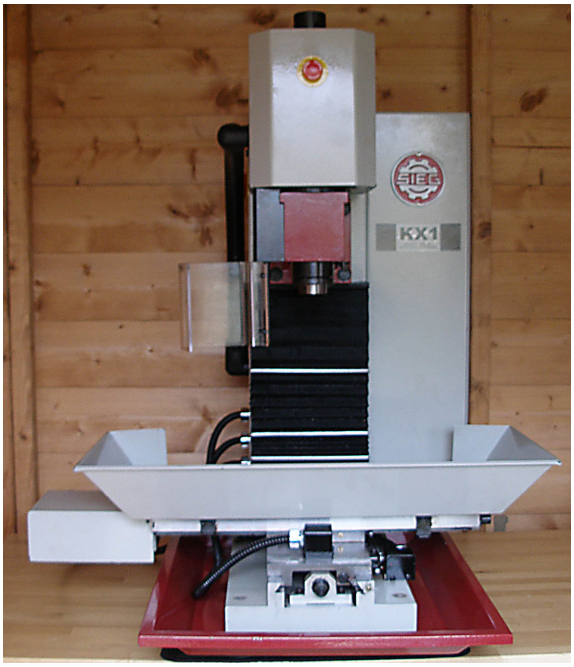


2. A close up of one of the fixing bolts.

3. There are four adjustable feet supplied

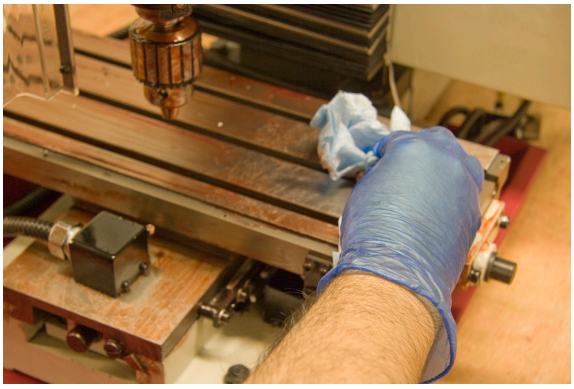
4. Fit the feet to an approximately equal height. The feet will be adjusted and finally tightened when the machine is on the bench. Tighten the nuts sufficiently to stop the tray falling off. The tray should fit into the holes in the bottom of the machine. When the machine has been lifted onto the bench, level the machine by using the adjustable feet and finally tighten the foot nuts against the bottom of the tray.

5. Place a rubber mat on the bench where the machine feet will go. The mat will stop the machine vibrating and moving around. The bench should be a minimum of 600mm deep if placed against the wall. You can of course pull the bench forward a bit so the front of the bench is 600mm from the wall.



6. Lift the machine on to the bench (the KX1 shown here is a two person lift) placing on the rubber mat.

7. Remove the four screws holding the guard so you can get at the machine to clean all the preservative off. Use WD40 or a similar product to remove the preservative. Then use SAE 30 hydraulic oil or equivalent to protect the surface. Safely dispose of any wipers used. Replace the guard after cleaning

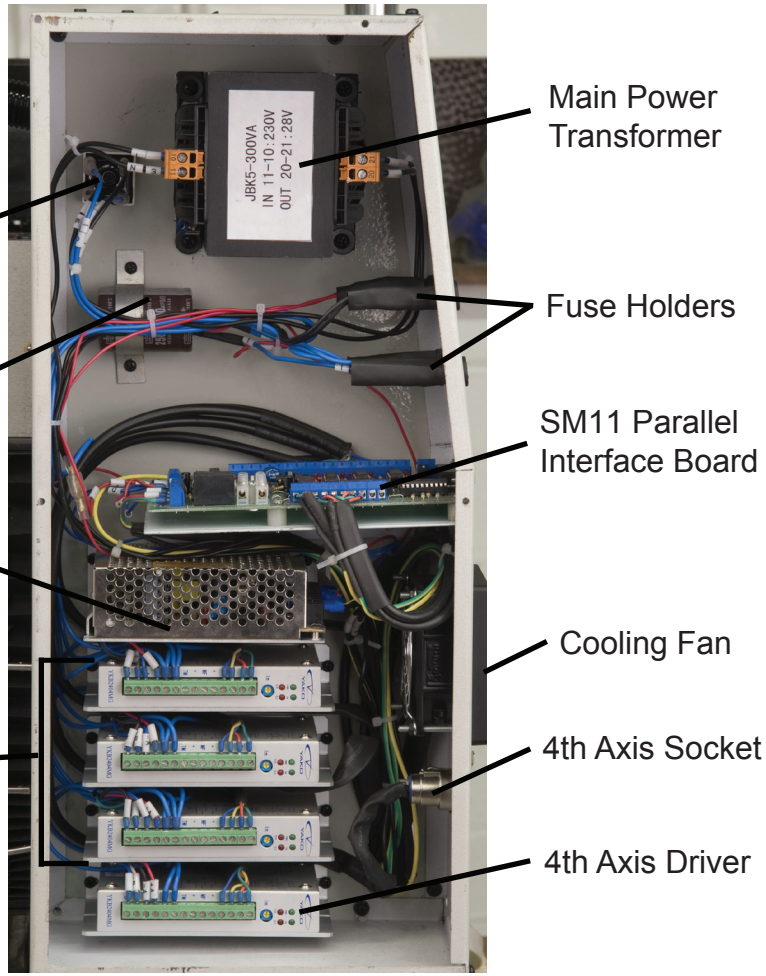


8. Use protective clothing when cleaning the preservative of the machine.

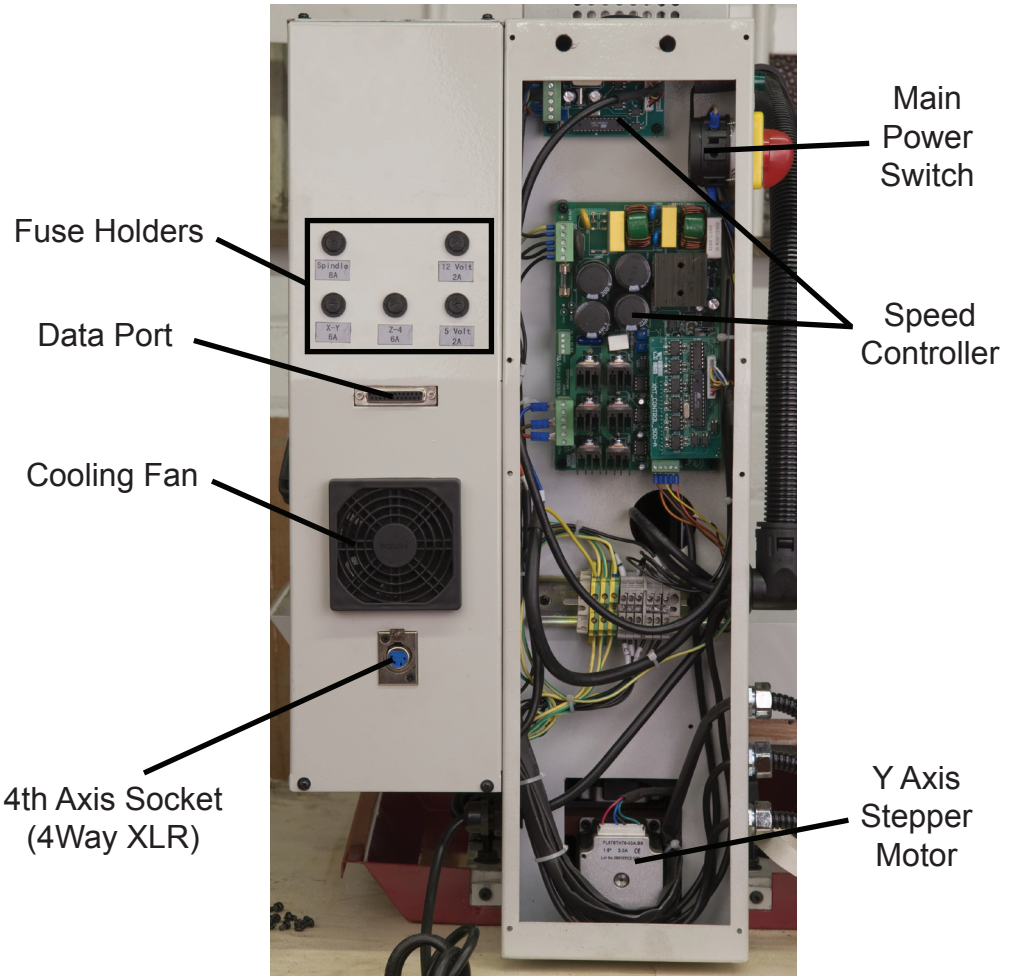
Although these pictures show the smaller KX1 CNC Mill, the same principles apply to the larger KX3 CNC Mill - except you will probably require a hoist to lift the heavier machine!

2. Electronics Layout

Right Side View



Rear View



3. Conventions used in this guide

To help make the software section of this Quick Start Guide easier to understand, keyboard input will be shown in a different way to screen input using a mouse or touch screen.

3.1 Keyboard Symbols:

Enter or Return key

Tab key

Cursor keys

Page Up and Page Down keys

Shift key Control Key Alt key

Escape key F5 Function key S character key

A passage of text might include: +; this means Shift+Left arrow or hold down the Shift key while pressing Left Arrow (cursor) key on the keyboard.

3.2 Mach3 Commands

3.2.1 Main Menu bar commands

You may be directed to <Config/Ports and Pins>. This indicates that you should open the Config menu at the top of the screen and click on the Ports and Pins option (which opens a dialogue box with various options)

3.2.2 Screens

Below the main menu bar, there is a row of 6 buttons which open different screens in Mach3. These will be indicated thus: **Program Run Alt-1** (Alt 1 indicates this screen can also be accessed using the +1 keys on the keyboard)

3.2.3 Buttons

Most of the Mach3 screen buttons have appropriate text on them so when you see ***Zero X***, or ***Set Tool Offset***, click the screen button with that name.

3.2.4 Data Entry Fields

Many of the data display buttons in Mach3 will also allow you to enter data from the keyboard. For example, if you are asked to enter +30.0000 into the ***Z-Axis DRO***, click on the Z-Axis Digital Read Out field on the screen, type in +30.0000 and press to accept the data. Notice how the background colour of the button changes when you click on it, letting you know it's ready for you to type some data in and changes back again when you press .

4. Install Mach3 now

Installing the Mach3 software from the CD or the Small CNC Support website will include:
Configuration files tailored specifically to Sieg KX Series machines.
Customised versions of the Mach3 screens for your machine.

You do not need the CNC mill connected yet. If you are just starting it would be better not to have it connected. You can install Mach3 from the CD or from a later version you downloaded from the Small CNC Support website.

4.1 Run Setup.exe

Run the **Mach3_Setup.exe** file now and you will be guided through the usual installation steps for a Windows program such as accepting the license conditions and selecting the folder for Mach3. On the Setup Finished dialog you should ensure that **Load Mach3 Driver** and **Install English Wizards** and also **Initialise System** are checked in the dialog, and click Finish. You will now be asked to reboot before running any Mach3 software.

4.2 The vital re-boot

This reboot is vital. If you do not do it then you will get into great difficulties which can only be overcome by using the Windows Control Panel to uninstall the driver manually. So please reboot now.

4.3 Which desktop icon??

So you have rebooted! The installation wizard will have created several desktop icons for the main programs. **Mach3 Loader** is the **only** one we are interested in since it will ask which machine Profile you wish to use. KX1 Mill and KX3 Mill are shortcuts which run a defined Profile specific for those machines.

4.4 Now test the system

It is now highly recommended to test the system. Mach3 is not a simple program. It takes great liberties with Windows in order to perform its job; this means it will not work on all systems due to many factors. For example, the QuickTime system monitor (qtask.exe) running in the background can kill it and there will be other programs which you probably are not even aware are on your system that can do the same. Windows can and does start many processes in the background; some appear as icons in the system tray (bottom right of screen) and others do not show themselves in any way.

Other possible sources of erratic operation are local area network connections and it is advised to disable these. It cannot be stressed enough that you use a dedicated machine to control this Sieg product.

It is now worthwhile to run the DriverTest.exe file. Use Windows Explorer (right-click Start) and by right-clicking on the DriverTest.exe file you can drag this shortcut onto your desktop. You can find the DriverTest.exe file in the Mach3 directory. Double click the Driver Test icon that you set up. Its screen shot is in Fig 1.

You can ignore all the boxes with the exception of the Pulse Frequency. It should be fairly steady around 25,000 Hz but yours may vary, even quite wildly. This is because Mach3 uses the Windows clock to calibrate its pulse timer and, over a short time scale, the Windows clock

can be affected by other processes running on the computer. So you may actually be using an "unreliable" clock (the Windows one) to check Mach3 and so get the false impression that Mach3's timer is unsteady.

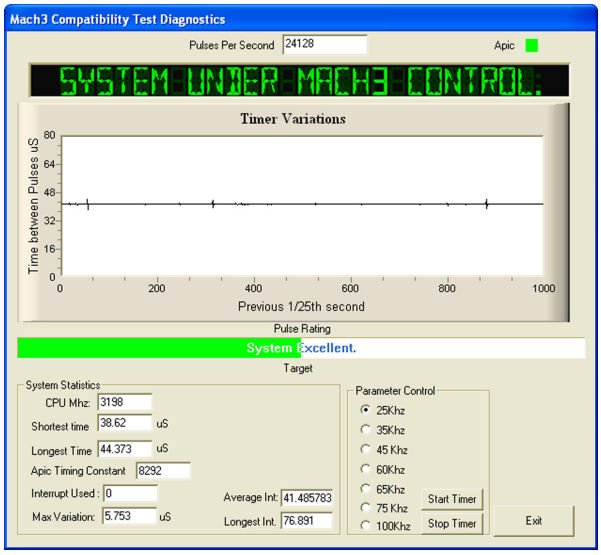


Fig. 1

Basically, if you see a similar screen to Fig. 1 with only small spikes on the Timer Variations graph and a steady Pulse Frequency, everything is working well so close the Driver Test program and skip to the section below.

You may have one of two things happen to you when running the test which may indicate a problem.

1. "Driver not found or installed, contact Art.", this means that the driver is not loaded into Windows for some reason. This can occur on XP systems which have a corruption of their driver database. Reloading Windows is the cure in this case. Or, you may be running Win2000. Win2000 has a bug/feature which may interfere with loading the driver. It may need to be loaded manually - see section 3.2.5 in the Mach3 User Guide.
2. When the system says, taking over…3…2…1.. and then reboots, one of two things has occurred. Either you didn't reboot when asked (told you!!) or the driver is corrupted or unable to be used in your system. In this case refer to the reference manual and remove the driver manually, then re-install. If the same thing happens, please notify ArtSoft using the e-mail link on www.artofcnc.ca and you will be given guidance.

A few systems have motherboards which have hardware for the APIC timer but whose BIOS code does not use it. This will confuse Mach3 install. A batch file SpecialDriver.bat is available in the Mach3 installation folder. Find it with Windows Explorer and double-click it to run it. This will make the Mach3 driver use the older i8529 interrupt controller. You will need to repeat this process whenever you download an upgraded version of Mach3 as installing the new version will replace the special driver. The file OriginalDriver.bat reverses this change. This is a rare occurrence these days with the new motherboards but it may be necessary to use this driver.

4.5 Registering the Mach3 software

The Mach3 software supplied on the CD is a demo version. It is a fully functioning program except that it's limited to 500 lines of G-code and some of the wizards are disabled. It is recommended that you purchase a Mach3 software licence via the link on:

http://www.smallcncsupport.com/

5. Connecting and Starting the KX Series CNC Mill

5.1 Connecting the Computer

Now that you are happy the driver is running correctly, it's time to connect the computer lead (supplied with your KX CNC mill) to the 25 way D connector on the back of the KX CNC Mill, see Fig. 2.

Fig. 2

Connect the other end of the lead to the printer connector on your PC. Do not connect through a printer, a scanner or other peripheral. Mach3 requires a dedicated parallel port.

5.2 Switching on

1. Switch the computer on first and allow to boot up.
2. Start Mach3: Open the **Mach3 Loader** program on the desktop and select either the KX1 or KX3 mill profile (to suit your machine), and click the OK button, Fig. 3. If you installed Mach3 from the CD or the smallcncsupport website, these profiles will be present by default.

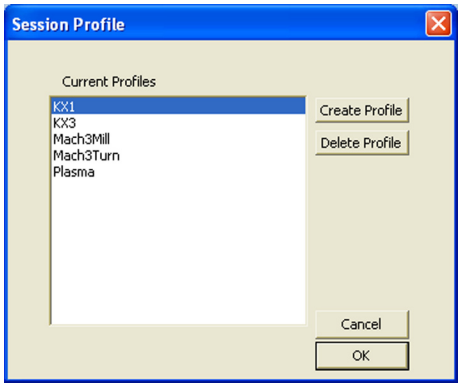


Fig. 3

Switch the KX mill on. This will now load the correct custom screen for your machine.

5.3 Reset the machine

Press the red ***Reset*** button, Fig. 4. The message "Press Reset ..... Emergency Mode Active" should clear.

If not, check that the emergency stop button has been released and chuck guard is closed and try again.

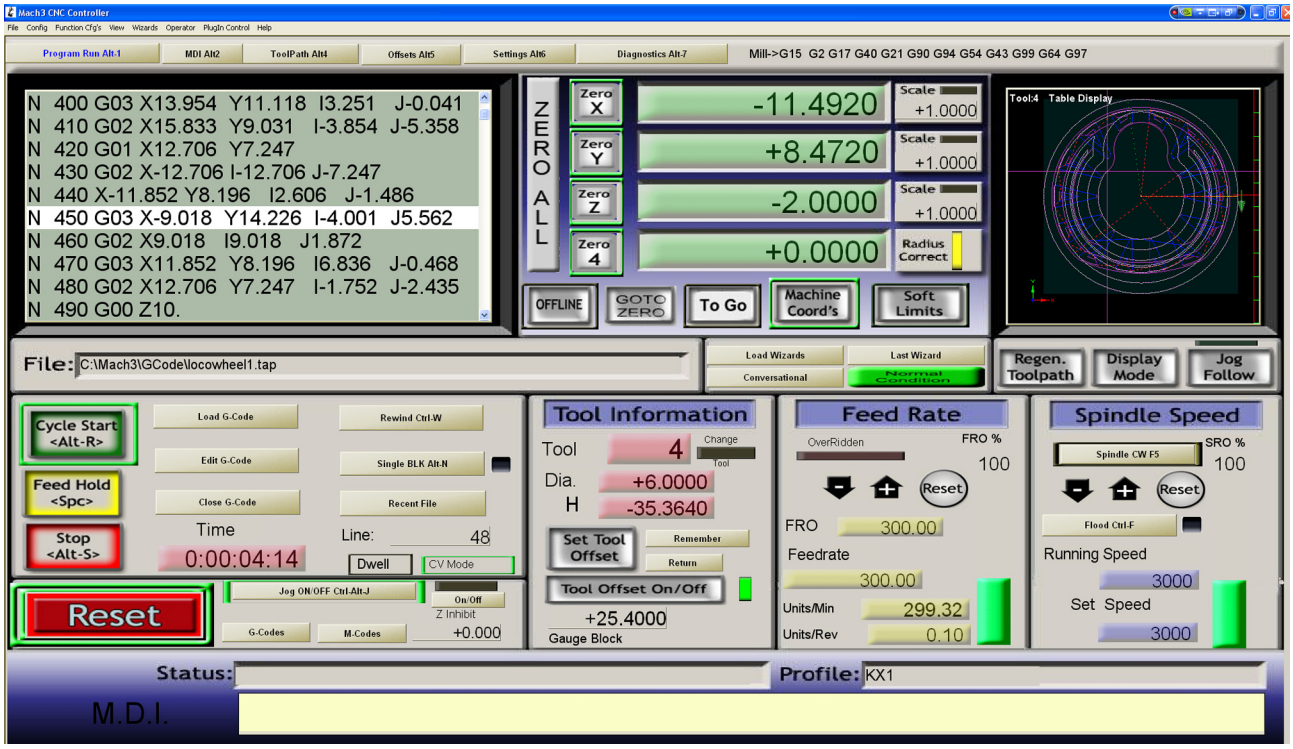
Fig. 4

6. Mach3 Screen Functions

Below is a screen shot of the main Mach3 front screen (shown with a file running) - known as the **Program Run Alt-1** screen. This is the correct custom screen you should now see (subject to updates).

Most of the information needed to setup and run appears on this screen. It may look complex but as it is designed in modules, the commands are actually grouped into separate function blocks known as Control Families.

Each control family will be taken in turn and explained in detail.



There are seven main families and two minor ones.

Main ones are:

Reading from left to right, top row:

- The G code window
- The axis Digital Readouts [DRO's]
- The toolpath screen.

Bottom row:

- The file loading and run commands
- All the tool information and settings
- Feed rates
- Spindle controls.

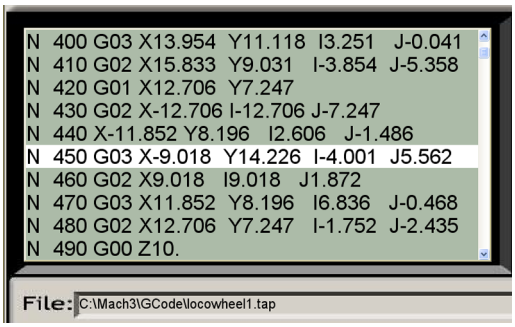
Minor families:

- All screen and wizard command buttons
- The MDI commands.

These families will be covered by separate larger screen shots and full explanations of each feature.

6.1 The G Code Window

This is the easiest family to understand - when a file is loaded it's contents appear in this window.

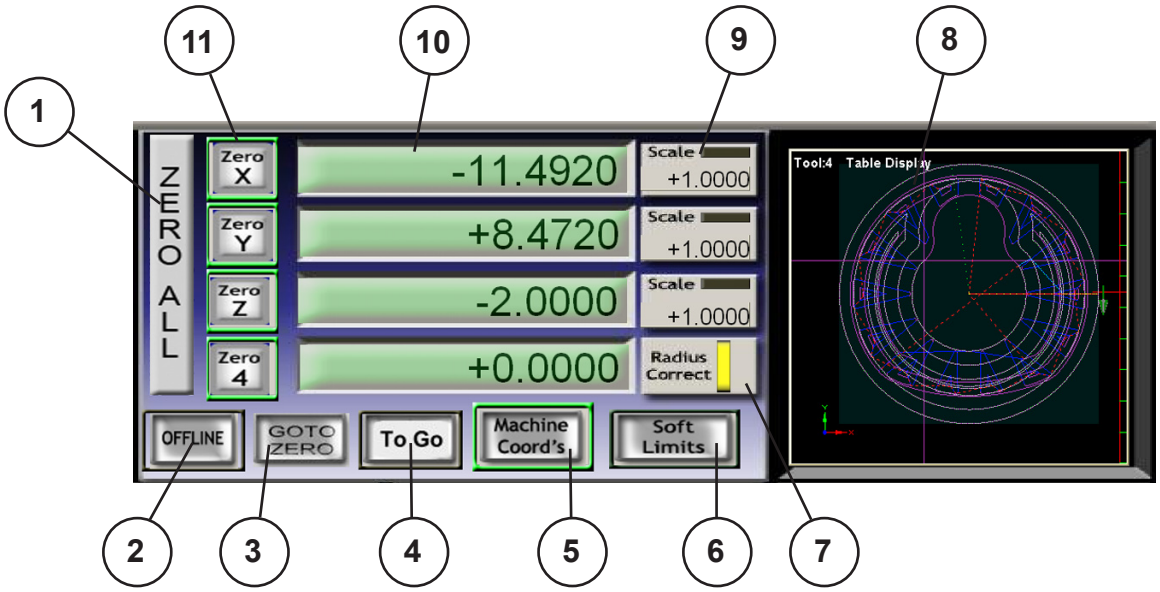


Mach3 will scroll through the file looking for any errors and then return to the beginning.

Any errors will show in the Status bar at the bottom of the program run screen.

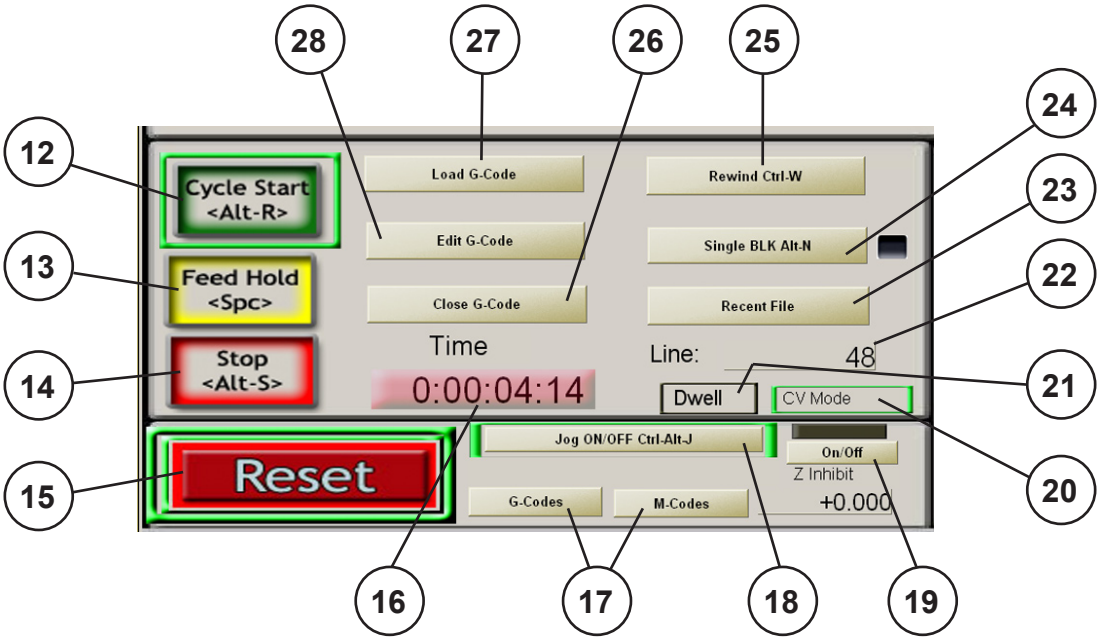
The file bar below the G Code window always displays the loaded file and it's location

6.2 The DRO's and Toolpath



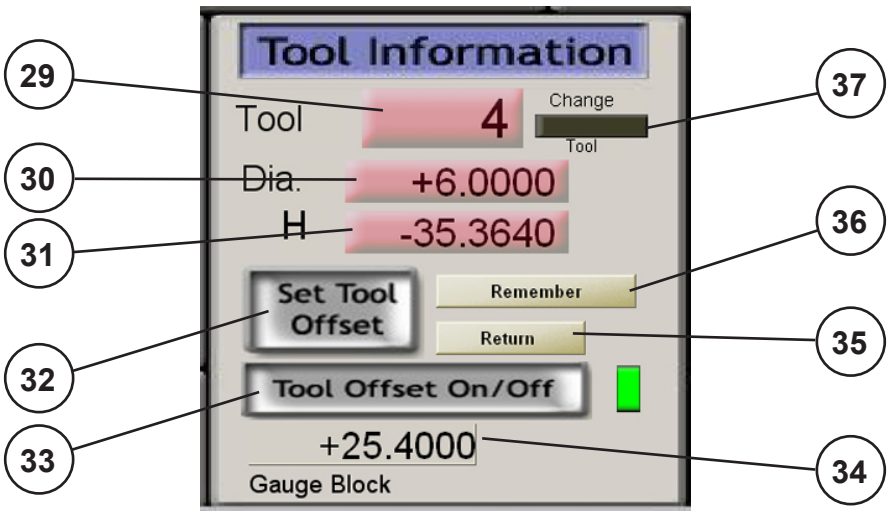
1	Zero All Axis	Zero's all the axis together when pressed.
2	Offline	Disables the parallel port to allow a file to run without the machine running.
3	Goto Zero	Returns the machine to its Zero Origin position.
4	To Go	Full title is - Distance to go - and is an inverse of what the DRO's state.
5	Machine Coord's	Switches between Machine Coords and Work Coords.
6	Soft Limits	Soft limits are a software version of limit switches and can be set under <Config/Homing/Limits>.
7	Radius Correction	For setting the radius of the work on the 4th axis so that the feed rate is correct. The actual Setting Box is on the **Settings Alt6** page.
8	Tool Path window	Complex, so see description of the Screen and Wizard Command Module.
9	Scale Window	Normally set at 1.00 but any value can be entered. If 0.5 is typed in to X and Y then the resulting file will be half size but full depth in the Z.
10	DRO's	Shows the values of each axis. You can also manually enter values into these.
11	Zero Buttons	One for each axis to zero the display.

6.3 Run and File Functions



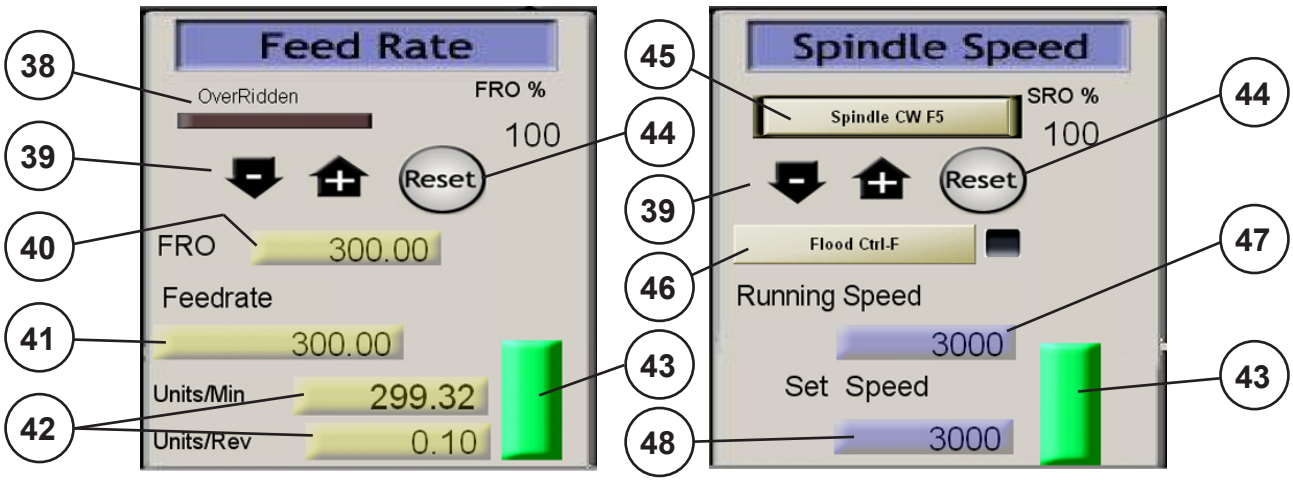
12	Cycle Start	Start button to run program, also obtained from Alt + R on the keyboard.
13	Feed Hold	Enables the program to be paused until Cycle Start is pressed again, shortcut key is the spacebar.
14	Stop Button	Stops the program, shortcut key is Alt + S.
15	Reset	Used to reset the machine at startup or from an emergency stop. Shortcut key is Esc.
16	Elapsed Time	Timer for program run time.
17	G & M Codes	Opens a new screen to list accepted G and M codes understood by Mach3.
18	Jog Control	When lit, allows the arrow keys ← ↑ ↓ → and page up / down Page Up Page Down to be used as jog keys.
19	Z Inhibit	Type a positive number in the DRO box and switch On, then when the code is run there will be no Z moves lower than the number input.
20	CV Mode	When lit, by default, Mach3 looks-ahead in the code to smooth out the toolpath.
21	Dwell	Lights up when a dwell time [G04] is called by the program.
22	Line	Displays the current line number as it runs through the code.
23	Recent File	Brings up a list of the most recent files used.
24	Single BLK	Single block or line, when active it runs through the code one line at a time using Cycle Start.
25	Rewind	Rewinds the code back to the start point.
26	Close File	Closes the current file. Opening a new file will do the same and over write, not Append.
27	Load G Code	Opens a new file.
28	Edit G Code	Opens the current file up in Notepad for editing, also updates the file on saving and closing.

6.4 Tool Information



29	Tool Number	Lists the tool number when a program is running. It is also used to enter a tool number when setting the tool height offset as described later in this Quick Start guide.
30	Tool Diameter	Lists the diameter either entered directly or from the tool table.
31	Tool Height	Lists the tool height offset from the tool table.
32	Set Tool Offset	Press to enter the tool offset into the tool table when the tool is at the correct position, again described later in this Quick Start guide.
33	Tool Offset	Toggle on / off to switch on the tool offsets in the Z DRO.
34	Gauge Block	Enter the height of a gauge block or setting block if used. This will automatically added to the tool offset value.
35	Remember	Click on this function before moving off position for any reason like changing a broken tool and it will remember the position.
36	Return	Brings up a menu to return to the default position or you can type in an alternative.
37	Change Tool	An LED that lights up when it's expecting a tool change. Once changed, press ***Cycle Start*** to continue.

6.5 Feed Rate & Spindle Speed



38	OverRidden	Flashes when the Feed Rate is greater than or less than 100%.
39	+ & - Signs	Allows incremental override steps.
40	FRO	Feed Rate OverRide, shows the true feed rate.
41	Feedrate	Shows the programmed feed rate.
42	Units	Two DRO's to show Units /Min and Units / Rev.
43	Slider	Feed Rate Over Ride slider control. Drag with mouse or touch screen to alter feed rate.
44	Reset	Returns over ride rate back to 100%.
45	Spindle	Switches the spindle on / off either by clicking or using the F5 hot key.
46	Flood	Switches coolant on (if fitted)
47	Running Speed	Displays the true spindle speed.
48	Set Speed	Shows the programmed spindle speed and also a speed can be entered here.

6.6 MDI Function

The Status bar shows any errors present in Mach3 . The Profile shows what machine it is setup for.

The MDI box which stands for Manual Data Input is a simple way to make manual moves one line at a time.

Using this function it is possible to enter one line of code to make the machine perform a move or Function. You must have knowledge of G-code to use this or damage to the machine can occur.

Use of this function is described in section 3.5 in the Mach3 User Guide.

6.7 Wizards and Screen Display

Load Wizards will open up a menu of Wizards that will perform most of the simple day to day operations done by CNC machines without having to write the code for them. Wizards are Mach3 terminology for what industry term conversational programming.

Most of the Wizards are free to use but some produced by Newfangled Solutions require an extra licence to generate code. These can also be easily accessed by the ***Conversational*** button.

Last Wizard will return you to the last one used and the green ***Normal Condition*** is just a system ready light.

Wizards are explained in more detail in section 3.6 in the Mach3 User Guide.

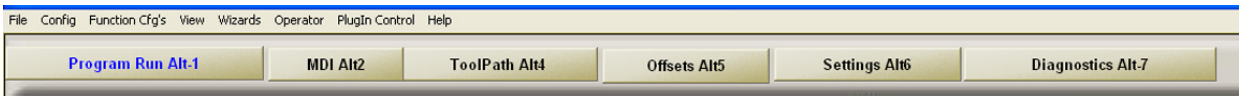
Regen. Toolpath is used when the code has been updated to force a screen redraw.

Display Mode toggles between the max size of the travels and the size of the job.

Jog Follow moves the screen to a central tool instead of the tool moving on the work, more suitable to fixed head machines.

Clicking on the toolpath screen whilst the job isn't running and holding the left mouse button down enables the screen to be rotated in 3D. The mouse wheel allows zoom in and zoom out.

6.8 Main Menu and Mach3 Screens Buttons



Across the top of the screen is the normal Windows type file layout. All these are covered in the Mach3 User Guide. The lower Screen buttons are the different screens that can be accessed in Mach3.

The **Program Run Alt1** screen is the main screen and the one we have covered here.

MDI Alt2 screen contains a duplicate MDI screen and a teach facility [see section 3.5.2]

Toolpath Alt4 screen contains a larger toolpath window for ease of use and also a set of DRO's labelled Program Limits that tell you after a file has been loaded how big the travels are in all 6 directions. It's advised you check these to make sure the job will fit the machine travels.

Offsets Alt5 screen contains all the functions to setup your work as regards the Origin.

The **Settings Alt6** page contains over ride limits so you can jog off if you hit a limit switch and the radius correction DRO for the 4th axis. The rest is more for fault finding.

The **Diagnostics Alt7** page is purely for fault finding if anything goes wrong and there will be a simple check guide on the smallcncsupport website for registered users to assist in checking machines.

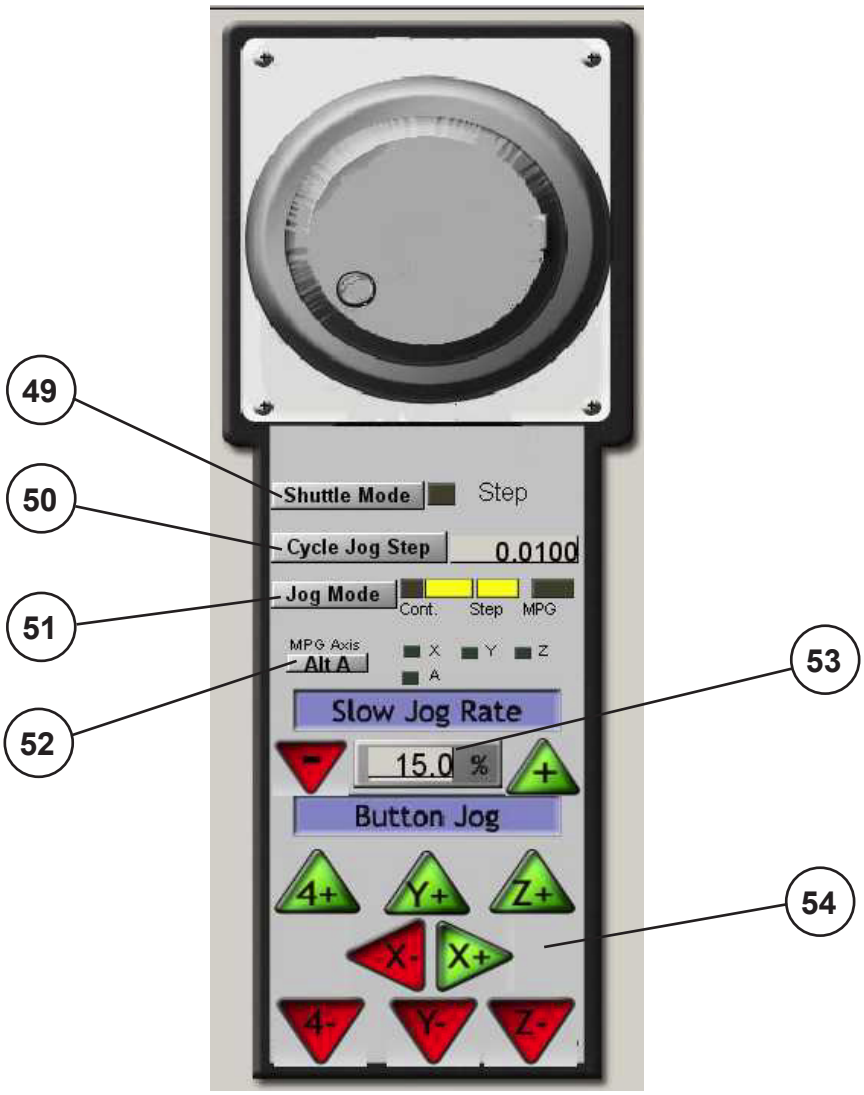
This is only a simple quick start guide and it is strongly advised you read the main Mach3 User Guide where all of what has been described here is cover in far greater depth.

6.9 Jog screen

The jog screen is available when the tab key is pressed and shows to the right of the main screen. This screen sets the feed rate for jogging as a percent of the maximum speed as set in <Config/Motor Tuning>.

In normal use, the machine, when jogged, moves at this slow jog rate.

If any jog key is pressed whilst the shift key is held then it will move at maximum speed.



49	Shuttle Mode	May be used if a Contour Design USB Shuttle is used as a handwheel.
50	Cycle Jog Step	Sets the step increments in units of 1.0000, 0.1000, 0.0100, 0.0010, 0.0001 and then back to 1.0000
51	Jog Mode	Toggles between Continuous and Step. Cont. moves the selected axis continuously whilst a jog key is pressed and Step allows jogging moves to be in steps of whatever units are selected in 50.
52	MPG Axis	Provides a visual check on what axis is selected when 49 is in operation.
53	Slow Jog Rate	Displays / sets slow jog speed described above. This can be altered with the arrow keys to either side or it can be typed in.
54	Jog Buttons	Click to jog an axis. The 4+ and 4- arrow buttons are for jogging the 4 axis.

7. Coordinates

Machining coordinates are normally entered into a program as X Y Z e.g. X1.0 Y-10.0 Z25.0, but what does this mean?

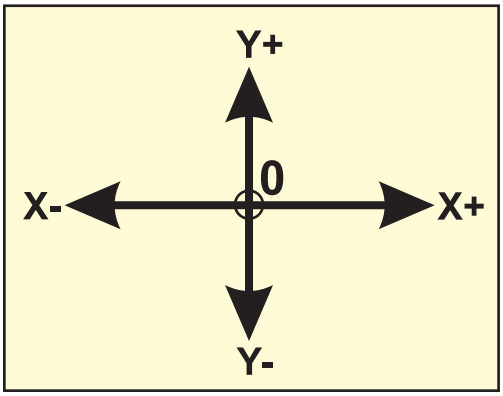


Fig. 5

Imagine you are looking down on the top of the workpiece in a vice on the table of the machine - see Fig. 5. The point where the two lines cross is known as the origin, or X0.0 Y0.0.

If the tool moves to the left of the origin, the X coordinate will be negative. If the tool moves to the right of the origin, X will be positive.

If the tool moves from the origin towards the back of the machine (Up in the diagram) then Y will be positive and conversely, Y will be negative of it moves towards the front.

It will be helpful later on if you begin by thinking of the ***tool moving*** in the X and Y directions even though it's the table that will be moving and not the tool!

The Z zero position is usually on the top surface of the component to be milled. Any movement above the job is in the Z+ direction and any movement below the surface of the job is in the Z- direction.

7.1 Check the Axis Direction

With a new machine, it's quite possible the axes of the machine will be moving in the wrong direction since the motor direction is difficult to determine at the time the machine is built. If the direction of any axis is reversed, it's a simple job to change. Proceed as follows to check and, if necessary, reverse the axis directions:

- Press the Tab key on the keyboard and the **Jog** Flyout will appear on the right of the screen.
- Use the jog buttons ***X-***, ***X+***, ***Y-***, ***Y+***, ***Z-*** and ***Z+*** at the bottom of the flyout to move the table and head. The table should move as follows: ***X-*** = right, ***X+*** = left, ***Y-*** = in (towards the column) and ***Y+*** = out. The head should move: ***Z-*** = down and ***Z+*** = up. If all axes are moving the in right directions, there is nothing to change and you can skip to the next section. If one or more axis is moving the wrong way, continue to the next step.
- On the Mach3 menu select <Config/Ports and Pins> and select the ***Motor Outputs*** tab at the top of the window - see Fig 6.

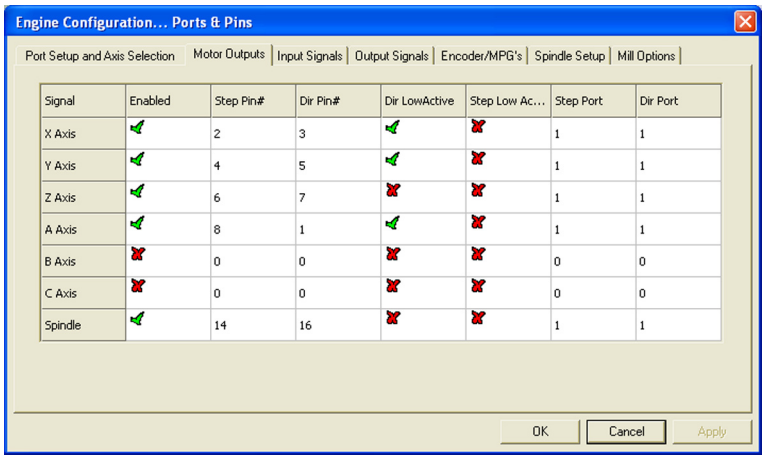


Fig. 6

- In the column marked **Dir LowActive** change the tick to a cross (or vice versa) on the axis concerned to reverse the motor direction.
- Exit and restart Mach3 and check that the alteration has taken place.

Once all three axis are moving in the right direction, the machine is ready for use.

7.2 Limit Switches

Your KX Series CNC Mill is fitted with limit switches on all 3 axes. The purpose of these switches is to prevent each axis from travelling beyond the machine's safe limits, possibly causing damage to the machine. When a limit switch is triggered, the following happens:

- The machine stops moving
- The ***Reset*** button flashes red and green
- There will be a scrolling message reading "Press Reset - Emergency mode active"
- The ***Status*** line reads "Limit switch triggered"

Pressing ***Reset*** allows you to over-ride and jog off the limit switch. If you have driven a long way onto the limit switch, you may need to press ***Reset*** again before you can jog off the switch.

8. Setting a datum

You will need to find the exact position of two edges (or a centre) of the component you wish to machine to act as a datum (otherwise known as the Origin. There are various ways to find this; the most popular is an edge finder or wiggler. A wiggler consists of a ball jointed probe that when running spins the tip in a circle - Photo 1. As the tip touches the work the circle gradually reduces as it gets closer until at one point there is no movement then immediately after this the tip rolls along the edge of the work and 'flicks' out. This is the contact point of the edge of the wiggler and the work.

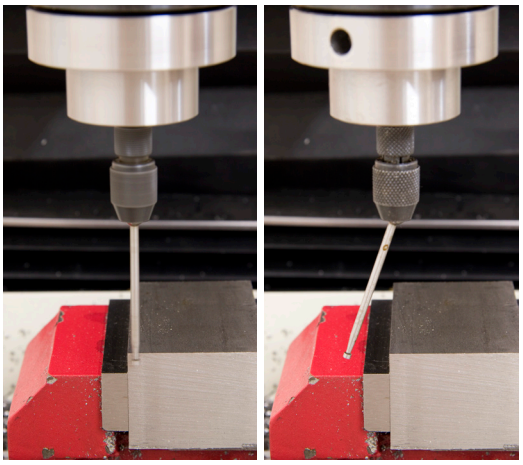


Photo 1 - A Wiggler in action

Wigglers are usually run at about 600 RPM but usually perform well over quite a large speed range.

Fit the wiggler into a holder - a standard end mill holder where the tool is held in by a grub screw is recommended. Next, the tool holder needs to be fitted to the spindle. Screw the tool into the drawbar and tighten with a spanner at the top and bottom of the spindle.

The drawbar of the KX1 CNC Mill is self extracting - Photo 2. This means there is a collar at the top of the spindle that stops the drawbar from being removed from the machine. When you undo the drawbar, it pushes against the collar and ejects the tool holder from the MT2 taper.

Photo 2 - The KX1 Drawbar

The KX3 CNC Mill is fitted with a standard drawbar which needs to be slackened a few threads and tapped to release the R8 or MT3 taper.

If you are using the metric system, it is recommended that you use an edge finder with a 6mm end and if using imperial, use a 0.200in end.

8.1 Zeroing the machine

Press the ***Zero X***, ***Zero Y*** and ***Zero Z*** buttons to the left of the machine DRO's (Fig. 7), this will set all the axes to read 0.0000.

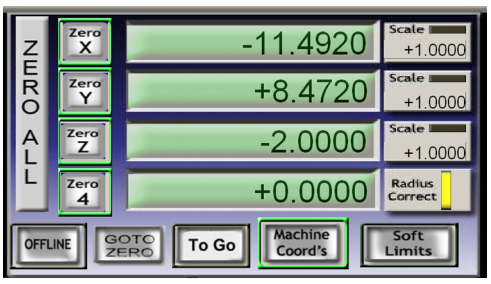


Fig 7

8.2 Using the jogging control

Press the Tab key on the keyboard and the **Jog** Flyout will appear on the right of the screen, Fig 8. First we'll zero the Y axis on the front of the work. Use the jog buttons ***X-***, ***X+***, ***Y-***, ***Y+***, ***Z-*** and ***Z+*** at the bottom of the flyout to move the wiggler close to the front of the work.

Fig 8

Remember:
When jogging, work as if the tool direction is moving, not the table!
If you forget, the table will move in the wrong direction!

Start spindle: type 600 into the ***set speed*** box on the **Program Run Alt-1** screen and press ↵ on the keyboard to accept the entry. The spindle will only start when you click on the ***Spindle CW*** button or press F5 on the keyboard. (To stop the spindle repeat the same command.) The end of the wiggler should now be running in a slightly eccentric circle.

Click the ***Y+*** button to jog towards the edge of the work.

HINT:
You can also jog using the keyboard:
← = X- and → = X+
↑ = Y+ and ↓ = Y-
Page Up = Z+ and Page Down = Z-
Holding the ⇧ key + one of the above will jog the axis at high speed.
Holding the Ctrl key + one of the above will move the axis a by the *Step* distance entered in the Jog flyout.

You can alter the jogging speed (feed) by clicking the ***Slow Jog Rate*** buttons on the flyout. You can change from continuous movement (***Cont.***) to single step (***Step***) by clicking on the ***Jog Mode*** button and change the single step distance by clicking the ***Cycle Jog Step*** button or entering a distance directly into the ***Step*** field. Jog until the wobbler flicks out.

Press F5 to stop the spindle.

At this point, click in the ***Y axis DRO*** and type in the radius of the wiggler. Since the wiggler is in front of the work, enter the wiggler **Radius** as a **Negative** value and press ↵ to accept this value. (If you were using the wiggler at the rear of the work, you would enter a positive value)

Next we'll find zero for the X axis using the left side of the work. Start the spindle again and jog to the edge of the work until the wiggler flicks out and enter the radius of the wiggler in the ***X axis DRO*** and press ↵. Since the wiggler is to the left of the work, this value must also be **Negative**.

Stop the spindle and raise the wiggler above the work and remove it from the spindle. Press ***Zero Z*** at this point so the ***Z axis DRO*** reads 0.0000 above the work. Now press the ***Goto Zero*** button. This sets the spindle above the work and central to the bottom left corner you set up to. At this point we have now set the X, Y, and Z axis clear of the work.

This is known as the **Work Offset** and relates to the work offset being equal to the **Work Zero** or **Origin Point**.

The origin point is a very important point when you are drawing a part to machine; in short it's the start point.

There are another set of offsets called Machine Offsets or coordinates. These are for more experienced users who need to do multiple parts from one point. They are described in the main Mach3 User Guide but we are not going to deal with these in this Quick Start Guide but will make them equal the Work Offsets so there is no conflict.

This will be done after the tool height offsets have been set.

8.3 Setting the tool height offsets

Using the Page Up and Page Down keys on the keyboard or the ***Z+*** and ***Z-*** jogging keys on the ***Jog*** flyout, raise the spindle above the work a distance greater than the longest tool you plan to use. Clearance is needed between the bottom of the tool and the work to miss any clamps etc. This is called the **Clearance Height** or **Clearance Plane**. A good figure here is 30mm as this should be sufficient to clear the clamps - see Photo 3.

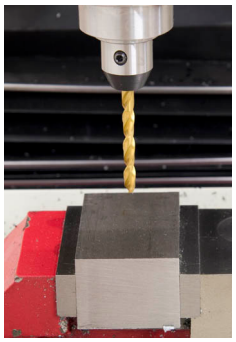


Photo 3

Once the longest tool is fitted, again ***Zero Z*** is pressed to remember this point.

Fit the tool to be used into the machine noting the tool number you are using or what the program needs. Click in the ***Tool*** number box in the Tool Information module, enter the tool number and press ↵ to save it. Then enter the tool diameter in ***Dia.*** and again press ↵ to save.

Jog the tool down until it just touches the work.

A time honoured method is to use a sheet of cigarette paper under the tool until it just traps it - Photo 4. At this point click the ***Set Tool Offset*** button. This enters the Z value into the Tool Table and turns the ***Tool Offset On/Off*** to On (indicated by a green LED) and shows 0.000 in the ***Z axis DRO*** as the tool is now at the working Z zero point.

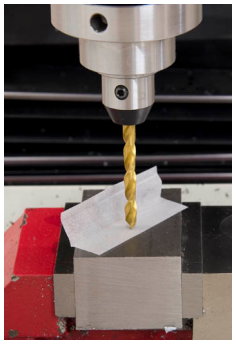


Photo 4

Click the ***Tool Offset On/Off*** button to turn it off. The green LED will go out and the ***Z axis DRO*** will now be reading the tool height offset.

Press the ***GOTO ZERO*** button and the tool will rise clear and go to its preset clearance plane.

CAUTION: If you fail to turn off the ***Tool Offset On/Off*** button ***before*** pressing the ***GOTO ZERO*** button, there is a risk for your tool to plough into your workpiece and cause damage.

Repeat this process for all the other tools you are using in the program.

At this point, all the work and tool offsets have been set. Go to the **Offsets Alt5** screen and click ***Save Tool Offsets*** to open the tool table - Fig 9 below.

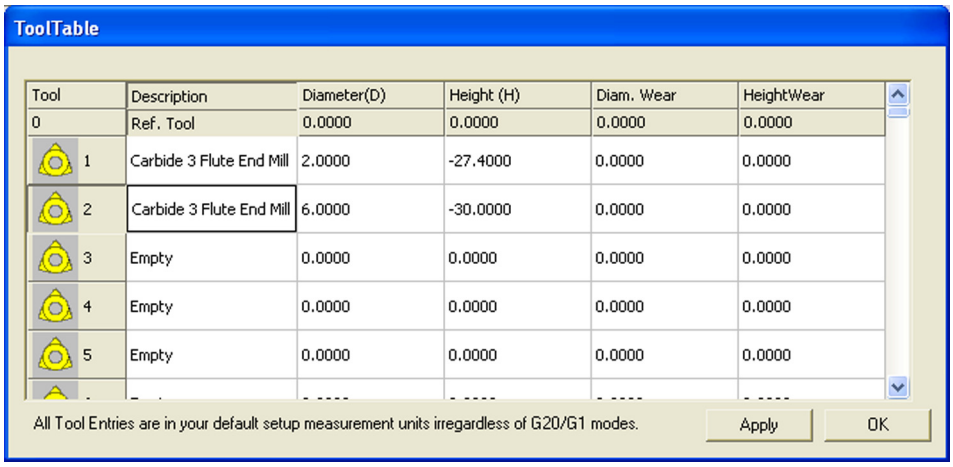


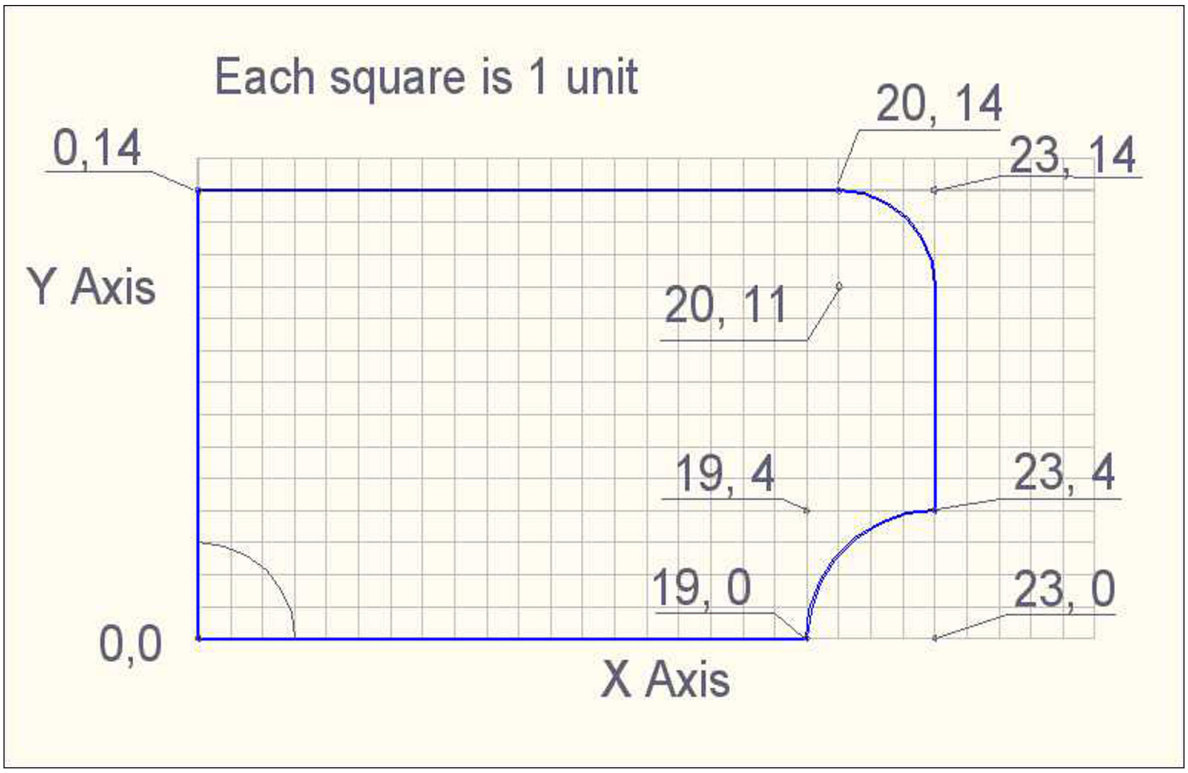
Fig. 9

It's not essential but you can enter a few words as a description. To save the tool table, click ***Apply*** and ***OK***. This table will then load when you next open Mach3.

The last job of setting up is to make the machine coordinates match the work coordinates.

Click the ***Tool Offset On/Off*** to turn it off and the green LED will go out. The ***Z axis DRO*** will now display the tool height offset so click ***ZERO ALL*** one last time to set Z back to 0.0000 and the machine is now in a full zero state.

9. Working with G Code



G-code is the name given to the instructions required by the Sieg KX series mills using Mach3 software. This code follows a loose Industry Standard in that many machines have special codes but with Mach3 the common ones are adhered to.

The four main moves are G00, G01, G02 and G03. The first two are linear moves and the latter two are arcs. Most shapes are a collection of these four moves.

G00 is a linear move at rapid rate. Rapid is defined as the maximum speed the machine can move. An example of this is: G00 X25.0. Assuming the tool is at zero then the tool will move plus 25 units in the X axis at maximum speed.

G01 is a linear move at a defined feed rate and can be expressed as: G01 X25.0 F 250.0. Again assuming the tool is at zero then this command will move the tool plus 25 units in the X axis at 250 units per minute.

G02 is a clockwise arc move and is defined as X, Y, I, J where X and Y are the end points of the arc, I is the incremental centre in the X plane and J is the incremental centre in the Y plane, both from the start point.

G03 is an anticlockwise move with the same definitions as the G02 move.

Below is the simple shape of a box with a radius on two of the corners and we will examine this shape and the code needed to trace the shape.

The origin point or 0,0 is in the bottom left and all moves will be made from this start point moving round in a clockwise direction.

9.1 G01 - Linear Move at a Defined Feed Rate

The first move will be in the Y axis to a point 14 units from 0,0 and we will move at a feed rate of 250 units per minute so out first line will look like:

```
G01 X0.0 Y14.0 F250.0
```

Next we need to move along the X axis for 20 units to get to the start of the first arc.

```
G01 X20.0 Y14.0 F250.0
```

9.2 G02 - Clockwise Arc

The next move is a G02 clockwise arc with the end point of the arc at X23.0 and Y11.0.

I relates to the centre of the arc in the X direction from the arc's start point, so since they are both the same, **I** = 0.0.

J is the centre of the arc in the Y direction from the arc's start point and since that was 14.0, the centre needs to be 11.0 and therefore **J** = -3.0. The line for the Arc then reads:

```
G02 X23.0 Y11.0 I0.0 J-3.0 F250.0
```

Arcs cause the beginner the most problems and there are two main points to remember, X and Y define the END point of the arc, I and J define the CENTRE incremental to the start point.

The shape then moves in the Y axis to the start of the second arc.

```
G01 X23.0 Y4.0 F250.0
```

9.3 G03 - Anticlockwise Arc

The last arc is anticlockwise so it's defined as G03. The end point is X19.0 and Y 0.0

The centre of the arc in X is still 0.0 and the centre in Y is -4.0 relative to the start. This gives the full line as:

```
G03 X19.0 Y0.0 I0.0 J-4.0 F250
```

To complete the shape the line then has to go back to its origin point.:

```
G01 X0.0 Y0.0 F250.0
```

This is just the basic code needed to follow the shape in the drawing, There are no Z axis moves or selecting the tools. At this point the code doesn't know what units it is working in, imperial or metric. Normally a CAM system generates these header and footer moves that define the rest of the program but as this is only a primer we will leave this to the owner to follow on.

9.4 The Safety Line

To get back to our code and putting it together with a couple of extra lines we get:

```
G21 G17 G90 G40 G49
G00 X0. Y0.
G01 X0.0 Y14.0 F250.0
G01 X20.0 Y14.0 F250.0
G02 X23.0 Y11.0 I0.0 J-3.0 F250.0
G01 X23.0 Y4.0 F250.0
G03 X19.0 Y0.0 I0.0 J-4.0 F250
G01 X0.0 Y0.0 F250.0
G28
M30
```

The first line sets the code to metric (G21), working in the X / Y plane (G17), absolute mode (G90) and G40 and G49 cancel any tool offsets that might still be running from the last program, this line is called the safety line and most programs such as dolphin and V Carve insert these lines automatically.

The next line forces the tool to go to the origin point if it's not already there. The G28 the end returns the axis to Zero and the M30 is the End of Program command.

It is important that this small example is followed through and understood as the whole method of writing and generating G-code is based on these simple moves.

With Mach3 running, load the program by clicking ***Load G-code*** on the **Program Run Alt-1** screen. Navigate to the GCode folder in Mach3 (e.g. C:\Mach3\GCode) and select the file **Demobox.tap**.

The program will now load in the G-code window and a preview will appear in the Toolpath screen. (If you like, you can view or edit the file by clicking the ***Edit G-code*** button which will open the file in Notepad)

Making sure the Z axis is clear of the table and the axis are zeroed with room to work, press cycle start and you can see it follow the shape in the drawing.

10. Ready to cut our first part

For the first time user it's advisable to do what is known as air cutting to get used to the machine and its settings.

At the side of the large red ***Reset*** button is a button called ***Z Inhibit***. It has an entry field and a two state button, ***On/Off***. If you type 6.000 into the DRO and ↵, then click the ***On/Off*** button so that the light comes on, it means that it will run a program but will ignore any Z commands with a value of less than +6.000 so the tool will be always 6.000mm above the work.

10.1 Cutting our first job

In this example we will use a file that has been previously generated in V Carve and is in the GCode folder, inside Mach3, the file is called Sieg_Logo.tap.

This is a simple engraving file that can be cut in any soft metal such as brass or aluminium or even MDF board.

The size of the part is confined into a circle 80mm in diameter and is designed to be cut with either a Vee cutter such as an engraving cutter or router cutter or even using a centre drill so no special cutters are needed for this first job. The origin point [0,0] is in the centre of the circle.

The depth of the engraving is 1mm deep so any material over 3mm will be ideal. It's advisable to use a piece larger than 80mm in diameter so that clamps can be used to hold it well out of the way of the tool.

10.2 Load the G Code

With Mach3 running, load the program by clicking ***Load G-code*** on the **Program Run Alt-1** screen. Navigate to the GCode folder in Mach3 (e.g. C:\Mach3\GCode) and select the file **Sieg_logo.tap**.

This file will run on either the KX1 or KX3 without any modifications.

10.3 Prepare the work.

The piece of material to be used should now be clamped onto the bed of the machine central to the travels of the X and Y axis. The origin or zero point is central to the work and can be found by working to a pair of diagonal crossed lines from corner to corner which will be as accurate as needed for this exercise.

Whilst learning it is also handy to mount your work on a sacrificial sheet of aluminium or MDF to give a degree of safety if you make any errors as it will give time to stop the machine before it damages the table.

10.4 Load the tool.

Fit the cutter into an appropriate collet, collet chuck or end mill holder depending on the size of the cutter and what tool holding equipment is available. Do not use a drill chuck to hold a milling cutter as these can and do work their way out, causing damage to the table and work. The rest of the tool loading is the same as covered in this Quick Start Guide earlier but we will go over it again. The tool used is tool number 20 and this has been coded into the program.

Jog the cutter over the work blank until it lines up with the centre mark you have previously made, at this point press ***Zero X*** and ***Zero Y*** at the side of the DRO's. This will set the origin point.

Because for this job we are only using one cutter we don't have to worry about setting all tools and dealing with just one tool makes it easier. Using the Page Up and Page Down keys jog the Z axis to a reasonable clearance plane above the work, for this job say 10mm and make sure all clamps are out of the cutting area.

In this position press ***Zero Z***. Then carefully jog down to the top of the work and using a strip of cigarette paper just trap the paper. This point is now the top of the work or the parts Z 0.00.

Type in 20 into the ***Tool*** number as this is the tool used in the program then press enter.

When typing values into DRO boxes it's important that you press enter and check the value is locked as if it's not pressed it remembers the previous value, often with dire results.

At this point click the ***Set Tool Offset*** button. This enters the Z value into the Tool Table and turns the ***Tool Offset On/Off*** to On (indicated by a green LED) and shows 0.000 in the ***Z axis DRO*** as the tool is now at the working Z zero point.

Click the ***Tool Offset On/Off*** button and the green LED will go out and the ***Z axis DRO*** will now be reading the tool height offset.

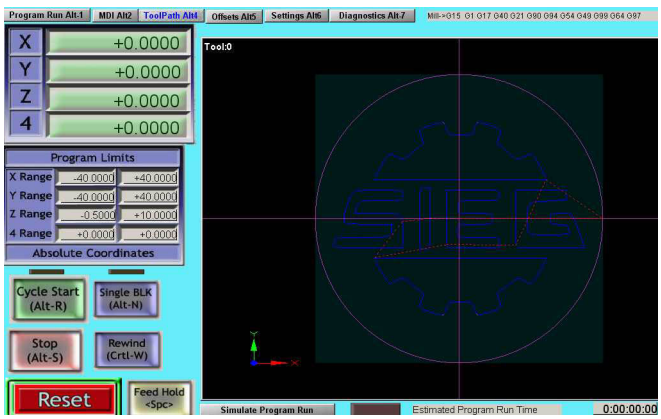
Press the ***GOTO ZERO*** button and the tool will rise clear and go to its preset clearance plane.

This now puts us in the correct position to start the job.

- The file is loaded.
- The tool is loaded.
- Work origin has been set.
- Tool offset has been set.

One last check we can do to satisfy ourselves that everything is correct is to see what the extents of the program are.

If we go to the toolpath screen from the top menu bar or Alt4, we get the following view.



This gives a bigger picture of what we are cutting but more importantly it shows the Program limits screen.

Looking at this shows how much room or travel is required in all axis and allows us to double check we have the room or we are not cutting too deep. In this case the maximum Z cut will be -0.5mm or 0.5mm deep INTO the part as it's a minus figure

You can now go back to the **Program Run Alt1** screen and press ***Cycle Start***.

It will load the tool then ask you to press ***Cycle Start*** again.

This is a safety measure to give you chance to check you have the right tool loaded.

The spindle will start and the tool will now feed into the work and commence to cut the Sieg logo. At the end of the cut it will stop the spindle and return to X0.0 Y0.0 Z10.0 and you will have cut your first part.

Happy Cutting !

11. CAM Software

On the install CD supplied with the machine there are two demo programs that can generate G Code. One is called Dolphin and is a full 2½D CAD and CAM package that is very powerful. The second program is called V Carve and is a sign making and engraving package but it can also do simple shapes.

Both these packages have tutorials on them and further details can be obtained from their respective web site with additional tutorials and videos.

12. Forum Based Support

12.1 SIEG CNC Machines (for machines running Mach3 only):

http://www.smallcncsupport.com/

This is the official support site. It provides on-line support on issues relating to your machine. To obtain the support, you must first be a bona fide owner of a SIEG CNC machine designed to be controlled by Mach3. Secondly, you must register your machine.

12.2 Registration Process

On the home page there is a log in box for registered users. Just underneath this is a link for new users to register. To register, you will need the Serial Number and Registration Number that came with your machine.

These two numbers ensure that only genuine owners can take advantage of the support. The support is forum based. As owners become more conversant with their machines we expect that all the owners will help each other.

As these machines get more popular we hope to add to the list with projects and ideas. Please use the feedback form to let us know what you would like to see.

12.3 Mach3 Support

Mach3 forum based support is available from:

http://www.machsupport.com/forum

For mach3 support issues relating to SIEG CNC machines, scroll down the page to the **Sieg Machines** forum in the **Third party software and hardware support forums** section. This site also has a good selection of downloadable video's and additional support.

12.4 C11G Breakout Board Support

The first point of contact for support with the C11G breakout board is via:

http://www.smallcncsupport.com/

Upon identification of a fault you will be referred to CNC4PC for guidance.

12.5 Other useful links

There are excellent Yahoo user groups for Mach3 and the C11G breakout board which can be found at:

http://groups.yahoo.com/group/mach1mach2cnc/

http://groups.yahoo.com/group/cnc4pc/

They are free to join and the members are very helpful.

E.&O.E.

13. Set-up Checklist

Following, is an abbreviated list of operations you need to carry out when setting up a job on your KX Series Mill.

Until you become more familiar with Mach3, we suggest to keep a copy of this page near the machine to act as a reminder.

- Start Computer and allow to boot
- Run the Mach3 Loader program
- Select the profile for your machine and Mach3 will then load
- Switch on the mill

13.1 Setting the Work Offset

- Click the ***Reset*** button to clear Emergency mode
- Fit edge finder or wiggler to spindle
- Click ***Zero X***, ***Zero Y*** and ***Zero Z*** to zero the DRO's
- Enter a spindle speed of 600rpm and start the spindle
- Jog the Y-Axis until the wiggler flicks out
- Enter wiggler radius into ***Y-axis DRO*** (+or -) and press ↵ to accept (see page A-8-2 for further details)
- Jog the X-Axis until the wiggler flicks out
- Enter wiggler radius into ***X-axis DRO*** (+ or -) and press ↵ to accept
- Raise spindle and remove wiggler and Click ***Zero Z***
- Click ***Goto Zero*** - this is the Work Offset or Origin Point

13.2 Setting the Tool Height Offsets

- Raise spindle above the work greater than longest tool
- Press ***Zero Z***
- Fit the longest tool to be used on this program
- Enter the tool number and diameter
- Jog the X and Y Axes placing the tool over the work
- Jog the Z-Axis down until it just touches the work
- Click the ***Set Tool Offset*** button
- Click the ***Tool Offset On/Off*** button to switch it off
- Click ***Goto Zero***
- Repeat the above for any other tools required in the program
- Go to the **Offsets Alt5** screen and click ***Save Tool Offset***
- Click ***Apply*** and ***OK*** to save the tool table
- Click the ***Tool Offset On/Off*** button to switch it off again
- Return to the **Program Run Alt1** screen
- Click ***Zero All*** button to set a full zero state

Index

Mach3
Configured

KX1 CNC Milling Machine

Technical Guide

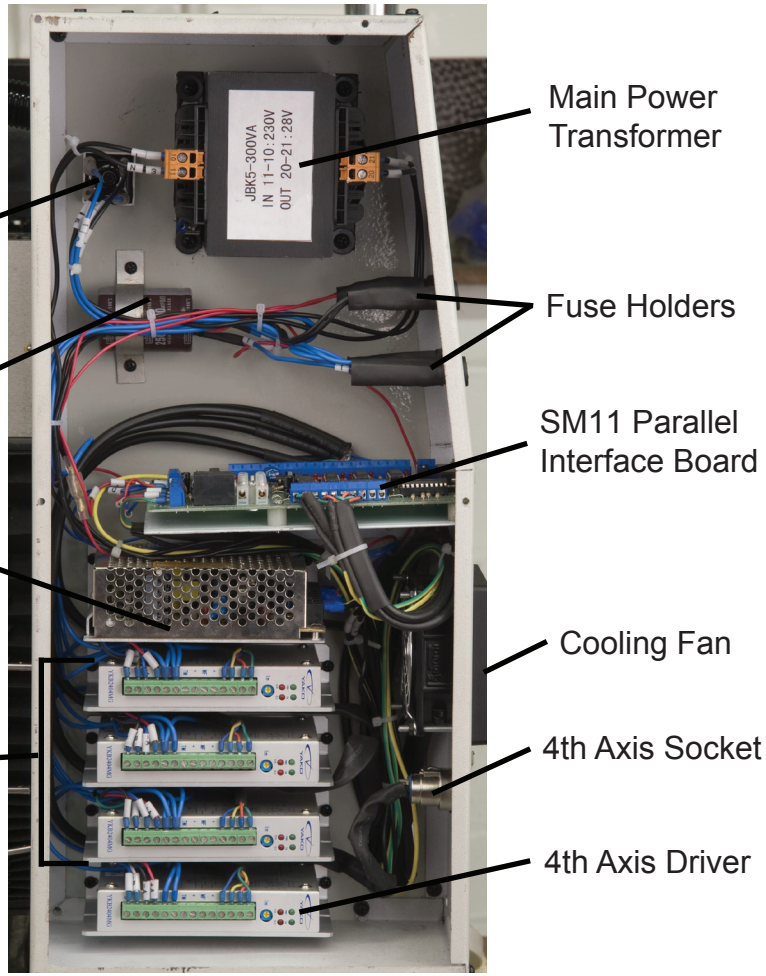
1. KX1 CNC Milling Machine Specifications

End Milling Capacity	10mm
Face Milling Capacity	20mm
Drilling Capacity	10mm
Table Size	400x145mm
Lubrication	Press Button Oilers*
Table travel - X axis	260mm
Table travel - Y axis	110mm
Head travel - Z axis	180mm
Head-Table Distance	70-250mm
Ballscrew Size (Dia./Pitch)	12mm x 4mm
X axis Motor	1.35 Nm
Y axis Motor	1.35 Nm
Z axis Motor	2.2 Nm
Throat	140mm
No. of slots on table	3 (8mm)
Positional Accuracy	0.01mm
Spindle Taper	MT2
Drawbar Thread	M10
Spindle Motor	500w Brushless DC
Spindle Speed	100-7000rpm
Max. Spindle Motor Torque	1.8Nm @ 1500RPM
Power Requirement	110v AC 60Hz
Overall Dimensions (w/d/h)	630x630x630mm
Max Space Required (w/d/h)	910x 630x730mm
Shipping Dimensions	76x76x79cm
Weight (Net/Gross)	86/120kg

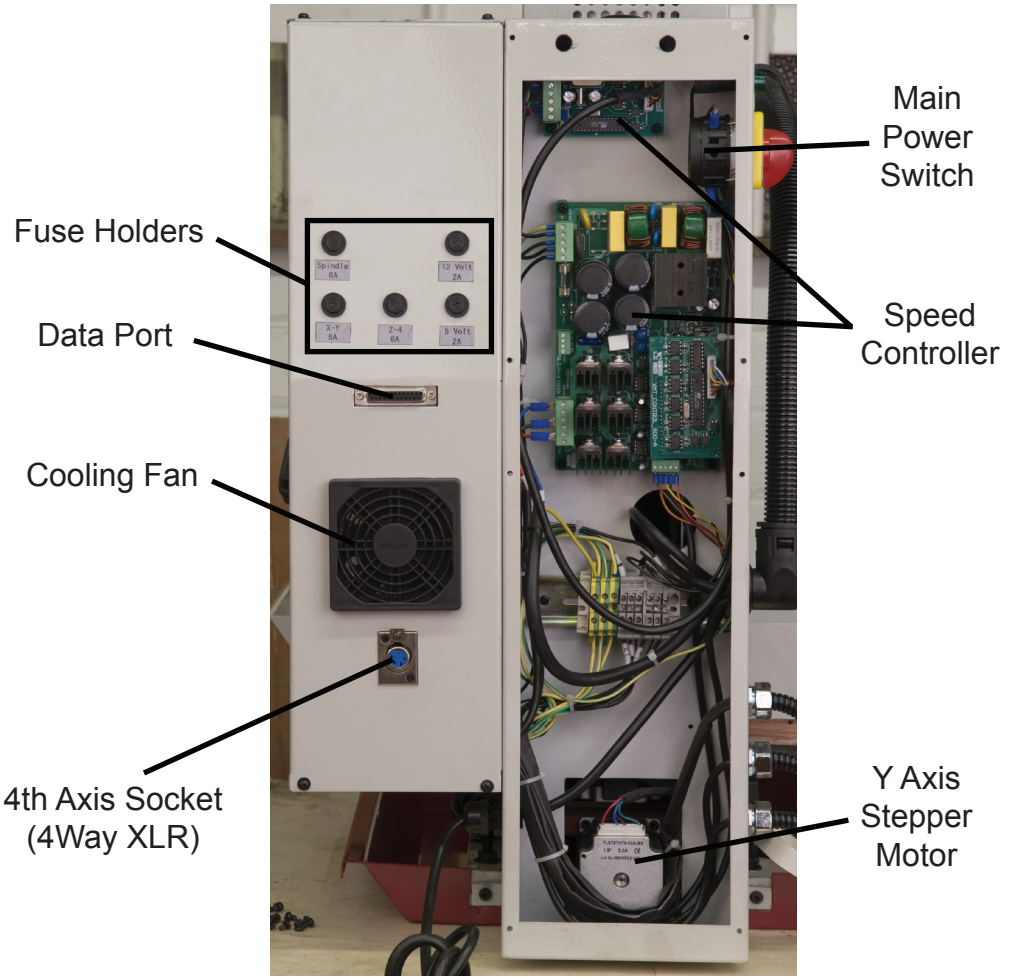
* Oil Daily with SAE 30 grade hydraulic oil or transmission fluid

2. Electronics Layout

Right Side View



Rear View

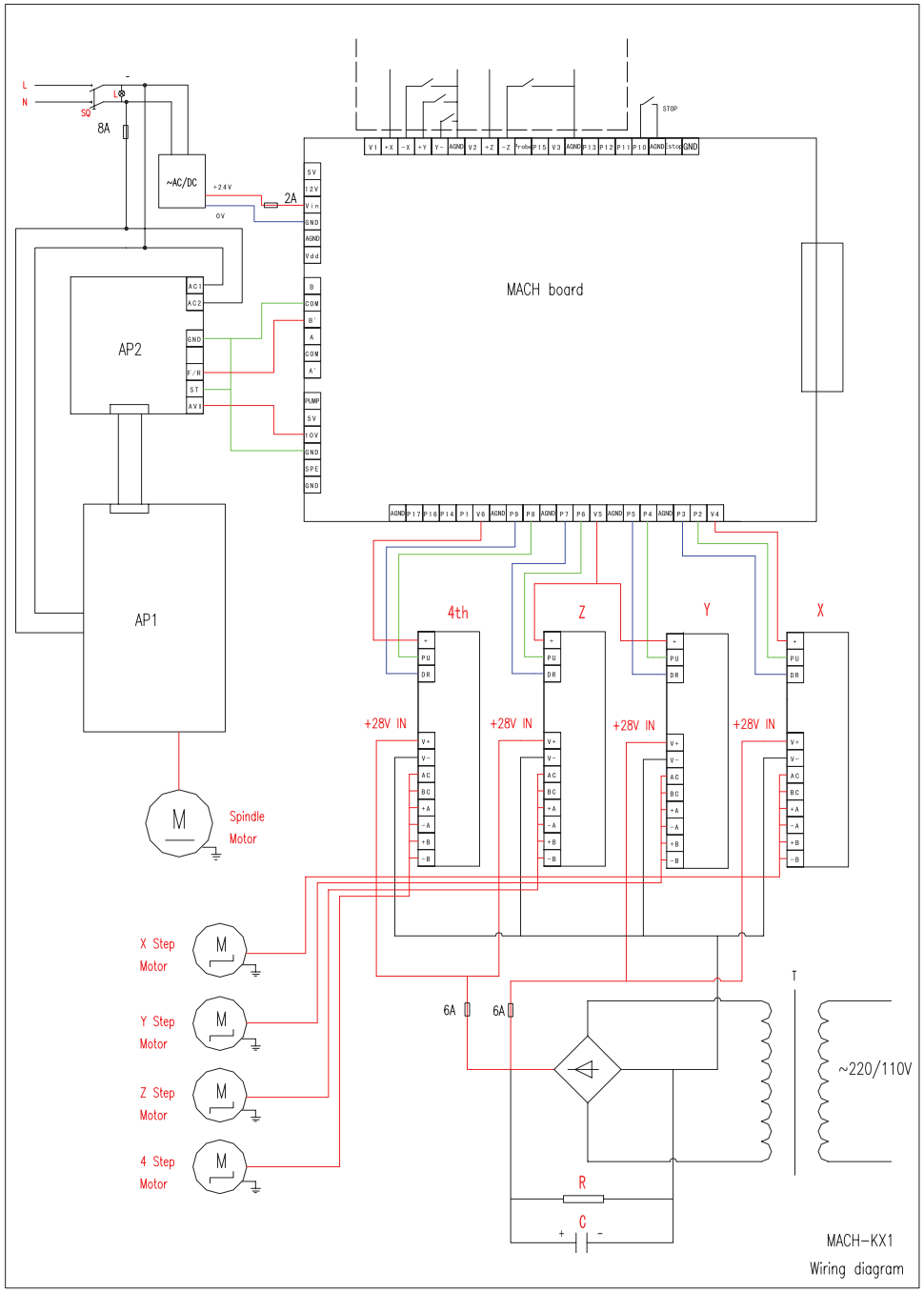


3. Toolkit and Accessories

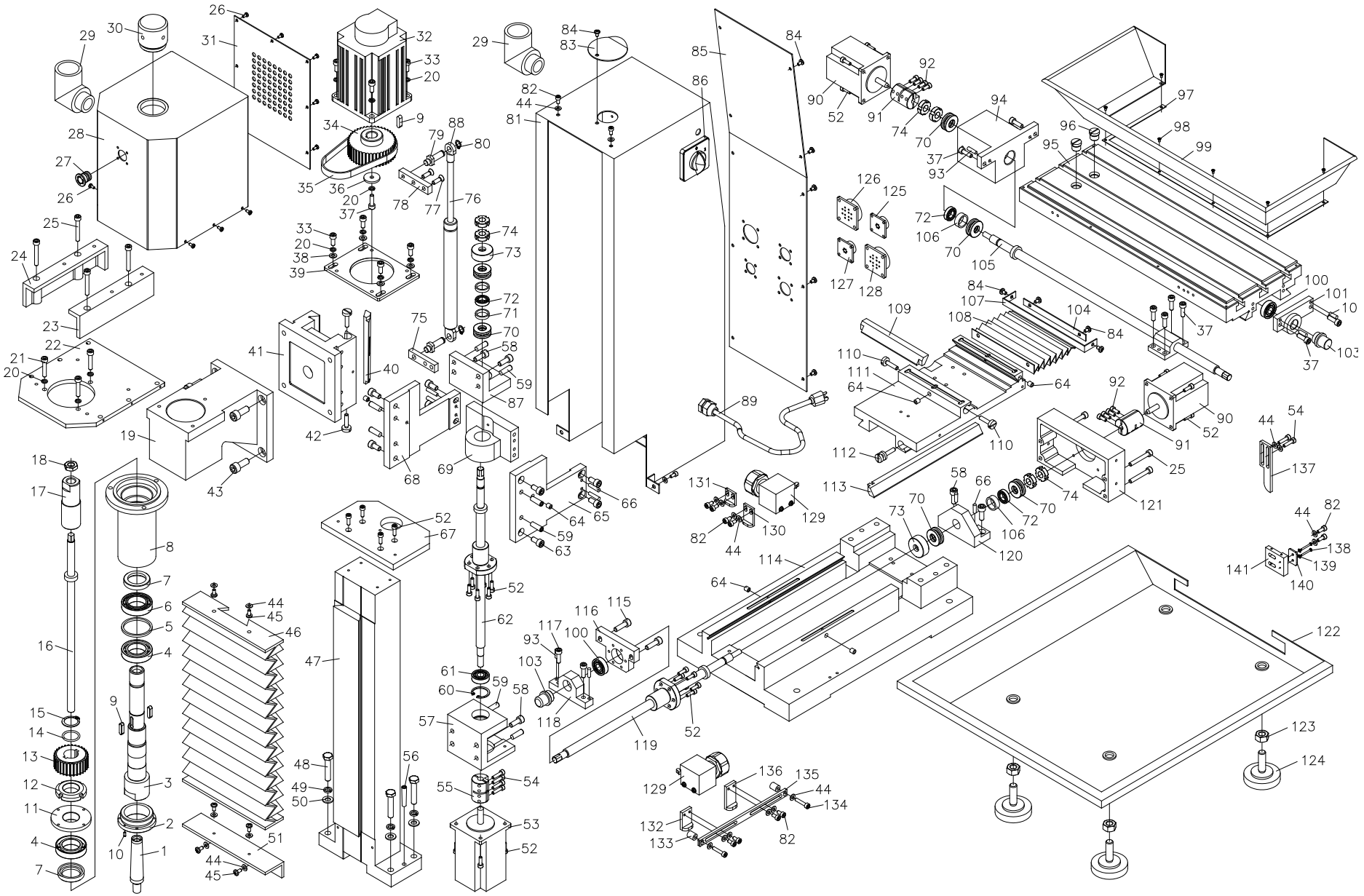
Contents:

1x 27mm AF Open Ended Spanner
1x 17/19mm AF Open Ended Spanner
1x 14/17mm AF Open Ended Spanner
1x 5.5/7mm AF Open Ended Spanner
1x 38-42mm C Spanner
1x 8mm AF Square Tee Wrench
1x 3mm AF Hex Key
1x 6mm AF Hex Key
4x Adjustable Feet
1x Data Cable (D type 25w male - 25w male)
1x Plastic Oil Bottle
2x 8mm Tee Nuts
1x Serial Number / Registration Number sheet (required for on-line support)
Spare Fuses: 2x 2A 250v; 2x 6.3A 250v; 1x 8A 250v (all 20mm quick blow glass fuses)
1x User Manual

4. KX1 circuit diagram



5. KX1 Exploded diagram



6. Spare Parts List

No.	Description	Qty
1	MT2 Drill Chuck Arbor (not supplied)	1
2	Sleeve nut	1
3	Spindle	1
4	Angular Contact Ball Bearing 71905C TA P5 (25x42x9mm)	2
5	Washer	1
6	Ball Bearing 61905-ZZ (=6905-ZZ) (25x42x9mm)	1
7	Oil seal	2
8	Spindle sleeve	1
9	Key 5*18	3
10	Screw M3*6	2
11	Nut for spindle	1
12	Nut M24*1.5	1
13	Spindle belt wheel	1
14	Rubber seal	1
15	Spring check ring	1
16	Draw bar	1
17	Taper extractor sleeve	1
18	Nut M10	1
19	Spindle sleeve bracket	1
20	Light spring washer	12
21	Screw M5*25	3
22	Base board	1
23	Belt wheel box	1
24	Belt wheel box	1
25	Screw M5*35	8
26	Screw M3*6	13
27	Emergency stop switch	1
28	Outer cover assembly	1
29	Right-angle cable tube	2
30	Dust proof cover for spindle	1
31	Back cover	1
32	Brushless motor	1
33	Screw M5*12	8
34	Motor belt wheel	1
35	Timing belt	1
36	Small washer	1
37	Screw M5*16	9
38	Washer 5	4
39	Motor mounting board	1
40	Z axis tapered gib strip	1
41	Spindle mount	1
42	Gib adjusting screw	2
43	Screw M8*25	4

44	Washer 4	24
45	Screw M4*8	6
46	Slideway cover	1
47	Column	1
48	Bolt M8*40	4
49	Spring washer 8	4
50	Washer 8	4
51	Lower cover support	1
52	Screw M4*12	28
53	Stepper motor	1
54	Screw M4*16	6
55	Z-axis coupling	1
56	Taper pin 6*35	2
57	Z-axis ball screw support	1
58	Screw M6*16	6
59	Taper pin 6*20	8
60	Circlip 24	1
61	Ball Bearing 609-ZZ (9x24x7mm)	1
62	Z-axis ball bearing screw assembly	1
63	Screw M6*12	8
64	Oil cup 6	6
65	Side support II	1
66	Taper pin A4*20	6
67	Column upper board	1

68	Side support I	1
69	Ball bearing nut seat	1
70	Thrust Ball Bearing 51100 (10x24x9mm)	6
71	Washer	2
72	Ball Bearing 61800-ZZ (=6800-ZZ) (10x19x5mm)	3
73	Bearing protect sleeve	2
74	Nut M10*1	6
75	Gas strut bottom mounting	1
76	Gas strut	1
77	Screw M5*16	2
78	Gas strut top mounting	1
79	Gas strut pivot	2
80	Circlip	2
81	Column cover	1
82	Screw M4*8	14
83	Ball screw access cover	1
84	Screw M4*6	15
85	Back cover board	1
86	Switch	1
87	Bearing seat	1
88	Gas strut end	1
89	Power line	1
90	Stepper motor	2
91	Coupling sleeve	2

92	Screw M4*10	8
93	Taper pin A4*16	4
94	X axis motor mount	1
95	Work table	1
96	Screw	2
97	Rubber strip	1
98	Screw M3*6	6
99	Guard for work table	1
100	Ball Bearing 609-ZZ (9x24x7mm)	2
101	Bearing seat	1
102	Taper pin A4*14	2
103	Small sleeve	2
104	Cover support II	1
105	X axis ball screw assembly	1
106	Washer	2
107	Cover support I	1
108	Slideway cover	1
109	X axis tapered gib strip	1
110	Screw	2
111	Saddle	1
112	Screw	1
113	Y axis tapered gib strip	1
114	Base	1
115	Screw M6*20	2
116	Y axis ball nut mount	1

117	Screw M5*20	2
118	Y axis bearing seat	1
119	Y axis ball screw	1
120	Y axis bearing mount	1
121	Y axis motor mount	1
122	Swarf tray	1
123	Nut M10	4
124	Adjustable foot	4
125	Five-core plug	1
126	Nineteen-core plug	1
127	Five-core plug	1
128	Sixteen-core plug	1
129	Travel switch	2
130	X-axis left bump block	1
131	X-axis right bump block	1
132	Y-axis right bump block	1
133	Plank mat block	2
134	Screw M4*20	2
135	Y-axis stop mount	1
136	Y-axis left bump block	1
137	Z-axis upper bump block	1
138	Screw M2*18	6
139	Washer 2	6
140	Mounting block	3
141	Travel switch seat	3

LIMITED 90 DAY WARRANTY

Harbor Freight Tools Co. makes every effort to assure that its products meet high quality and durability standards, and warrants to the original purchaser that this product is free from defects in materials and workmanship for the period of 90 days from the date of purchase. This warranty does not apply to damage due directly or indirectly, to misuse, abuse, negligence or accidents, repairs or alterations outside our facilities, criminal activity, improper installation, normal wear and tear, or to lack of maintenance. We shall in no event be liable for death, injuries to persons or property, or for incidental, contingent, special or consequential damages arising from the use of our product. Some states do not allow the exclusion or limitation of incidental or consequential damages, so the above limitation of exclusion may not apply to you. THIS WARRANTY IS EXPRESSLY IN LIEU OF ALL OTHER WARRANTIES, EXPRESS OR IMPLIED, INCLUDING THE WARRANTIES OF MERCHANTABILITY AND FITNESS.

To take advantage of this warranty, the product or part must be returned to us with transportation charges prepaid. Proof of purchase date and an explanation of the complaint must accompany the merchandise. If our inspection verifies the defect, we will either repair or replace the product at our election or we may elect to refund the purchase price if we cannot readily and quickly provide you with a replacement. We will return repaired products at our expense, but if we determine there is no defect, or that the defect resulted from causes not within the scope of our warranty, then you must bear the cost of returning the product.

This warranty gives you specific legal rights and you may also have other rights which vary from state to state.

3491 Mission Oaks Blvd. • PO Box 6009 • Camarillo, CA 93011 • (800) 444-3353