CNC machine tools

Outrata, J. and Jowe, J.

While the specific intention and application for CNC machines vary from one machine type to another, all forms of CNC have common benefits. Here are but a few of the more important benefits offered by CNC equipment.

The first benefit offered by all forms of CNC machine tools is improved automation. The operator intervention related to producing workpieces can be reduced or eliminated. Many CNC machines can run unattended during their entire machining cycle, freeing the operator to do other tasks. This gives the CNC user several side benefits including reduced operator fatigue, fewer mistakes caused by human error, and consistent and predictable machining time for each workpiece. Since the machine will be running under program control, the skill level required of the CNC operator (related to basic machining practice) is also reduced as compared to a machinist producing workpieces with conventional machine tools.

The second major benefit of CNC technology is consistent and accurate workpieces. Today's CNC machines boast almost unbelievable accuracy and repeatability specifications. This means that once a program is verified, two, ten, or one thousand identical workpieces can be easily produced with precision and consistency.

A third benefit offered by most forms of CNC machine tools is flexibility. Since these machines are run from programs, running a different workpiece is almost as easy as loading a different program. Once a program has been verified and executed for one production run, it can be easily recalled the next time the workpiece is to be run. This leads to yet another benefit, fast change over. Since these machines are very easy to set up and run, and since programs can be easily loaded, they allow very short setup time. This is imperative with today's just-in-time (JIT) product requirements.

Motion control - the heart of CNC

The most basic function of any CNC machine is automatic, precise, and consistent motion control. Rather than applying completely mechanical devices to cause motion as is required on most conventional machine tools, CNC machines allow motion control in a revolutionary manner2. All forms of CNC equipment have two or more directions of motion, called axes. These axes can be precisely and automatically positioned along their lengths of travel. The two most common axis types are linear (driven along a straight path) and rotary (driven along a circular path).

Instead of causing motion by turning cranks and handwheels as is required on conventional machine tools, CNC machines allow motions to be commanded through programmed commands. Generally speaking, the motion type (rapid, linear, and circular), the axes to move, the amount of motion and the motion rate (feedrate) are programmable with almost all CNC machine tools.

A CNC command executed within the control tells the drive motor to rotate a precise number of times. The rotation of the drive motor in turn rotates the ball screw. And the ball screw drives the linear axis (slide). A feedback device (linear scale) on the slide allows the control to confirm that the commanded number of rotations has taken place3. Refer to fig.1.

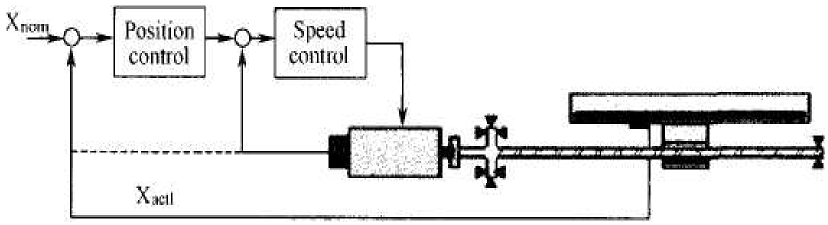


Fig.1 typical drive system of a CNC machine tool

Though a rather crude analogy, the same basic linear motion can be found on a common table vise. As you rotate the vise crank, you rotate a lead screw that, in turn, drives the movable jaw on the vise. By comparison, a linear axis on a CNC machine tool is extremely precise. The number of revolutions of the axis drive motor precisely controls the amount of linear motion along the axis.

How axis motion is commanded - understanding coordinate systems

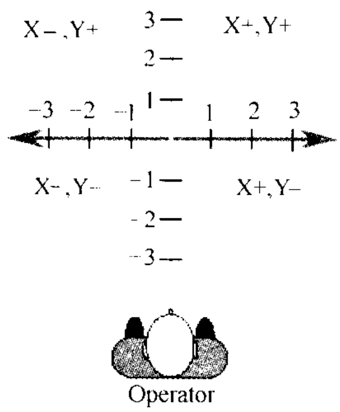
It would be infeasible for the CNC user to cause axis motion by trying to tell each axis drive motor how many times to rotate in order to command a given linear motion amount4. (This would be like having to figure out how many turns of the handle on a table vise will cause the movable jaw to move exactly one inch!) Instead, all CNC controls allow axis motion to be commanded in a much simpler and more logical way by utilizing some form of coordinate system. The two most popular coordinate systems used with CNC machines are the rectangular coordinate system and the polar coordinate system. By far, the more popular of these two is the rectangular coordinate system.

The program zero point establishes the point of reference for motion commands in a CNC program. This allows the programmer to specify movements from a common location. If program zero is chosen wisely, usually coordinates needed for the program can be taken directly from the print.

With this technique, if the programmer wishes the tool to be sent to a position one inch to the right of the program zero point, X1.0 is commanded. If the programmer wishes the tool to move to a position one inch above the program zero point, Y1.0 is commanded. The control will automatically determine how many times to rotate each axis drive motor and ball screw to make the axis reach the commanded destination point. This lets the programmer command axis motion in a very logical manner. Refer to fig.2, 3.

Understanding absolute versus incremental motion

All discussions to this point assume that the absolute mode of programming is used6. The most common CNC word used to designate the absolute mode is G90. In the absolute mode, the end points for all motions will be specified from the program zero point. For beginners, this is usually the best and easiest method of specifying end points for motion commands. However, there is another way of specifying end points for axis motion.



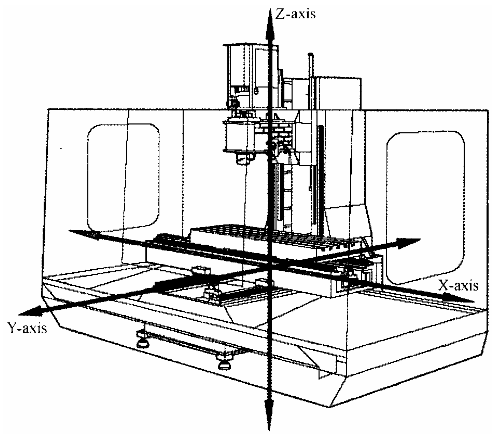


Fig. 2 view of X, Y grid from above

Fig. 3 X, Y, and Z-axis lines

In the incremental mode (commonly specified by G91), end points for motions are specified from the tool's current position, not from program zero. With this method of commanding motion, the programmer must always be asking "How far should I move the tool?" While there are times when the incremental mode can be very helpful, generally speaking, this is the more cumbersome and difficult method of specifying motion and beginners should concentrate on using the absolute mode.

Be careful when making motion commands. Beginners have the tendency to think incrementally. If working in the absolute mode (as beginners should), the programmer should always be asking "To what position should the tool be moved?" This position is relative to program zero, NOT from the tools current position.

Aside from making it very easy to determine the current position for any command, another benefit of working in the absolute mode has to do with mistakes made during motion commands. In the absolute mode, if a motion mistake is made in one command of the program, only one movement will be incorrect. On the other hand, if a mistake is made during incremental movements, all motions from the point of the mistake will also be incorrect.

Assigning program zero

Keep in mind that the CNC control must be told the location of the program zero point by one means or another. How this is done varies dramatically from one CNC machine and control to another8. One (older) method is to assign program zero in the program. With this method, the programmer tells the control how far it is from the program zero point to the starting position of the machine. This is commonly done with a G92 (or G50) command at least at the beginning of the program and possibly at the beginning of each tool.

Another, newer and better way to assign program zero is through some form of offset. Refer to fig.4. Commonly machining center control manufacturers call offsets used to assign program zero fixture offsets. Turning center manufacturers commonly call offsets used to assign program zero for each tool geometry offsets.

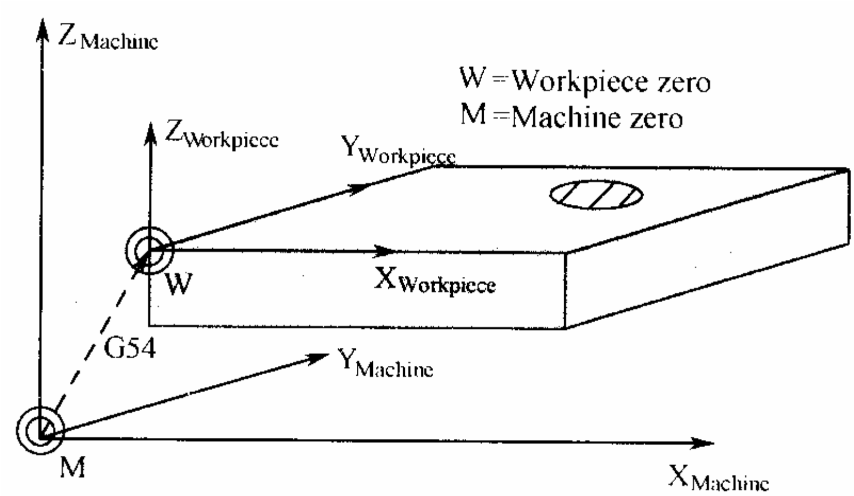


Fig.4 assign program zero through G54

Flexible manufacturing cells

A flexible manufacturing cell (FMC) can be considered as a flexible manufacturing subsystem. The following differences exist between the FMC and the FMS:

1. An FMC is not under the direct control of the central computer. Instead, instructions from the central computer are passed to the cell controller.

2. The cell is limited in the number of part families it can manufacture.

The following elements are normally found in an FMC:

- Cell controller
- Programmable logic controller (PLC)
- More than one machine tool
- A materials handling device (robot or pallet)

The FMC executes fixed machining operations with parts flowing sequentially between operations.

High speed machining

The term High Speed Machining (HSM) commonly refers to end milling at high rotational speeds and high surface feeds. For instance, the routing of pockets in aluminum airframe sections with a very high material removal rate1. Refer to fig.5 for the cutting data designations and for mulas. Over the past 60 years, HSM has been applied to a wide range of metallic and non-metallic workpiece materials, including the production of components with specific surface topography requirements and machining of materials with hardness of 50 HRC and above. With most steel components hardened to approximately 32-42 HRC, machining options currently include:

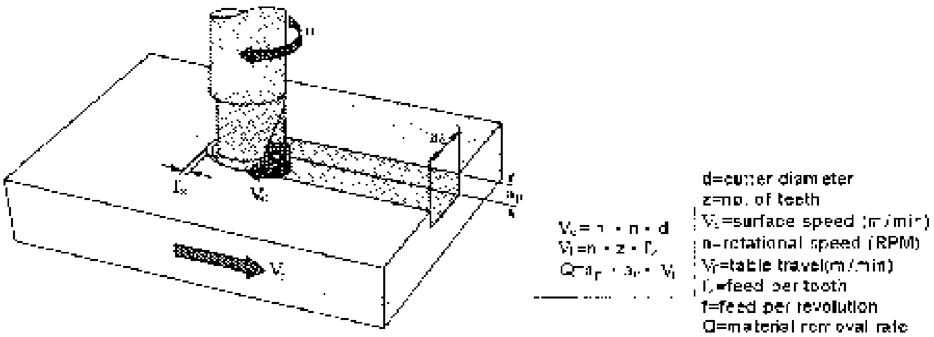


Fig.5 cutting data

rough machining and semi-finishing of the material in its soft (annealed) condition heat treatment to achieve the final required hardness = 63 HRC machining of electrodes and Electrical Discharge Machining (EDM) of specific parts of dies and moulds (specifically small radii and deep cavities with limited accessibility for metal cutting tools) finishing and super-finishing of cylindrical/flat/cavity surfaces with appropriate cemented carbide, cermet, solid carbide, mixed ceramic or polycrystalline cubic boron nitride (PCBN).

For many components, the production process involves a combination of these options and in the case of dies and moulds it also includes time consuming hand finishing. Consequently, production costs can be high and lead times excessive.

It is typical in the die and mould industry to produce one or just a few tools of the same design. The process involves constant changes to the design, and because of these changes there is also a corresponding need for measuring and reverse engineering .

The main criteria is the quality level of the die or mould regarding dimensional, geometric and surface accuracy. If the quality level after machining is poor and if it cannot meet the requirements, there will be a varying need of manual finishing work. This work produces satisfactory surface accuracy, but it always has a negative impact on the dimensional and geometric accuracy.

One of the main aims for the die and mould industry has been, and still is, to reduce or eliminate the need for manual polishing and thus improve the quality and shorten the production costs and lead times.

Main economical and technical factors for the development of HSM

Survival

The ever increasing competition in the marketplace is continually setting new standards. The demands on time and cost efficiency is getting higher and higher. This has forced the development of new processes and production techniques to take place. HSM provides hope and solutions...

Materials

The development of new, more difficult to machine materials has underlined the necessity to find new machining solutions. The aerospace industry has its heat resistant and stainless steel alloys. The automotive industry has different bimetal compositions, Compact Graphite Iron and an ever increasing volume of aluminum3. The die and mould industry mainly has to face the problem of machining high hardened tool steels, from roughing to finishing.

Quality

The demand for higher component or product quality is the result of ever increasing competition. HSM, if applied correctly, offers a number of solutions in this area. Substitution of manual finishing is one example, which is especially important on dies and moulds or components with a complex 3D geometry.

Processes

The demands on shorter throughput times via fewer setups and simplified flows (logistics) can in most cases, be solved by HSM. A typical target within the die and mould industry is to completely machine fully hardened small sized tools in one setup. Costly and time consuming EDM processes can also be reduced or eliminated with HSM.

Design & development

One of the main tools in today's competition is to sell products on the value of novelty. The average product life cycle on cars today is 4 years, computers and accessories 1.5 years, hand phones 3 months... One of the prerequisites of this development of fast design changes and rapid product development time is the HSM technique.

Complex products

There is an increase of multi-functional surfaces on components, such as new design of turbine blades giving new and optimized functions and features. Earlier designs allowed polishing by hand or with robots (manipulators). Turbine blades with new, more sophisticated designs have to be finished via machining and preferably by HSM . There are also more and more examples of thin walled workpieces that have to be machined (medical equipment, electronics, defence products, computer parts).

Production equipment

The strong development of cutting materials, holding tools, machine tools, controls and especially CAD/CAM features and equipment, has opened possibilities that must be met with new production methods and techniques5.

Definition of HSM

Salomon's theory, "Machining with high cutting speeds..." on which, in 1931, took out a German patent, assumes that "at a certain cutting speed (5-10 times higher than in conventional machining), the chip removal temperature at the cutting edge will start to decrease...".See fig.6.

Given the conclusion:" ... seems to give a chance to improve productivity in machining with conventional tools at high cutting speeds..."

Modern research, unfortunately, has not been able to verify this theory totally. There is a relative decrease of the temperature at the cutting edge that starts at certain cutting speeds for different materials.

The decrease is small for steel and cast iron. But larger for aluminum and other non-ferrous metals. The definition of HSM must be based on other factors.

Given today's technology, "high speed" is generally accepted to mean surface speeds between 1 and 10 kilometers perminute, or roughly 3 300 to 33 000 feet per minute. Speeds above 10 km/min are in the ultra-high speed category, and are largely the realm of experimental metal cutting. Obviously, the spindle rotations required to achieve these surface cutting speeds are directly related to the diameter of the tools being used. One trend which is very evident today is the use of very large cutter diameters for these applications - and this has important implications for tool design.

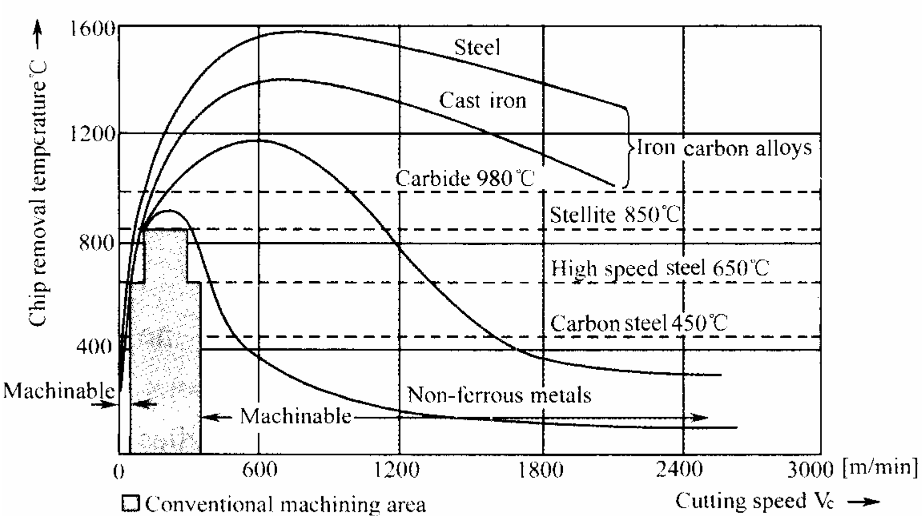


Fig.6 chip removal temperature as a result of the cutting speed

There are many opinions, many myths and many different ways to define HSM.

Maintenance and troubleshooting

Maintenance for a horizontal MC

The following is a list of required regular maintenance for a Horizontal Machining Center as shown in fig.7. Listed are the frequency of service, capacities, and type of fluids required. These required specifications must be followed in order to keep your machine in good working order and protect your warranty.

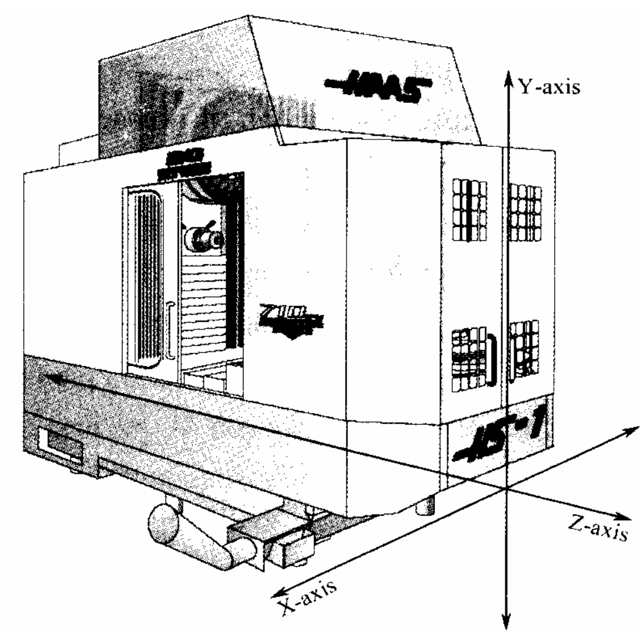


Fig. 7 horizontal machining center

Daily

Top off coolant level every eight hour shift (especially during heavy TSC usage).

Check way lube lubrication tank level.

Clean chips from way covers and bottom pan.

Clean chips from tool changer.

Wipe spindle taper with a clean cloth rag and apply light oil.

Weekly

- Check for proper operation of auto drain on filter regulator. See fig. 8

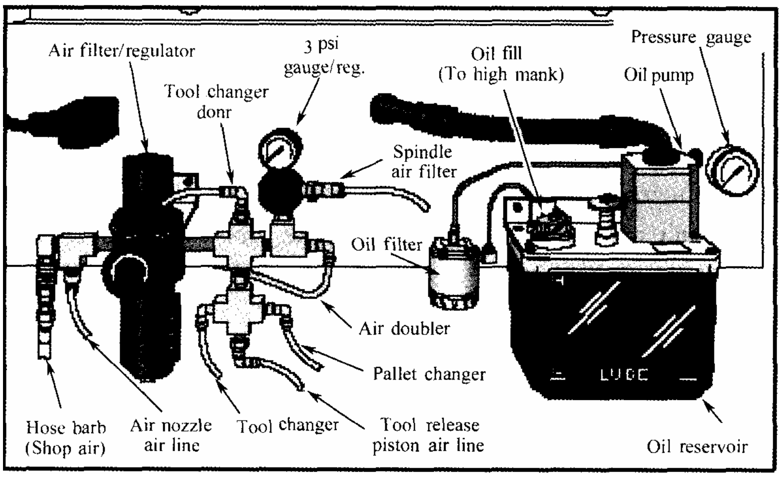


Fig. 8 way lube and pneumatics

On machines with the TSC option, clean the chip basket on the coolant tank.

Remove the tank cover and remove any sediment inside the tank. Be careful to disconnect the coolant pump from the controller and POWER OFF the control before working on the coolant tank . Do this monthly for machines without the TSC option.

Check air gauge/regulator for 85 psi.

For machines with the TSC option, place a dab of grease on the V-flange of tools. Do this monthly for machines without the TSC option.

Clean exterior surfaces with mild cleaner. DO NOT use solvents.

Check the hydraulic counterbalance pressure according to the machine's specifications.

Place a dab of grease on the outside edge of the fingers of the tool changer and run through all tools".

Monthly

Check oil level in gearbox. Add oil until oil begins dripping from over flow tube at bottom of sump tank.

Clean pads on bottom of pallets.

Clean the locating pads on the A-axis and the load station. This requires removing the pallet.

- Inspect way covers for proper operation and lubricate with light oil, if necessary.

Six months

Replace coolant and thoroughly clean the coolant tank.

Check all hoses and lubrication lines for cracking.

Annually

- Replace the gearbox oil. Drain the oil from the gearbox, and slowly refill it with 2 quarts of Mobil DTE 25 oil.
- Check oil filter and clean out residue at bottom for the lubrication chart.

Replace air filter on control box every 2 years.

Mineral cutting oils will damage rubber based components throughout the machine.

Troubleshooting

This section is intended for use in determining the solution to a known problem. Solutions given are intended to give the individual servicing the CNC a pattern to follow in, first, determining the problem's source and, second, solving the problem.

Use common sense

Many problems are easily overcome by correctly evaluating the situation. All machine operations are composed of a program, tools, and tooling. You must look at all three before blaming one as the fault area. If a bored hole is chattering because of an overextended boring bar, don't expect the machine to correct the fault.

Don't suspect machine accuracy if the vise bends the part. Don't claim hole mis-positioning if you don't first center-drill the hole.

Find the problem first

Many mechanics tear into things before they understand the problem, hoping that it will appear as they go. We know this from the fact that more than half of all warranty returned parts are in good working order. If the spindle doesn't turn, remember that the spindle is connected to the gear box, which is connected to the spindle motor, which is driven by the spindle drive, which is connected to the I/O BOARD, which is driven by the MOCON, which is driven by the processor. The moral here is don't replace the spindle drive if the belt is broken. Find the problem first; don't just replace the easiest part to get to.

Don’t tinker with the machine

There are hundreds of parameters, wires, switches, etc., that you can change in this machine. Don't start randomly changing parts and parameters. Remember, there is a good chance that if you change something, you will incorrectly install it or break something else in the process6. Consider for a moment changing the processor's board. First, you have to download all parameters, remove a dozen connectors, replace the board, reconnect and reload, and if you make one mistake or bend one tiny pin it WON'T WORK. You always need to consider the risk of accidentally damaging the machine anytime you work on it. It is cheap insurance to double-check a suspect part before physically changing it. The less work you do on the machine the better.