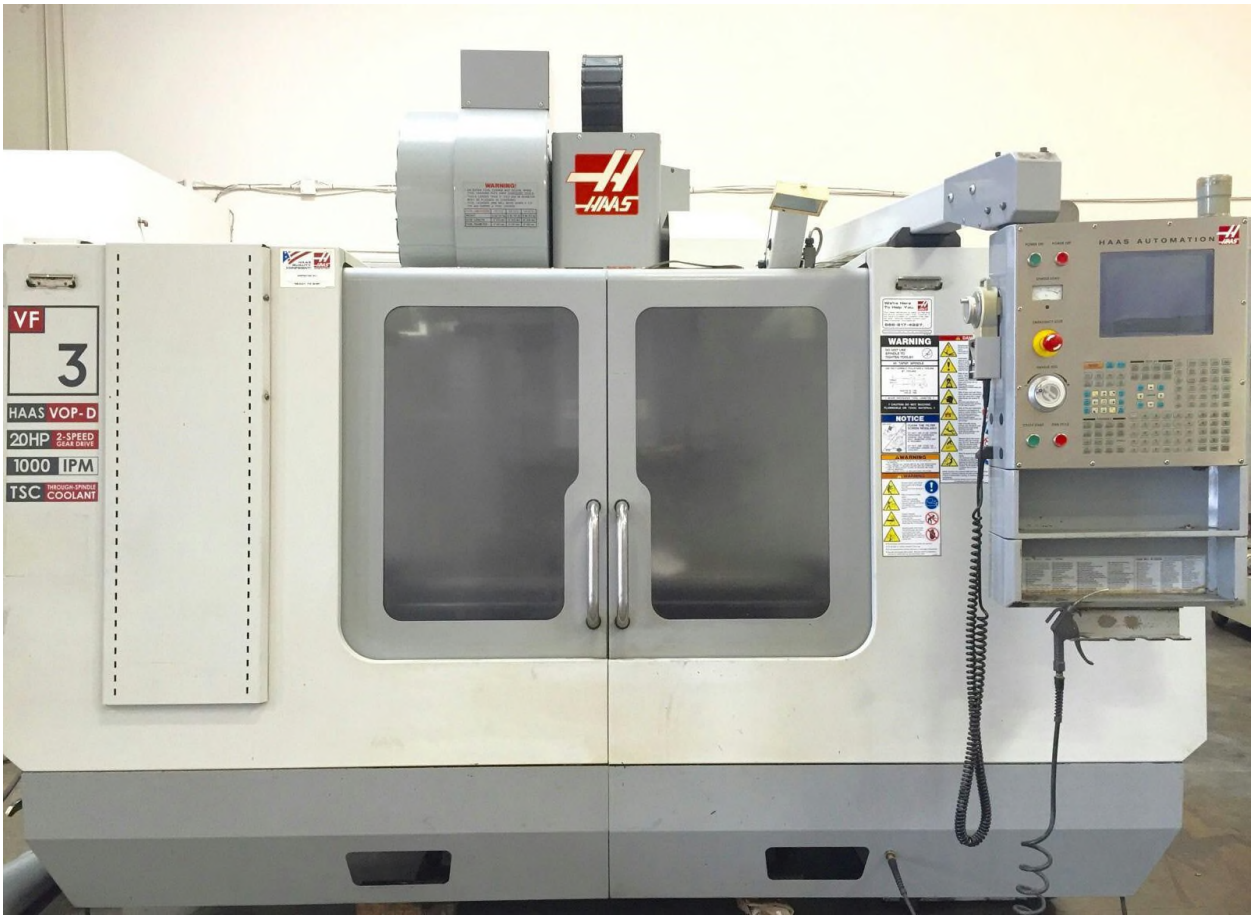


SETTING UP AND OPERATING A VMC

Explaining to someone how to drive a car will help that person understand how to drive.

But until this person drives for a good length of time, he or she won't be an experienced driver.

The same can be said about setting up an operating a CNC machine.

You can read lots of information on how to setup and operate a CNC machine.

You can have someone explain to you what's all involved with CNC setup and operation.

But until you get some actual on-the-job," hands on experience", a lot of what you read and hear can be as clear as mud.

Clear as mud was one of my CNC teacher's favorite sayings.

But as soon as you get some “hands on” CNC setup and operating experience, the mud will begin to clear.

The procedures mentioned for setting up a CNC machine were performed on a Haas vertical machining center.

Machine setup and operation is basically the same regardless of the machines age.

The main difference is, new Haas machines have an electronic probe that is used for setting work offsets and tool offsets.

Much of what is written was learned from my schooling at moraine park technical college.

I also talk about machining techniques I learned while setting up and operating CNC machines at work.

Some information mentioned is from the Haas VMC operational manual and internet.

I also include links that you can click on that will bring up YouTube videos on many different subject that I cover.

THE ENGINEERING DRAWING

If you are given an engineer drawing and told to write a CNC program from it, you must first determine what all needs to be machined.

Next you must determine what tools are needed to machine the part features.

You must decide in what order you want to machine the part features in, from the first tool that will start machining the part to the last tool that will finish machining the part.

Operations to be performed on the part may include spot drilling and drilling holes, reaming drilled holes, milling outside contours, tapping drilled holes, boring holes, milling pockets, countersinking etc.

If you were writing the program and not using a CAD CAM software, you must calculate the speed and feed to use for each tool.

If you are using CAD CAM software to write the program, the software will calculate the feeds and speeds for all tools that will be machining the part.

There is just a little information that you will need to enter so the software can calculate the correct feeds and speeds.

The engineer drawing will list the type of material to machine the part out of.

The engineer drawing will show the part dimensions, finish requirements, tolerances and any datums associated with the part.

The part's zero origin shown on the engineering drawing must be used when writing the program and when setting up the part and inspecting the part.

DOCUMENTING THE MACHINING PROCESS

If you are given an engineer drawing and told to write a CNC program and then set up the CNC machine and run the part, you must document the following:

Document the program.

Document the CNC machine setup procedure.

Document the tools required to machine the part.

Document any special fixtures or tool drawings required to set up the CNC machine.

Document any feed and speed changes that may have been made while running the first part.

Documenting all pertinent information on machining the part will make the machining process consistent throughout future runs and it will also help eliminate quality issues.

THE TOOL DRAWINGS

Each tool required to machine the part may include a tool drawing.

A tool drawing may include all or some of the six things listed below.

The tool #

The components that make up the tool.

The insert number or letter, this is the size of the insert.

The insert grade, the insert grade is the quality of the insert.

The dimensions of the tool.

A picture of the tool.

THE TOOL LIST

The tool list will be in the CAD CAM software for the program that will be used to machine the part.

The tool list is a list of all the tools that will be machining the part.

The tool list gives a description of each tool, such as whether the tool is a drill, tap, end mill etc.

The diameter of the tool, the tool's length, the tool's insert identification number and the tools insert grade may be listed.

The type of material the tool is made from is normally either high-speed steel or carbide.

THE SETUP SHEET MAY INCLUDE ALL OR SOME OF THESE 9 THINGS LISTED BELOW

The type of work holder that will be used to hold the stock material while the part is being machined.

The location where the work holder will be positioned to the machine table.

Where program zero is located on the part.

The program number to load into the machine.

The name of the part.

The part #

The operation #

A picture of the part.

Special instructions for setting up the part.

FEED AND SPEED

Using the correct feeds and speeds allows the tools to cut with better precision, resulting in a smoother surface finish.

Using the correct speeds and feeds extends tool life.

Using the correct speeds and feeds puts less strain on the CNC machine.

If the feed isn't correct the tool can wear prematurely.

Feed is the distance the tool is going to machine the part at in one minute.

In the program, feed is written as inches per minute / **I.P.M**

If the speed isn't correct the tool can also wear prematurely.

Speed is the number of revolutions that the tool turns in one minute.

In the program, speed is written as revolutions per minute / **R.P.M**

FACTORS THAT DETERMINE WHAT FEED AND SPEED TO USE ARE:

The type of material that is going to be machined.

The depth the tool is going to machine at.

The number of flutes on the tool.

The diameter of the tool.

The type of machining that is going to be performed.

How rigid the part / stock material is held in the vise or fixture.

The rigidity of the tool.

The type of material the tool is made from, such as high-speed steel or carbide.

BEGIN SETTING UP THE CNC MACHINE AFTER THESE ITEMS ARRIVE

THE WORKHOLDER / The work holder accurately positions and holds the stock material while the part is being machined. Accurately positioning the stock material allows the machine to know precisely where X0 Y0 / program zero is located on the part. The machine needs to know where X0 Y0 is located on the stock material so that the tools can machine the part features at the correct location on the part. The type of work holder that you use will depend on how the part is going to be a machined and by the shape of the part. Seven types of work holders used on vertical machining centers are listed below.

VISE / A machine vise is the most commonly used work holder. The vise accurately positions and securely holds the stock material while the part is being machined.

FIXTURE / A fixture is the second most used work holder. A fixture accurately positions and securely holds the stock material while the part is being machined. A fixture is used when the shape of the part will not allow it to fit in a machine vise or when the vise gets in the way of the machining of the part.

V BLOCK & CLAMPS are sometimes used when machining round stock.

TOMBSTONE / A tombstone may be used when machining is required on multiple sides of the stock / part. The horizontal machining center must have a B axis so that the machine can rotate the tombstone to a different plane / side of the part and then machine it.

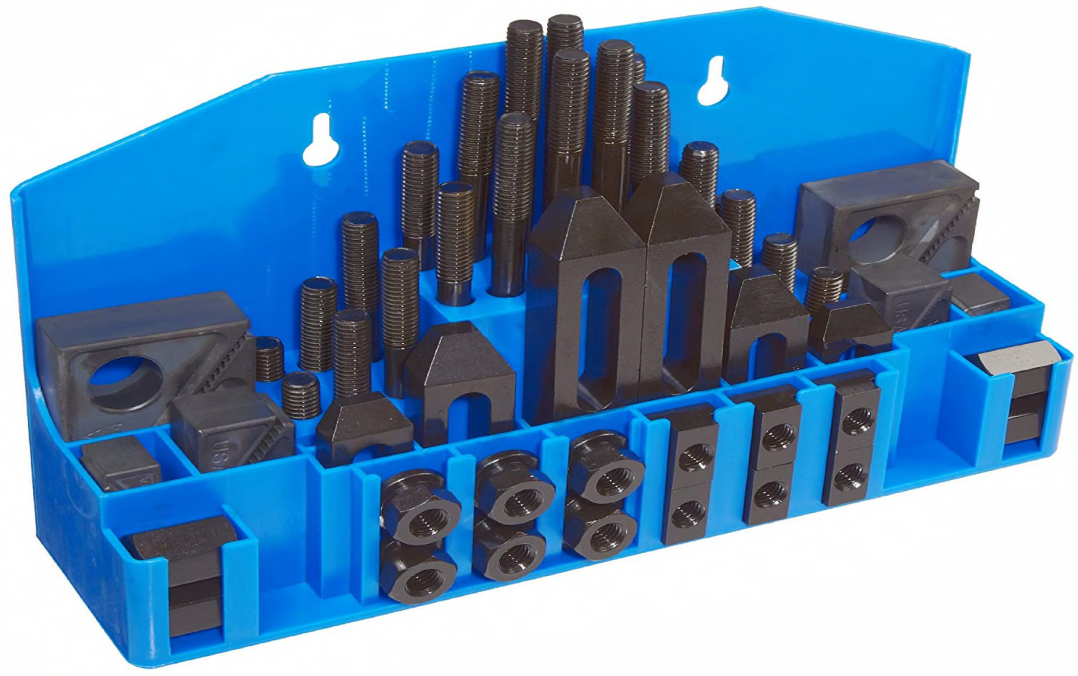
THREE JAW CHUCK / A three-jaw chuck can be mounted on a vertical machining center table. The three-jaw chuck accurately positions and securely holds the round stock material while the part is being machined.

ANGLE PLATE / The stock material is clamped to the angle plate. The angle plate is secured to the machine table using step blocks, clamps, studs, t nuts and nuts.

STEP BLOCKS, STRAP CLAMPS, STUD BOLTS, T NUTS and **NUTS** are often used when mounting large or small aluminum or steel stock to the machine table.

CLAMP SET

A clamp set will include step blocks, strap clamps, stud bolts, t nuts and nuts.



Step blocks have little steps machined into them.

The steps allow you to join two step blocks together so that you can adjust the height of them.

To set the two step blocks at the correct height, grab a matching pair of step blocks, mesh the steps from one step block into the steps of the other step block.

Place the joined pair of step blocks on the machine table, next to where you want to clamp the stock material.

Place the end of the strap clamp that has the steps machined into it on the step block, the step block supports this end of the strap clamp.

The strap clamp must be set at the correct height so that it angles down slightly towards the stock material, this slight angle will make it possible for the strap clamp to securely clamp the stock material down to the table.

The other end of the strap clamp holds the stock material down to the table; this end of the strap clamp is where you run a stud bolt through; the stud bolt gets screwed into the t nut.

Before you can screw the stud bolt into the t nut; you need to slide a t nut into one of the t slots that is machined into the machine table.

The end of the stud bolt that sticks out of the top of the strap clamp gets a nut screwed onto it; after tightening this nut, the strap clamp will securely hold the stock material down to the machine table.

NOTE: You may not have to join 2 step blocks together.

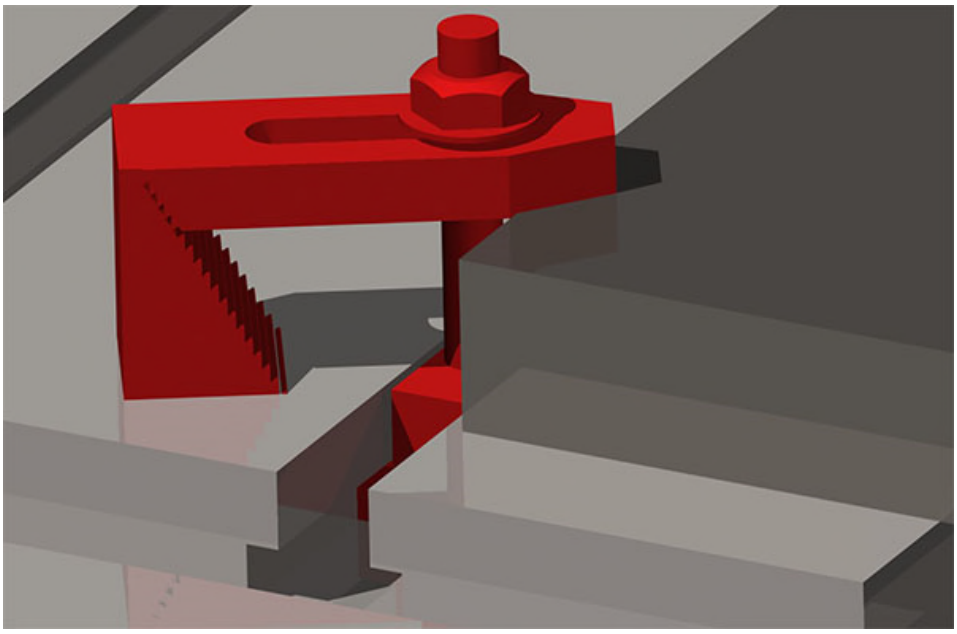
You may only have to use 1 step block, that is, if the step block is high enough so that it can angle the strap clamp down slightly towards the stock material.

The steps that are machined into the strap clamp are meshed into the steps that are machined into the step block.

The strap clamp must angle down slightly towards the stock material or it will not be able to clamp the stock material securely.

https://youtu.be/cS_KkHOe8dc

STEP BLOCK AND CLAMP VIDE



In the picture above we see one corner of the stock material being clamped down to the table. The parts that are used to clamp this one corner of the stock material down to the table are a step block, a strap clamp, a t nut, a stud bolt and a nut. The t nut is barely visible, the t nut is slid into the t slot that is machined into the table. The device that is holding the stock material down to the table is a strap clamp.

WORKHOLDER VIDEO'S

https://youtu.be/J1VtofzVG24

https://youtu.be/M5otomckPYU

FIVE OTHER THINGS YOU NEED TO SET UP THE CNC MACHINE

1. **TOOLS** / The end mills, spot drill, drills, all the tools that will machine the part.

2. **CNC PROGRAM** / You can write a program in MDI or send a program to the CNC machine from a computer. You may also be able to plug a jump drive / flash drive into a USB port that is located on the CNC controller and then run a program.

3. **TOOL LIST** / The tool list shows you what tools are required to machine the part.

4. **STOCK MATERIAL** / The stock material is the type of material that the part is going to be machined out of.

5. **SETUP SHEET** / The setup sheet are the instructions for setting up the CNC machine for a specific job run.

POWERING ON THE CNC MACHINE

Before powering on the CNC machine, you may have to flip the main power breaker switch to the ON position.

There must be a constant 70 to 100 psi of clean, dry air flowing to the CNC machine.

Twist the **EMERGENCY STOP** knob to the right to release it.

Press the green colored **POWER ON** button and you will hear a beep sound.

The correct term is probably a **key** instead of a **button**. But in the writing of this instructional handbook I call the things that you press, that get the machine to do something, buttons.

Wait 30 seconds or so for the Haas display to appear on the screen.

Press the **RESET** button a couple times to clear out all the alarms.

Press the **POWER UP RESTART** button, tool #1 will be inserted into the spindle and the machine will begin to home itself.

If tool #1 is already in the spindle the machine will just home itself.

THE EMERGENCY STOP BUTTON

You release the emergency stop button when you power up the machine.

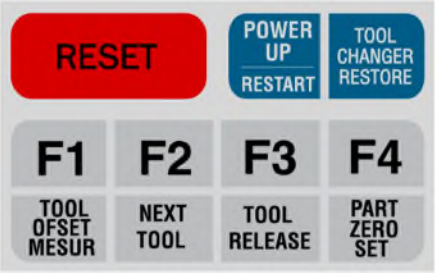
You press the emergency stop button when you turn the machine off.

You press the emergency stop button if shit starts hitting the fan.

https://youtu.be/NJ0DP8RcJbw
THE EMERGENCY STOP BUTTON

SHUTTING OFF THE CNC MACHINE

To shut the machine down press the **POWER UP / RESTART** button, press the **EMERGENCY STOP** button, press the **POWER OFF** button.



https://youtu.be/g5gNEp0j4Os
POWERING ON AND OFF

https://youtu.be/TyA_5_3hms0
STARTUP AND SHUTDOWN

http://youtu.be/QcegOHQzrZg
POWERING ON AND OFF

https://youtu.be/9-Ln4MSKCsg

POWERING ON AND OFF

https://youtu.be/5v5b8xwVtp8

TOOL SETUP

One of the first steps in setting up a CNC machine is to round up all the tools that are going to machine the part.

The tool list will show you what tools are needed to machine the part.

Make sure the tools are the correct type, such as a drill, end mill, reamer etc.

Make sure the tools are sharp and in good condition.

Make sure the tools are the correct diameter.

Make sure the tools are the correct material, such as carbide, high speed steel etc.

The tools must be inserted into the tool collets deep enough so that they do not wobble or loosen.

Make sure the tools are long enough so that they can machine the part correctly.

The L.O.C / length of cut that the tools need to be is listed on the tool list or in the CAD CAM program.

Some tools in the program have to have a precise length of cut or they will not be able to machine the part feature deep enough.

The tools / tool holders are inserted into the correct tool pockets according to the instructions outlined on the setup sheet.

If you write the program, place similar machining operations that use the same tool next to each other in the program. Doing this will prevent the machine from having to cycle through several tools before the tool is inserted into the spindle, thus saving time.

THE RETENTION KNOB / PULLSTUD BOLT

The retention knob is also called a pull stud bolt.

The pull stud bolt is the only part that holds the tool holder in the spindle.

The pull stud bolt can occasionally break, it should be inspected every 6 months.

The manufacturers of pull stud bolts say you should torque pull stud bolts to a specific foot pound, I've never seen anyone torque a pull stud bolt.

Use a pull stud wrench or pull stud socket and ratchet and securely tighten the pull stud bolt into the tool holder, just don't go ape shit on it with a breaker bar.

Three reasons why you may need to change or remove a pull stud bolt

1. Once in a rare moon you may have to remove the pull stud bolt because you need to place a rod in the tool holder and hit the rod with a hammer to remove a stuck tool.

2. Say your shop has Haas and Fadal CNC machines, Haas CNC machines use a long pull stud bolt and Fadal CNC machines use a short pull stud bolt. If you want to put a tool holder that has a long pull stud bolt into a Fadal machine you will first have to remove the long pull stud bolt from the tool holder and replace it with a short pull stud bolt. Once you replace the tool holder with a sort pull stud bolt you can install the tool holder in a Fadal machine spindle.

3. You may need to change a pull stud bolt because it is either broken or damaged.

RETENTION KNOB / PULL STUD BOLT VIDEO'S

https://youtu.be/-eCI6lUFqNk

https://youtu.be/GXVICVNtepQ

https://youtu.be/YtQH8iBQ9xM

https://youtu.be/OS-sMAZzaZ0

Pictured above is a multi-tooth spanner wrench, the wrench is placed on the collet clamping nut.

Turning the wrench clockwise tightens the end mill in the collet.

Turning the wrench counterclockwise loosens the end mill from the collet.

The green colored device towards the bottom right is a tool loading / unloading fixture.

Use a tool loading / unloading loading fixture when installing or removing an end mill from the collet / tool holder.

INSTALLING AN END MILL INTO THE COLLET / TOOL HOLDER

Find the correct size end mill that you need.

Find a collet that fits the shank of the end mill.

The shank of the end mill should slide easily into the collet.

If you must use force to get the end mill to fit the collet, the collet is too small.

If the shank of the end mill is too small for the collet the collet will not hold the end mill securely.

After you find the correct size collet for the end mill, carefully remove the end mill from the collet.

Be careful removing the end mill so that you don’t cut yourself, end mills are razor sharp.

Snap the collet into the collet clamping nut.

Screw the collet clamping nut on the tool holder threads, but don’t tighten it yet.

Place the tool holder assembly on the tool loading fixture.

Carefully insert the end mill into the collet.

Do not have the end milling sticking out of the collet any farther then necessary or tool vibration may occur, but the tool must stick out far enough so that it can machine the part feature correctly.

Place the multi-tooth spanner wrench on the collet clamping nut and turn the wrench clockwise until it doesn’t turn anymore, the end mill will lock itself in the collet.

The tool / tool holder can now be inserted into the machine spindle and be touched off so that the end mill's, tool length offset can be entered in the machine.

REMOVING AN END MILL FROM THE COLLET / TOOL HOLDER

Place the tool holder on the tool loading / unloading fixture.

Place the multi-tooth spanner wrench on the collet clamping nut and turn the wrench counterclockwise.

Carefully remove the end mill out of the collet.

If the end mill is snug and it doesn't want to come out of the collet; place a rag over the end mill, carefully grab the rag and pull the end mill out of the collet.

There's less chance of getting cut by placing a rag over the end mill.

TOOL HOLDER VIDEOS

https://youtu.be/IPWGV_EGAHw

https://youtu.be/r_MJf8_VYVY

https://youtu.be/3GazwUq9WF4

https://youtu.be/WKikm6cQKh0

https://youtu.be/6zD6CNBdqMk

THE 4 MOST COMMON CAT 40 TOOL HOLDERS

1. Set Screw End Mill Tool Holder
2. Drill Chuck Tool Holder
3. RE Collet Tool Holder
4. Solid Body / Face & Shell Mill Tool Holder

1. SET SCREW END MILL TOOL HOLDER

Use a set screw tool holder for roughing end mills. Using a set screw tool holder helps prevent the end mill from being pulled out of the tool holder as its machining. The set screw must line up with the keyway notch or the flat that's been ground into the shank of the end mill. Tighten the set screw securely. If you use a roughing end mill in a RE collet the tool may pull out of the collet and machine the part feature to deep. There is also a chance that the tool may break.

2. DRILL CHUCK TOOL HOLDER

Drills under .500 of an inch in diameter are used most often with drill chuck tool holders. Do not use a reamer in a drill chuck tool holder.

3. **RE COLLET TOOL HOLDER**

End mills, reamers, spot drills and larger diameter drills are used most often with RE collet tool holders.

4. **SOLID BODY / FACE AND SHELL MILL TOOL HOLDER**

HOW TO PUT A TOOL HOLDER / TOOL INTO THE SPINDLE

You must be in either **MDI, HANDLE JOG** or the **ZERO RETURN** mode for the **TOOL RELEASE** button to work.

Carefully grab the tool holder with the tool mounted in it.

Keep your fingers away from the cutting edge of the tool or you may cut yourself.

Align the tool holder key slots with the two keyways that are in the spindle.

Press the **TOOL RELEASE** button that's located by the spindle, do not press the **TOOL RELEASE** button that's located on the control panel.

It's safer and easier to press the tool release button that's located next to the spindle.

Keep your fingers away from where the tool holder gets inserted into the spindle or your finger may get pinched.

Release the **TOOL RELEASE** button and the machine will pull the tool up and lock it into the spindle.

If the tool holder does not go into the spindle correctly because you didn't have it aligned correctly then securely grab the tool holder, press the **TOOL RELEASE** button and repeat the steps mentioned above until the tool holder is inserted into the spindle correctly.

ATC FWD / ATC REVERSE / NEXT TOOL BUTTONS

The **ATC FWD** button, the **ATC REVERSE** button or the **NEXT TOOL** button will activate the automatic tool changer.

Pressing **ATC FWD** or **ATC REVERSE** or the **NEXT TOOL** button will cause the machine to remove the tool out of the spindle and place the next tool that's in the carousal into the spindle.

HOW TO REMOVE A TOOL FROM THE SPINDLE

To remove the tool holder / tool from the spindle, securely grab the tool holder.

Keep your fingers away from the cutting edge of the tool or you may cut yourself.

Press the **TOOL RELEASE** button that's located by the spindle and remove the tool holder.

Do not remove the tool by pressing the **TOOL RELEASE** button that's located on the control panel, it's safer and easier to use the **TOOL RELEASE** button that's located next to the spindle.

CALLING UP A TOOL FROM THE TOOL CAROUSEL

There will be times when you need to call up a tool from the tool carousel and have the machine put the tool in the spindle.

You may need to remove a tool from the carousel, so that you can put a different type of tool holder or tool in the machine.

Before calling up the tool make a note of what tool number is written on the carousel, next to tool that you want to put in the machine spindle

To call up the tool that has a number 1 written next to it and have the machine insert this tool into the spindle, follow the steps below.

Press the **MDI** button.

Enter **P1**

P1 means pocket #1

Press the **ATC FORWARD** button or the **NEXT TOOL** button.

The tool that is in **pocket #1** will be placed in the spindle.

Be careful not to take a tool out of the tool carousel that you have already measured the tool length offset on, a tool that will be used to machine the part.

http://youtu.be/kkGQuBYEGMM
LOADING TOOLS IN SPINDLE

http://youtu.be/dTY-vYi5BxE
TOOL SETUP USING A PROBE

https://youtu.be/7sM_IgO6N3I
TOOL HOLDERS

https://youtu.be/r_MJf8_VYVY

SETTING UP AND LOADING TOOLS

THE MACHINE VISE

Before installing the vise, remove the chips off the table with a brush, air gun and a t slot tool.

The t slot tool is used to remove the chips out of the t slots, the t slots are the slots that are machined into the machine table.

Wipe the table clean with a rag.

Use a fine grit stone and oil to smooth out the machine table, doing this will allow the vise to sit as flat as it can against the table.

If possible, place the vise at the center of the table.

Now you must align the vise so that it is square to the table.

You want the bolts that hold the vise to the machine table to be loose enough so that you can move the vise around with your hands.

Mount a dial indicator in the spindle, do not turn the spindle on!

Jog down in Z until the **tip** of the dial indicator is about 1/8th of an inch above the front edge of the fixed vise jaw. **Note:** the **tip** is also called the **probe.** See page 25

You do not want the indicator tip to be touching the fixed vise jaw.

Jog the table in the X axis to see how much the indicator tip varies in distance from the left side of the fixed vise jaw to the right side of the fixed vise jaw.

Move the vise around and get it close to being square to the machine table.

When you jog the X axis, you want the dial indicator tip to be roughly the same distance from one side of the fixed vise jaw to the other side of the fixed vise jaw.

As soon as you get the vise close to being square, within .015 or so, slightly tighten the vise bolts.

Jog the tip of the indicator down in Z so it’s about one eighth inch below the top edge of the fixed vise jaw.

Jog the Y axis until the tip of the indicator touches the fixed vise jaw and the needle of the indicator turns about 90 degrees.

Zero out the indicator.

Jog the X axis to see how square the vise is to the table.

CCW needle movement on the dial indicator means that side of the fixed vise jaw is angling away from you, it needs to be moved a little bit towards you.

CW needle movement on the indicator means that side of the fixed vise jaw is angling towards you, it needs to be moved a little bit away from you.

Think about what is happening as the needle on the indicator is moving, this will help you determine which side of the vise needs to be hit with the hammer.

If the vise isn't square, lightly hit the end of the vise that needs to be adjusted, keep doing this until you get the vise square with the table.

Make small adjustments at a time, about half of what the dial indicator reads.

As soon as you get the vise square to the machine table, to within a thou or less, tighten the 2 vise bolts.

After tightening the vise bolts, recheck the vise once more with the dial indicator to make sure that the vise is still square.

Occasionally, loosen the vise bolts, lift the vise up and clean under it, at this same time you may also want to stone the bottom of the vice.

NOTE: You can use some of the steps mentioned above to square up a piece of steel or aluminum stock material before you clamp it down to the table.

https://youtu.be/FelgkWIXKr0
INSTALLING A SET OF PARALLELS

https://youtu.be/P4QkO-m3t9w
INSTALLING A VISE

https://youtu.be/cPvOKX5PeBo
MACHINIST STONE

https://youtu.be/cn05fX55pqc

SQUARING A VISE

https://youtu.be/r50TYp98Vgk

SQUARING A VISE

RUNNING THE HAAS SPINDLE / WARM-UP PROGRAM

If the CNC machine has not been used for several hours you must warm up the spindle before machining a part.

Press the **LIST PROGRAMS** button.

Cursor to the Haas **O2020** spindle warm-up program.

Press the **SELECT PROG** button.

Press the **MEMORY** button.

The spindle warm-up program will be displayed by the left side of the screen.

It doesn't matter whether there's a tool holder in the spindle.

To start the spindle warm-up program, first make sure the doors are closed and then press the **CYCLE START** button, the spindle warm up program will begin.

CREATING A SPINDLE WARM-UP PROGRAM IN MDI

If the CNC machine your using does not have a spindle warm-up program, you can warm up the spindle by entering a short program in MDI.

Press the **MDI** button**.**

Enter S1500 M3

It doesn't matter whether there's a tool holder in the spindle.

Press the **CYCLE START** button.

The spindle should be turning clockwise at 1500 RPM

Let the spindle warm-up program run for approximately 15 minutes and then press the **SPINDLE STOP** button.

The spindle is now warmed up and ready to machine parts.

https://youtu.be/BZSLc42XbjY
YouTube Video SPINDLE WARM UP

JOG MODE

Jog mode allows you to jog one, two or all three axes, X, Y, Z to a specific location.

THERE ARE TWO WAYS TO JOG AN AXIS

The first way to jog an axis is to press the **HANDLE JOG** button.

Press one of the **HANDLE JOG INCREMENT** buttons, press either the .0001 button, the .001 button, the .01 button or the .1 button. Be careful when using the .1 button, it will jog the machine table / spindle at a high feed rate.

HANDLE JOG INCREMENT BUTTONS

After pressing one of the **HANDLE JOG INCREMENT** buttons, press one of the axis jog buttons, displayed below. Press the axis button that you want to jog in.

AXIS KEYPAD JOG BUTTONS

After you press the axis button that you want to jog in, you're ready to jog.

To start jogging an axis, turn the **HANDLE JOG WHEEL** either + for clockwise jogging or – for counterclockwise jogging.

Each click on the **HANDLE JOG WHEEL** will move the selected axis .0001 .001 .010 or .100 of an inch, be careful when using the .1 button, it will jog the machine table / spindle at a high feed rate.

THE SECOND WAY TO JOG AN AXIS

The second way to jog an axis is used when you want to manually machine the stock material. It is also used when you are touching a tool off to the electronic tool probe, to set a tool length offset or a work offset.

Press the **HANDLE JOG** button.

Press one of the **HANDLE JOG INCREMENT** button.

Press either the .1 the 1. the 10. or the 100 **HANDLE JOG INCREMENT** button.

.1 is .100 thousand of an inch per minute.	Jogs the axis at .100 of an inch per minute.
1. is inch per minute.	Jogs the axis at one inch per minute.
10. is 10 inches per minute.	Jogs the axis at 10 inches per minute.
100. is 100 inches per minute.	Jogs the axis at 100 inches per minute.

On the axis keypad, press the axis button that you want to jog in.

As you are pressing in the axis button, the axis will jog at the feed-rate that you selected from the **HANDLE JOG INCREMENT** buttons.

https://youtu.be/UoGj9crlXIU

THE JOG BUTTON

WHY YOU NEED TO SET THE WORK OFFSETS

The program zero location on the part is where you set the G54 X and Y work offsets at.

In absolute programming all tool machining motion begins from the program zero location that has been assigned to the part.

The machine home position is a fixed X, Y and Z starting point on the CNC machine, the machine home position is where all tool motion is measured from.

This technical stuff really isn't all that important to you, what is important is that you set the G54 X and Y work offsets correctly.

After you set the X and Y work offsets the machine will know where program zero is located on the part, relative to the machine home position.

The CNC machine must know the work offset distance so that all the tools can move to the correct locations on the part / stock material and then machine the part features.

The X work offset is often located at the left side of the stock material, this is where the stock material bumps up to the bump stop.

The Y work offset is often located against the solid jaw. There is less chance that the Y work offset location will be bumped out of position when the part is against the fixed / solid jaw of the vise.

Below are step by step instructions on how to set the X and Y work offsets
I explain how to set the work offsets using an edge finder, newer Haas machines use an electronic probe to set the work offsets. An electronic probe is quicker and more accurate than an edge finder. But for most machining jobs an edge finder will be plenty accurate.

SETTING THE G54 X WORK OFFSET ON A HAAS VMC

Put an edge finder in the machine spindle.

Press the **MDI** button.

Enter **S1000 M3**

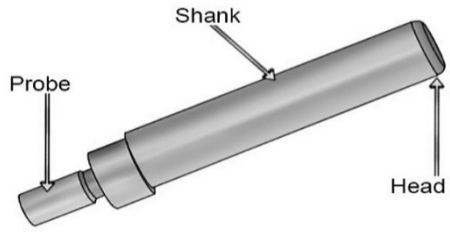
Press the **CYCLE START** button.

The edge finder should be turning clockwise at 1000 rpm.

Tap on the probe with your finger so that it wiggles as its turning.

If the doors must be closed before the spindle can be turned on, offset the probe from the shank by moving the probe off center from the shank, move the probe off center from the shank by pushing on it with your finger.

Offset the probe from the shank before you close the doors and turn the spindle on**.**



Pictured above is an edge finder / The probe is also referred to as the tip.

You want the probe to wiggle as it's turning.

The instructions that I explained to you about getting the probe to wiggle first is how I do it. You can also get the probe to wiggle by jogging the probe a little ways into the edge of the stock. You then back the probe away from the edge and jog it back into to the edge to find the center of the edge, the work offset location.

Keep pressing the **OFSET** button until the work offset page displays.

Scroll over and highlight the **G54 X WORK OFFSET.**

Press the **HANDLE JOG** button.

Press the **.01 HANDLE JOG INCREMENT** button.

Press the **Z AXIS JOG** button.

Turn the **HANDLE JOG WHEEL** and jog down in Z until the edge finder is a couple inches above the stock material / part.

Press the **X AXIS JOG** button.

Jog the edge finder approximately a half inch away from the left side of the X edge of the stock.

You may also have to jog the Y axis so you can position the edge finder to where it needs to be. You will then be able to jog the edge finder to the stock materials X edge.

Press the **Z AXIS JOG** button.

Keep jogging the edge finder down until the bottom of the probe is approximately .100 thousands below the top of the stock material.

Press the **.001 HANDLE JOG INCREMENT** button.

Press the **X AXIS JOG** button.

Slowly turn the **HANDLE JOG WHEEL** clockwise until the probe touches the edge of the stock material, as soon as the probe and shank run true to each other, stop jogging.

Press the **.0001 HANDLE JOG INCREMENT** button.

Slowly turn the **HANDLE JOG WHEEL** clockwise, as soon as the probe separates from the shank, stop turning the **HANDLE JOG WHEEL.**

Write down the X work offset value that is displayed on the screen.

Again, jog over in the X axis to make sure that the X work offset value that displayed on the screen before displays the same value.

If the X work offset value displays the same value, to within .001 or so, your good.

Do not move the X axis**.**

If the work offset value is off by more than .001 then repeat steps starting at / Press the **X AXIS JOG** button.

Press the **Z AXIS JOG** button.

Jog up in the Z axis until the bottom of the probe is approximately .100 above the stock.

Press the **X AXIS JOG** button.

Press the **.1 HANDLE JOG INCREMENT** button.

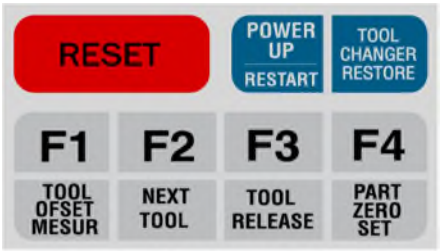
Turn the **HANDLE JOG WHEEL** clockwise one increment / one click to the right.

The middle of the probe should position itself exactly over the X edge of the stock material.

Look closely to make sure that the probe is positioned exactly over the X edge of the stock material.

On the display screen make sure that the **G54 X WORK OFFSET** is highlighted.

Press the **PART ZERO SET** button**.**



The G54 X work offset distance / location has now been recorded into the machine.

SETTING THE G54 Y WORK OFFSET ON A HAAS VMC

Put an edge finder in the machine spindle.

Press the **MDI** button.

Enter **S1000 M3**

Press the **CYCLE START** button.

The spindle should be turning clockwise at 1000 rpm.

Tap on the probe with your finger so it wiggles as its turning.

If the doors must be closed before the spindle can be turned on, then before closing the doors and pressing the **CYCLE START** button, offset the probe from the shank by pushing on the probe with your finger.

You want the probe to be off center from the shank so that the probe wiggles as the edge finder is turning.

Keep pressing the **OFSET** button until the work offset page displays.

Scroll over and highlight the **G54 Y WORK OFFSET**.

Press the **HANDLE JOG** button.

Press the **.010 HANDLE JOG INCREMENT** button.

Press the **Z AXIS JOG** button

Turn the **HANDLE JOG WHEEL** and lower the edge finder until it is approximately a couple inches above the stock material / part.

Press the **Y AXIS JOG** button.

Jog the edge finder approximately a half inch behind the back side of the Y edge of the stock.

You may also have to jog the X axis so you can position the edge finder to where it needs to be.

You will then be able to jog the edge finder to the stock materials Y edge.

Press the **Z AXIS JOG** button.

Jog the edge finder down in Z until the bottom of the probe is approximately .100 below the top of the stock material / part.

Press the **.001 HANDLE JOG INCREMENT** button.

Press the **Y AXIS JOG** button.

Keep jogging the edge finder slowly towards you, in the Y axis, until the probe touches the edge of the stock material.

As soon as the probe and shank run true to each other stop jogging.

Press the **.0001 HANDLE JOG INCREMENT** button.

Continue jogging the edge finder slowly into the Y edge of the stock until the probe and the shank separate, stop jogging as soon as the probe separates from the shank.

Write down the Y work offset value that's displayed on the screen.

Jog over in Y again to make sure that the Y work offset value that displayed on the screen before displays the same value.

If the Y work offset value displays the same value, to within .001 or so, your good.

Do not move the Y axis.

Press the **Z AXIS JOG** button.

Press the **.010 HANDLE JOG INCREMENT** button.

Jog up in Z until the bottom of the probe is approximately .100 above the stock.

Press the **Y AXIS JOG** button.

Press the **.1 HANDLE JOG INCREMENT** button.

Turn the **HANDLE JOG WHEEL** one increment to the right.

Look closely to make sure that the center of the probe is positioned exactly over the X edge of the stock material.

On the display screen make sure that the **G54 Y WORK OFFSET** is highlighted.

Press the **PART ZERO SET** button.

The G54 Y work offset distance / location has now been recorded into the machine control.

Once you get experienced at setting the X and Y work offsets you will be able to set them in a couple minutes.

https://youtu.be/AyMsFtwzrmI
SETTING WORK OFFSETS WITH AN EDGE FINDER

https://youtu.be/_kZlQdr7iJ0
SETTING TOOL AND WORK OFFSETS

http://youtu.be/X9vhZhcpN3I
WORK OFFSETS

https://youtu.be/58Hd5e-36hE
SETTING WORK OFFSETS

https://youtu.be/Swn45rMa8W0
SETTING WORK OFFSET WITH PROBE

https://youtu.be/sw1X2YNzRnA
SETTING WORK OFFSET WITH PROBE

https://youtu.be/JzpjVG8SE8g
USING A TOOL PROBE ON A VMC

https://youtu.be/hq45VlHtw0o
HOW TO FIND A CENTER OF A HOLE

HAVE THE G54 X AND Y WORK OFFSETS BEEN SET CORRECTLY

The safety program below is often used by newbies to see if the X and Y work offsets have been set correctly. Experienced CNC setup / operators have a good idea that the work offsets have been set correctly if the tool positions itself to the correct location on the part before it starts machining the part feature.

Put a small diameter drill in the spindle or a small diameter tool that comes to a point.

By using a small diameter drill or a small tool that has a pointed end, it's easy to know if the drill positions itself over the X0, Y0, G54 work offset location on the part.

Press the **MDI** button.

Enter **GOG54X0Y0**

Press the **WRITE ENTER** button.

Turn the **RAPID SWITCH** to **5%**

Press the **CYCLE START** button.

Be ready to press the **FEED HOLD** button if it looks like a crash may occur.

After the small drill or pointed tool has positioned itself to the program zero location on the part press the **HANDLE JOG** button.

Press the **.010 JOG INCREMENT** button.

Press the **Z AXIS JOG** button.

Turn the **HANDLE JOG WHEEL** and jog the drill down until it's approximately .100 above the part.

If the G54 X and Y work offsets have been set correctly then the drill should have positioned itself at the program zero location on the part, which in this case happens be located at the left, back corner of the part.

Look at the G54 work offset values that are displayed on the screen, X should display a zero and Y should display a zero.

If the work offsets have been set correctly then all the tools in the program will know where program zero is located on the stock material / part.

If the G54 work offset is not correct, then put the edge finder back in the spindle and set the X and Y G54 work offsets over again.

TOOL LENGTH OFFSET INFORMATION

NOTE: Tool length offset, and tool height offset mean the same thing.

When you set up the CNC machine to run a program and machine the part you must measure the tool length offsets for all the tools that are going to machine the part.

The tool length offset values for each tool are listed on the tool length offset page.

The tools used for machining the part vary in length.

The CNC machine must know how much to adjust itself for these different variations in tool lengths.

After all the tool length offset values have been entered in the machine, the tools will be able to machine all the part features to the correct depths.

The machine, in a sense, acts as if all the tools are the same length.

For a tool to cut to the correct depth the machine must know the distance from the bottom of the tool, starting from machine home, to the program Z0 location on the part.

The top or bottom of the stock material is usually the program Z0 location.

The H address code in the program tells the machine how far to offset the tool.

If the CNC machine reads H1 in the program, it's going to offset tool #1 by the distance that was measured and recorded into the CNC machine for tool #1

Measuring the tool length offset for a tool takes a minute or less and it's easy to do.

SETTING A TOOL LENGTH OFFSET ON A HAAS VMC

Below I explain how to set a tool length offset using a tool setter gauge like the one pictured below.

Newer Haas machines have an electronic probe that is used for setting tool length offsets.

Keep pressing the **OFSET** button until you see the work zero offset page.

Make sure the G54 Z value is set to 0

If it isn't set to 0, then highlight the Z value and enter 0

Press the **F1** button to lock the 0 value into the machine control.

Double check to make sure the G54 Z work offset value is now set to 0

TOOL SETTER GAUGE

HOW TO MEASURE A TOOL LENGTH OFFSET

Place the tool setter gauge on the stock material or on the machine table, where you place the tool setter gauge depends on where program Z zero is located on the part.

If program Z zero is located at the top of the part, then place the tool setter gauge on the top of the stock material.

In this example you will measure the tool length offset for tool #1

Load tool #1 into the spindle.

Press the **HANDLE JOG** button.

Press the **.001 HANDLE JOG INCREMENTAL** button.

Press the **Z AXIS JOG** button.

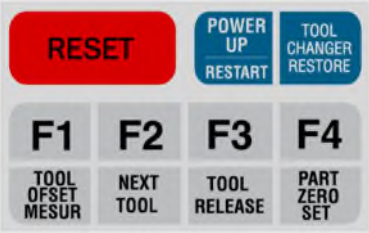
Slowly jog tool #1 down until it just touches the top of the tool setter's plunger and the needle moves a hair.

Press the **.0001 HANDLE JOG INCREMENTAL** button.

Keep jogging the tool slowly down until the needle on the tool setter's gauge reads zero.

Keep pressing the **OFSET** button until the tool length offset page displays.

Keep scrolling up or down until the tool number that your setting matches the tool length offset number.



Press the **TOOL OFSET MESUR** button.

Enter – 2.000 inches or enter whatever value you need to enter that will put the tool at the Z zero location on the part.

You enter 2.000 inches because 2 inches is the height of the tool setter gauge and Z zero is located at the top of the stock material / part.

Press the **WRITE ENTER** button.

If the height of the tool setter gage is 4 inches you would enter -4.000 and then would press the **WRITE ENTER** button.

After pressing the **WRITE ENTER** button make sure that there is a 2-inch difference between the top tool length offset number and the bottom tool length offset number that is displayed on the screen.

After pressing the **WRITE ENTER** button look at the screen to see if the tool length offset number that was entered looks correct, make sure it's a negative number and that the number does not look like it is way off, compare it to the other tool length offset values that are displayed on the screen.

If the machine asked you to bring up the next tool to be measured you can press Y for yes, that is, if you have another tool that needs to be measured.

Measure the tool lengths for all the tools that are going to machine the part.

Do not use the **JOG HANDLE WHEEL** to move the tool up after pressing the **TOOL OFSET MESUR** button.

You may accidently turn the **JOG HANDLE WHEEL** in the wrong direction and crash the tool into the tool setter's plunger and damage the tool setter gauge.

Press and hold in the **+ Z** arrow button that's located above the **JOG LOCK** button and the tool will safely move up and out of the way.

Or press the **ATC FORWARD** button or the **NEXT TOOL** button while in MDI and the tool will move up to the tool change position.

TOOL LENGTH OFFSET SAFETY PROGRAM

This safety program is often used by newbies, it will let them know whether the tool length offsets are set correctly.

An experienced CNC setup person may skip this safety program and will just keep a finger on the feed hold button.

If a tool does not move down in the Z axis to the correct location on the part before machining occurs, they press the **FEED HOLD** button and then find out what the problem is.

If a crash occurs or is about to occur, they press the emergency stop button.

If a tool did not move down to the correct Z height above the part then the tool's, tool length offset value is probably wrong.

The program below will let you know if the tool length offset value that is entered in the machine control for tool #1 is correct.

If tool #1 isn't in the spindle, then press the **MDI** button and enter **T1**

Press the **ATC FORWARD** button or the **NEXT TOOL** button and tool #1 will be inserted into the spindle.

Press the **MDI** button.

Enter **5%GOG43H1Z2**.

Press the **WRITE ENTER** button.

Press the **CYCLE START** button.

Keep a finger on the **FEED HOLD** button, but don't press it.

If the tool moves closer than 2 inches above the part, quickly press the **FEED HOLD** button.

The tool should stop precisely 2 inches above the stock material.

If the tool did not stop precisely 2 inches above the part then you need to re-measure the tool length offset for tool #1, because its offset value isn't correct.

Look closely to make sure the tool is precisely 2 inches above the stock material.

Slide the 2-inch side of a 1,2,3 block under the tool to see if the tool is exactly 2 inches above the stock material.

Run this safety program for all the tools that are going to machine the part.

Each time you run this safety program you will change the tool height offset number.

For example, the second tool that you check to see if the tool length offset is correct, that is, if it is tool #2, would have H2 written in the safety program.

H2 tells the CNC machine that tool #2 will be offset by the distance that was entered in the machine, this offset distance can be viewed on the tool length offset page.

Tool #2 offset value was entered in machine control when the tool was touched off to the tool setter gauge and the **TOOL OFSET MESUR** button was pressed.

http://youtu.be/BFY6J-vUhMw

TOOL LENGTH OFFSETS

https://youtu.be/J7dCwBkUNNU

TOOL LENGTH OFFSETS

BEFORE CLAMPING THE STOCK MATERIAL IN THE WORKHOLDER

Before clamping the stock material in the vise or fixture measure the stock to see if it has been cut to the correct size.

Measure the length, width and thickness of the stock.

If there are any burrs on the stock, file them off before clamping the stock material in the vise or fixture.

Before clamping the stock in the vise or fixture check the stock material with a machinist square to make sure that it is square.

The person who cut the stock material may have not cut it square.

There may be .010 thousands or more size variation from one end of the stock material to the other end.

It will be difficult or impossible to clamp the stock material so that it sits flat or square in the vise or on the parallels.

Make sure the vise or fixture is bolted securely to the table.

Make sure the parallels are the correct height and width.

Before placing the stock material on the parallels, inspect the parallels to make sure that there aren't any burrs or nicks on them, if there are, filing them off.

CLAMPING THE STOCK MATERIAL IN THE VISE OR FIXTURE

Clamp the stock material securely in the vise or fixture.

Certain precision parts may need this procedure done / After clamping the stock material in the vise, tap the material down with a mallet so the part sits tight and flat against the parallels.

Tug on the parallels to make sure that they don't move.

If the parallels do not move, do not tighten the vise anymore, if you tighten the vise more, the stock material may move, and it may not sit flat and tight against the parallels.

After clamping the stock material in the vise or on parallels, place a machinist square up against the side of the stock material to make sure that the stock is not tilted in the vise or on the parallels.

LOCATING THE STOCK FROM A BUMP STOP

After clamping the part in the vise, you may have to move the bump stop down so that it is out of the way. This will prevent the bump stop from getting crashed into by any of the tools that will be machining the part.

MARKING A PART FEATURE

To help verify that a part feature is going to be machined at the correct location you can scribe lines on the stock material.

The distance to a part feature usually starts from the edge of the stock material and goes to the middle of the part feature.

Set your caliper to the X distance that's shown on the print.

Scribe a line on the stock material with your caliper, this scribed line is the X axis distance to the middle of the part feature.

Set your caliper to the Y distance that's shown on the print.

Scribe a line on the stock material with your caliper, this scribed line is the Y axis distance to the middle of the part feature.

With a black magic marker place a dot at where the two lines intersect, this is the center of the part feature, this location is where the tool should move to before machining occurs.

The part feature that you usually want to scribe your X and Y lines to is where the spot drill will spot drill the first location where a hole is going to be drilled.

Many times, the spot drill is the first tool to be used, that is unless the stock material is going to be face milled first.

If the stock material is going to be face milled first then mark the front edge of the part at either the right or left side, where the first spot drilling will occur.

If the spot drill does not move to the location that you marked with the magic marker then quickly press the **FEED HOLD** button before the spot drill starts spot drilling.

If the spot drill moves to where the first hole is supposed to be spot drilled at then chances are good that all the other spot drill locations will be correct.

If the spot drill did not move to the correct location on the part, it may be that:

1. The stock material is not oriented correctly in the vise or the stock may be cut to the wrong size.

2. One or both work offsets is incorrect.

3. There's a problem with the program.

USE THIS METHOD IF THE FIRST TOOL IS A SPOT DRILL

Another method that I occasionally use to verify that the work offset is correct is to change the spot drills, tool height offset value in the machine.

I'll raise the spot drills tool height offset up, a positive .100 thousands of an inch.

When the spot drill spots the first hole location it will only leave a tiny mark in the steel or aluminum.

After the first hole has been lightly spot drilled, stop the machine and check to see if the spot drill has spot drilled at the correct location.

If the spot drill has spotted at the correct location then home the machine and change the spot drills tool height offset down a negative -.100, back to its original value, and then run the program.
If the spot drill moves to the correct location where the first spot drilling will occur, then chances are good that all the other locations that are going to be spot drilled will be at the correct locations.

Another method that I use to see if a tool is moving to the correct location on the part is to turn the coolant off as the tool is approaching the part feature. If the coolant is turned on its hard to see where the tool is at because the coolant is hiding the tool. If the tool looks like it has moved to the correct location on the part, I quickly turn the coolant back on before the tool starts machining the part feature.

BEFORE RUNNING THE PROGRAM / MACHINING THE PART / CHECK LIST

The stock material has been securely mounted to the table, vise or fixture.

The work offsets have been entered in the machine control.

The tool length offsets have been entered in the machine control.

The program Is good.

Press the **LIST PROGRAMS** button.

Scroll to the program that you want to run.

Press the **SELECT PROGRAMS** button.

Make sure that the correct program displays on the screen.

Set the **RAPID** to **5%**

Set the **FEED** to **5%**

Keep the **SPEED / RPM** set to **100%**

Make sure that the coolant valve is open.

Before machining the part add some positive tool diameter offset and add some positive tool length offset to the tool's that will be machining part features that have tight tolerances.

Before machining the part add .010 positive tool diameter offset to each tool that will be machining pockets and contours.

Adding .010 positive offset to the tool's diameter will prevent the tool from removing to much material from the part feature.

Add .010 positive tool height offset to each tool that will be machining part features that have depth, such as pockets and steps.

Adding .010 positive tool height offset to the tools will prevent the part features from being machined to deep.

As you gain more machining experience you will have a better idea on how much positive tool diameter and positive tool height offset to add to a tool before machining the part, you may only add .003 instead of .010

IT'S TIME TO MACHINE THE PART

Before pressing the **CYCLE START** button have a copy of the engineering drawing in your hand and a list of the tools that are going to machine the part.

Have the tools listed in order, from the first tool that's going to machine the part to the last tool that's going to machine the part, this way you will know what kind of machining the tool is going to do before it starts machining the part feature.

To start machining the part, press the **CYCLE START** button.

Make sure that there is plenty of coolant hitting all the tools as they are machining the part.

Press the **EMERGENCY STOP** button if shit starts hitting the fan.

Keep a finger on the **FEED HOLD** button and be ready to push it if a problem occurs.

Make sure that the first tool that's going to machine the part moves to the correct location on the stock material before it starts machining.

If the first tool in the program does not move to the correct location on the part, press the **FEED HOLD** button**.**

Send the spindle and table back to machine home and find out why the first tool moved to the wrong location on the stock material / part.

REASONS WHY THE TOOL MOVED TO THE WRONG LOCATION ON THE PART

The G54 work offset or work offsets are not set correctly.
The stock material is not oriented correctly in the vise or fixture.
The stock material is the wrong size.
There's a problem with the program.

After you have figured out why the tool did not move to the correct location on the part, start the program over again.

Press the **CYCLE START** button.

Just because you may have performed a dry run and proved out the program doesn't mean that everything is good.

Some feeds and or speeds may not be correct or there could be some other problem with the program or setup, so use caution!

On the display screen pay attention to the distance to go and to the program code.

Pay attention to the speed and feed of the tools to make sure that they are correct.

Use your ear as an indicator that something may not be correct with the way the part is being machined.

After you get the CNC machine dialed in and its machining the part as good as it can be machined, document any feed or speed changes that you made so you will have a record of which ones work best when running the program / machining the part.

Document any other changes you may have made to the program or to the set up.

After you are sure that the part has been machined correctly, turn the **RAPID** and **FEED** up to **100%** and start mass producing widgets / parts.

THE FEED HOLD BUTTON

You may press the feed hold button after starting a program to verify that the tool has moved to the correct location before the tool starts machining the first part feature.

You may press the feed hold button to verify that the tool isn't about to crash into something.

You may press the feed hold button because you need to measure the diameter or the depth of a part feature, to do this, press the feed hold button, press the spindle stop button, press the coolant button to turn the coolant off, open the doors and measure the diameter or depth of the part feature.

You may press the feed hold button because you must adjust a tool offset.

You may press the feed hold button because you must remove chips from the machine.

You may press the feed hold button because you must remove a piece of metal that has been machined off the stock material.

https://youtu.be/WWW5r5C0YnU
THE CYCLE START & FEED HOLD BUTTONS

WHAT TO DO IF THE MACHINE CRASHES

If the part, vise, fixture or machine table gets crashed into by the tool, the G54 X and or Y work offsets may have been moved out of position.

The program zero location on the part may not relate to the work offset values that are stored in the machine.

You may have to realign the stock material, vice or fixture with a dial indicator so that its square to the machine table.

The G54 X and or Y work offsets will then have to be re-entered into the machine.

THE RECOVER / RESTORE BUTTON

The **RECOVER RESTORE** button accesses a step by step recovery procedure that you use if a tool change is interrupted by a crash, jam, malfunction or if the power goes out while the machine is running.

TO STOP THE MACHINE FROM MACHINING THE PART

Wait for the next tool change or for the tool to retract out of the hole or cut and then press the **FEED HOLD** button.

Pressing the feed hold button will stop all axis motion but the spindle will continue to turn.

Press the **SPINDLE STOP** button to stop the spindle from turning.

Press the **COOLANT** button to turn the coolant off.

Open the door and remove the chips or do whatever you need to do.

TO START MACHINING AGAIN AFTER PRESSING THE FEED HOLD BUTTON

Close the doors and press the **SPINDLE CLOCKWISE** button, the spindle will start turning clockwise.

Press the **COOLANT** button and the coolant will start flowing.

Make sure that the coolant is hitting the tool.

Press the **CYCLE START** button and watch the tool to make sure its machining correctly.

WHAT TO DO AFTER MACHINING THE FIRST PART

After machining the first part leave the part in the vise or fixture.

De-burr the part features that have tight tolerances.

Measure the part features to see if they are in tolerance.

If a part feature has had to much material removed from it and the dimension is out of tolerance the part is probably scrap.

If a part feature has not had enough material removed from it and the dimension is out of tolerance, the part feature will need to have more material machined off it.

Bring up the tool offset page.

Highlight the tool that machined the out of tolerance part feature, make sure that the tool number matches the offset number that you are using.

Make an offset adjustment to the tool’s diameter or to the tool’s length.

If both the diameter and length of the tool need to be adjusted, then adjust both.

Make the offset adjustment to the tool that machined the out of tolerance dimension on the part.

After adjusting the tool offset or offsets, rerun the tool over again and the dimension or dimensions on the part feature will be machined to the correct size.

TOOL DIAMETER OFFSET INFORMATION

To make an end mill machine a smaller diameter pocket then what is shown on the engineering drawing you must add some positive offset to the tool's diameter, .003 to .010 thousands of an inch are normally added to the tool's diameter offset numerical value.

To make an end mill machine a larger diameter pocket then what is shown on the engineering drawing you must add some negative diameter offset to the tool's diameter, negative -.003 to -.010 thousands of an inch are added to the tool's diameter offset numerical value.

Let's say that the CNC machine is going to machine a 1.000-inch diameter pocket.

The 1.000-inch diameter pocket has a tolerance of plus or minus .001 thousands of an inch.

Before machining the part, add .010 of positive offset to the tool's diameter offset numerical value.

Adding .010 positive offset to the tool's diameter will prevent the pocket from being machined to big.

Now theoretically, if the end mill is exactly the size that the program calls for and the program is correct and there is no tool vibration, the part feature should be machined to the correct size. You would not have to add diameter offset to the tool, unfortunately in the real world of machining you must add some positive diameter offset to the end mill or the part feature may be machined too large.

As you gain more machining experience you will have a better idea on how much positive tool offset to add to the tool before machining the part, you may add only .003 instead of .010

OTHER REASONS WHY YOU OFFSET THE TOOL'S DIAMETER

You may offset the tool's diameter a bit so that you can make slight adjustments for tool deflection.

You can also do a fine finish pass on a part feature by offsetting the tool a little bit.

HOW TO MAKE AN UNDERSIZE END MILL CUT TO SIZE

The program calls for tool #1 to be a .500 diameter end mill and all you have is a re-sharpened end mill that measures .475 in diameter.

Keep pressing the **OFFSET** button until the tool diameter offset page displays.

Highlight tool #1 and offset #1

Add negative -.025 to tool number one's diameter offset numerical value.

Adding negative -.025 will make the .475 diameter end mill machine like it's a .500 diameter end mill.

The .475 diameter end mill will machine the part features to the correct size.

When replacing a worn tool with a new tool, the wear offset column should be cleared.

To clear the wear offset column, cursor to the wear offset value for the tool that you want to zero out and enter 0

Press F1 to change the offset value to 0

ADJUSTING THE DEPTH OF A TOOL

Features on the part that may need their dimensions adjusted in the Z axis are drilled holes and milled pockets.

If a part feature has not been machined deep enough or it has been machined to deep you will need to bring up the tool offset page.

Keep pressing the **OFSET** button until the tool offset page displays.

Highlight the tool that machined the incorrect depth.

Make sure that the tool number matches the correct offset number, tool #1 will usually use offset #1

If a part feature needs to be machined .010 deeper, then enter negative -.010 to the tool's, tool length offset numerical value, you are adding a negative -.010 value to the tool length offset.

Rerun the tool over again and the tool will machine the part feature to the correct depth.

If a part feature has been machined to deep, the part may be scrap.

Add the required positive offset value that will allow the tool to machine the part feature to the correct depth, add this positive offset value to the tool's, tool length offset numerical value.

The next part that you run will machine the part feature to the correct depth.

After running the tool over again, measure the feature or features that were previously machined too deep or too shallow to make sure that they are now at the correct depth.

SURFACE FINISH

Make sure the surface finish on the part is in tolerance to what's stated on the print.

If there's a problem with the surface finish you may have to adjust the feed or speed of the tool.

You may have to tweak the work holder so that the stock material is more rigid in the vise or fixture, this may help dampen vibration.

If you do not have a nice surface finish on the part the tool may be dull.

HOW TO MANUALLY FACE MILL THE STOCK

Yes, you can also do manual machining with a vertical machining center, here's how.

In this example you're going to face mill .010 thousands off the top of the stock material.

Put the face mill that you want to use into the spindle.

Set the correct RPM for the face mill, I'm going to have the end mill turn at 1000 rpm.

Press the **MDI** button.

Enter **S1000 M3**

Press the **CYCLE START** button.

The spindle should be turning clockwise at 1000 RPM

Manually jog the face mill a couple inches to the left side of the stock.

Jog the face mill down until its .010 of an inch below the top surface of the stock.

Press the **HANDLE JOG 10. INCREMENT** button.

Press the negative - X axis button.

You pressed the -X button because you want to face mill to start cutting from the left side of the stock material and move to the right side of the stock material.

Turn the coolant on by pressing the **COOLNT** button.

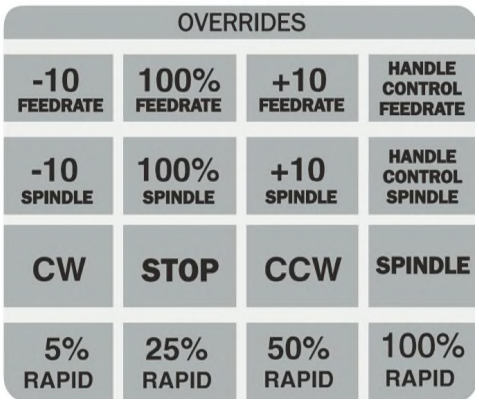
Press the **JOG LOCK** button and the face mill will automatically machine .010 of an inch off the face of the stock at 10 inches per minute.

To stop machining the part, press the **JOG LOCK** button or wait for the machine to run out of axis travel and it will stop automatically.

After machining the part, press the **SPINDLE STOP** button and the spindle will stop turning.

Press the **COOLNT** button and the coolant will stop flowing.

OVERRIDE BUTTONS



The **OVERRIDE** buttons allow you to vary the rapid traverse motion as well as vary the programmed feeds and speeds while the machine is running.

SPINDLE SPEED OVERRIDES

While the machine is running the spindle speed can be varied from 10% to 150% by using the **SPINDLE OVERRIDE** buttons.

Each time you push the **-10 SPINDLE** button the spindle speed / the RPM decreases by 10%

Press the **100% SPINDLE** button to set the adjusted spindle speed back to the programmed spindle speed.

Each time you push the **+10 SPINDLE** button the programmed spindle speed / the RPM increases by **10%**

FEED RATE OVERRIDES

While the machine is running the program the **FEEDRATE** / **IPM** can be varied from **10% to 200%**

Press the **FEEDRATE OVERRIDE** buttons to adjust the feed-rate.

Each time you push the **-10 FEEDRATE** button the feed-rate that is written in the program decreases by 10%

Pressing the **100% FEEDRATE** button will set the feed-rate back to the feed-rate that is written in the program.

Each time you press the **+10 FEEDRATE** button the feed-rate that is written in the program increases by 10%

Press the **CW** button and the spindle will start turning in a clockwise direction.

To stop the spindle from turning press the **STOP** button

Press the **CCW** button and the spindle will start turning in a counterclockwise direction.

OPERATOR RESPONSIBILITIES MAY INCLUDE THE FOLLOWING

Correctly load the stock material in the vise or fixture and press the cycle start button.

While the part is being machined the previously machined part will most likely need to have the burrs removed.

Measure the dimensions on the part with your machinist tools to make sure they are within tolerance.

Pack the parts according to company packing instructions.

Keep your work area clean and organized.

Follow company safety rules, wear ear plugs, safety glasses and safety shoes.

Keeping the CNC machine clean.

Turn on the chip auger, the chip auger pushes the chips onto the conveyor.

Turn on the conveyor so the metal chips can fall into a chip barrel.

Turn off the conveyor and chip auger at the end of your shift.

Empty and or replace the chip barrel when it gets full.

Count the quantity of parts that you made and document this information.

Machine the required amount of parts per shift that the company requires.

Adjust tool offsets when a dimension goes out of tolerance or is close to going out of tolerance.

Replace broken and dull tools and worn tool inserts.

Add coolant to the CNC machine.

Add oil to the CNC machine.

At the end of your shift pull the tool out of the spindle; leaving the tool holder in the spindle can cause the inside of the spindle and tool holder to rust.

THE BASIC STEPS TO MACHINING A PART

Power on the CNC machine and let the machine home itself.

Write a program in MDI or send a program to the CNC machine from a USB drive or from a computer.

Load the correct tools into the machine.

Enter the tool length offsets values into the machine by touching the tools off to a tool setter gauge or an electronic probe.

Set the part's program zero location; the G54 X and Y work offset values into the machine.

Run the program / part.

Shut the machine off.

Clean the CNC machine and work area.

TOOLS USED FOR MACHINING

Face Mills are used for quickly milling the top of the work piece to the correct thickness while leaving a nice surface finish.

End Mills are used for milling pockets and machining outside contours on the work piece / 2D machining.

Ball & Bullnose End Mills are typically used for 3D machining; these end mills are often used to machine molds.

A Spot Drill is used for accurately spotting the location where a hole will be drilled.

Drills under a half inch in diameter tend to move off their location while they start drilling the hole.

When a spot drill is used first, followed by the drill, the drill will not move off its location, it will drill the hole precisely where it's supposed to be drilled at.

The spot drill does not wander off its location because it is short and stubby, it will not bend and wander off its location like a little diameter drill will.

Therefore, the location where the hole is going to be drilled is first spot drilled and then drilled.

Spot drills often spot and chamfer at the same time, when the drill, drills the spot where the spot drill had spot drilled, the hole is already chamfered.

Drills are used for drilling different size holes in the work piece.

Reamers machine a drilled hole to a precision diameter size. A reamer will machine a smoother and rounder hole compared to what a drill can do. A reamer can machine a hole’s diameter size to within .001 of an inch or less.

Taps machine threads in drilled or milled holes.

SHOP SAFETY

Never grab an end mill by its cutting edge.

Slivers are a bitch, one way to help avoid them is to not be a CNC machinist.

Be careful of sharp burrs on the edges off machined surfaces.

Keep a couple band aids in your wallet so you don't have to hunt for one after you get cut.

Do not crawl inside the CNC machine!!

THE AUTO ALL AXES BUTTON

Pressing the **AUTO ALL AXES** button will move all axes to machine zero. The Z axis will position first to machine zero and then the X and Y axis will position to machine zero.

THE ORIGIN BUTTON

Pressing the **ORIGIN** button will zero out various displays and timers on the control.

THE ZERO SINGL AXIS BUTTON

Enter either X, Y or Z and then press the **ZERO SINGL AXIS** button, the axis that you entered will move to machine zero.

THE HOME G28 BUTTON

Pressing the **HOME G28** button will cause all axes to rapid back to machine zero, machine zero is the machines home position, its starting location from where it first moves from.

The Z axis / the spindle will rapid up to machine zero first and then the X and Y axis / the table will rapid to machine zero.

THE MACHINE HOME POSITION

At the beginning of setting up a CNC machine, the machine must be homed.

You press the **POWER UP RESTART** button when powering on the machine.

When you press the **POWER UP RESTART** button the three axes, the spindle and table are moved to the furthest positive locations on the machine until the limit switches are reached.

G28 moves the spindle and table quickly, it’s a two-step motion process.

First the spindle moves all the way up in the Z axis and then the machine table moves all the way back away from you in the Y axis and all the way to the right in the X axis, this location is machine home.

From the machine home position, the only direction the table and spindle can move is in a negative X, a negative Y and a negative Z direction.

The machine home position is a fixed X, Y and Z starting location.

Machine home is where all tool motion is measured from.

The CNC machine goes to machine home when it is doing a tool change or when it reads a G28 code in **MDI** or in the program.

When entering a G28 code watch closely for a possible crash.

You can use the **RAPID OVERRIDE** buttons to slow down or speed up the axis's rapid movements, speed up the table and spindle rapid movements or slow them down.

Slowing down the **RAPID OVERRIDE** will give you time to visually verify that a crash will not occur when the machine is going to its home position.

To send the X axis / the machine table to its machine home position

Press the **X AXIS JOG** button.

Press the **ZERO RETURN** button.

Press the **HOME** button.

The X axis will return to its machine home position.

To send the Y axis / the machine table to its machine home position

Press the **Y AXIS JOG** button.

Press the **ZERO RETURN** button.

Press the **HOME** button.

The Y axis will return to its machine home position.

To send the Z axis / the spindle to its machine home position.

Press the **Z AXIS JOG** button.

Press the **ZERO RETURN** button.

Press the **HOME** button.

The Z axis will return to its machine home position.

CNC ALARMS

If the machine detects improperly formatted code, commands that it cannot executed or if it detects another machine fault the machine will not be able to run the program.

If the machine is running a program and an alarm occurs, the machine will stop.

After detected an alarm the **DISPLAY SCREEN** will display the word **ALARM**.

Pressing the **ALARM MESGS** button will display a list of any alarms the machine has encountered.

Most of the time you can fix the problem yourself, but not always.

When an alarm occurs, the machine will not operate until the error has been corrected and the **RESET** button is pressed.

Before running the program, you must diagnose and correct the problem that caused the alarm.

You may have to edit the program in MDI or edit the CAD CAM program and then resend the program back to the machine.

An alarm will also occur when the machine detects low air pressure or if the oil reservoir needs to be filled.

https://youtu.be/NNk2E24sYVw

ALARM AND MESSAGES

THE RESET BUTTON

The reset button is pressed when powering on the CNC machine.

Pressing the reset button clears out any active alarms that may be present.

If you're running the spindle warm-up program and you think it has run long enough, you can press the reset button to stop the spindle warm-up program.

After editing a program, press the reset button and the cursor will move back to the beginning of the program.

To run the program, press the **MEMORY** button and then press the **CYCLE START** button.

Pressing the reset button will stop all axes motion.

Pressing the reset button will prevent a tool change.

Pressing the reset button will turn off the spindle and the coolant pump.

Do not press the reset button while the program is running, it may be difficult to continue running the program from the location from where it had stopped at.

If the CNC machine alarms out, the program cannot be restarted until the reset button is pressed.

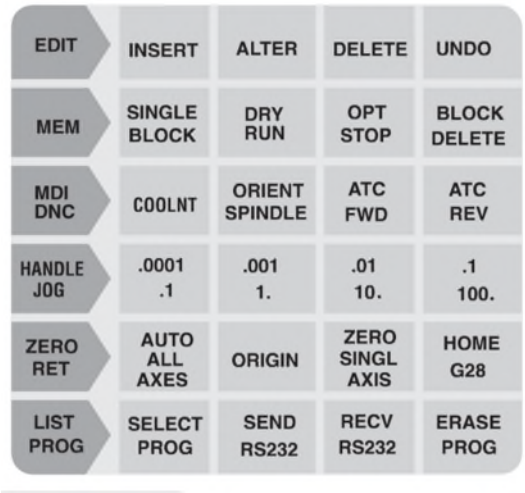
If a program is started immediately after the reset button is pressed the machine control will have forgotten the program code in the look-ahead buffer. This will cause several lines of the program to be skipped. If you continue running the program, serious problems may occur due to missing program code. As an example, if a tap, try's tapping and the hole that has not been drilled yet the tap is going to break. This is just one example of something bad that might happen.

Pressing the **RESET** button when an alarm has occurred will cancel the alarm.

https://youtu.be/TpKX9Fk4ciw

THE RESET BUTTON

THE OPERATION MODE BUTTONS



THE EDIT BUTTON

When the **EDIT** button is pressed you can either store CNC programs to the machine's memory or you can edit a program.

You can also add a program, delete a program or retrieve a program from the machine’s memory.

Pressing the **EDIT** button will display the current active program.

After pressing the **EDIT** button, you can **INSERT**, **ALTER**, **DELETE** or **UNDO** any of the selected text that is listed on either the **PROGRAM DISPLAY PAGE,** the **LIST PROGRAM PAGE** or in **MDI**.

Cursor to the location in the program that you want to edit by pressing the **CURSOR** buttons.

THE MEM BUTTON

When the **MEM** button is pressed, a program stored in memory can be run in either **AUTO MEMORY** or in **SINGLE BLOCK**. You can also do a sequence N block search to find a line of code in the program.

THE MDI / DNC BUTTON

When the **MDI / DNC** button is pressed you can manually input data into the machine by using the keypad.

Enter the CNC code and coordinates that you need to enter and then start running the program by pressing the **CYCLE START** button.

You can also edit programs without having to use the **EDIT MODE**, you can enter lines of code without disturbing the main program.

Data in MDI will remain in the machine when you are switching between modes or when you turn off the power.

A program in MDI can be saved as a regular program in the machines memory by placing the cursor at the beginning of the first line of code and typing the letter O followed by four random numbers, such as O6854

Press the **ALTER** button and the new program will be added to the program list, the program in MDI will be cleared.

The entire MDI program may be erased by pressing the **ERASE PROG** button while in MDI.

You can write a program in MDI, after writing a program in MDI press the **CYCLE START** button to start running the program / machining the part.

Almost anything that can be done within a CNC program can also be done in MDI, including G00, G01, G02, and G03.

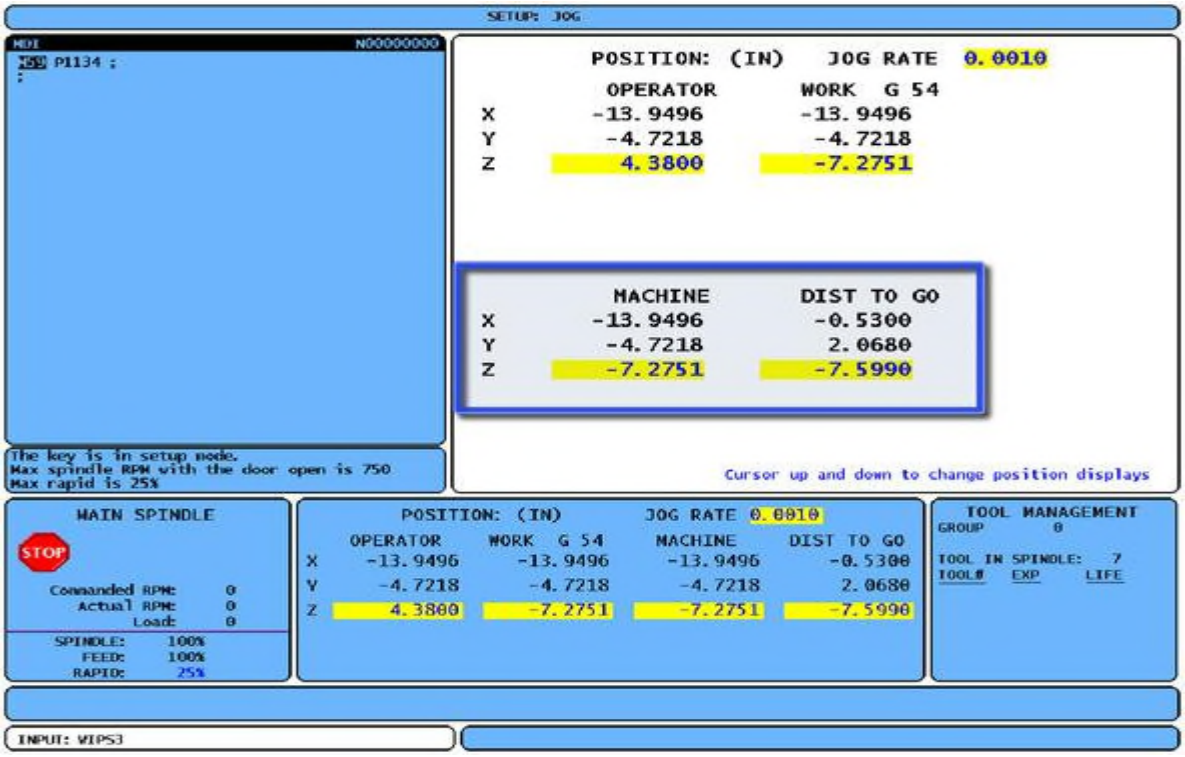
THE ZERO RET BUTTON

When the **ZERO RET** button is pressed the X, Y and Z machine axes can be sent back to the machine home position.

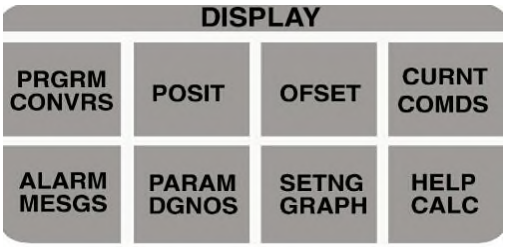
THE LIST PROG BUTTON

When the **LIST PROG** button is pressed you can send a program, select a program, receive a program or delete a program.

POSIT / POSITION PAGE



These eight **DISPLAY KEYS** provide access to all the different screens and operational information that you can view.



Some of the **DISPLAY** buttons will display different screens when they are pressed multiple times.

The display page that you are viewing is listed at the upper, left corner of the screen.

THE POSIT BUTTON

Pressing the **POSIT** button will show you all four position screens.

After pressing the **POSIT** button, pressing the arrow up or down buttons will show you each of the four position screens in a separate, large display.

The **WORK POSITION SCREEN** shows the distance the tool is from the work offset location / the program zero location on the part. The distance the tool is from the parts work offset location is displayed for the X axis, the Y axis and the Z axis. After powering up the machine G54 is the default coordinate position.

The **MACHINE POSITION SCREEN** shows how far away the tool is from machine zero. The distance the tool is away from machine zero is displayed for the X axis, the Y axis and the Z axes.

The **DIST TO GO POSITION SCREEN** shows the travel distance remaining for the programmed move that the tool is positioning to or machining to before it reaches its end point. This endpoint is the distance that the tool is going to move in one line / block of program code.

The **POSITION OPERATOR SCREEN** displays the distance you have jogged the X, Y or Z axes. This is not the distance that the X, Y or Z axis is from machine zero, except when the machine is first powered on. When you are on the **POSTION OPERATOR SCREEN** you can select either the X, Y or Z axis and then press the **ORIGIN** button. After pressing the **ORGIN** button the axis that you selected will zero itself out. This zeroed out axis is then used only for reference positioning.

You can define an axis with a numerical value.

Enter X

Enter -3.125

Press the **ORGIN** button.

Use this method for reference only.

The -3.125 number will not change until you press the **ORIGIN** button or until you power off the machine**.**

If you're jogging an axis and you want to see a zero reference all you do is switch to **MEM** or **MDI** and then switch back to **JOG** to zero out the axis.

http://youtu.be/Uhnvp1NNYxQ

POSITION BASICS

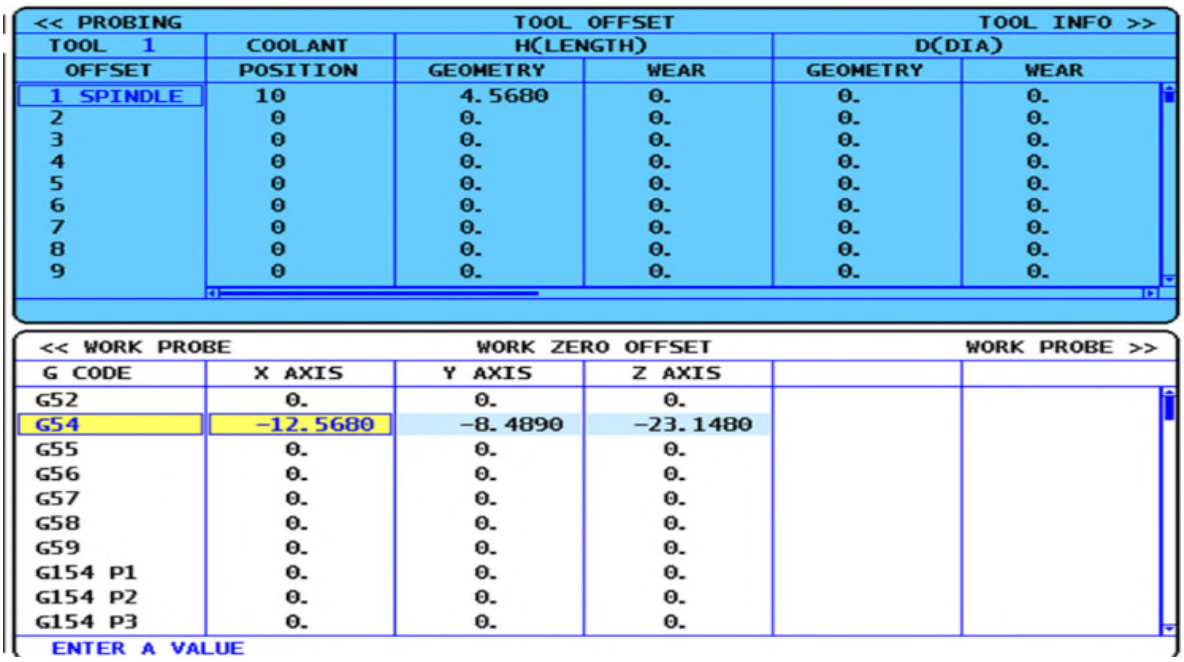
THE ORIGIN BUTTON

The **ORIGIN** button is often used when you are in the **HANDLE JOG** mode

When setting up the machine the **ORIGIN** button may be pressed to zero out the operator position coordinates so that the setup person can move an axis a precise distance.

The **ORIGIN** button is also pressed when you want to zero out tool length offset values that are not being used anymore.

THE TOOL OFFSET PAGE



Each tool in the machine will have its own defined tool length offset value that is stored in the tool length offset register.

Each tool may have its coolant tube position listed.

Small adjustments that are made to the tools, tool length and to the tools, tool diameter are listed in the tool wear offset columns.

If you move the cursor to the right in the tool register there may be additional information about the tool, such as how many flutes the tool has, the diameter of the tool and the type of tool it is, such as a drill, an end mill, tap etc.

THE OFSET BUTTON

You may press the **OFSET** button to adjust the tool length offset. Adjusting the tool length offset on a tool allows you to adjust how deep or shallow you want the tool to machine to.

You may press the **OFSET** button to adjust the tool diameter offset, adjusting the tools, tool diameter offset allows you to adjust how big or small the end mill will machine the part feature.

You may press the **OFSET** button because you must make an adjustment for tool wear.

You may press the **OFSET** button to adjust the coolant nozzle position for a tool.

You may press the **OFSET** button because you need to adjust the X, Y or Z work offset values.

After you press the **OFSET** button, cursor over to the field that you want to adjust.

Pressing the **OFSET** button again or the page up key will show the work offset values for the X, Y and Z axes.

Pressing the **OFSET** button again or the page up key will toggle you back and forth between the work offsets and tool offsets.

If the **ORIGIN** button is pressed while the tool offsets are displayed, the control will prompt the user to **ZERO ALL (Y/N)**

Entering a **Y** will zero out all the tool offsets and coolant positions.

You can also remove work offset values by pressing the **ORIGIN** button.

Bring up the work offset page, press the **ORIGIN** button, enter **Y** for yes and all the old work offset values will be cleared.

REMOVING OLD TOOL LENGTH OFFSET VALUES

Before setting tool length offsets, clear out all the tool length offset values that aren't being used anymore, doing this can help prevent a machine crash.

Do a tool change to bring the tool to the machine zero position.

Bring up the tool offset page.

Highlight the tool length offset number that you want to remove.

Press the **TOOL OFSET MESUR** button.

The tool length offset value will turn to zero.

Scroll to the next tool that you want to zero out, again press the **TOOL OFSET MESUR** button.

Another method that you can use to zero out tool length offset values that are not being used anymore is to highlight the tool on the tool offset page and then enter the opposite of the number that shows on the screen. Press the **ENTER** button and the tool length offset value will turn to zero.

THE CYCLE START BUTTON WILL BE IGNORED WHEN:

(1) The **FEED HOLD** button has been pressed.

(2) The **EMERGENCY STOP** button has been pressed.

(3) The **RESET** button has been pressed.

(4) The **MODE SELECT** switch is set to the wrong position.

(5) A **SEQUENCE BLOCK NUMBER** is being searched for.

(6) An **ALARM** has occurred.

When **MEMORY PROTECT** is turned on the program cannot be edited.

When **MEMORY PROTECT** is turned off, the program can be edited.

THE SINGLE BLOCK BUTTON

You turn on **SINGLE BLOCK** by pressing the **SINGLE BLOCK** button. Each time you press the **CYCLE START** button only one line of program code will be executed.

THE OPT STOP / OPTIONAL STOP BUTTON

When the machine reads a **M01** code in the program and the **OPTIONAL STOP** button is turned on, the machine will stop before the next tool is called up. Pressing the **CYCLE START** button will insert the next tool into the machine spindle and the machine will start machining again. **M01** is used when checking out a new program to make sure each tool is machining correctly. This allows you to open the doors and take a measurement on a part feature. If the dimension is within tolerance press the **CYCLE START** button and start machining the part again. If the dimension is out of tolerance, then adjust either the tool diameter or tool length offset or both if need be. Run the tool over again and the part feature will be machined to tolerance.

https://youtu.be/kC1ItR-WgwY

OPTIONAL STOP

THE BLOCK DELETE BUTTON

To turn on **BLOCK DELETE** press the **BLOCK DELETE** button**.** A block of program code that has a slash / in front of it will be ignored.

https://youtu.be/BSF54F8vn14

BLOCK DELETE

https://youtu.be/y-ffAeKMN20

BLOCK SKIP

PERFORMING A DRY RUN

Many people do not use the **DRY RUN** function anymore. Most programs nowadays are written using CAD CAM software like Mastercam. The software shows you how the part will look after it gets machined and lets you know if a crash will occur or if the part has not been machined correctly. Instructions on how to perform a dry run are written below.

Dry run is turned on by pressing the dry run button while in **MEM** or **MDI** mode.

After turning on dry run you can quickly check a new program for program and setup errors.

No actual machining will occur on the part when performing a dry run.

Dry run can only be turned on or off when a program is stopped in a reset condition.

The first push of the dry run button turns dry run on, the second push of the dry run button turns dry run off.

Dry run will make all the programmed tool changes.

Dry run will make all the X, Y and Z axis moves.

A dry run will let you know if there are any bad X, Y, or Z moves in the program.

Running a new program in single block allows you to locate and correct program and setup errors.

If there's something wrong with the program the tool could crash into either the machine table, the stock material or the vise or fixture.

Press the **RAPID 5%** button.

Keep pressing the **OFFSET** button until the work offset page displays.

Highlight **G54 WORK OFFSET**.

Highlight **Z**

Enter a positive value that is higher than the deepest programmed Z depth that is in the program, 2 inches is a good value to use.

You want the tools to be moving around about 2 inches above the part this way, this will allow you to see if the tool movements are conforming to what the parts supposed to look like after it gets machined.

After entering 2 inches for the G54 Z work offset height, press the **WRITE ENTER** button.

Press the **MEMORY** button**.**

Press the **CURRENT COMMANDS** button.

Press the **LIST PROGRAMS** button.

Scroll to the program that you want to dry run.

Press the **SELECT PROGRAMS** button.

Make sure to correct program displays on the screen.

Press the **SINGLE BLOCK** button and single block will be activated.

Make sure the coolant is turned off.

Keep pressing the **CYCLE START** button until you have cycled through the entire program.

Each time you press the **CYCLE START** button you are initiating one line of CNC program code.

Make sure everything looks correct and no alarms have occurred.

Watch what the tool is doing each time you press the **CYCLE START** button.

Make sure that the tool movements conform to what is written in the CNC program.

Make sure that all the tools move to the correct locations on the part that are supposed to get machined.

If an end mill is supposed to mill the outside contour of the part, make sure the end mill movements conform to what the parts outside shape is supposed to look like.

When dry run is turned on the feed rates and rapids in the program are ignored.

Adjust the feed rate override and rapid overrides to a comfortable setting, this will allow you to quickly run through the entire program to see if there are any setup or program errors.

When dry run is turned on you cannot tell the difference between rapid motions and cutting motions / feed-rate.

For extremely long machining cycles you toggle dry run on and off.

Once you’re sure that the block of code is tool machining motion you turn dry run back on, doing this allows the tool machining movements to be completed quickly, repeating these procedures allows you to quickly check a lengthy program and verify that all the cutting commands are correct.

Remember no actual machining will occur, the tools are moving above the part.

CNC DRY RUN VIDEOS

https://youtu.be/MEx9KBhUTeA

https://youtu.be/Orh_Zndvsic

THE INSERT BUTTON

Press the **INSERT** button when you want to insert your typed text to the right side of the current cursor location.

Press the **INSERT** button when you want to paste program code from the clipboard into the program that you're working on.

THE ALTER BUTTON

To change the text that's next to the cursor, to the text that is displayed in the input buffer, press the **ALTER** button.

To place an MDI program into the program list, press the **ALTAR** button.

To replace highlighted text with the text that you want to enter, press the **ALTER** button.

When you're in **BLOCK EDIT,** pressing the **ALTER** button will shift a block of code from one location in the program to another location in the program.

THE DELETE BUTTON

Pressing the **DELETE** button deletes the text that the cursor is on.

Pressing the **DELETE** button deletes any highlighted text.

You can **DELETE** a block of code in the program by using **BLOCK EDIT**.

THE UNDO BUTTON

Pressing the **UNDO** button allows you to undo up to 10 edit changes.

Pressing the **UNDO** button will not undo anything that was done in **BLOCK EDIT**.

THE LIST PROG BUTTON

The **LIST PROGRAMS** button is used to list programs, send programs, receive programs or delete programs.

Pressing the **LIST PROG** button will show you all the programs that are in the machine's memory.

CREATING A NEW PROGRAM NUMBER / PROGRAM

Press the **MDI** button

To create a new program, enter the letter **O** and then enter 4 different numbers, as an example I'm using **6854** for my four program numbers.

Your new program will look like this, **O6854**

To create program **O6854**

Press the **WRITE ENTER** button.

The new program can now be created on the line following the program number.

All further program entries are made by keying in a letter followed by a numeric value and then pressing either the **WRITE ENTER** button, the **INSERT** button or the **ALTER** button.

The **ALTER** and **INSERT** buttons are explained on page 62

The **WRITE ENTER** button is explained on page 66

THE SELECT PROG / PROGRAM BUTTON

Pressing the **SELECT PROG** button makes the highlighted program on the program list the active program.

There will be an asterisk* in front of the active program.

When the machine is in **MEM** mode and you press the **CYCLE START** button the program will start, and the part will begin to be machined.

Cursor up or down and highlight the program that you want to make the active program.

Press the **SELECT PROG** button and an asterisk* will move in front of the program.

To bring up a program in the control, enter the program number that you want and then press the **SELECT PROG** button.

THE ERASE PROG / PROGRAM BUTTON

The **ERASE PROG** button is pressed when you want to delete a program from memory or from MDI.

To delete a program, you must be in the **LIST PROG** mode.

The programs will be listed by their program numbers.

Use the up or down cursor keys to highlight a program number or type the program number in the input line and press the **WRITE ENTER** button.

Pressing the **ERASE PROG** button will delete the program.

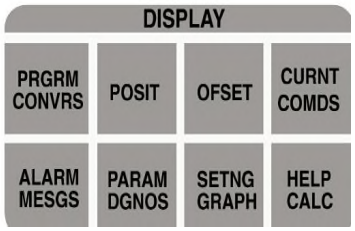
All the programs that are listed may be deleted by selecting **ALL** at the end of the list and then pressing the **ERASE PROG** button.

Use caution when cursor selecting a single program or **ALL** programs.

After pressing the **ERASE PROG** button, you will be prompted with the message **DEL** 0nnnn (**Y/N)** to erase programs.

If you press **Y** for yes, all the programs will be deleted.

The **UNDO** button will not recover programs that where deleted.



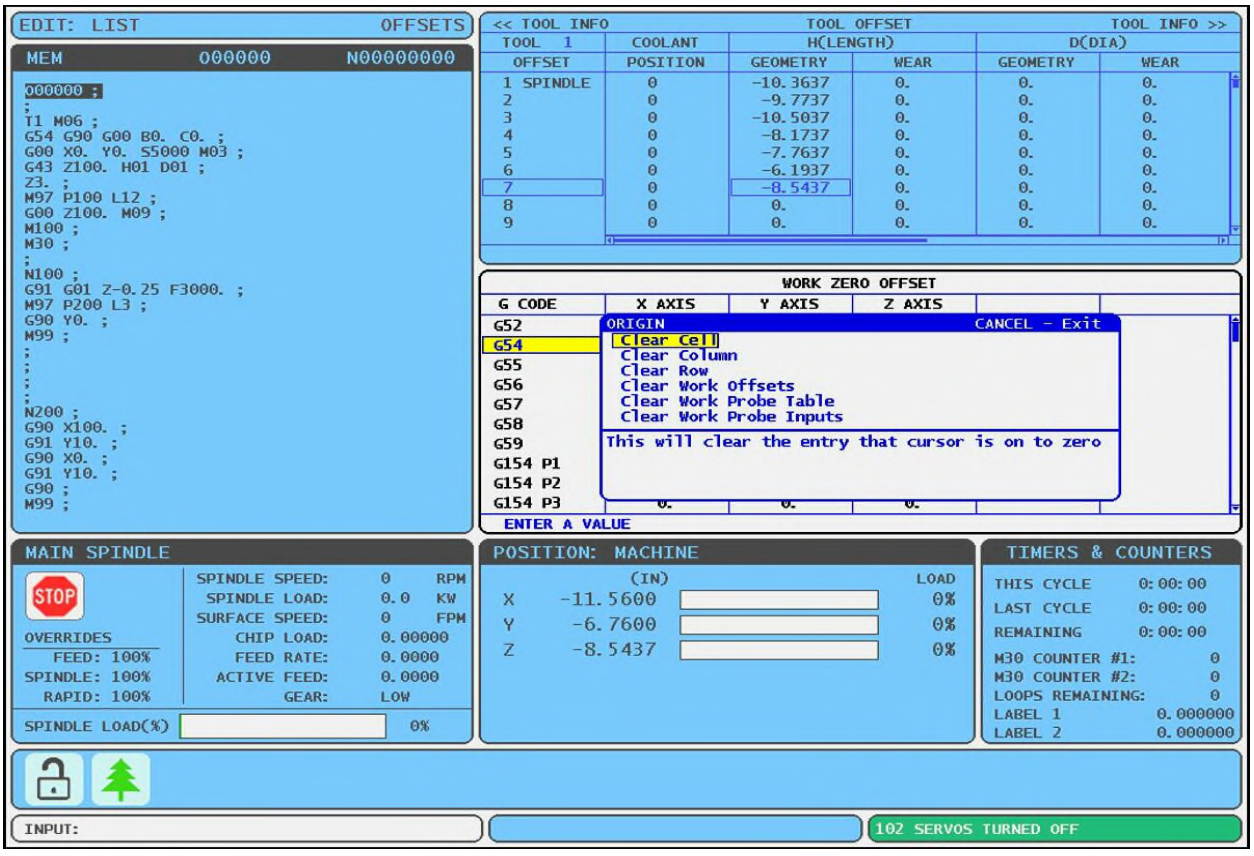
THE PRGM CONVRS DISPLAY BUTTON

To find a program press the **PRGRM CONVRS** button. Pressing the **PRGRM CONVRS** button allows you to cursor through and view the active program on the right side of the display screen while also viewing the same program as it's running on the left side of the screen. To bring up program review, press **F4,** make sure you're in MEMORY / MEM mode.

BACKGROUND EDIT

BACKGROUND EDIT allows you to edit a program while in **MEM** mode from the **PRGRM** display page while a program is running. Type in the program number that you want to edit and press **F4**. You can then edit an existing program, a new program or the program that is presently running. Edits to the program that is currently running will not take effect until that program cycle ends with an **M30** or by pressing the **RESET** button and then running the program over again.

THE CURRENT COMMANDS PAGE



To view the current command page, press the **CURNT COMDS** button.

The current commands page shows 15 lines of the active program.

The feed-rate and spindle speed are displayed.

If you adjust the feed and or speed override buttons the current spindle speed and feed-rate will be displayed on current command page.

For example, if the program has tool #1 turn on at 1000 rpm, 1000 rpm for tool #1 will be displayed on the current command page. If you adjust the spindle speed override by +10 percent the current command page will now show you that tool #1 is turning at 1100 RPM.

Next example, if the program has tool #1 start machining the part at 100 FPM, 100 FPM for tool #1 will be displayed on the current command page. If you adjust the feed-rate override by +10 percent the current command page will now show you that tool #1 is moving the tool at 110 FPM.

The current commands page also shows spindle load, axis loads, surface speed, chip load, and spindle CW, CCW or STOP commands.

Current axis positions are shown, cursor up and down and you can view the operator coordinates, the work coordinates, the machine coordinates or the distance to go coordinates.

TO START RUNNING A PROGRAM OTHER THAN AT IT'S BEGINNING

There will be times when you must start a program in the middle of the program or towards the end of the program.

The reasons for this may be that the power went out while the part was being machined or a tool broke while the part was being machined or you had to edit the program.

Cursor to the **BLOCK NUMBER** that you want to start running the program from.

Press the **MEM** button.

To start machining the part, press the **CYCLE START** button.

If you start the program from a line of code that's going to tap a hole and the hole hasn't been drilled yet; you are going to crash the tap into the stock. You must know what you're doing when you start a program that does not start at the beginning of the program.

Here is a video on how to start machining a job in the middle of the program or towards the end of the program. https://youtu.be/ePlyXeXYZBE

THE WRITE ENTER BUTTON

Pressing the **WRITE ENTER** button acts as a general purpose enter key.

Anytime you need to add or change information in the control you press the **WRITE ENTER** button.

Pressing the **WRITE ENTER** button takes numbers and or text that are on the input line and inserts them next to the cursor.

THE CANCEL BUTTON

When entering text to the input line or when entering text in the **EDIT** mode the **CANCEL** button functions like the backspace key on a computer keyboard.

PROGRAM MANAGMENT

http://youtu.be/M5pDTaWkaAQ
PROGRAM MANAGEMENT

http://youtu.be/rBIYXTFZFNk
SETUP_ RUN AND EDIT

https://youtu.be/pms_MSZ5Ywc
PROGRAM EDIT

THE CONTROLLER AND WHAT IT'S BUTTONS DO

http://youtu.be/Ru_46sirK2A
HAAS MILL BUTTON TOUR

https://youtu.be/ipEVaN4h6Wo
THE HAAS CONTROL PANEL

http://youtu.be/3pHQNsvE4vA
CNC BUTTON TOUR

https://youtu.be/Fstoz4-vrGU
BASIC FUNCTIONS

THE CURSOR

https://youtu.be/N0IzDtWoZhc
THE HAAS CURSOR

INDEX

Page 9

Powering on the CNC machine

Page 10

The EMERGENCY STOP button

Shutting off the CNC machine

Page 11

Tool setup

The retention knob / pull stud bolt

Page 12

Three reasons why you may need to change or remove the pull stud bolt

Page 13

Installing an end mill into the collet / tool holder

Page 14

Removing an end mill from the collet / tool holder

Page 15

The four most common cat 40 tool holders are:

The set screw end mill tool holder

The drill chuck tool holder

Page 16

The RE collet tool holder

The Solid Body / Face Mill and Shell Mill tool holder

Page 17

How to insert a tool holder into the spindle

ATC forward / ATC reverse / Next Tool button

How to remove a tool / tool holder from the spindle

Page 18

Calling up a tool from the tool carousel

Page 19 and 20

The vise

Page 21

To run the Haas spindle warm - up program

To create a spindle warm up program in MDI

Page 22

Jog Mode

The two ways to jog and axes

The Handle Jog Increment buttons

The Axis Keypad Jog buttons

The Handle Jog Wheel

Page 23

The second way to jog and axes

Page 24 and 25 and 26

Why you need to set the work offsets

Setting the G54 X work offset on a Haas VMC

Page 27 and 28

Setting the G54 Y work offset on a Haas VMC

Page 29

Have the G54 X and Y work offsets been set correctly?

Page 30

Tool length offset information

Setting the tool length offsets on a Haas VMC

Page 31 and 32

Tool setter gauge

How to measure a tool length offset

Page 33

Tool length offset safety program

Page 34

Before clamping the stock material in the work holder

Page 35 and 36

Clamping the stock in a vise or fixture

If you are locating the stock from a bump stop

Marking a part feature

Page 36

If the spot drill did not move to the correct location on the part, it may be that:

You can use this method if the first tool is a spot drill.

Page 37

Before running the program / machining the part / checklist

Page 38

Before machining the part, add some positive tool diameter offset and some positive tool length offset to the tools that will be machining part features that have tight tolerances.

It’s time to machine the part

Page 39

Reasons why the tool moved to the wrong location on the part

The FEED HOLD button

Page 40

What to do if the machine crashes

The Recover / Restore button

To stop the machine from machining the part.

Page 41

To start machining again after pressing the Feed Hold button

What to do after machining the first part

Page 42

Tool diameter offset information

To make an end mill machine a smaller diameter pocket

To make an end mill machine a larger diameter pocket

Other reasons why you may offset the tool's diameter

Page 43

Making an undersized end mill machine to size

Adjusting the depth of a tool

Page 44

Surface Finish

Machining a part in manual mode

Page 45

Override buttons

Spindle Speed Overrides

Page 46

Feed Rate Overrides

Operator responsibilities may include the following

Page 47

Basic steps to machine a part

Page 48

Tools used to machine the part

Page 49

Shop Safety

The AUTO ALL axes button

ORIGIN button

The ZERO SINGL axis button

The HOME G28 button

Page 50

To send the X axis / the machine table to its machine home position

To send the Y axis / the machine table to its machine home position

Page 51

To send the Z axis / the spindle to its machine home position

CNC Alarms

Page 52

RESET button

Page 53

The Operation Mode buttons

EDIT button

The MEM button / Memory button

The MDI / DNC button / Manual Data Input / Direct Numerical Control button

Page 54

The HANDLE JOG button

The ZERO RET / zero return button

Page 55

LIST PROG / List program button

POSIT / Position page

DISPLAY keys

Page 56

The POSIT button / Position button

Page 57

ORIGIN button

The Tool Offset page

Page 58

The OFSET button

Removing old tool length offset values

Page 59

The CYCLE START button will be ignored when

SINGLE BLOCK button

The OPT stop button / Optional Stop button

Page 60 and 61

The BLOCK DELETE button

Performing a Dry Run

Page 62

INSERT button

ALTER button

DELETE button

Page 63

UNDO button

The LIST PROG button

Creating a new program number / program

THE SELECT PROG / program button

Page 64

The ERASE PROG / program button

The PRGM CONVRS DISPLAY button

Page 65

BACKGROUND EDIT

The Current Commands page

Page 66

To start a program other than at its beginning

The WRITE ENTER button

CANCEL button

Page 67

PROGRAM MANAGMENT / YouTube videos

The CONTOLLER and what its buttons do / YouTube videos

The CURSOR / YouTube videos

Written by Ken Maier

kmmoraine6854@outlook.com

Schooling / 2-year CNC / Tool & Die Program

Job experience / 8 years CNC Setup / Operator