Haas Automation Inc.

TIPS AND TRICKS

TOOLING AND WORKHOLDING

Contents

SECTION 1 - WORKHOLDING

Here are some tips and tricks on setting up a Haas CNC milling machine. The first thing to determine is how do you want to hold the work piece to the machine. There are three basic types of work holding used in milling operations, a mill vise, clamps, and a chuck.

First off, here is a note of caution. Before placing any type of work holding on your machine table, great care must be taken to be sure that the table is CLEAN AND FREE OF CHIPS AND OTHER DEBRIS.

TIP: Chips and other debris caught between the fixture and the machine table will cause damage to both. Having swarf caught between the fixture and table can cause the fixture to rock and the parts machined will be inaccurate.

Also, make sure that whatever you set on the table is clean, free of debris. You should always use a honing stone to rub the locating surface. This will ensure that the locating surface does not have any burrs or dings that may damage the table. If you plan to leave your work holding on the table for any length of time, a light coat of rust-preventative oil or WD-40 will help keep your table and work holding free of rust and corrosion.

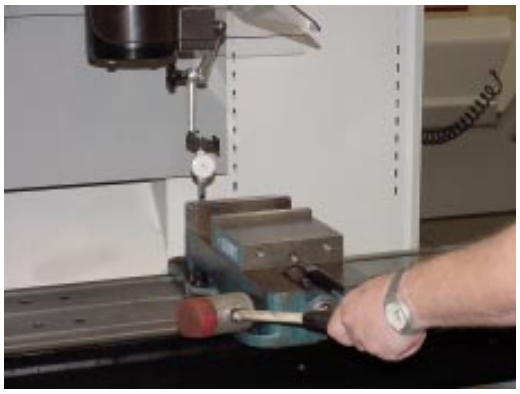
The most common method of holding a work piece for machining is a mill vise. For precision work, the vise must be set so that the clamping surfaces are parallel to the X or Y-axis of machine travel. This is done using an indicator. When indicating a mill vise, this simple procedure can make it quick and easy.

1. Set the mill vise on the table and put your tee-nuts and bolts in position.
2. Tighten the bolt on the right side of the vise and just “snug” the bolt on the left side.
3. Place the magnetic base anywhere on the bottom of the Z-axis head. To insure that the indicated readings are accurate the magnetic base should be mounted on a solid non-movable part of the head. Jog the machine axes to bring the indicator tip to the right side of the vise on the clamping surface you want to indicate. Set the tip of the indicator so it begins to register on the indicator dial and set zero.

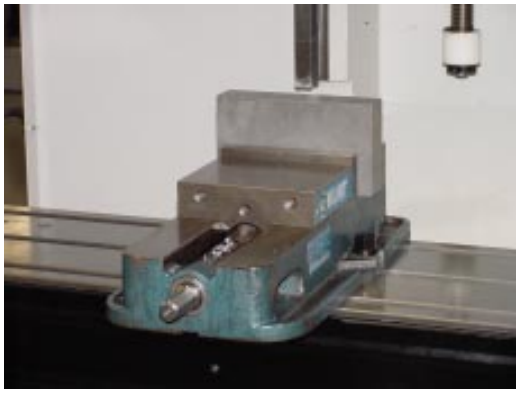
4. Jog the indicator across the clamping surface and stop at the left side of the vise. Determine which direction the vise needs to be moved and tap the vise until your indicator moves back to zero. Note: with the right bolt tight, the vise will rotate around this point. Jog the indicator back to the right side of the vise, and re-set zero. Jog back to the left side and tap the vise until your indicator reads zero. You should be very close by now. Repeat the previous steps until you indicator stays at zero across the entire indicating surface.

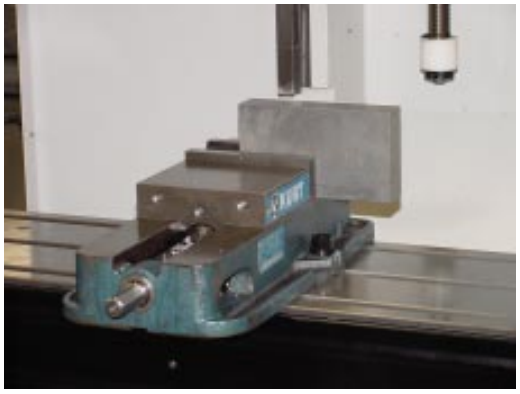
5. When the indicated reading is perfectly flat across the vise jaw, tighten the left bolt first, then, tighten the bolt on the right side. Run your indicator across the surface one last time to make sure is still parallel with the machine travel.

TIP: Use a soft or dead blow hammer to tap the fixture or vise into position. Using a ball peen hammer or other hard objects may damage the fixture.



Make sure, when locating a part in a mill vise, that you center your part in the vise. You don't want a large portion of the work piece hanging off the side of the vise. This will cause the movable vise jaw to twist and pinch the work piece, greatly reducing the clamping force. If you try to drill a hole in the overhanging material, the Z-axis thrust may cause the material to push down where drilling and push up on the other side of the work piece. If it is necessary to drill through a work piece that is held in a vise, use step jaws. Step jaws allow you to locate the work piece up, off the bottom of the vise. This will provide clearance below your work piece so you can drill through without drilling into the vise. If you only have hardened steel jaws without a step cut in them, you can use a set of parallel bars inside the vise to set the work piece on to keep it off the bottom of the vise. Always verify that the parallel bars are the same size to insure that your work piece is sitting flat.

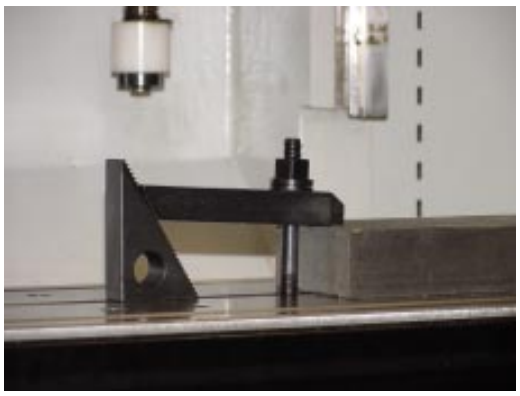




TIP: Most precision mill vises have a Key slot and keys on the locating surface. Since all Haas milling machines have precision tee slots machined and lined up with the X-axis, you can use the keys on the vise to locate the vise in the tee slot. This will locate your vise square to the table. If your vise doesn't have keys, you can make a sub-plate for the vise with keys or dowel pins on the bottom to locate in the tee slots. On the top, you can cut holes for attaching the sub-plate to the machine table and tap holes for attaching the vise to the sub-plate. Locate the sub-plate in the tee slots and bolt it to the table. Set the vise on the sub-plate and indicate it as described above. Now, every time you use that vise, just locate the sub-plate in the tee slots; bolt it down, and your ready to go. For high-precision work, you will still have to check the indication and make small adjustments.

When using clamps to hold a part with downward force, ALWAYS make sure that the clamp is lower where it contacts the part, and higher in the back. Most downward-force clamps use a jackscrew or serrated block that meshes with serration's on the clamp to support the end of the clamp opposite the end making contact with your part. The serrated end of the clamp must be higher than the contact end of the clamp. If it is not higher, the clamp will make contact with the edge of your part and not the top. This will greatly reduce the amount of clamping force holding the part, and probably create a dent at the intersection of the top surface and the side surface of the part. If you use the jackscrew style clamp, make sure that the jackscrew does not sit directly on the mill table. Always use a thick piece of shim stock or other material to protect the mill table from damage.

Improper use of clamp



Correctly posioned clamp

TIP: When using clamps in a production situation, periodically check the jackscrew adjustment to make sure that your clamp is still higher on the jackscrew end than the contact end. In any case, the hold-down bolt should be as close to the material being clamped as possible to transfer the maximum clamping pressure to the material.

When clamping on a cylindrical surface is required, a 3-jaw chuck mounted on your machine table may be the best way to go.

TIP: if your cylindrical surface is finished, put a set of soft jaws on your chuck. Use an end mill to machine your jaws to the exact diameter of the surface you want to clamp on. Just remember that you should always have the chuck clamped when you are machining the jaws. A piece of raw bar stock or a hex nut work well. Anything that will allow the jaws to be tightened and leave room for your cutter to cut to the desired depth. This is also true if you are machining soft jaws on a mill vise. The vise should always be tightly clamped before any type of machining is performed.

SECTION 2 - SETTING OFFSETS

In many cases, a CNC programmer has defined Z-zero in the program to be the top of raw stock. Quite often raw stock is not flat or parallel.

TIP: If you need to set tool length offsets from the top of raw stock but need to have precise measurements on your tool lengths, take a light skim cut on the stock. Now you can measure your cutters from a flat, clean surface. Or you can set the tool length offsets from the fixture where the part will be sitting, then increment the work coordinate offset Z-axis a positive value the amount equal to the thickness of the part.

To set the tool length offsets, jog the tool down toward Z-zero. When you get close, slide a sheet of paper between the tool and the work piece. Carefully, move the cutter down to the top of the part, as close as possible, and still be able to move the paper. Switch to the smallest increment in the handle jog mode. Now slide the paper back and fourth while slowly jogging down. You will begin to feel the tension on the paper. Press OFSET key, press PAGE UP as many times as necessary to get to the CLNT (LENGTH) (RADIUS) page for the tool you are setting. Cursor to the GEOMETRY column and down to the tool number you are setting. Press TOOL OFSET MESUR key. This will take the Z absolute machine position located in the bottom left of the screen and enter it in the tool length column for the tool in the spindle number position.

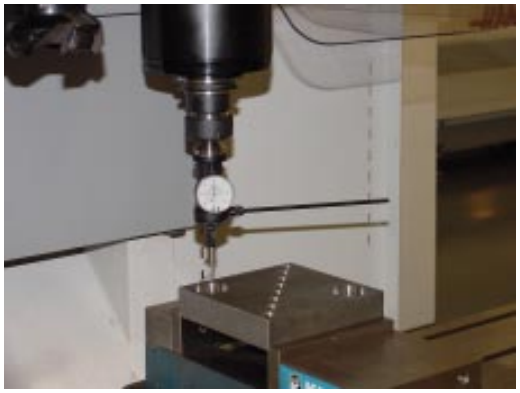
TIP: When setting TLO (tool length offsets) the absolute machine position for the Z-axis should be registered when pressing the TOOL OFSET MESUR button. If this is not the case, Setting 64 in the control should be turned off.

When setting work coordinate offsets, you must locate X and Y zero accurately. Remember that you are measuring the centerline of the spindle to a location on a part or a fixture. If it is the edge of a part or fixture, an edge finder is the most common tool used. An edge finder is composed of two concentric cylinders, spring loaded together. To use it, place the edge finder in a collet chuck and offset the two halves slightly so that there is a wobble as it spins. Then, jog the part into the wobbling end of the edge finder slowly. The edge finder will center up, and then break out of concentricity suddenly. At that point, jog the edge finder plus in the Z-axis to raise it above the work piece. Now, jog the axis you are finding by the amount of the radius of the edge finder. Make sure that you are on the page labeled “WORK ZERO OFFSET” and that your cursor is on the correct line in the G-CODE (G54 etc.) column. Cursor across to the correct axis, press the PART ZERO SET button and confirm the entry went into the right column.

TIP: If you are setting your TLO from the part, you will only need to set the work coordinate offset for the X and Y-axis. The TLO will be compensated for by the tool length offsets.

TIP: 1000-1500 RPM is a good spindle speed range when using an edge finder.

If you need to find the centerline of a hole or a round part feature, an Indicol is a helpful tool. An Indicol is a type of holder for dial-test indicators. It has a “C-clamp” style clamp to attach the Indicol to a tool holder in the machine spindle. The Indicol also has two or three adjustable arms and a clamp at the end to hold a dial test indicator. The adjustable arms allow you to position the indicator to spin the same diameter as the hole. To find the centerline of a hole, position your indicator tip just above the hole and manually spin the tool holder with the Indicol attached. You will be able to see if your indicator tip is spinning the same approximate diameter as your hole and how far off center you current position is. Adjust your X and Y-axis as close as possible before moving your indicator down into the hole. Once you’re close, jog the Z-axis down so the indicator tip is inside the hole and adjust the arms so that the indicator begins to give a reading. Spin your indicator so that it is making contact with the surface of the hole in one of four quadrants (X+, X-, Y+, or Y-). Now, set your indicator to zero and rotate it 180 degrees. The amount of indicator movement is 2X the amount you need to adjust that axis. If your indicator moves minus .016, then you need to jog the axis .008 in the plus direction. Now, rotate your indicator 90 degrees and re-set zero. Rotate your indicator 180 degrees and find the amount and direction that the other axis needs to be adjusted. Remember, the amount the indicator moves is 2X the amount that you will have to jog the axis to find the centerline of the hole. This procedure can be tricky on small diameter holes, but it is very accurate. You really can find the exact centerline of a hole, within .0001 of an inch, in each axis.



TIP: A huge time saver for finding the center of a hole or round part feature is a co-axial indicator. The indicator fits into a collet chuck and you use it while the spindle is turning. Manufacturers claim that you can use their indicators at speeds up to 800 RPM, but 50 to 100 RPM works well. If the spindle is rotating too fast, it is difficult to tell which axis needs to be adjusted. A restraining arm allows the face of the indicator to remain stationary while the spindle rotates. With each rotation of the spindle, the indicator dial will show the amount it is off center. You simply jog the machine axes while watching the indicator movement. This saves time because you can start the rotation while the indicator is off center as much as .250 inches and you can literally dial it in within seconds.

SECTION 3 - TOOL HOLDERS

Selecting the right tool holder for the job is as important as selecting the right cutter for the job. You should always use the shortest tool holder possible for all machining applications. In addition, the tool should be set in the tool holder as far as possible. This will increase the tool holder's grip on the tool and reduce vibration. The shorter the distance from the spindle nose to the tool tip, the more rigid your set-up will be. Increased rigidity means less vibration when cutting. Haas Automation, Inc. recommends that any tool holder running at 10,000 RPM or higher be balanced to G2.5, or better, at the maximum RPM. You can buy pre-balanced tool holders but they should be balanced again with the cutter set in the holder.

Tool should be held in the holder with as little of the tool as necessary left unsupported.

TIP: Balancing tool holders will only improve machining conditions. It will prolong the life of your machine's spindle and your cutting tools. It will also improve part surface finish and dimensional accuracy. Lower quality surface finish and spindle damage can occur if the balance of the tool holder, with the tool in it, is not within the G2.5 specification.

TIP: If your situation requires running the spindle at speeds in excess of 10,000 RPM and balancing the tool holders is necessary, avoid using endmill holders with setscrews. Endmill holders will not allow the cutter to run true (concentric to the spindle) due to the unidirectional clamping force applied by the setscrew. The best types of holders for high-speed use are shrink fit holders, collet chucks with balanced nuts and collets or hydraulic collets. These types of holders will apply even clamping pressure on the tool and TIR is almost zero.

TIP: For high-speed operation, round shank tooling should not have Weldon flats on them. Weldon flats will cause unbalance due to the uneven weight distribution. Tool should be held in the holder as short as possible.

SECTION 4 - CUTTING TOOLS

When selecting cutting tools for a job, the first thing to consider is what type of operation needs to be performed. Here is a quick description of the basic cutting tools most often used in milling operations.

Drill

A drill is used to create a round, cylindrical hole in a work piece. Drilled holes can be “through holes” or “blind holes”. A “blind hole” is not cut entirely through a work piece. Quite often, an engineering blue print will specify a drilled hole to be drilled to “full diameter depth”. This means that the hole diameter must be a specified depth without regard to the angled tip of the drill. When you measure your tool length offset, you are measuring the length of the drill and it’s tip. So how deep do you drill the hole so that the full diameter depth is correct? Well, you need to know how long the drill point is.

TIP: The length of the drill point is determined by the tool point angle and the drill diameter. Constant values can be multiplied by the drill’s diameter to calculate how long the drill point angle is. Most standard high-speed steel drills have a drill point angle of 118 degrees. For a drill with a drill point angle of 118 degrees, multiply the drill’s diameter by .3. For a drill with a drill point angle of 135 degrees, multiply the drill’s diameter by .207. For a drill with a drill point angle of 141 degrees, multiply the drill’s diameter by .177. These constant values will calculate the drill point length within a few thousandths of an inch.

Center drill

A center-drill is a small drill with a pilot point. It is used to create a small hole with tapered walls. When a hole’s location must be held to a close tolerance, use a center drill first and then use a twist drill to finish the hole. The tapered walls of the center-drilled hole will keep the twist drill straight when it begins to drill into the work piece.

TIP: Many Machinists use this rule of thumb. If the tolerance of the diameter of a center-drilled hole is not critical, drill as deep as you want this diameter to be. On a standard, 60-degree center drill below .375-inch diameter, the diameter produced will be close to the depth you drilled. When you get to larger center drills, .375 and above, the depth to diameter ratio becomes larger. You could be off as much as .080-. 100.

Reamer

A reamer is designed to remove a small amount of material from a drilled hole. The reamer can hold very close tolerance on the diameter of a hole, and give a superior surface finish. The hole must be drilled first, leaving .005 to .015 of an inch stock on the walls of the hole for the reamer to remove.

TIP: The ideal situation for best accuracy for hole size and location when reaming is to process the hole with the following steps: the hole is first drilled, then bored, then reamed.

TIP: Stock allowance for a reamed hole will depend on the size of the hole. A general rule is that for holes under ½” stock of less than 0.0150” on diameter, for holes over ½” stock of 0.030” on diameter. The type of workpiece material and the method used to create the hole will affect the stock allowance.

TIP: A reamer produces the best, most uniform surface finish when it is fed into, and out of the hole (G85 canned cycle). Many people try to save time by using G81 canned cycle. This canned cycle will feed into a hole and rapid out. It is quicker but will usually leave a helical swirl mark on the cylindrical surface of the hole. Although, this swirl mark is only a cosmetic flaw, and doesn't affect the size of the hole, the appearance of the hole may be rejected by some customers.

Tap

A tap is used to create screw threads inside of a drilled hole. NOTE: Great care must be taken when using a milling machine to perform a tapping operation.

TIP: If you are using a machine with rigid tapping, feed rate (in inches per minute) = thread pitch X revolutions per minute. Also, You should never tap more than 1.5 X the tap's major diameter. Threaded connections will not increase in strength if the contact length is more than 1.5 times the diameter of the fastener. If you need threads that are deeper, machine tap them first and hand-tap them to finished depth. If you tap deeper than 1.5 X the hole diameter, your chances of breaking the tap increase dramatically. Chip control becomes a problem. When tapping blind holes, always drill as deep as possible to avoid packing chips below the tap. Using a spiral flute tap will bring the chips up, out of the hole. To further reduce tapping headaches, make sure all holes to be tapped are free of chips and use a tapping fluid specifically designed for the type of material you are cutting.

TIP: Tap drill size is the size of the hole required for a specific tap. For 75% effective threads the formula that will determine the correct drill size is:

D - 1/N
D = major diameter of the tap
N = number of threads per inch
A tapped hole with 75% of thread depth has only 5% less strength than 100% thread and takes only 1/3 of the cutting force of a 100% thread.

End Mill

An end mill is shaped similar to a drill, but with a flat bottom. It is used, primarily, to cut with the side of the tool to contour the shape of a work piece.

TIP: Programming an end mill to cut contour or pocket tool paths using cutter compensation (G41 and G42) allows you much more flexibility in adjusting the size of machined features. Using cutter compensation allows you to adjust the amount of stock removal. As an end mill wears, minor offset adjustments allow you to make every part the same size. You may also use a different size end and have the machine cut the same part features as with the end mill originally programmed for that tool path.

Bull End Mill

A bull end mill is the same as a regular end mill except that there is a radius on the corner where the flutes meet the bottom of the endmill. This radius can be any size up to ½ of the tool's diameter.

TIP: Bull end mills are effective for producing a corner radius between a wall and a floor on a given part feature. They also add to the strength of an end mill. When machining hard, tough to cut materials, the sharp corners on a standard end mill tend to chip and wear faster than an end mill with a corner radius. The radius on a bull end mill provides a more gradual shearing entry in to the work piece.

Ball End Mill

A ball end mill is a bull end mill where the corner radius is exactly ½ the tool's diameter. This gives the tool a spherical shape at the tip. It can be used to cut with side of the tool like an end mill.

TIP: The primary purpose of a ball end mill is to machine lofted surfaces. The spherical shape of the tool is able to move along any undulating surface and cut anywhere along the cutter's "ball end". As a ball can roll over a surface, a ball end mill can be used to cut any such surface.

Insert End Mill

An insert end mill is the same as a standard end mill but with replaceable carbide inserts.

TIP: Insert end mills are designed to remove metal at higher rates than solid carbide. They come in large range of diameters and are able to cut at a deeper depth of cut. This is fantastic but, when using these cutters, it is a good idea to calculate the horsepower required to make a cut. Piece of cake… on your Haas control there is a button on the front titled "help/calc". Press this button once to get the help menu, press it again to get the calculator functions. Use the page up/page down keys to scroll between three pages: Trigonometry Help, Circular Interpolation Help, and Milling Help. Each one of these pages has a simple calculator in the upper, left hand corner. On the Milling Help page, you can solve three equations:

1. SFM = (cutter diameter in.) * RPM * 3.14159 / 12
2. (Chip load in.) = (feed in. per min.) / RPM / # of flutes
3. (Feed in. per min.) = RPM / (thread pitch)

With all three equations, you may enter all but one of the values and the control will compute and display the remaining value. To calculate the Horsepower required for a cut, you must enter values for RPM, feed rate, number of flutes, depth of cut, width of cut, and choose a material from the menu. If you change any of the above values, the calculator will automatically update the required horsepower for the cut you intend.

The next thing to consider when choosing cutting tools for a job is what material you are going to cut. The most common materials cut in the metalworking industry can be divided into two categories, non-ferrous and ferrous. Non-ferrous materials include aluminum and aluminum alloys, copper and copper alloys, magnesium alloys, nickel and nickel alloys, titanium and titanium alloys. Common ferrous materials include carbon steel, alloy steel, stainless steel, tool steel, and ferrous cast metals like iron. Non-ferrous metals are softer and easier to cut, with the exception of nickel and titanium. Ferrous metals, on the other hand, are generally harder in composition and tougher to cut.

Cutting tool material is one of the biggest decisions you'll have to make when choosing a cutting tool. Most all of the cutters described above are available in three basic materials, High Speed Steel, Solid Carbide, and Carbide Insert Style. Most all of the basic cutting tool materials can be used to cut most all materials. It really boils down to performance. High Speed Steel cutting tools has very high toughness but lack wear resistance. Carbide, on the other hand, has a very high wear resistance but chips and breaks easily. Carbide will always be able to cut materials at higher speeds and feeds, but is more expensive. Carbide insert cutting tools are very useful in high production situations because the inserts are designed with multiple cutting edges on each insert. When they become worn out, you index the inserts to the next cutting edge and when all cutting edges are used, you only replace the inserts and not the whole tool.

TIP: if you are using a high-speed steel drill; always use a center drill to get the hole started. Then, drill the hole. This will ensure that the drilled hole is in the correct location. If you are using a carbide drill, it is not necessary to center drill first because carbide drills are ground with a self-centering tip. Using a carbide drill to drill a hole that is already center drilled will damage the drill. The outer cutting edges will contact the tapered walls before the tip of the drill begins to cut. This will shock the outer cutting edges and cause the drill to chip. Carbide drills must begin to cut at the tip before the outer cutting edges.

Each one of these cutting tool materials is available with a variety of different coatings to enhance their performance. The three coatings most widely use today are titanium nitride (TiN), titanium carbonitride (TiCN), and titanium aluminum nitride (TiAlN). TiN coating is easily recognized by its gold color. The advantages of TiN coating are increased surface hardness, increased tool life, better wear resistance, and higher lubricity, which provides less friction and reduces edge build-up. TiN coating is mostly recommended for machining low alloy steel and stainless steel. TiCN coating is gray colored compared to TiN, and even harder. Its advantages are increased cutting speed and feeds (40% to 60% higher compared to TiN), bigger metal removal rates, and superior wear resistance. TiCN coatings are recommended for machining all material types. TiAlN coating appears gray or black and is primarily used to coat carbide. It can work at very high temperatures, 800 degrees Celsius, which makes it ideal for high speed machining without coolant. Pressurized air is recommended to remove chips from the cutting zone. It works well on hardened steels, titanium and nickel alloys, as well as abrasive materials like cast iron and high silicon aluminum.

When selecting end mill tools, the number of flutes, or cutting edges, is an important factor. The more flutes an end mill has, the smaller, or shallower, the flutes are. The solid center section of an end mill is approximately 52% of the end mills diameter on a two-flute end mill. The center section of a three-flute end mill is 56% of its diameter, and a four or more flute end mill has a center section that is 61% of its diameter. This means that the more flutes an end mill has, the more rigid it will be in the cut. Two flute end mills are recommended for soft, gummy materials such as aluminum and copper. Four flute end mills are recommended for harder, tougher steel materials.

SECTION 5 - SPEEDS AND FEEDS

Cutting speed refers to the speed at which the cutting edge of the cutter moves with respect to the work measured in surface feet per minute (SFM). Feed is the rate at which the work moves into the cutter measured in inches (or millimeters) per minute (IPM). Feeds and speeds affect the time to finish a cut, tool life, finish of the machined surface and power required of the machine. The material to be cut and the material of the tool mostly determine the cutting speed. To calculate the proper spindle speed in revolutions per minute (RPM), multiply the suggested SFM by 3.82 and divide that sum by the diameter of the cutter being used. 3.82 is a constant factor used for transposing SFM to RPM. The feed rate depends on the width and depth of cut, finish desired and many other variables. To calculate the desired feed rate, multiply feed per tooth by number of teeth and rpm of the spindle.

TIP: To find the right speed for any task, refer to the Machinery's Handbook or other reference. Most all cutter manufacturers can provide general guidelines for their cutters based on the material to be cut. Many of them will even come to your shop to see the exact application and make suggestions for proper cutters, coatings, and cutting speeds.

TIP: Although manufacture references for tool speed and feed are provided for your convenience, they are intended for reference as a starting place to cut. In many situations the numbers given are under ideal conditions and will not always work. Experience will be valuable to tune the cutter to the conditions of the cut. Chatter and vibration may occur, to overcome these conditions alteration of the speeds and feeds will be required.

TIP: The Haas Control has as a standard feature a calculator that will assist the operator by doing calculations for trigonometry, circular interpolation, and milling. To access this feature, press the HELP/CALC button twice, then page up or down to the calculator you want to access. Enter the prompted data and the control will do the math for you.

Setting up a CNC milling machine to produce excellent quality parts in the shortest possible time requires two things. The first is a lot of common sense. The second thing is thorough knowledge of all the topics discussed in this article. There are many excellent sources of information on all of these topics. Haas Automation, Inc has an applications department that can answer all of your questions regarding their machines and the specific problems you may have with a machining application. In addition, cutting tool manufacturers can provide answers to questions about their products. Lastly, the Internet is a vast source for information on any subject.

The Haas Controller has features that will allow the user to monitor and control machine function by recording and storing data about the tools the machine is using. The Control monitors tools according to tool number and records the spindle load, feed time and usages, for each tool, storing this information for the user's convenience.

TIP: Using the Tool load page located in the current commands screens (when in current commands, any mode, page up once to the tool load data screen). The operator can select any of the above options for tool load response by changing Setting 84. If there is no limit set for a tool there will be no response by the machine. The operator can have the machine perform either of the following actions when spindle load conditions exceed the value entered in the LIMIT% column.

- Alarm condition
- Feed hold condition
- Beep (beeper on the control pendant will sound)
- Autofeed condition

Using Setting 84, the user can prevent many of the common problems that occur during the machining process. Here are some examples of how this feature is useful.

Cutters and inserts wear, and over time will require changing. The increase of the spindle load will be one of the results of this; having the machine monitor its operation is a useful benefit in this situation.

Insufficient coolant flow may cause galling or welding of material or swarf onto the tool, this will inhibit chip evacuation and the cutting action of the tool will be impaired. This will result in an increase of the spindle load; having the machine monitor the operation is a useful benefit in this situation.

Having an uneven depth or width of cut will increase the spindle load during only parts of the cut; selecting Autofeed in Setting 84 will reduce the feed rate of the machine to maintain a specified maximum that is set on the Tool Load page of the current commands. Parameters 299, 300 and 301 will control the amount of reduction and recovery time.

The operator can have the machine generate an alarm #174 Tool Load Exceeded when the spindle load exceeds the user set value for the Limit % for the tool in the cut.

TIP: Cutting tools wear over time and keeping a tool in tiptop condition is an important consideration and can yield higher production rates. Over time you can track performance of a specific tool. Once you know the number of times a tool will be able to cut a part and still maintain it's usefulness is all of the information you need. For example, if you know that a tool will fail after 27 uses, this information will allow you to set the machine to stop after 25 or 26 uses for a change of tool or inserts. You can do this by setting the number of uses you want the tool to have in the Alarm column on the Tool Life screen of the current commands.

TIP: Using the Tool life page located in the current commands screens (when in current commands, any mode, page up twice to the tool life data screen). The operator can have the machine stop at in an alarmed state when a tool is used for a specified number of times. To do this, enter the number of usages in the alarm column on this page. When the tool is used the specified number of times the machine will alarm with a tool usage alarm #362, at this time the operator can press reset, change the inserts or the tool and zero the accumulated tool usage number in the usage column on this page.

TIP: To clear the values that are stored in the tool data screens, move the cursor to the appropriate line and column then press the ORIGIN button on the keypad. If you want to clear all of the data in a column, move the cursor to the top of the column that you want to zero then press the ORIGIN key.