DENFORD

Total Commitment to Education and Training WorldWide.

Turning Courseware for CNC Machines.

Notes

Contact Details

Address:	Denford Limited, Birds Royd, Brighouse, West Yorkshire, HD6 1NB, UK.
Telephone:	General Enquiries +44 (0) 1484 712264 Customer Services +44 (0) 1484 722733
Fax:	+44 (0) 1484 722160
e-mail:	for customer services and technical support contact, customer_services@denford.co.uk

Contents

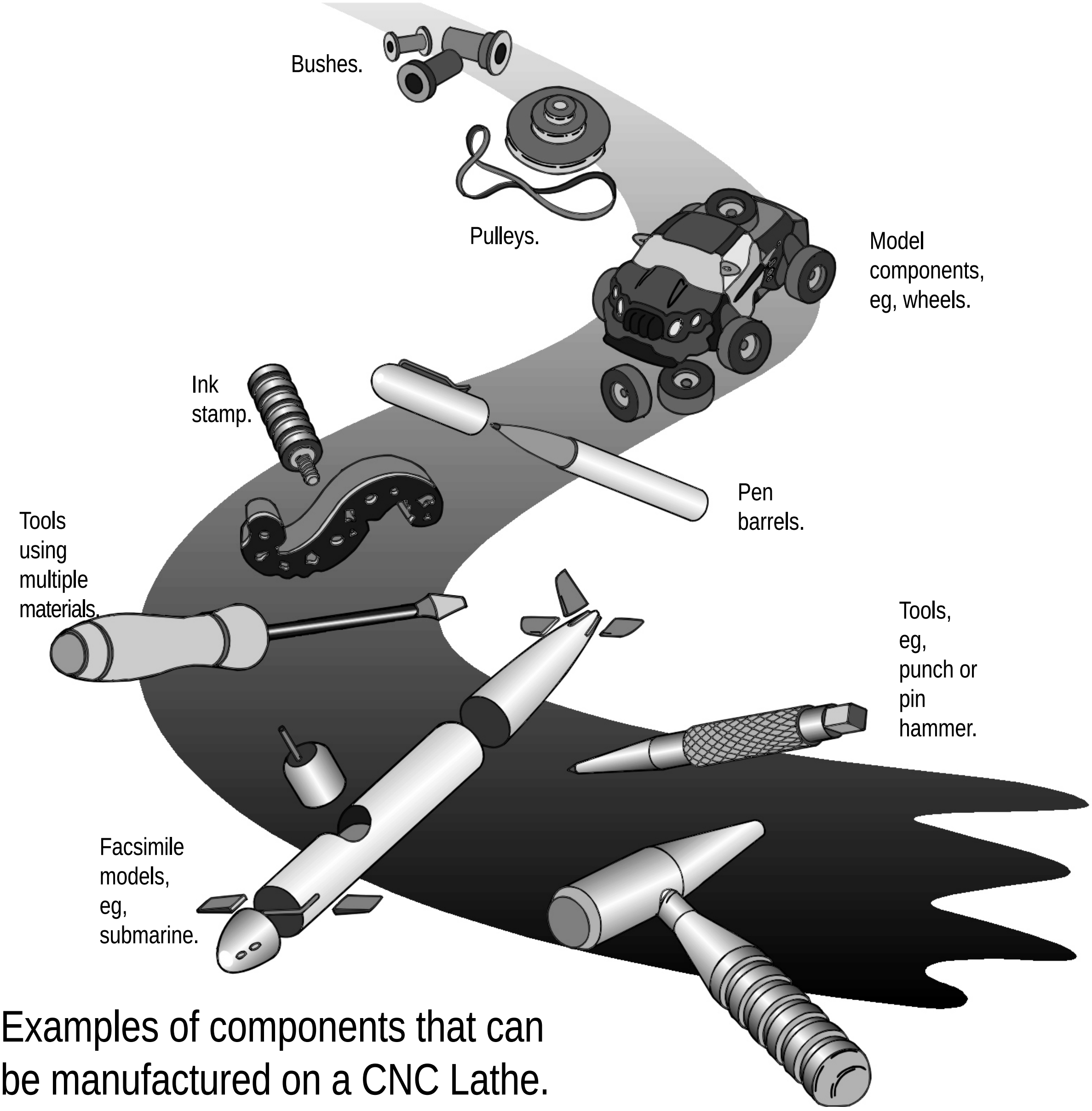
Contents

What can be made on a CNC Lathe?

Most mechanical devices we use today contain at least one cylindrical component - for example, pistons, pulleys, bushes, wheels and connecting rods. In components such as these, the quality of fit and finish are paramount to the success of the designs in which they are used.

In order to achieve such a high quality of precision, the components are turned, manufactured using machine tools called lathes.

Where large numbers of identical components are required, the process is usually automated, using computers to drive the lathe. Machine tools such as these are called CNC (Computer Numerically Controlled) lathes. The movement of the 2 axes on a cnc lathe can be used to generate very complicated shapes, from simple straight lines to complex curves, producing 3 dimensional shapes.

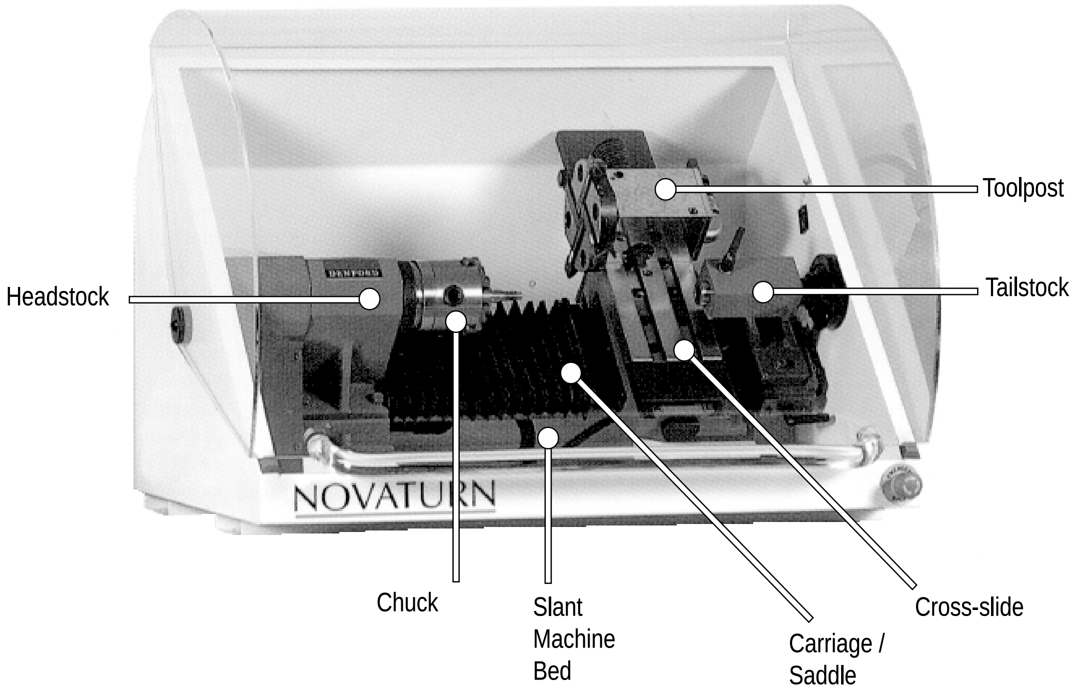


Examples of components that can be manufactured on a CNC Lathe.

Parts of a CNC Lathe

Basic Lathe Parts.

The main parts of the CNC lathe are the machine bed, headstock, cross-slide, carriage, turret, tailstock, drive motors, ballscrews, hydraulic and lubrication systems and the MCU (machine control unit).



Identification of Basic Lathe Parts.

Lathe Sizes.

Lathes sizes are categorised by two factors:

1) The Swing of the lathe. This represents the largest diameter billet that could be machined.
2) The distance between the centres. This represents the longest billet that could be machined. The distance is calculated along the spindle centre line from the headstock spindle nose to the rearmost part of the tailstock.

Parts of a CNC Lathe

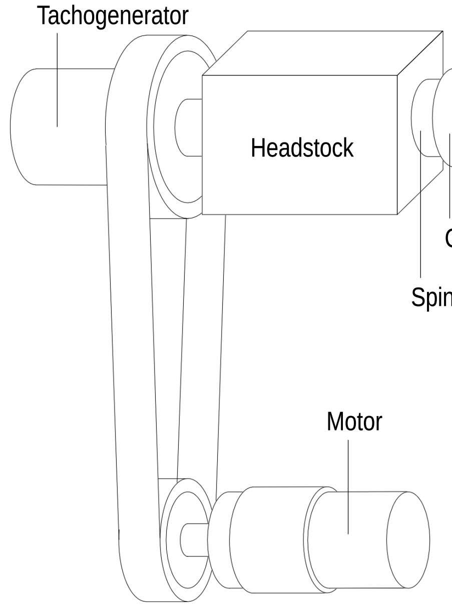
The Machine Bed.

The bed is usually made from high quality cast iron, well suited to absorb the shock created from heavy machining cuts. It is common to find beds of slant design in a CNC lathe, angled from 30° to 45°, providing easier access for the loading and unloading of both tools and workpieces. Angling the bed also allows any chips and coolant to fall directly to the base of the machine cabinet. Parallel surfaces are machined into the front of the cast bed, providing mounting tracks for the hardened bed ways. The machine bed is fixed, running parallel to the machine spindle centreline.

The Headstock.

The headstock contains the spindle drive motors, gearing and chuck. The chuck is the area of the lathe where the work material, called the workpiece, or billet, is held. The headstock transmits the maximum power and torque from the motor to the spindle. CNC machines are available with a variety of drive motor sizes, ranging from around 7 to 75 horsepower and spindle speeds from around 30 to 5500 rpm. The spindle speed is usually programmable in 1 rpm increments.

The diagram right shows a spindle driven by a three-phase AC or DC motor. In three-phase AC drives, the spindle speed selection is obtained through use of a gearbox. Depending on the design of the gearbox, a number of different fixed speeds can be selected. However, most CNC lathes use AC or DC servo motors, since their speeds are infinitely variable between their designated speed ranges, through use of a tachogenerator. To achieve the most favourable torques for particular machining operations and also to change the spindle speed range, AC and DC drives frequently incorporate a reduction transmission with two to four stages.



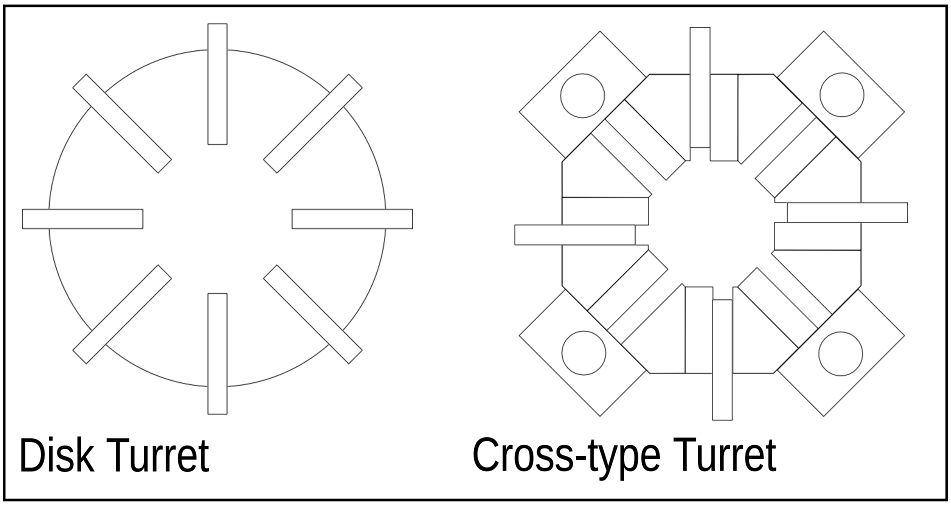
CNC Lathe Work Spindle.

The MCU - Machine Contol Unit.

The MCU allows the operator to enter, edit, store, graphically display and export any part programs. Additionally, the MCU can perform comprehensive diagnostics and run the parogram manually or automatically.

Parts of a CNC Lathe

The Turret.



It is rare that workpieces can be completely machined without a tool change. The turret is the component that holds all the various single and multi-point cutting tools used to machine the workpiece. The type, style and number of turrets on individual CNC lathes differ, according to the size of the machine and its specifications. The most common types are the drum turret and the square multitool turret - each constructed to hold up to eight or more inner and outer turning tools. Calling a tool change from the CNC program has the effect of causing the tool to rotate until the required tool is at its correct working portion. Since most toolchangers use a bi-directional indexing system, together with rapid traverse rates approaching 100 m/min, non-cutting time can be dramatically improved on machining operations that demand frequent tool changes.

The Tailstock.

The tailstock is positioned at the opposite end of the headstock, running parallel and in line with the spindle centreline. Tailstocks are primarily used for drilling, boring operations and supporting long workpieces. On some lathes that turn small components, it is normal for the tailstock to be omitted.

CNC lathes can be equipped with different types of tailstock; a manual tailstock similar to a standard manual lathe, an automatically controlled tailstock, or a swing-up tailstock. The tailstock travels on its own hardened and ground bearing ways. This allows the carriage to move past the tailstock when a short workpiece is being held. It also eliminates the need to extend the quill of the tailstock to its maximum distance, maintaining the rigidity of the part.

The automatically controlled tailstock can be moved via CNC program command, or manually using the switches on the MCU. Positioning, clamping and release of the tailstock to the bearing ways is achieved using hydraulic pressure. Many tailstocks are fitted with motion sensors to prevent them from colliding with any indexing tools.

The swing-up tailstock adds flexibility since it can swing-up to support workpieces for external machining, then swing away to allow internal operations such as deep hole drilling and boring.

Principles of turning a component on a Lathe

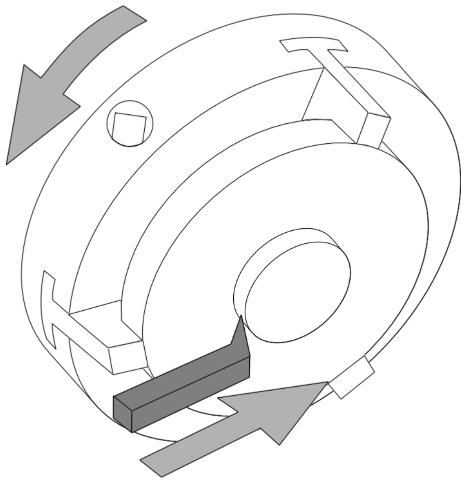
Principles of turning a component on a lathe.

The billet is held securely in a the lathe chuck. The chuck rotates the billet and a cutting tool, mounted on a toolpost, is moved around, across and into the rotating billet, removing material using a wedge type cutting action.

The path that the cutting tool follows determines the shape of the component. All components manufactured on lathes are composed of either cylindrical or flat shapes.

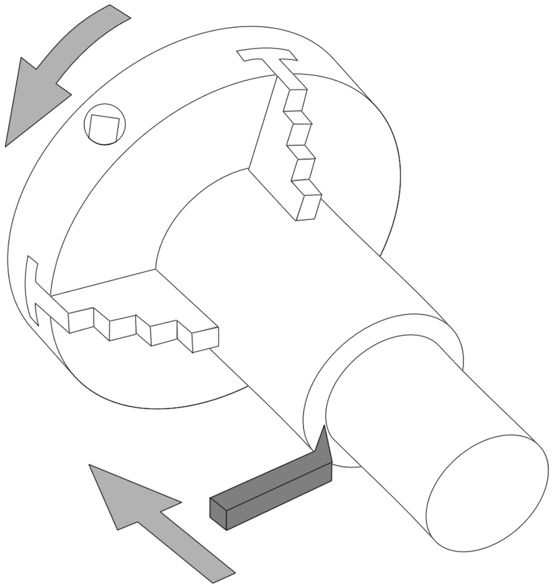
Example - Flat shape - Facing.

The cutting tool moves in a straight line across the end of the billet, at right angles to the chuck spindle.



Example - Cylindrical shape - Parallel Turning.

The cutting tool moves along the side of the billet in a straight line parallel to the chuck spindle.



Types of Lathe Operation

Operation.	Description.
Rough Turning	The process of removing excess material from the workpiece in a minimum time, by using a high feedrate and a high depth of cut.
Finish Turning.	The process of generating a high quality, smooth surface finish, by using a high spindle speed, a low feedrate and a low depth of cut.
Shoulder Turning.	The process of generating a stepped cylindrical shape. Types of shoulder include, square, bevel and radius.
Taper Turning.	The process of generating a conical shape by turning a cylinder whilst gradually reducing the diameter along its length.
Facing.	The process of machining the ends of the workpiece to produce a flat surface square with the Z axis.
Knurling.	The process of embossing a diamond shaped pattern onto the workpiece, used as a surface for gripping.
Grooving.	The process of reducing the diameter of the workpiece over a very narrow surface.
Forming.	The process of turning convex, concave or any other irregularly shaped surface.
Drilling.	The process of machining or enlarging a hole, using a drill.
Boring.	The process of turning internal surfaces, using a single point cutting tool.
Counter-boring.	The process of enlarging a hole a certain distance from one end of the workpiece, rather than enlarging the entire length of the drilled hole.
Reaming.	The process of producing holes having a smooth surface finish, good dimensional accuracy and close tolerances, required for precision assembly.
Thread Cutting.	The process of machining specified external threads.
Parting Off.	The process of cutting the finished workpiece from the billet, following all the required turning operations.

Axis Configuration and Identification

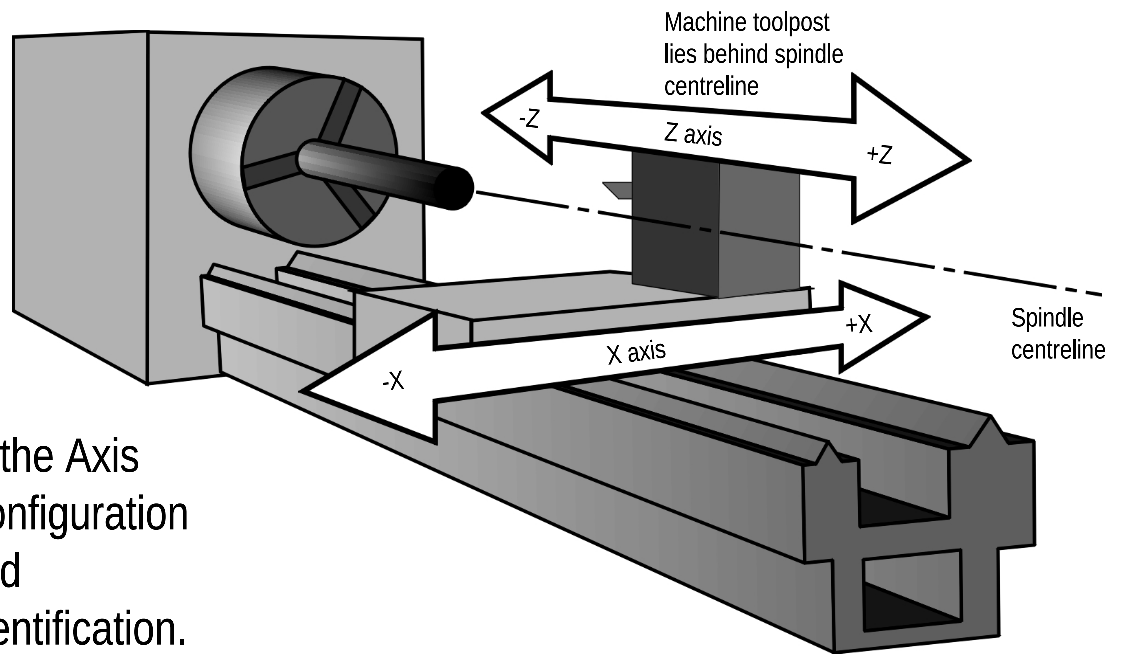
All machine tools have more than one slide. It is important that each slide is identified individually. There are two planes in which movement can take place on a lathe:

- Longitudinal.
- Transverse.

Each plane is assigned a letter and is referred to as an axis:

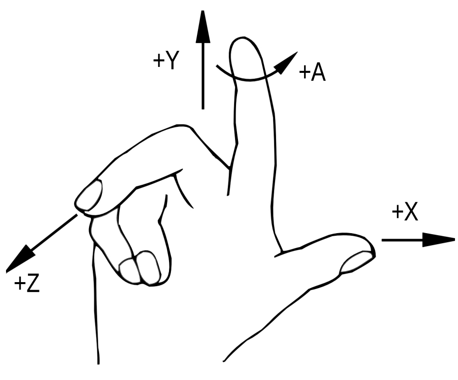
- Z Axis - This axis is always parallel to the main spindle of the machine.
- X Axis - This axis is always parallel to the work holding surface and always at right angles to the Z axis.

Additionally, each axis is identified with a plus (+) or minus (-) sign, according the direction of travel and co-ordinate system in use.



Lathe Axis Configuration and Identification.

The co-ordinate system used for designating the axes is the conventional 'Righthand co-ordinate system', also called the 'Clockwise rotating co-ordinate system'. A labelling of the axes is a righthand co-ordinate system whenever the fingers of the right hand are aligned with the positive X axis and are then rotated (through the smaller angle) towards the positive Y axis, then the thumb of the right hand points in the direction of the positive Z axis. This system is used when the toolpost is positioned behind the spindle centreline. Otherwise, the orientation is a 'lefthand co-ordinate system'.

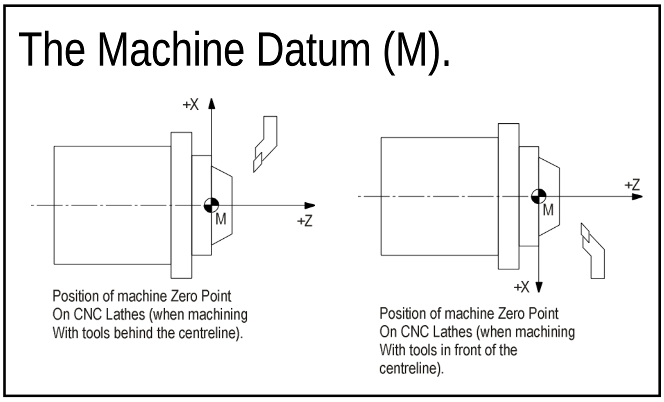


Righthand Co-ordinate System.

CNC Machining Datum Points

On CNC machines, all tool movements are controlled using co-ordinating systems. Their accurate positioning within the machine tool is established using various datums (co-ordinate points from which a series of measurement can be taken).

The Machine Datum (M).



Position of machine Zero Point On CNC Lathes (when machining With tools behind the centreline).

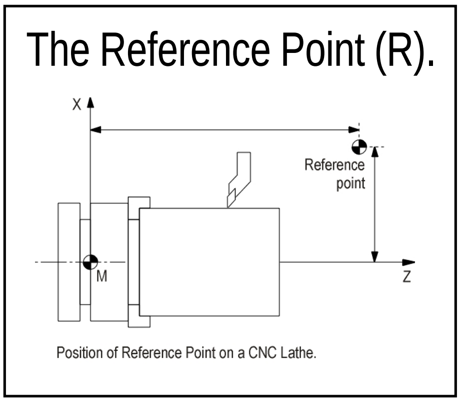
Position of machine Zero Point On CNC Lathes (when machining With tools in front of the centreline).

The *machine datum* is the zero point for the co-ordinate systems and reference points within the machine. The machine datum is positioned inside the programmable area of movement and its position is set by the machine manufacturer. It is from the machine datum that the machine calculates any programmed movements. It is often referred to as the *machine zero point*, or *zero datum*, designated with the letter *M*.

On CNC lathes, the machine datum is generally positioned along the spindle centreline and sometimes at the centre of the spindle nose face. The main spindle axis (centreline) represents the Z axis, the face determines the X axis. The directions of the positive X and Z axes point towards the working area, so when the tool traverses in a positive direction it always moves away from the workpiece. In some cases the machine datum may be moved, or offset to a new position, to suit a particular machining exercise. This is done by using the *zero offset* facility.

However, some CNC machine manufacturers align the machine datum point with their machine reference point on one or both slide axes - this can make it easier to set some values when setting the tool offsets. Check your CNC lathe operator manual for precise details about datum positions and how to home your particular CNC machine.

The Reference Point (R).



Position of Reference Point on a CNC Lathe.

The *reference point* is used for calibrating and controlling the measuring system of the slides and tool traverses. The position of the reference point is determined by limit switches along the X and Z axes. Therefore, the reference point co-ordinates always have the same precisely known numerical values in relation to the machine zero point.

Homing the Machine

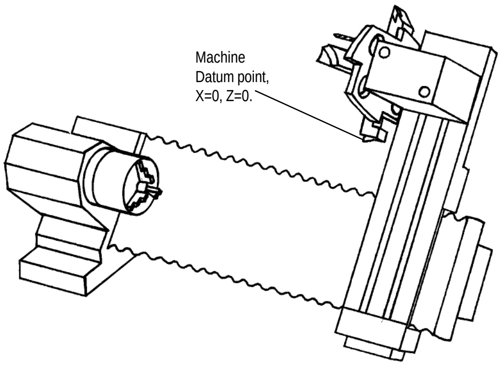
Homing the CNC machine.

After first starting the machine controlling software, the machine must be *datumed*, or *homed*. The *reference point* is approached from all axis directions to properly calibrate the slide measuring system, otherwise the machine will not know the co-ordinate position of any tools in relation to its *machine datum* and its working envelope.

The working envelope describes the maximum area of movement for the machine co-ordinate system.

If the current position of the axes is lost during a machining process, for example through an electrical failure or emergency stop procedure, all positional data will be lost. The machine must be rehomed to re-establish its positional values before any work can continue.

In the example shown below, the CNC lathe is first switched on, the machine datum point is set, by homing the machine. After homing the machine to find the full working envelope, the machine datum is confirmed as the tip of the currently selected tool, with the co-ordinates X=0, Z=0.



Homing the Machine.

Please Note - The diagram shows tools fitted to a toolpost which homes in the back right-hand corner of the machine (all Denford Automatic Toolpost and most manual toolpost machines) using a righthand co-ordinate system.

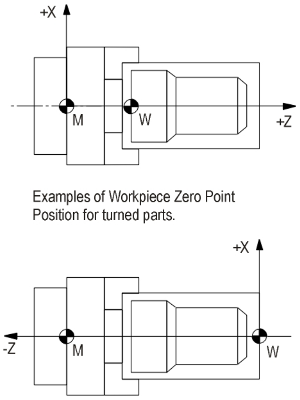
CNC Machining Datum Points

The Workpiece Zero Point (W).

The *workpiece zero point* is devised by the programmer when he writes the part program. It is often referred to as the *program datum*, or *program zero point*, designated with the letter *W*. All program co-ordinate movements are calculated relative to this point. It is usual to make the workpiece zero point and the machine datum the same point.

This point determines the workpiece co-ordinate system in relation to the machine zero point.

The Workpiece Zero Point (W).



Examples of Workpiece Zero Point Position for turned parts.

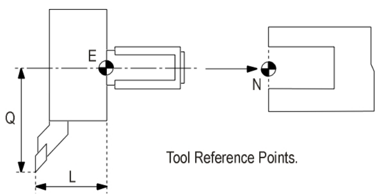
The workpiece zero point is chosen by the programmer, set by parameters in the CAD/CAM software, or input into the CNC machine when setting up the machine. The workpiece zero point can be freely chosen by the programmer to lie anywhere within the working envelope of the machine.

However, the workpiece zero point is usually positioned so that the dimensions in the workpiece drawing can be conveniently converted into co-ordinate values and orientation, when clamping, setting up the chuck and setting up the tools. Generally, the workpiece zero point is positioned along the spindle axis (centreline), aligned with the right-hand or left-hand face end of the finished contour.

The Tool Reference Point.

When machining a workpiece, it is essential to control the tool point or tool cutting edges in precise relationship to the workpiece along the machining path. Since all cutting tools have different shapes and sizes, the precise tool dimensions must be carefully established and input into the control system before attempting any machining. Any tool dimensions are related to a fixed tool setting point during presetting.

The Tool Reference Point.



Tool Reference Points.

The tool setting point, E, is located at a certain point on the tool holder. This setting point permits the measuring of any tool data away from the machine. Data such as tool length, tool point offset or tool nose radius are input into the data storage memory of the control system. The mate of the tool setting point is the socket point N on the tool carrier. When the tool or tool holder is inserted into the tool carrier, the setting point and the tool socket point exactly coincide.

Configuring the Tool Offsets

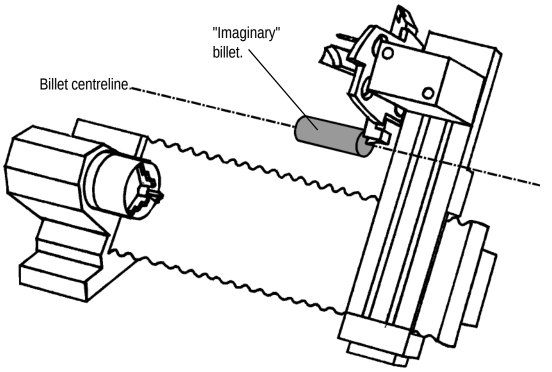
The Zero Offset.

The *zero offset* facility allows the machine datum to be moved temporarily. After the machine datum has been relocated, all slide movement measurements will be taken from this the new datum. A typical use for this facility is where two identical components are to be machined in one setting. The process of moving the machine datum point temporarily is commonly referred to as *setting the tool offsets*.

Configuring the Tool Offsets.

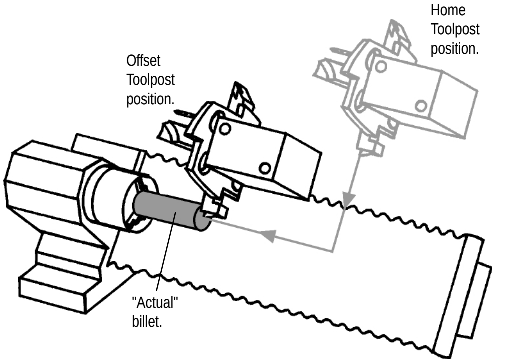
Configuring the tool offsets aligns each tool where the *actual* billet is positioned, so each tool can start cutting in the correct place.

The G-code program describes the path taken by the tools when cutting the part. All these movement commands are relative to a position set within the G-code program. This position is usually the face end of the billet along its centreline, with the co-ordinates X=0, Z=0.



Loading the G-code program.

When the G-code program is loaded into the machine, the machine datum point (ie, the tip of the currently selected tool) is also the starting position for the G-code program. The program will presume that the billet that we want to cut is also positioned here - see the diagram left.



Tool Offsets position.

However, the *actual* billet, held in the chuck, is positioned in a different area of the machine. Therefore, the starting position for the G-code program must be moved, or "offset", so it coincides with the face end of the *actual* billet along its centreline - see the diagram left.

This amount of offset between the machine datum point for the currently selected tool and the fixed position of the actual billet will differ according to the type of tool, since all the tools differ in size. The offsets must be set individually for every tool that will be used to make the part.

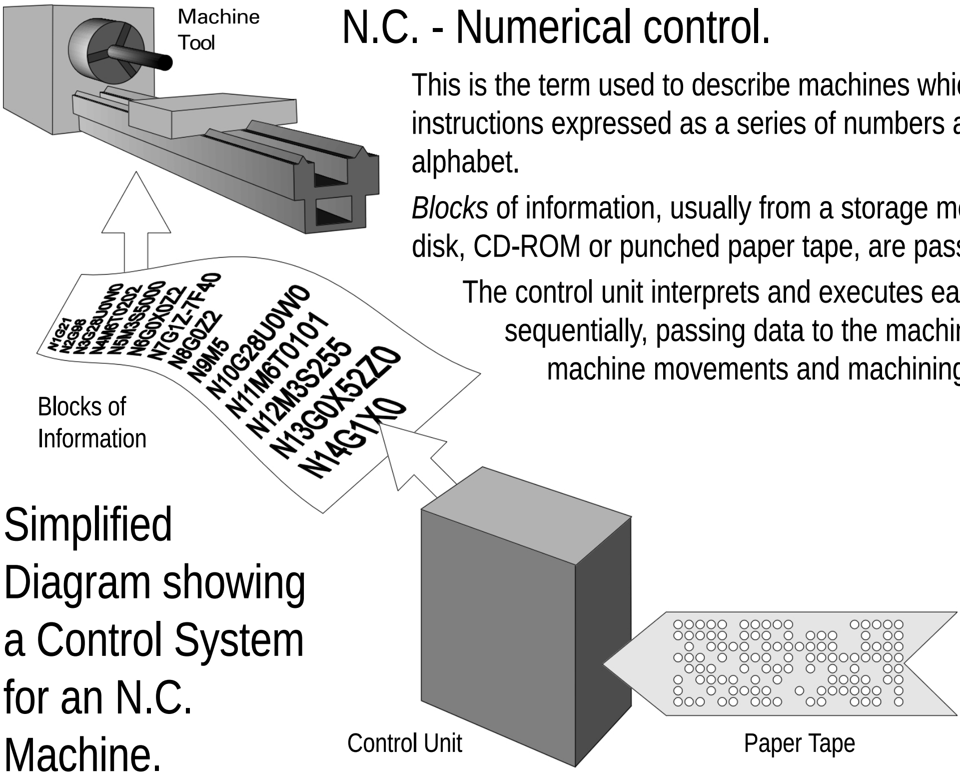
Control Methods

N.C. - Numerical control.

This is the term used to describe machines which are controlled by instructions expressed as a series of numbers and letters of the alphabet.

Blocks of information, usually from a storage medium such as floppy disk, CD-ROM or punched paper tape, are passed to the control unit.

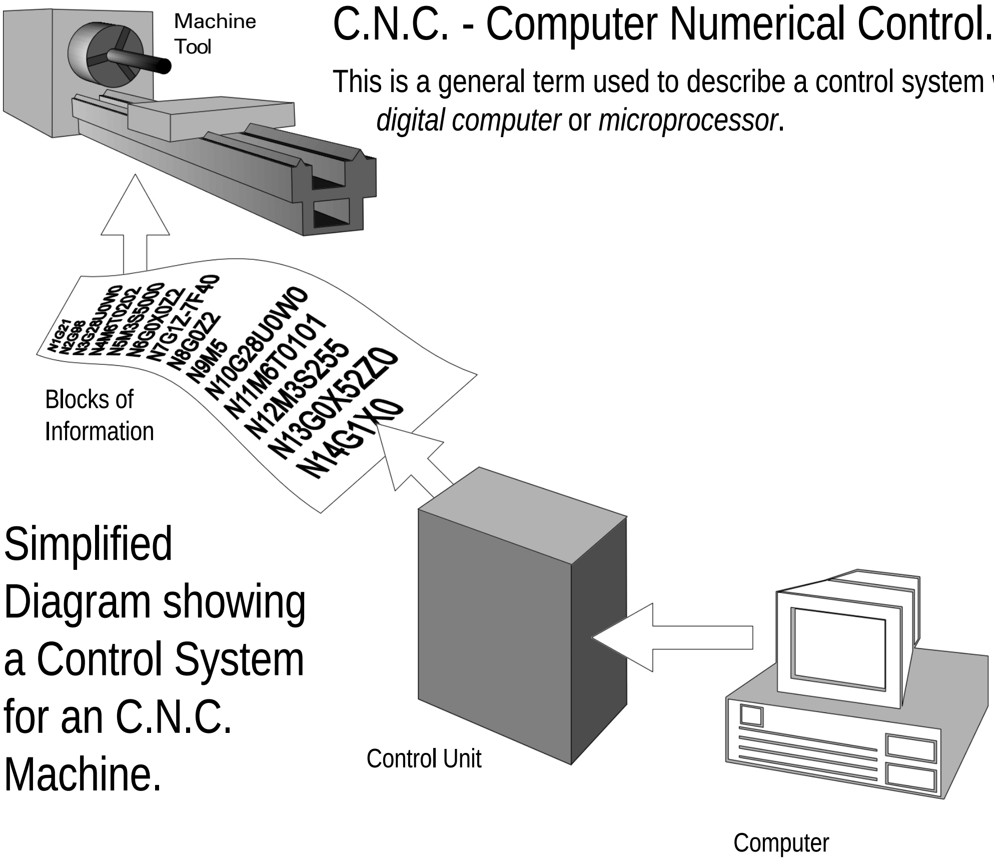
The control unit interprets and executes each block of information sequentially, passing data to the machine tool which defines machine movements and machining parameters.



Simplified Diagram showing a Control System for an N.C. Machine.

C.N.C. - Computer Numerical Control.

This is a general term used to describe a control system which includes a *digital computer* or *microprocessor*.



Simplified Diagram showing a Control System for an C.N.C. Machine.

Control Methods

Data, in the form of a coded program, may be entered in several ways.

For example:

- MDI (Manual Data Input).
- By prepared paper tape.
- By magnetic tape.
- From a floppy disc or CD-ROM.
- From a micro computer.
- CAM (Computer Aided Manufacture).

Computer Numerical control is adaptable to a wide range of manufacturing processes.

Applications include :

- Metal cutting
- Welding
- Flame cutting
- Woodworking

Machines of this type are capable of working long hours with minimum supervision and are ideally suited to mass and batch production. Additionally, the manufacturing industries which are adopting new CNC technology are able to achieve increased productivity with reduced operator involvement.

The 3 Jaw Chuck

The method of holding the workpiece is important, since unlike many other types of machine tool, the workpiece itself rotates at very high speeds, not the cutting tool. Similarly, any cutting forces imposed on the workpiece tend to be higher on CNC machines. The demand for heavier metal removal rates and higher spindle speeds also dictates the need for high performance clamping methods. Therefore the workpiece must be held both firmly and securely, to enable safe machining to take place.

The clamp used to hold the billet at the headstock end of the lathe is called the *Chuck*. Along with the high performance requirements already mentioned is a need for better and more secure gripping of the workpiece, quick-change jaws and specialised chucks that can hold different size workpieces.

The 3 Jaw Chuck.

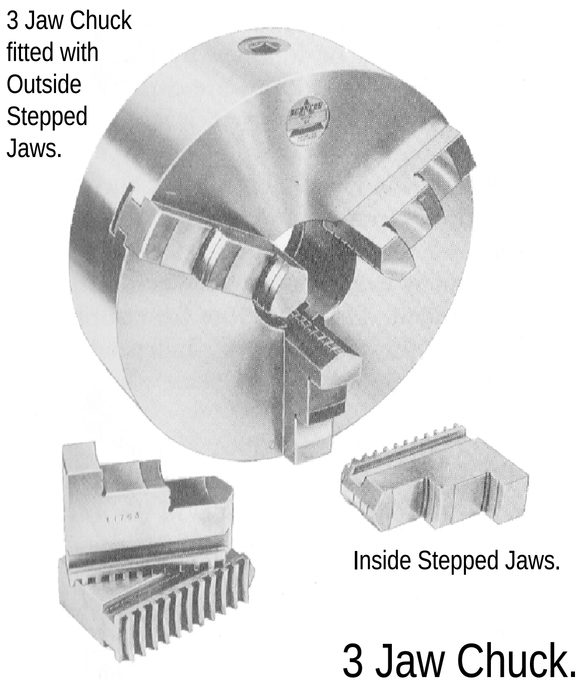
The self centring 3 jaw chuck is used to centrally hold cylindrical section billets, or billets with side faces divisible by three, such as hexagonal sections.

On manual versions of the 3 jaw chuck, the jaws are opened using a square ended chuck key. When the key is rotated in one of the slots provided on the side of the chuck, all three jaws open or close together simultaneously. Whilst closing the jaws you should always ensure that the centre line of your billet remains along the centre line of the chuck (the spindle centre line). Turning the key clockwise closes the jaws. Turning the key anticlockwise opens the jaws.

Always ensure that chuck key has been removed before using the lathe.

Self centring 3 jaw chucks are normally more accurate than other types of chuck. They are recommended for general turning operations using bar stock, forgings and castings which can be gripped from a turned diameter.

For high precision machining, it is recommended that all operations on one end of a billet are completed before altering the position of the billet in the chuck. Despite being self centring, it is unlikely that the same billet will be replaced in the chuck in exactly the same position.



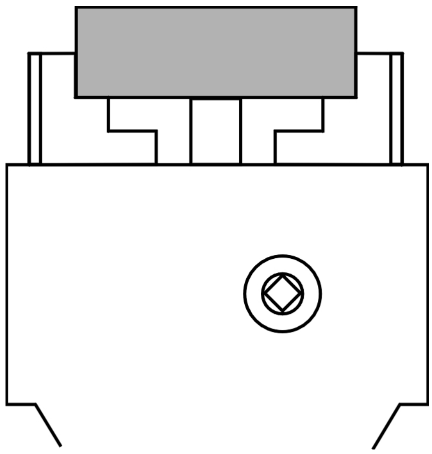
3 Jaw Chuck.

The 3 Jaw Chuck

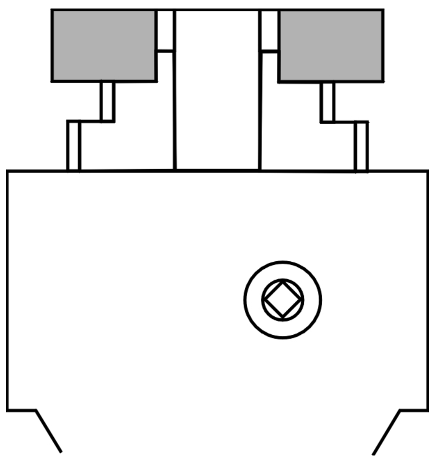
Each 3 jaw chuck has two sets of jaws:

- *Inside stepped jaws* - Positioned with the steps pointing towards the centre of the chuck. These allow solid billets of various dimensions to be held.
- *Outside stepped jaws* - Positioned with the steps pointing away from the centre of the chuck. These allow the inside edges of any bored cylinders to be held.

Both sets of jaws can be changed, when required. When changing a set of jaws, you must ensure that the numbers stamped on each jaw is match correctly with the numbered slots on the chuck body.



Holding a solid billet in a 3 Jaw Chuck using outside stepped jaws.



Holding a borred billet in a 3 Jaw Chuck using inside stepped jaws.

The 3 Jaw Chuck

Removing the Jaws on a manual 3 Jaw Chuck:

1) Isolate the machine power supply. Using the chuck key, wind out each of the jaws until they protrude beyond the outside edge of the chuck.
2) While continuing to unwind the jaws, gently pull jaw number 3 out until it slides free from the chuck.
3) Repeat this process with jaw number 2 and finally jaw number 1.

Engaging the Jaws on a manual 3 Jaw Chuck:

1) Isolate the machine power supply. Ensure that the jaw teeth and slots are clean.
2) Insert the chuck key, rotating it clockwise until the scroll tooth's outer edge appears at the top of the number 1 chuck slot. Then turn the chuck key anticlockwise a short distance so that the scroll tooth is no longer visible.
3) The chuck jaw marked number 1 should then be engaged into the chuck slot marked number 1. Push the chuck jaw until the thread locates on the scroll.
4) Rotate the chuck key clockwise about half a turn while holding the jaw gently in position.
5) To check that the scroll is properly located, try to pull the jaw outwards.
6) The process will need to be repeated if the jaw can be pulled free. Jaws, numbers 2 and 3 should be engaged in order, using the same process.

Locating a Cylindrical Work Piece in a manual 3 Jaw Chuck:

1) Isolate the machine power supply. Open the chuck jaws wide enough to accommodate the workpiece.
2) To ensure the workpiece is firmly held, insert the maximum amount of workpiece into the chuck, while checking that this will not interfere with the machining required.
3) Holding the workpiece in place, tighten the chuck jaws around it. This will locate the workpiece centrally in the chuck and ensure it is held parallel to the jaws.
4) The chuck jaws should be firmly tightened. Finished diameters should be protected by inserting a shim between the workpiece and the jaws.

The 3 Jaw Chuck

Locating and Clamping Workpieces with Large Diameters in a manual 3 Jaw Chuck:

1) Isolate the machine power supply. Open the chuck jaws wide enough to accommodate the workpiece.
2) Ensure that the jaws do not stand proud of the outer edge of the chuck. If the jaws are beyond the edge of the chuck, step jaws should be used.
3) Locate the workpiece on the face of the chuck or the face of the jaw.
4) While holding the workpiece in place, tighten the chuck jaws around it. This will locate the workpiece centrally in the chuck and ensure it is held parallel to the jaws.
5) Tap the workpiece with a soft hammer to ensure it is properly positioned and securely tighten the chuck.

Clamping a Hexagonal Bar in a manual 3 Jaw Chuck:

1) Isolate the machine power supply. Align the flats with the jaw faces when placing the workpiece in the chuck.
2) As the jaws are tightened, the workpiece should be gently rocked and twisted to ensure it will locate correctly. This movement will decrease as the jaws tighten.
3) When movement has ceased, the jaws should be securely tightened.

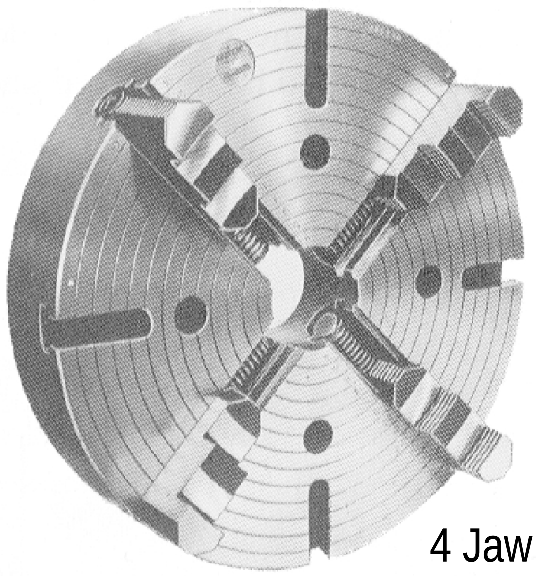
Clamping a Work Piece with an Internal Bore in a manual 3 Jaw Chuck:

1) Isolate the machine power supply. Adjust the jaws to accommodate the internal bore diameter of the workpiece.
2) Ensure the workpiece is properly located onto the face of the chuck or the face of the jaws.
3) To ensure the workpiece will be securely held, the maximum possible area of jaw should be in contact will the internal bore. Stepped jaws should be used to secure shallow bores.
4) Open the chuck jaws while holding the workpiece in position and allowing it to centralise.
5) Gently tap the workpiece to ensure it is properly seated and securely tighten the chuck.

The 4 Jaw Chuck

The 4 Jaw Chuck is used to hold square, rectangular and irregularly shaped billets. Each of the 4 jaws can be opened or closed independently, allowing parts to be held more securely, due to the greater pressures that can be exerted on each jaw when tightening. Although the independent 4 jaw chuck is quite versatile, it also takes longer periods of time to set-up.

The four jaw chuck has reversible jaws which can be independently adjusted. This allows both symmetrical and irregularly shaped billets to be held. Unlike the 3 jaw chuck, each of the 4 jaws must be set individually to secure the billet.



4 Jaw Chuck.

To reverse the jaws in a manual 4 jaw chuck:

1) Isolate the machine power supply.
2) Using the chuck key, wind out the jaws until they protrude beyond the outside edge of the chuck.
3) Continue to wind, while gently pulling the jaw until it slides clear from the chuck.
4) Reverse the jaw and replace it in the chuck slide, ensuring that it slides freely.
5) Push the chuck jaw in, ensuring the teeth can be felt.
6) Continue to push in lightly, rotating the chuck key until the jaw is positioned to hold the work piece.
7) Repeat the procedure for each jaw.

The 4 Jaw Chuck

Clamping a Symmetrical Workpiece in a manual 4 jaw chuck:

1) Isolate the machine power supply. Set each jaw to the same index line on the chuck face.
2) Each jaw should be equally adjusted until the approximate size of the workpiece is achieved.
3) To ensure the workpiece is firmly held, insert the maximum amount of workpiece into the chuck while ensuring that this will not interfere with the machining required.
4) Holding the workpiece in position, wind in both pairs of opposite jaws by equal amounts, until the workpiece is gently clamped.
5) Securely tighten the jaws. To prevent damage from jaws, a finished part of a workpiece needs to be protected.

Clamping an Irregularly Shaped Workpiece in a manual 4 jaw chuck:

The positioning and clamping of an irregularly shaped workpiece is a similar process although:

1) Depending on the shape of the workpiece, the jaws may not all be positioned on the same index line on the chuck face.
2) It may be necessary to reverse at least one jaw to securely hold a workpiece.

To Approximately set a Workpiece Centrally:

1) Isolate the machine power supply. Manually rotate the machine spindle.
2) Check by eye that the workpiece is set centrally on the chuck face. It may not be necessary to centralise the workpiece with more accuracy than this if the jaws are accurately set on the index rings and the workpiece diameter is significantly oversized.

To Centralise a Workpiece Accurately using a Scribing Block:

1) Isolate the machine power supply. The base of the scribing block should be placed on the flat bed of the lathe, with the scriber pointing towards the face of the workpiece.
2) Adjust the scriber point until it is level with the centre line of the workpiece.
3) Ensure scriber point will not foul chuck jaws.
4) Manually rotate the chuck, while sliding the scriber point so that it makes contact the highest outside point of the workpiece diameter.
5) Find the largest gap between the scriber point and the workpiece by rotating the chuck.
6) The nearest jaw should be adjusted by half distance of the gap between the workpiece and the scriber point. Securely tighten the opposite jaw.
7) Repeatedly check and adjust until the scriber point evenly touches the whole diameter of the workpiece.

The above procedures can also be used to set hexagons or similar shapes except that:

1) The point of the scriber should be moved to touch the highest corner of the workpiece.
2) The workpiece should be adjusted so that each corner comes into equal contact with the scriber point.

The 4 Jaw Chuck

Using a Scribing Block to Centrally Set an Irregularly Shaped Workpiece:

Irregularly shaped workpieces require marking out in order to obtain the necessary information to correctly set them on the chuck.

To set the workpiece centrally:

1) Isolate the machine power supply. Roughly set the workpiece centrally in the chuck.
2) The base of the scribing block should be placed on the flat bed of the lathe, with the scriber pointing towards the centreline face of the workpiece.
3) Manually rotate the chuck, while adjusting the scriber point so that it touches the circumference of the datum circle.
4) Rotate the chuck a further 180 degrees, taking note of the differences between the datum circle and the scriber point.
5) The jaw nearest the scriber point should be loosened and adjusted by half the distance between the marked line and the scriber.
6) The opposite jaw should then be tightened securely.
7) The process of checking and adjustment should be repeated until the scriber point is aligned with the datum circle line around the workpiece.

Balancing a Work Piece in the Chuck.

Turning irregularly shaped workpieces or eccentric diameters means that the weight of the workpiece in the chuck is not centralised. In such cases, it is necessary to equalise the weight on the chuck to ensure balance.

To balance the chuck:

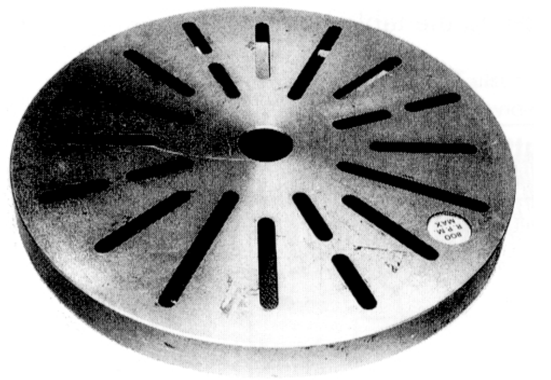
1) Isolate the machine power supply. Ensure the chuck can rotate freely.
2) Assess the weight required to balance the off-centre weight of the workpiece.
3) Firmly attach a balancing weight to the chuck.
4) Check the balance by manually rotating the chuck 90 degrees. Once it is steady, release the chuck.
5) If the chuck remains stationary, balance has been achieved.
6) If the workpiece rotates to the bottom, the amount of counter balance weight should be increased.
7) If the counter balance weight rotates to the bottom, it should be decreased.

When the chuck rotates slowly, only a small adjustment will be required to balance the workpiece on the chuck.

1) If the workpiece rotates very slowly to the bottom, the weight should be moved slightly out.
2) If the weight rotates very slowly to the bottom, the weight should be moved slightly in.
3) When the chuck remains stationary in any position, balance has been

Specialised Fixing Methods

Faceplates.



Faceplate.

Particularly awkward shapes can be held using a face plate. The workpiece is bolted firmly to the face plate, which is then attached to the spindle. Counterweights must be added since the centre of gravity of the billet will be offset compared to the spindle centre line. If the face plate is run out of balance, damage could occur to the headstock and headstock spindle bearings.

Countercentrifugal Chucks.

The countercentifugal chuck is one way that manufacturers have met the need to better the grip of the workpiece at high speeds. The countercentrifugal chuck reduces the centrifugal force developed by high rpm values. Counterweights pivot so that the centrifugal force tends to increase the gripping pressure, offsetting the outward forces developed by centrifugal force of the chuck jaws.

One of the disadvantages of these chucks is the tendency to increase the gripping pressure as the chuck slows down, which can damage the workpiece. An alternative method is to use elements of the chuck to lock the chuck jaws mechanically in their original position.

Countercentrifugal chucks can be obtained in a variety of sizes, from 200 - 450mm diameters, operating at spindle speeds of 5500rpm for a 200mm diameter chuck and 3500rpm for a 300mm diameter chuck. The repeatability of these chucks is 0.02mm.

Collet Chucks.

The collet chuck is ideal for holding square, hexagonal and round bar stock. The collet assembly consists of a drawtube, a hollow cylinder with master collets, and collet pads. Master collets are available with three or four gripping fingers and are referred to as either three-split or four-split designs. The four-split design has better gripping power but is less accurate. Collet chucks are front actuated and the collet pads are sized for the diameter being machined.

Tailstock

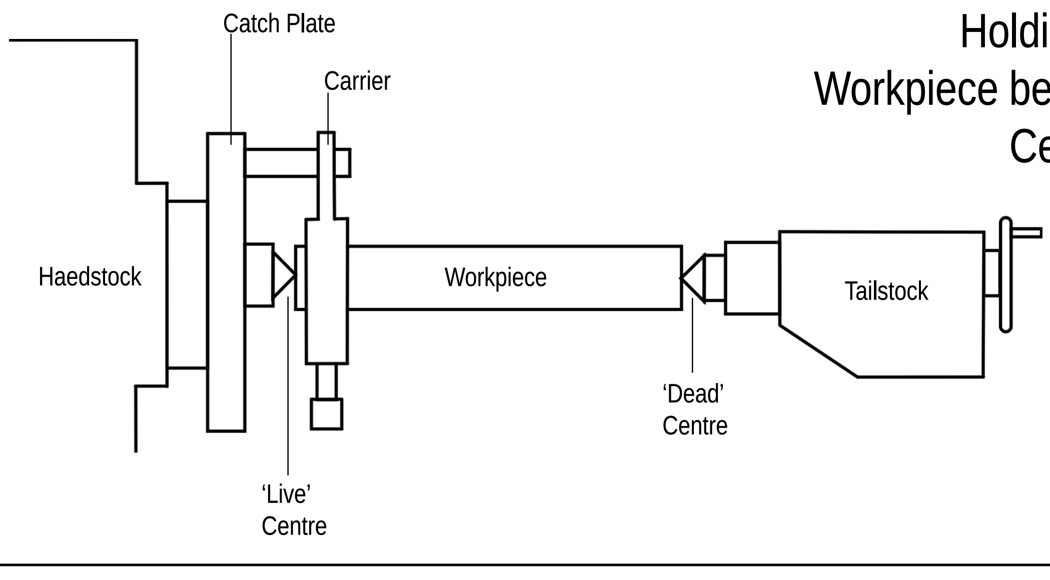
The tailstock is located at the opposite end of the lathe to the chuck and headstock, the nose of the tailstock running along the chuck and spindle centreline. It is primarily used to clamp centres, helping to support long workpieces. They can also be used to hold tools for operations such as boring and drilling, though most proprietary toolchangers on CNC lathes are more readily used for such operations.

Holding the Workpiece between Centres.

Long workpieces must be held between centres. This avoids the end of workpiece nearest to the tailstock from flexing, when the tool pushes against the side of the workpiece to cut away material.

The centres are held in the headstock spindle and at the opposite end of the lathe, in the tailstock. Before using centres, all the workpieces must be prepared by facing and drilling both ends to receive the centres. The headstock spindle uses a reducing bush to locate the 'live' centre (which can freely rotate on its bearings) or 'dead' centre (which must be lubricated since it does not move). Then the correct size of carrier is clamped to the workpiece. The carrier and workpiece are secured against the catch plate, which is attached to the spindle. A dead centre in the tailstock receives the opposite end of the workpiece.

Repeatability is available, when using centres, because workpieces can be replaced in exactly the same positions.



Holding the Workpiece between Centres.

Safety Issues

Chuck Clamping Forces.

Two devices can be used to measure the clamping force that is put on the workpiece. One measures the static gripping force, whilst the other measures the dynamic gripping force. The static gripping force is the force per jaw exerted by the chuck on the workpiece when the spindle is stopped. The dynamic gripping force is the force per jaw exerted by the chuck on the workpiece when the spindle is running.

Each individual chuck will have a specific clamping pressure.

When the chuck jaw clamping pressure is set, the pressure must not exceed the maximum pressure stamped on the chuck or warning plate. If a greater pressure is used, high stress forces are created in the chuck, which can lead to damage to the chuck, workpiece and machine. Front actuated chucks have a typical operational pressure of 200 - 500psi. Operating a chuck below 200psi will cause insufficient clamping force on the workpiece.

Always use maximum chuck clamping pressure unless the applied pressure will damage the workpiece.

Centrifugal Force and Speed Limitations.

Centrifugal force imposes speed limitations on all types of chucks. Centrifugal force will always increase as the speed of rotation of the chuck increases. This tends to throw the chuck jaws outward, decreasing the amount of clamping force on the workpiece. No chuck is entirely immune from this, therefore, all chucks have a maximum rotation speed. This speed must never be exceeded under any circumstance.

In addition to increasing with speed, centrifugal force increases as the jaws are moved outward from the centre line of rotation, since the jaws are 'made' heavier. Do not mount top jaws so that they extend beyond the diameter of the chuck and reduce the spindle speed when using special top jaw tooling configurations.

Automatic Chuck Systems

Changing the Chuck Jaws.

When it is necessary to frequently change the chuck jaws, to accommodate different sized workpieces, or when additional machining operations require a different method of holding the workpiece, a quick jaw-changing system can be used. Systems such as these can reduce the time taken to change jaws from around 30 minutes to 1-2 minutes, or less. Although initially expensive, the system can pay for itself in a very short space of time. There is less machine down time, so the productivity of the machine can improve quite dramatically.

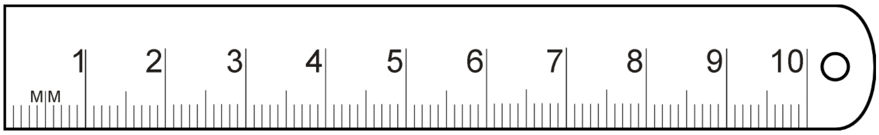
CNC lathes can also be equipped with fully automatic jaw, or chuck, changing systems. Some automated systems change on jaw at a time, whilst other systems may be capable of changing all jaws at once.

Steadyrest / Follower Rest.

When long, thin workpieces or shafts are machined, chatter vibration can often occur. Left unchecked, this chatter can damage both the cutting tool and surface finish of the part. Chatter can be reduced using a steadyrest, or follower rest.

The steadyrest is mounted to the CNC lathe bed and can be programmed to open or close automatically, providing feed through capabilities for the workpiece. The steadyrest uses constant hydraulic pressure applied to the support rollers, enabling it to adjust to the correct workpiece diameter. In addition to reducing chatter, the steadyrest also allows the machine to operate at higher and more efficient speeds and feeds.

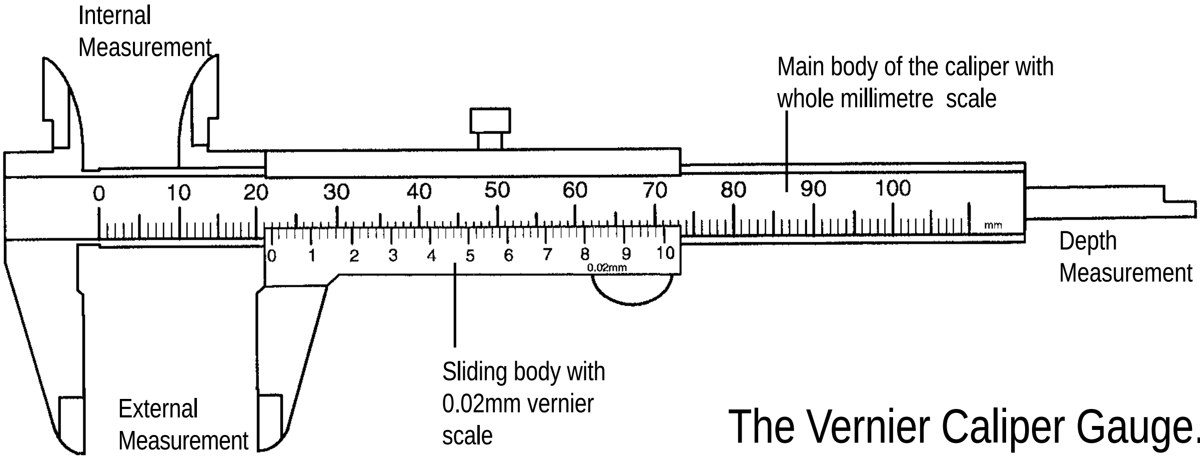
Measurement Tools



100mm/4” Straight Edged Rule.

Straight Edged Rules.

The straight edged rule is the most common and simplest all purpose workshop measuring tool. Straight edged rules are accurately marked with a metric (millimetres) scale on one face and an imperial (inches) scale on the opposite face. A good quality steel rule of 100mm/4” or 300mm/12” will serve as a straight edge for testing and basic marking out on small workpieces, when used with a scriber.



The Vernier Caliper Gauge.

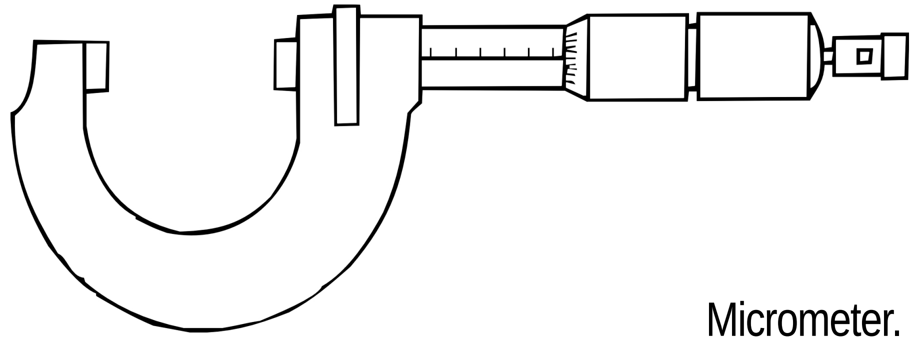
The Vernier Caliper Gauge.

Named after its inventor, the Vernier scale is incorporated into many types of measurement tool, the most common being the sliding caliper gauge, used for taking internal, external and depth measurements. Other measurement tools including the vernier scale include the depth gauge and the height gauge. Vernier calipers are made from fine alloy steels, accurately ground and finished to read up to 1/100th of a millimetre or 1/1000th of an inch.

Before using the vernier caliper gauge, it is important to check that the surfaces of both the gauge and the workpiece are clean and that all parts of the gauge can move freely.

A metric vernier has two main scales. The main body of the caliper houses the whole millimetres scale, whilst the smaller sliding body houses the 0.02mm vernier scale. Similar setups are available for imperial measurements. To read a metric vernier the whole millimetre value is determined from the highest value on its scale before aligning with the zero marker of the 0.02mm vernier scale. Where the lines on the 0.02mm vernier scale and the main body whole millimetre scale coincide determines the 1/100th mm value to be added.

Measurement Tools



Micrometer.

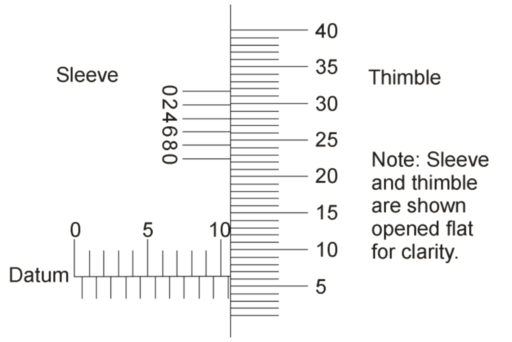
The Micrometer.

The micrometer is a complex precision made instrument, used for measuring the thickness of a workpiece when great accuracy is required. Micrometers are available with either metric or imperial graduations, allowing measurements to within 1000th of a millimetre or 1000th of an inch.

The micrometer must be zeroed before use, to ensure an accurate reading is obtained. The spindle is extended and the workpiece is positioned in the frame of the micrometer between the spindle and the anvil. The spindle is then closed onto the workpiece using the thimble. When the spindle nears the workpiece edge, the rachet stop is used until it 'clicks'. Readings are taken using the graduated segments on the thimble assembly.

Care must be taken not to tighten the spindle onto the workpiece, since the delicate internal threads of the micrometer may become damaged. Reading can also become distorted through holding the micrometer too long, since warmth from your hands can transfer and expand the metal assemblies. Always replace the micrometer to a safe clean area when it is not in use, preferably back in its box.

Example reading:
Above the datum, 10 whole millimetres are visible = 10.000 mm
Below the datum, 1 half millimetre is visible = 0.500 mm
Highest line on the thimble below the datum is 6 = 0.060 mm
Vernier line coinciding with the line on thimble is 4 = 0.004 mm
Reading of measurement is 10.564mm
When no lines coincide with the vernier, the intermediate thousandths can be estimated, ie, if the reading lies between 4 and 6 then the additional thousandths reading would be 0.005mm.



Example Micrometer Reading.

Feedback

A conventional, hand operated lathe. relies heavily on the experience of the operator.

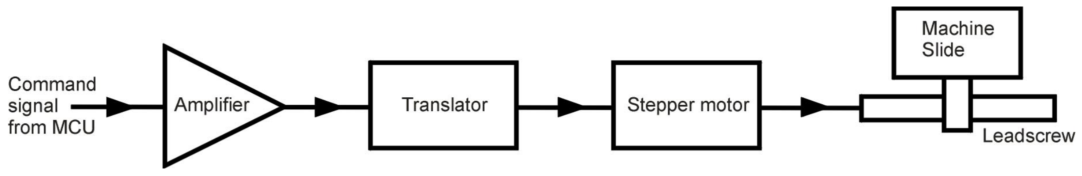
It is their responsibility to check that everything on the lathe is in the right position. The operator provides all this important information, called "*feedback*". If problems arise, the operator must use this information and act to adjust the machine.

Fully automatic CNC lathes can work very accurately, since they use a system which checks that the 2 different slides (the X and Z axes) of the machine are in the right place when an information signal is sent out asking them to move. The system detects if the slides are in the wrong position, at any given time, and the computer automatically corrects any mistakes. The process of sending information on the position of the slides back to the computer is called "*feedback*".

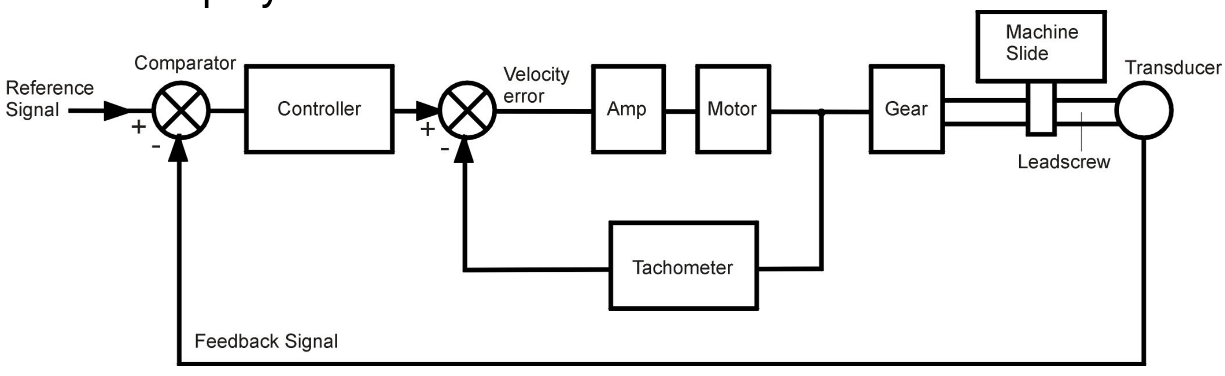
On a CNC lathe, the feedback information is provided by sensors placed in different areas of the machine. These sensors are sometimes called *"encoders"* or "*transducers*".

Any system which uses feedback is called a "*Closed loop*" system, whilst those which do not are called "*Open loop*" systems.

Open Loop System.



Closed Loop System.



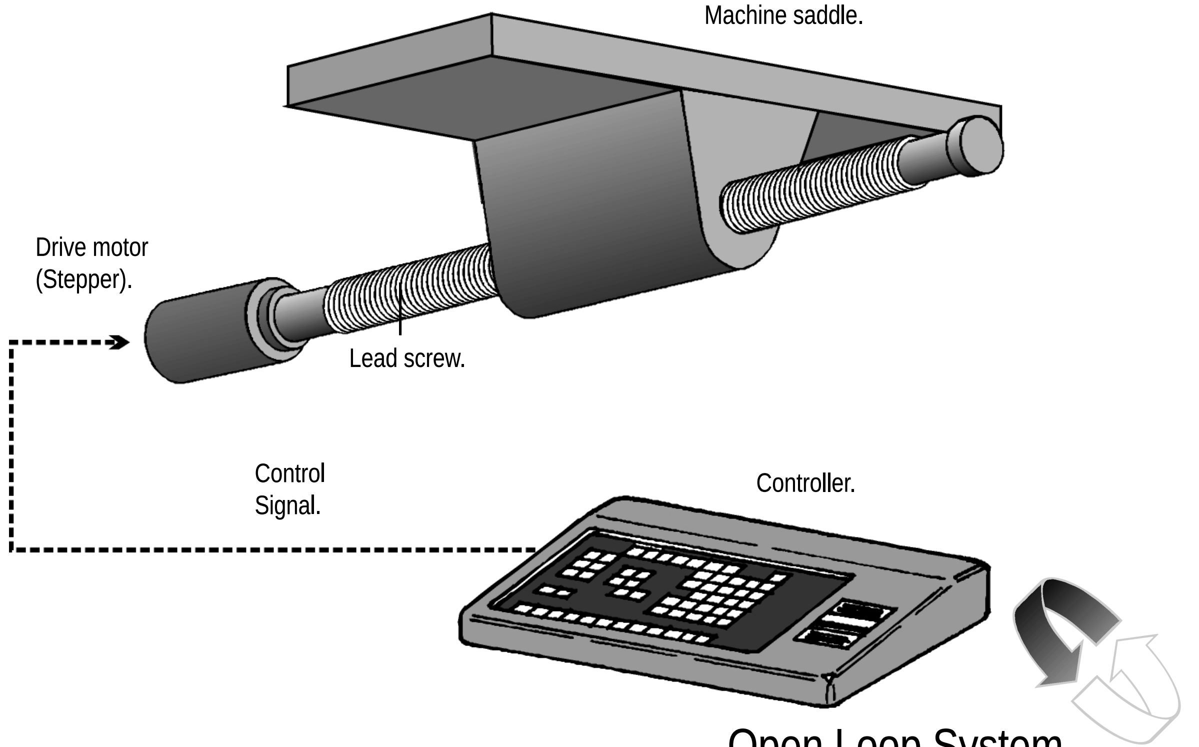
Open Loop Systems

Machine control systems which do not have the facility to provide "*feedback*" are called "*open-loop*" systems. The action of the controller has no information about the effect of the signals it produces. This implies that the neither the movement nor the velocity of the slide is being measured.

Open loop systems are of digital type, commonly use "*Stepper*" motors to drive the slides.

In an open-loop system, the controller sends out information instructing a slide to move a certain distance. This information is called the "*control signal*". This signal switches on the stepping motor, whose output shaft rotates through a fixed angle in response to an input pulse. The movement and velocity are controlled by the number of pulses and the pulse frequency respectively. The system is totally dependant upon the quality of the machine components, since no feedback is available regarding the accuracy of the system, ie, whether the slide actually moves the exact distance required. The diagram on page 31 shows an open loop control for a single axis of motion.

Open loop systems are satisfactory in applications where tight tolerances are not required. They are generally restricted to smaller CNC machines due to the limited power output availability of stepping motors - the pulses per second will restrict the speed of the drive.



Open Loop System.

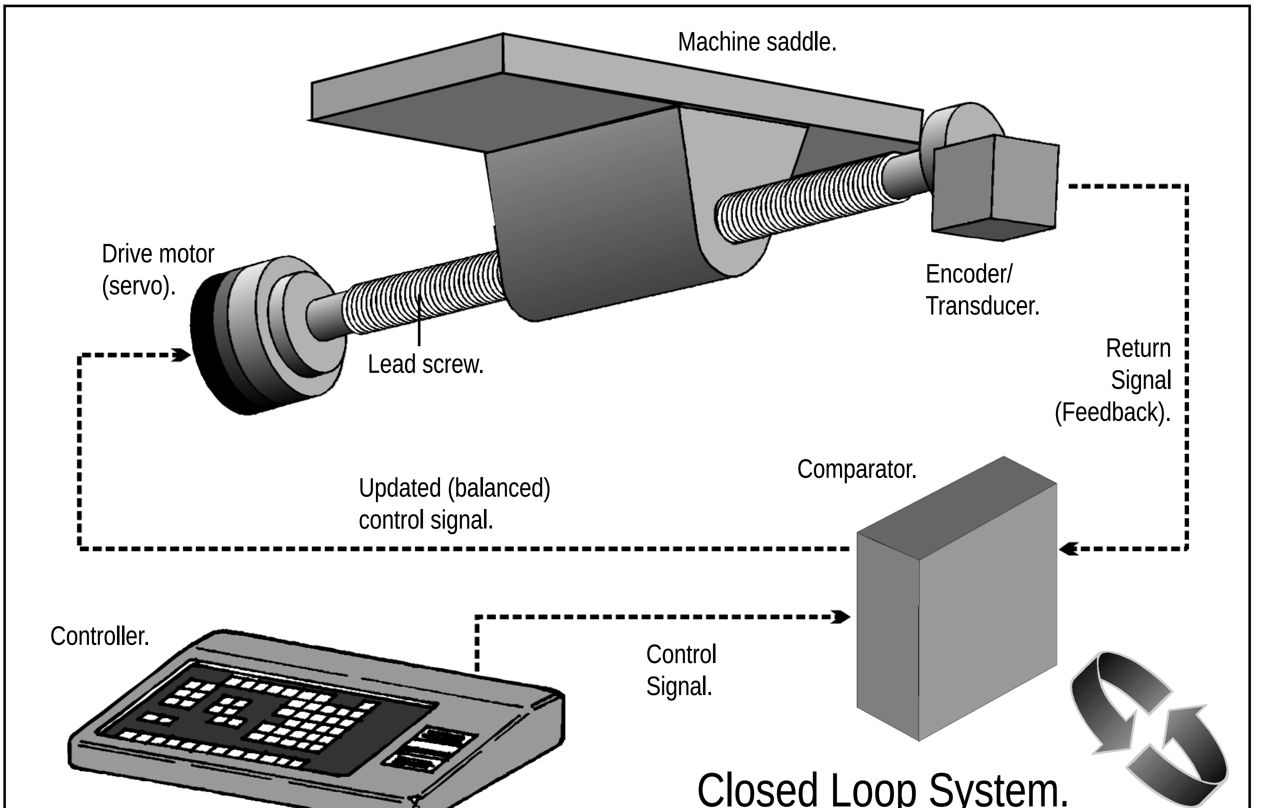
Closed Loop Systems

Machine control systems which do have the facility to provide "*feedback*" are called "*closed loop*" systems. Closed loop systems commonly use conventional variable-speed DC motors, called "*Servos*" to drive the slides. In order to keep track of the slide position, a servo must be fitted with a sensing device. The system measures the actual position and velocity of the slide and compares them with the desired values. Such a control can have capabilities of up to 0.0001mm resolution and speeds up to 10 m/min. The diagram on page 31 shows a closed loop control for a single axis of motion.

In a closed loop system, information is sent out to the machine instructing it to perform an operation. For example, move the Z axis from Z=0 to Z=50. This is called the "*control signal*". A sensor, called the "*encoder*", or "*transducer*", is used to count the number of turns of the servo motor. Therefore it always knows its position in relation to where it originally started.

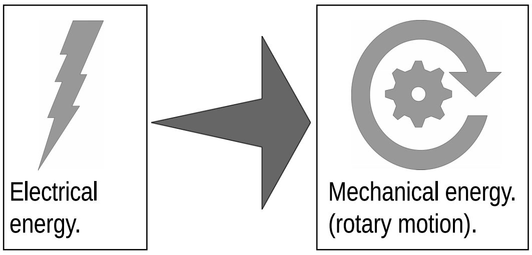
At the same time, information is fed back to the control unit. This information is called the "*return signal*" and the control unit is called the "*comparator*". The comparator compares the control signal with the return signal and balances any differences between the two. This difference between the two signals is known as "*following error*". The system is designed to eliminate, or reduce this error to a minimum.

For example, the feedback 1/4 of a second into the operation might indicate that the Z axis had not moved as far as the computer had expected it to. The next, or "*updated control signal*" sent out would instruct the stepper motor to spin slightly faster, in order to "catch-up" this difference. This process of comparing and balancing signals is carried out up to 500 times per second, constantly throughout the operation. By increasing the magnitude of the feedback signal (more pulses per revolution of the leadscrew) the loop will be made more sensitive.



Closed Loop System.

Stepper Motors



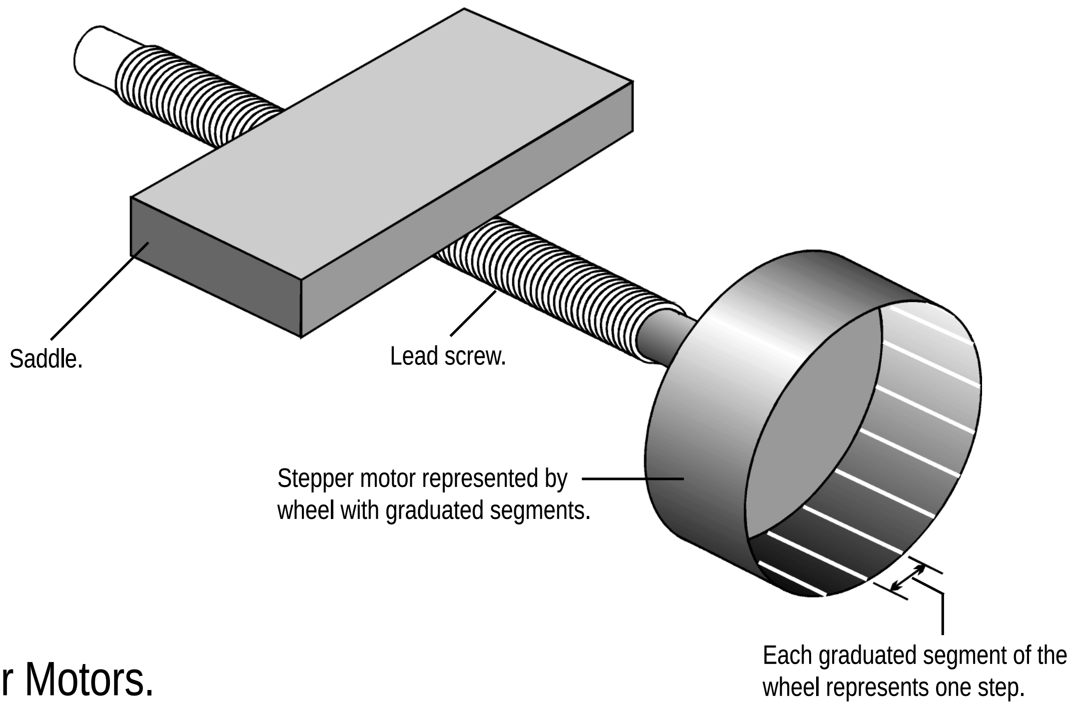
Electric motors are devices used to convert electrical energy to mechanical energy (rotary movement).

The stepper type of drive motor was fitted to earlier types of CNC machine and CNC machines using open loop control systems. The centre of the motor, called the rotor, is fastened to the leadscrew of the machine tool. When energised (on receipt of small electrical pulses), the motor rotates through small angular divisions, called steps, hence the name.

The distance moved depends upon the number of pulses received and the pitch of the leadscrew. A typical example is a 7° rotation per pulse.

Since each of these steps are the same, the leadscrew can be positioned quite accurately by simply counting the number of pulses required for each step.

Try to imagine the stepper motor is a wheel with graduated segments, one segment equating to one step. Every time the wheel moves one step forward, the saddle will move a certain distance, say 1/4 of a millimetre. If we wanted the saddle to move 4 millimetres, the wheel must move 16 steps. This is the basic principle behind the control offered by stepper motors.



Stepper Motors.

Stepper Motors

FIGURE 1.

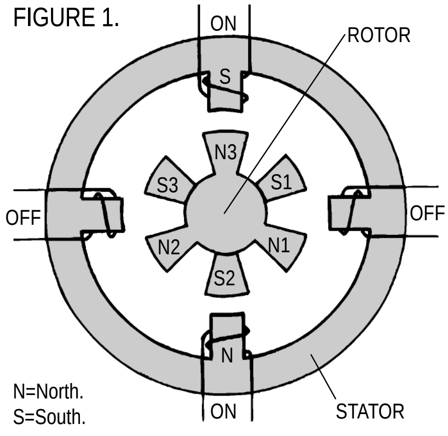


FIGURE 2.

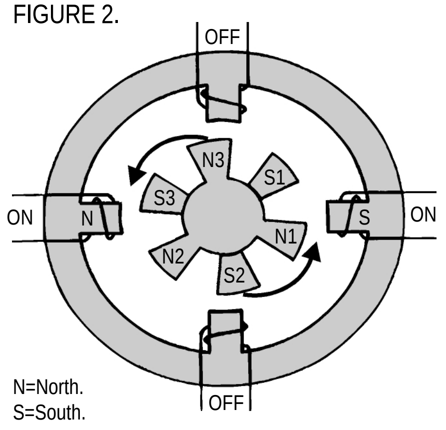
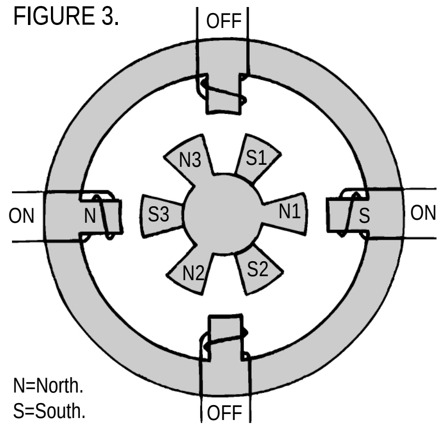


FIGURE 3.



A conventional d.c. motor is switched on by applying a controlling voltage signal across it and will continue to rotate until switched off.

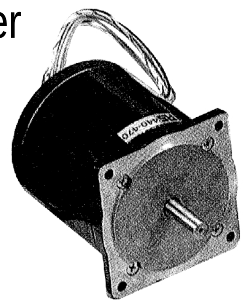
The stepper motor is different because it will move in a series of small steps. In order to do this, a stepper motor needs to be continually switched on and off, for each step required. Therefore, the control voltage signal for a stepper motor is a series of electrical pulses.

The stepper motor uses two basic parts in its construction, the *"rotor"* and the *"stator"*.

The rotor is made from a number of permanent magnets with fixed north and south poles. This is the part of the motor which rotates.

The stator is made from a number of electromagnets, which can be independantly switched on and off. This is the part of the motor which remains still.

Stepper Motor.



How does a stepper motor turn one step?

Note - the rotor and stator shown in the diagrams have been simplified to make the drawing clearer.

Figure 1 - The vertical electromagnet in the stator is switched on and holds the rotor firmly in position.

Figure 2 - The vertical electromagnet is switched off whilst the horizontal electromagnet is switched on. The "new" north and south poles created by this horizontal electro-magnet start to pull the rotor round (through magnetic attraction - N1 is attracted to S, whilst S3 is attracted to N).

Figure 3 - The horizontal electromagnet in the stator is now holds the rotor firmly in its "new" position.

Servo Motors

The servo type of drive motor incorporates power amplification and feedback, in order to balance the output signal with the input signal and hence follow a command signal accurately. Servo motors offer constant torque (twisting force) across most of the speed ranges, so the size of the motor remains relatively small.

The drives motors are usually connected directly to the leadscrew of the machine, so a stiff drive is always provided. In CNC machines where indirect systems are used, such as a pulley drive belt, the high accuracy and quality of the feedback can become compromised.

Servo Motors.

Positional Tranducers

The Role of Positional Tranducers.

A transducer is simply a device that converts one form of energy to another.

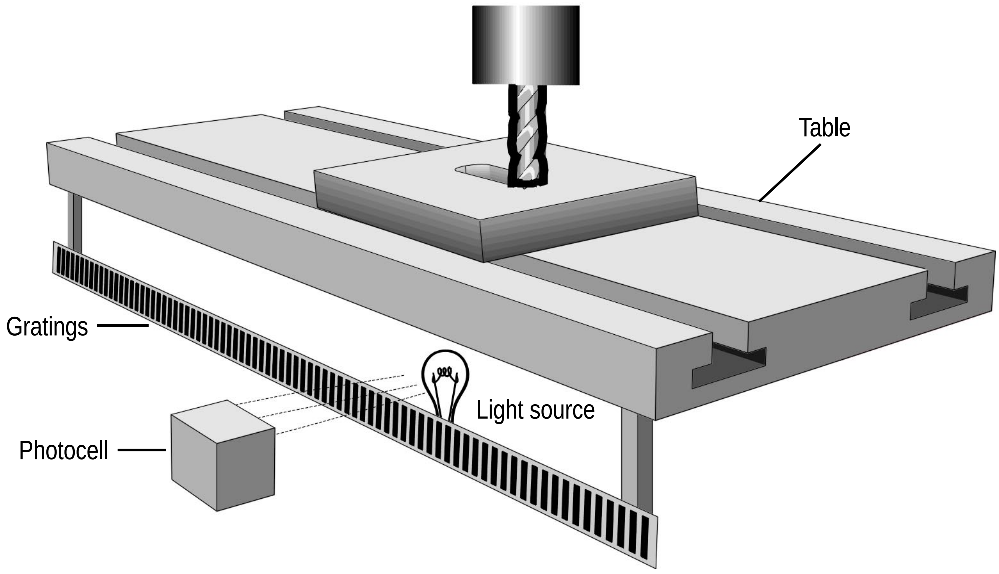
There are basically two types of position-measuring transducer that are used on CNC machine tools:

- Linear transducers.
- Angular transducers.

Linear Tansducers.

Linear transducers, more commonly used in CNC milling machines, work by recording the movement of the machine table. A linear scale is engraved onto a moving work table. The scale is a series of parallel lines. A beam of light is shone over the gratings so that as the table moves the beam of light is interrupted giving a pulsing effect. This pulse is ‘picked up’ by a photocell and information from it is sent back to the controller. By knowing the pitch of the engraved lines and counting the pulses it is possible to establish the distance the machine table has moved.

Linear tranducers.

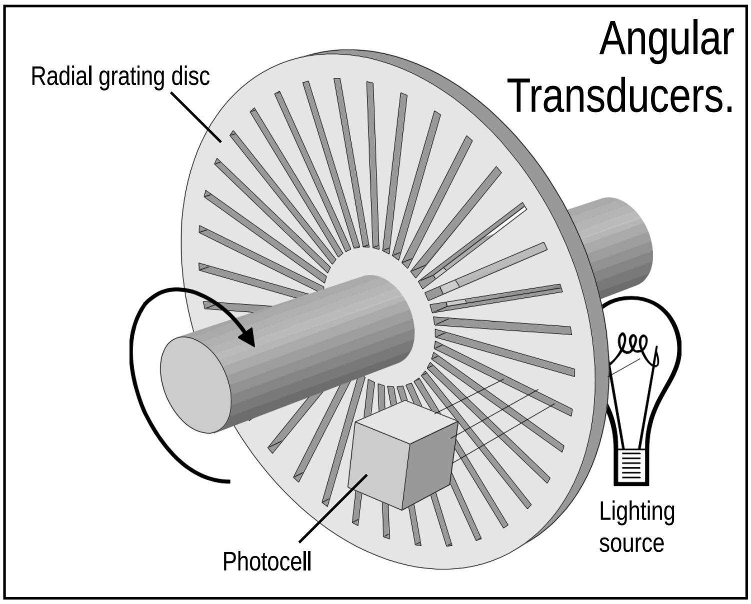


Positional Tranducers

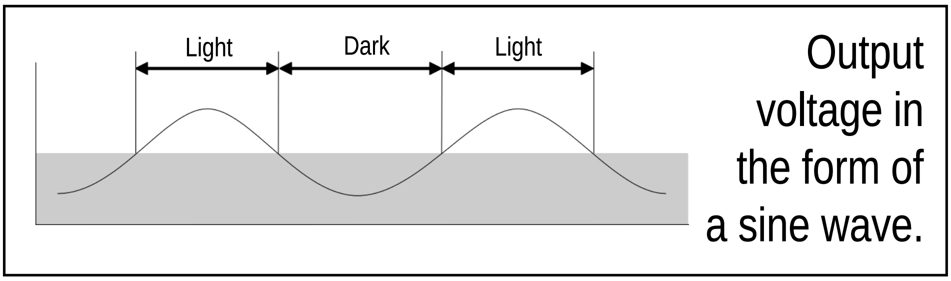
Angular Transducers.

Angular transducers measure angular rotation of the axis leadscrew. If we know the pitch of the leadscrew the movement of the table can be easily found.

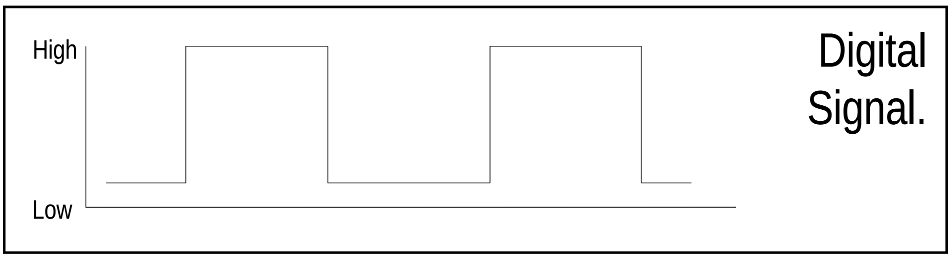
A radial grating disc is fastened onto the machine spindle. This disc has alternate transparent opaque areas. These areas pass in front of a beam of light as the leadscrew rotates, the intensity of the light onto the photocell varies.



As the photocell gives out a voltage that varies according to the intensity of the light beam the output is in the form of a sine wave, shown below....



The voltage at this point is said to be in analogue form. A Schmitt Trigger converts this analogue signal to a Digital signal which has a square form, shown below... The output signal can now be recognised as a series of pulses which represent movement of the leadscrew. By knowing the pitch of the leadscrew and the number of transparent areas on the engraved disc, the exact distance the slide moves can be determined by calculating the number of pulses.



Cutting Tools

Classification of Single Point Tools.

Lathe cutting tools need to be both hard and tough, due to the extreme forces that are exerted on their tips. There are four basic categories of single point cutting tool:

1) *Ground Tools* - The cutting edge is formed by grinding the end of a piece of steel stock.
2) *Forged Tools* - The cutting edge is formed by rough forging, before hardening and grinding the cutting edge.
3) *Brazed tip tools* - The cutting edge is formed by brazing a small tip of high grade material to a toolholder shank of lower grade material.
4) *Mechanically held tip tools* - The cutting edge is formed by mechanically holding a small disposable high grade material tip in a toolholder shank. Since only the very end of the tools are used to cut away material, interchangeable cutting tips are much more flexible, since each single worn tip can be simply replaced by a new one. Tips can also be ground for special operations and quickly changed when they are required. Both high speed steel and carbide bits are available for purpose made holders.

Tool Selection.

The selection of the cutting tool is probably one of the most important choices you will need to make to ensure successful operations. An incorrect choice of cutting tool could have many consequences relating to the following areas.

1) *Load* - If a cutting tool is not able to adequately sustain the load under which it is placed, it is liable to flex, causing chattering or breakage. Both of these would ruin a work piece.
2) *Profile Depth* - If the cutting tool selected is not of the correct geometry then it will be unable to cut the billet with enough accuracy to achieve the required tolerance.
3) *Removal Rate* - If the cutting tool selected is not able to achieve the required removal rate then the initial roughing process will be slow and laborious.

Directional Type Classification of Tools

Single point tools can also be classified in one of the following directional categories, based on the type of cutting performed:

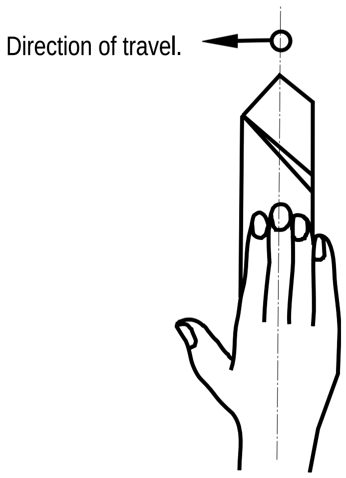
1) *Right hand type.*
2) *Left hand type.*
3) *Neutral type.*

This directional type classification is determined by laying your hand on the tool, palm facing downward, so that while your fingers are pointing towards the tool point, your thumb points to the side of the tool with the main cutting edge.

Right Hand Type Tool.

A right hand type tool moves from the right to the left, along the lathe bed, ie, from the tailstock end to the headstock end.

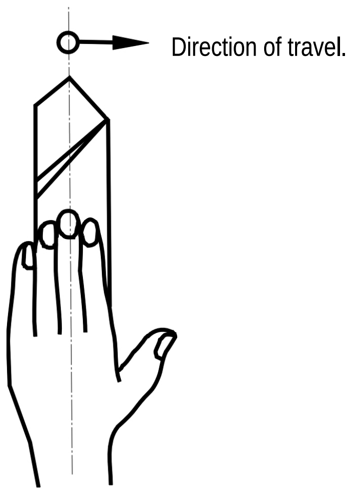
Right hand tools have their cutting edge on the left when viewed from the top with their nose pointing away from the operator.



Left Hand Type Tool.

A left hand type tool moves from the left to the right, along the lathe bed, ie, from the headstock end to the tailstock end.

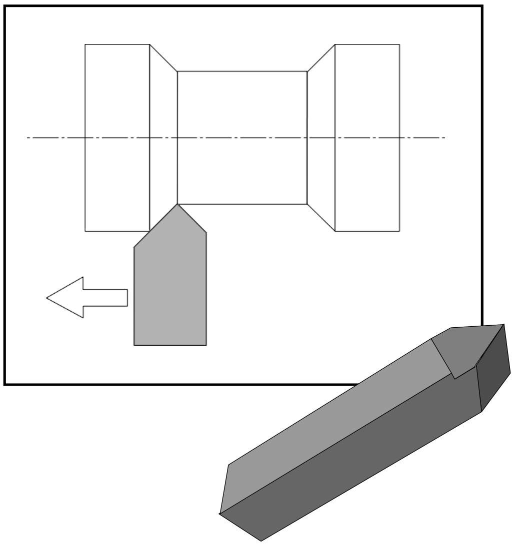
Left hand tools have their cutting edge on the right when viewed from the top, with their nose pointing away from the operator.



Neutral Type Tool.

A neutral type tool can move in either direction, from right to left, or from left to right. For this reason, they are constructed with zero back rake and side rake angles. In special cases, a small back rake angle is used. A neutral type tool is used for finish turning operations, with an approach angle of 45 degrees.

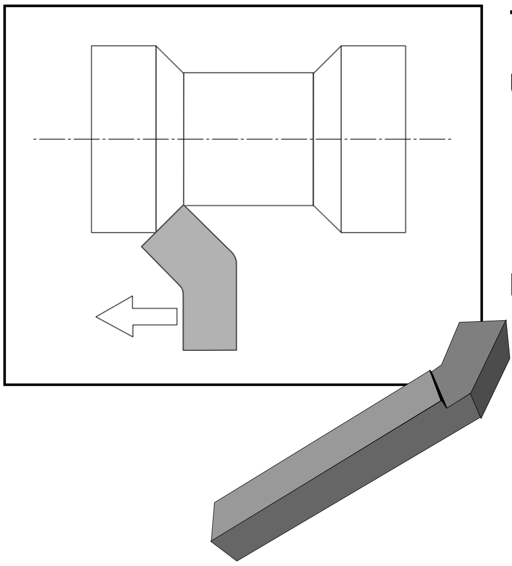
Basic Tool Shapes and Uses



The Heavy Duty Roughing Tool.

Used for: General Shaping - removing large amounts of material quickly and efficiently. Its thick shank and extremely large wedge angle make it very rigid allowing deep, fast cuts.

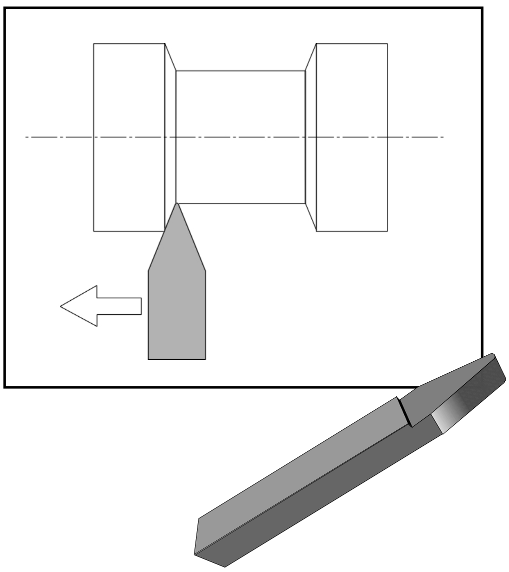
Notes: Safety implications - since cutting forces are likely to be high, great care must be taken to ensure all cutting angles are correct. Too large a cutting angle may result in the workpiece being damaged. Too small a cutting angle may result in the tool being damaged.



The 45 Degree Roughing Tool.

Used for: General Shaping - removing large amounts of material quickly and efficiently, though lighter cuts are taken when compared to the heavy duty roughing tool. It is more commonly used towards the end of the roughing process.

Notes: Safety implications - since cutting forces are likely to be high, great care must be taken to ensure all cutting angles are correct. Too large a cutting angle may result in the workpiece being damaged. Too small a cutting angle may result in the tool being damaged.

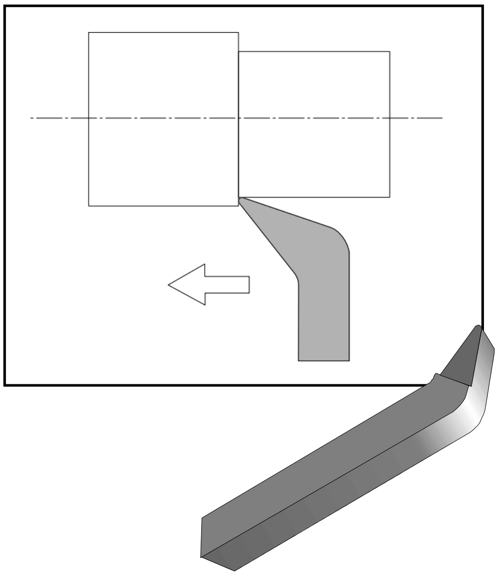


The Straight Round Nose Tool.

Used for: Finishing operations - removing minimal material to produce a high quality surface finish. The straight round nose tool can be used to either provide a bevelled edge or shoulder on a workpiece or as a finishing tool due to its round nose.

Notes: Quality implications - The smaller nose radius on this tool dictates the quality of finish, since this reduces the surface of the cutting area. It is important that the correct balance of angles is maintained, as they can effect the radius. A larger radius can result in chatter. A smaller radius can

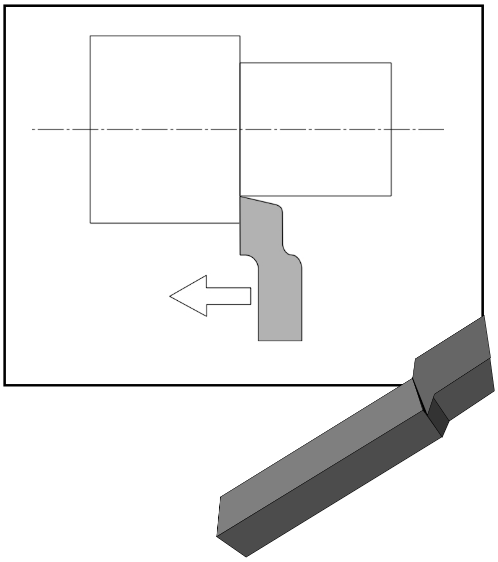
Basic Tool Shapes and Uses



The Cranked Round Nose Tool.

Used for: The cranked round nose tool is used to make sure that the billet is perfectly round by just travelling along the bottom edge of the billet and removing any material projecting from it. This guarantees a regular shape and should be used with all billets before any other machining takes place.

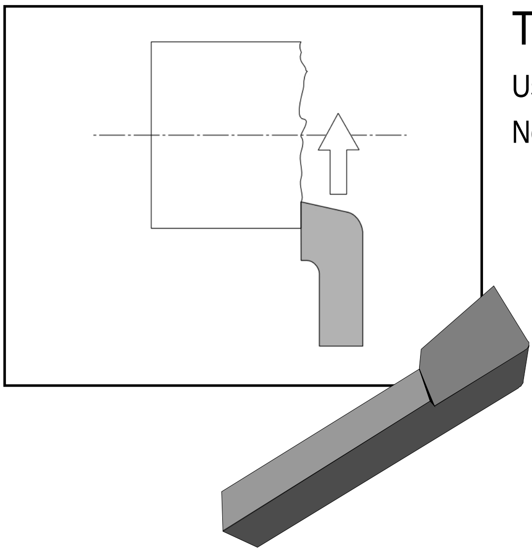
Notes: Safety implications - This tool is used when clearance angles are the prime consideration, for example, when machining close to the machine chuck. The cranked round nose tool can be used for most operations, so long as the correct cutting angles are ground and used.



The RH Knife Tool.

Used for: The knife tool is normally used for medium to light cuts, often to create a shoulder on a billet. The RH means that the tool is right handed and therefore that it cuts from the right. Left handed tools are also available.

Notes: Quality implications - Often used for finishing operations. When this is the case, the correct cutting angles must be maintained. Too small a cutting angle may result in rubbing to produce chatter and a poor surface finish.

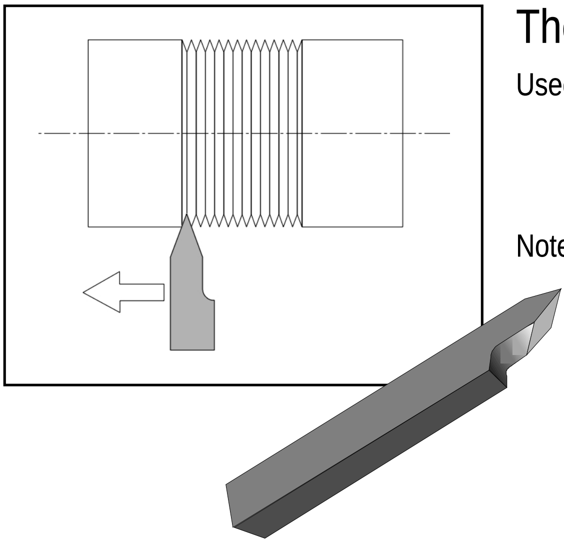


The Facing Tool.

Used for: Facing the workpiece or a turned shoulder.

Notes: Quality implications - When machining a shoulder, check that the tool nose radius is compliant with the tolerance required on the part. Correct cutting angles must be maintained to eliminate rubbing and chatter.

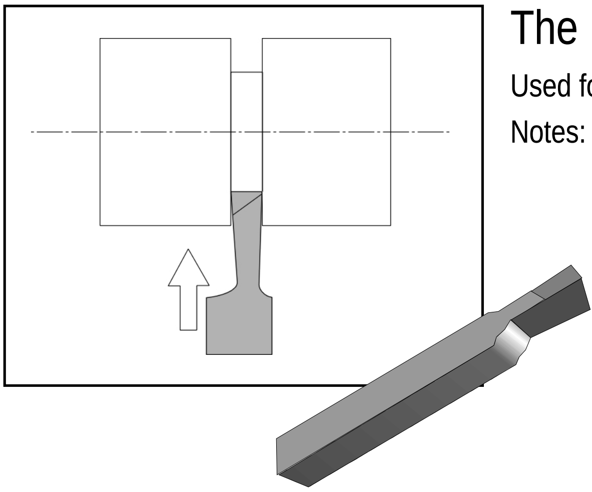
Basic Tool Shapes and Uses



The Threading Tool.

Used for: Producing outside threads on the workpiece. It must be used with a calculated spindle speed, feed rate and cutting depth as these values dictate the pitch of the thread.

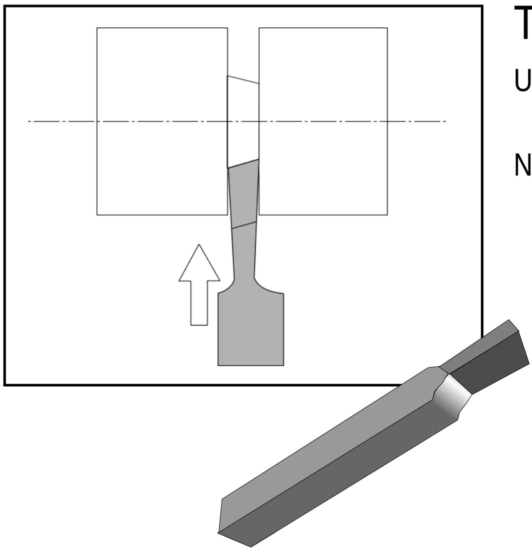
Notes: Quality implications - The cutting tool angles must be ground to specific values depending on the pitch of the thread to be cut. Correct cutting angles must be used to prevent chattering and shredding of thread flanks.



The Recessing Tool.

Used for: Producing recesses in the workpiece.

Notes: Quality implications - The cutting tool must be ground to specific width and length depending on the size of recessing required. Too large a clearance angle may result in chatter, a poor surface finish, possible breakage of the tool and damage to the workpiece. Too small a clearance angle may result in the tool overheating, damaging both the tool and the workpiece.



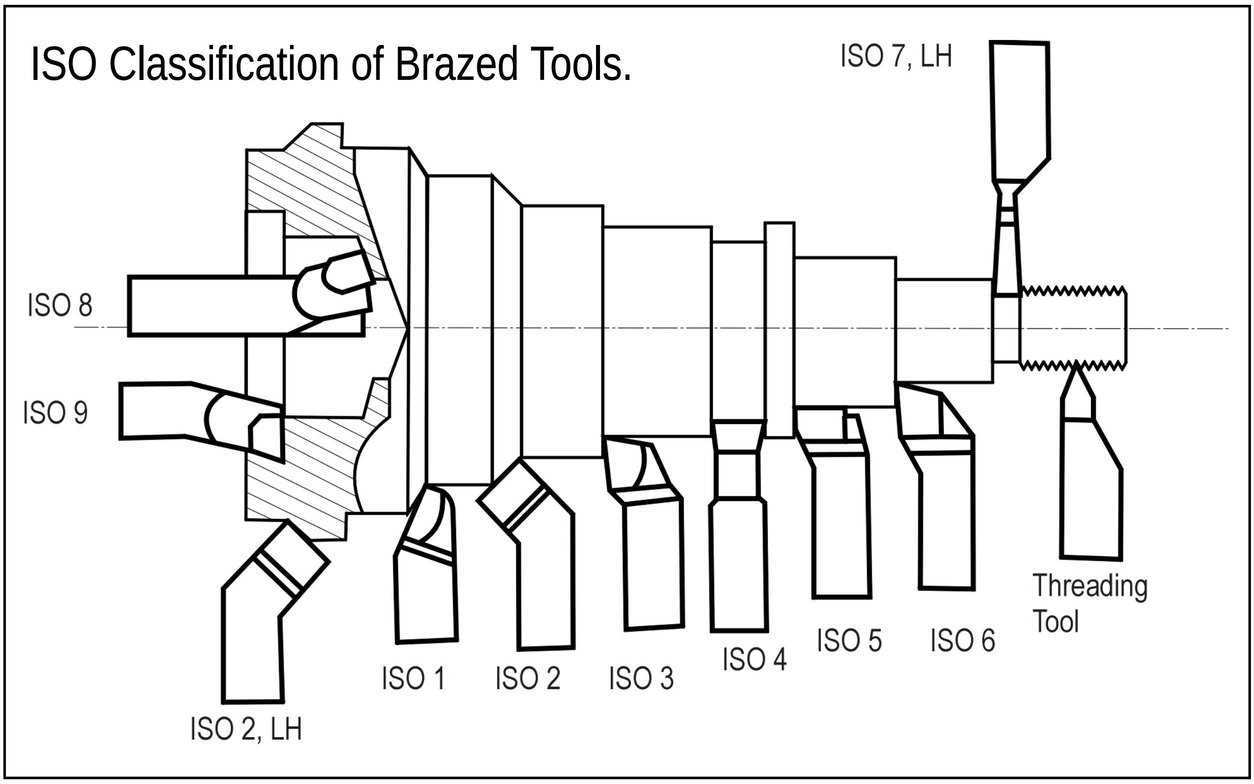
The Parting Tool.

Used for: Removing the finished part from the remainder of the workpiece.

Notes: Safety implications - The parting tool is shaped to allow the finished part to be cleanly cut from the remainder of the workpiece, without having to remove it from the lathe. Parting tools are normally ground so that a minimum amount of material is wasted in the operation. If the tool is undersized there is a greater chance of it breaking. Too large a clearance angle may result in the tool flexing and the part will not be cut away cleanly. Too small a clearance angle may result in the tool rubbing, overheating and both the tool and part being damaged.

Brazed Tip Tools

The International Standards Organisation has standardised nine types of brazed carbide tools, covering turning, shouldering, chamfering, grooving, parting and boring operations, as shown in the diagram below.



ISO Classification of Brazed Tools.

Advantages of brazed tip tools:

- They are suitable for general purpose work.
- They have a low unit cost.
- They require less space compared to mechanical toolholders.
- Special tools can be readily formed with non standard cutting edges.

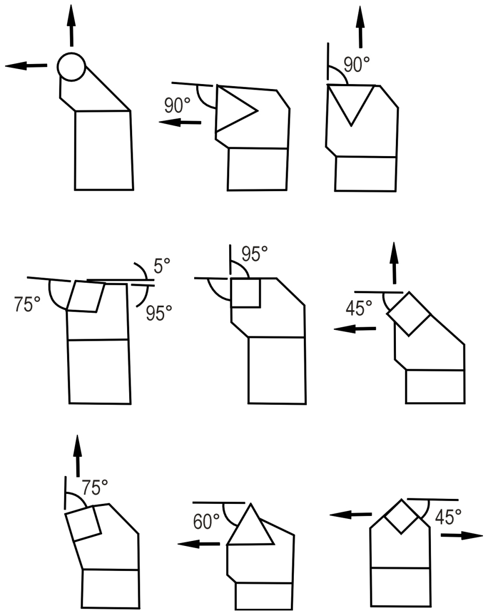
Disadvantages of brazed tip tools:

- They have high regrinding costs.
- Brazing of the carbide tip to the steel shank can cause residual stresses that cause breakage of the tip during resharpening or cutting procedures.
- Speeds and feedrates require careful monitoring to prolong tool tip life.

Mechanically Held Tip Tools

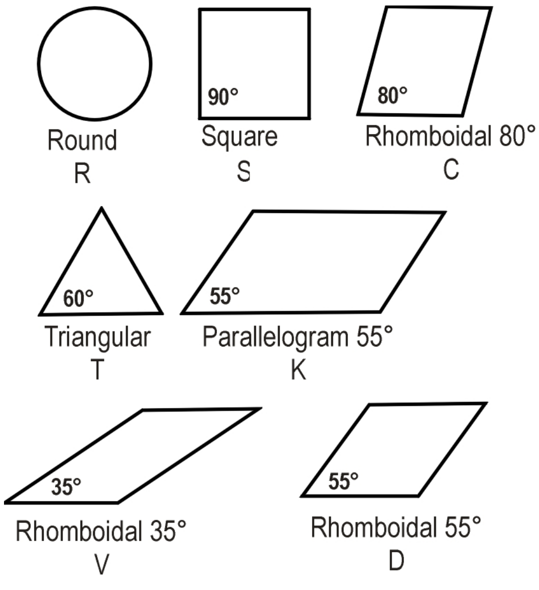
Tips are made from specially developed high grade materials, such as carbides, ceramics, cermets and diamond. The inserts are purchased in different sizes and shapes, ready for use, with cutting edges on a number of indexable planes. When all the cutting edges have been used, the tip is discarded and simply

Positioning the Tip to suit the Cutting Operation.



Inserts can be adjusted to provide various approach angles, according the type of toolholder used.

Standard Tip Shapes.



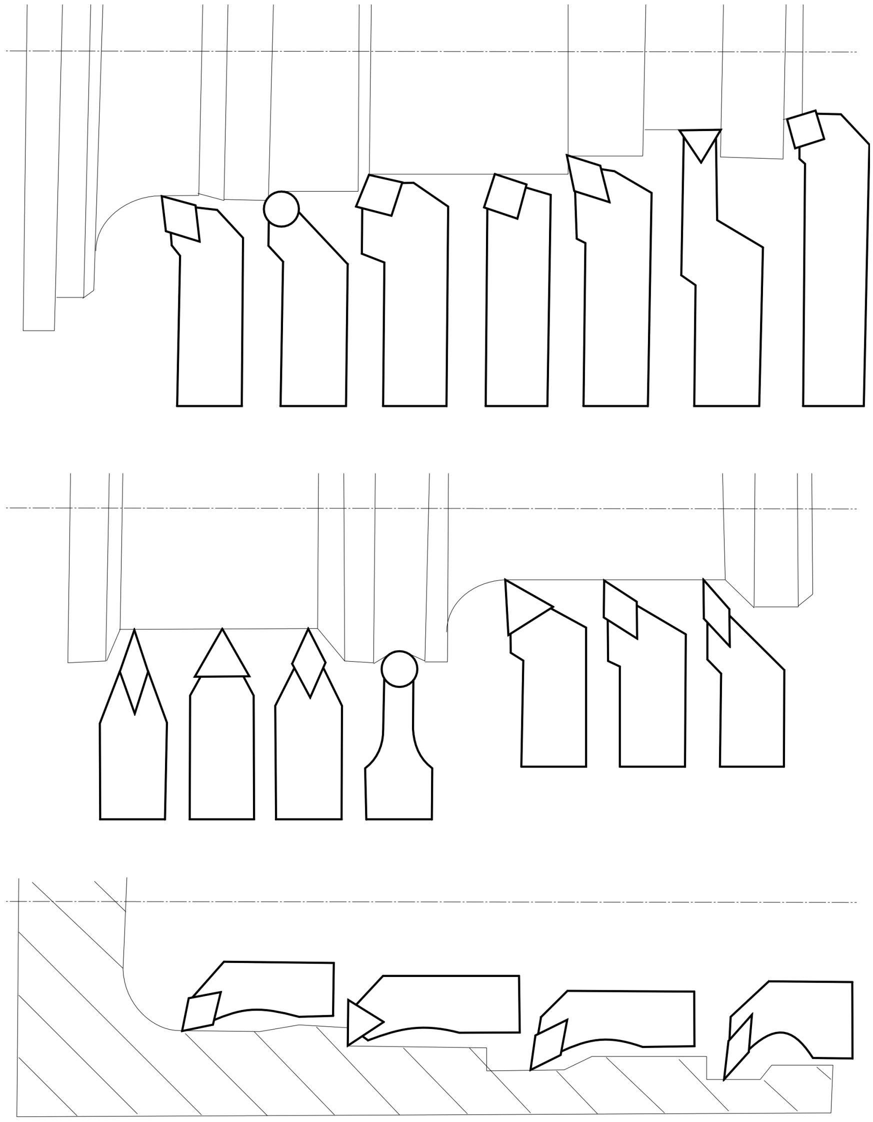
The inserts are available in a number of different shapes. The overall strength of the tool depends on the tool tip angles, round inserts being the strongest.

Mechanically Held Tip Tools

Advantages of mechanically held tip tools:

- No regrinding or resharpening is required.
- Since inserts can be indexed very accurately, tool tip and holder change times are greatly reduced.
- High speeds and feedrates are permissible.
- When using negative rake angles, both the top and bottom cutting edges can be used, doubling the number of cutting edges of the insert.
- Specially coated tips, such as titanium carbide can be readily used.

Use of Insert Shapes for
Various Turning Operations.



Mechanically Held Tip Tools

Selection of Indexable Insert.

Insert selection is mainly based upon the characteristics of the insert and the requirements of the application. Performance aspects and cost considerations should be taken into account in the selection process, together with the material cutting grade.

Selection criteria: Insert Shape.

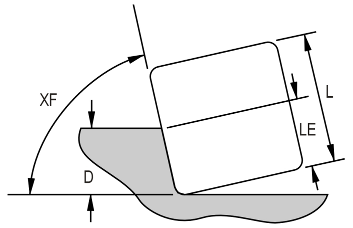
- In general, inserts with larger included angles should be chosen over those with smaller angles, in the following order: S (90°), C and W (80°), T (60°), D (55°), V (35°).
- Where applicable, the trigon shape W (80°) should be chosen over the C insert, since more cutting edges are available.
- Round inserts are an alternative to S inserts and may also be suitable for use in form turning.
- Negative rake inserts which are useable on both sides are more cost effective than one sided positive rake inserts.
- Positive rake inserts offer advantages in turning operations using thin walled parts and soft work materials.

Selection criteria: Cutting Edge Length.

The size of an indexable insert is governed by the maximum depth of cut, the approach angle, insert shape and geometry.

Approach angles vary between 75° - 105°. The effective cutting edge length is roughly equal to the maximum depth of cut.

Cutting Edge Length.



LE = D / sin XF

D = Depth of Cut.
LE = Effective Cutting Edge Length (width of cut).
L = Insert cutting edge length.
XF = Approach angle.

Selection criteria: Corner Radius.

The corner radius determines the strength of the cutting point, the maximum admissible feed and the suitable finish of the workpiece.

Generally, always try to select the largest possible corner radius.

The following maximum radii, depending on the feedrate, are recommended for roughing operations:

Radius r $\geq$ 1.6 x Feedrate f (for insert shapes C and S).

Radius r $\geq$ 2.5 x Feedrate f (for insert shapes D and T).

When finishing, good surfaces are achieved with:

- Higher cutting speeds.
- Inserts with sharp cutting edges.
- Positive rake angles and positive chip breaker geometries.
- Use of cermet tips.
- Rigid machining setups.
- Use of easily machinable work materials.
- Use of cutting fluids.

Tool Tip Materials

Stellites.

Stellite is a cobalt based alloy which contains little or no iron. The alloy, composed of 50% Cobalt, 33% Tungsten, 3% Carbon and 14% of various other materials can only be used for casting to shape or deposited as a hard facing. Stellite is naturally hard and requires no heat treatment. In fact, it is so hard that it can only be machined by grinding. Stellite is slightly softer than high speed steel but it keeps its hardness even when the cutting edge is glowing red hot. Due to the properties of Stellite, it is very expensive compared to high speed steel and as it is extremely strong and tough, would be suitable for use in standard tool holders at high values of positive rake angles.

Metallic Carbides.

Metallic carbides are harder and cheaper than Stellite, yet are capable of operating at the same temperature. Again the metal is so hard it can only be machined using Green grit silicon abrasive wheels. There are three categories of carbide:

1) *Tungsten Carbide* - This metal is used to machine materials like cast bronze and grey cast iron as it is very hard. As a result of the casting process Tungsten carbides have a hard and abrasive skin but a relative low tensile strength.
2) *Mixed Carbides* - These are mixtures of tungsten and titanium carbides. They are not as hard and abrasion resistant as straight tungsten carbide but can be much stronger and tougher and are mainly used to cut high strength materials.
3) *Coated carbides* - These are the most expensive type of carbide because it can run up to 30% faster than the recommended speed for tungsten and mixed carbides without any reduction in the life expectancy.

Ceramics.

Ceramics tips are harder but less brittle than those made from carbides. The most common form of ceramic is aluminium oxide which can be pure or mixed with other metallic oxides like chromic oxide. Ceramic tips can only be used if it is clamped on to an object as it can’t be brassed. Ceramic tips are very weak under tension and the edges chip easily. This metal is most commonly used for high speed, high quality finishing cuts.

Machines that use ceramic tooling must be powerful and very rigid to utilise the full power of this material’s special properties. Cutting speeds from 150-300 m/min are very common when using ceramic tooling, but vibration chatter will immediately make the tool unusable. The rake angle for ceramic tools is normally between -5 to -7 degrees and the chips are often seen flying off red hot due to the high power necessary when using ceramic tooling.

Taps and Dies - Manual Cutting of Threads

The Tap.

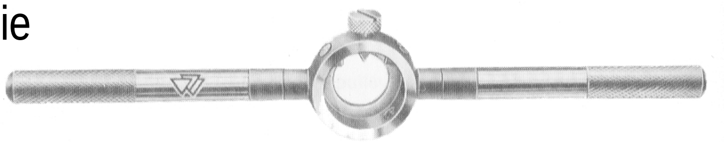
The tap is used for manually cutting internal threads. It is similar in shape to a screw, although it has narrow slits along its length to allow the surplus material to be ejected from the hole being tapped. Before tapping a work piece, ensure it is secured in a vice. The tap should then be turned inside the hole to be threaded, using a tap wrench to apply the necessary leverage. Alternatively, the work piece can be secured in a chuck and the tap turned by using the tailstock. The thread is cut by simultaneously turning the chuck anti-clockwise while turning the tailstock clockwise.

Note: For every two full turns of the chuck in an anti-clockwise direction, you must turn it half a turn clockwise to stop the tap from clogging.

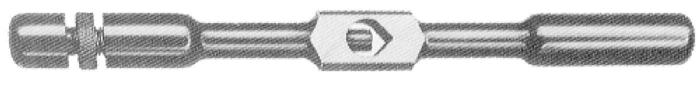
The Die.

The die is used for manually cutting external threads. It has internal cutting edges with narrow slits along its length to allow surplus material to be removed from the thread being cut. Before using the die, ensure the work piece is secured in a vice. The die should then be turned on the billet, using a spring-loaded centre or wrench to give the necessary leverage. Alternatively, the work piece can be secured in the chuck and the tailstock can be used to turn the die. The thread is cut by simultaneously turning the chuck anti-clockwise while turning the tailstock clockwise.

Note: For every two full turns of the chuck in an anti-clockwise direction, you must turn it half a turn clockwise to stop the die from clogging.

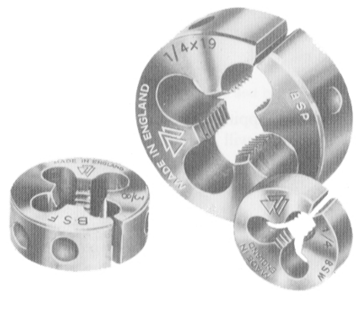


Manual Die Holder.



Manual Tap Holder.

Standard Set of Taps.



Standard Set of Dies.

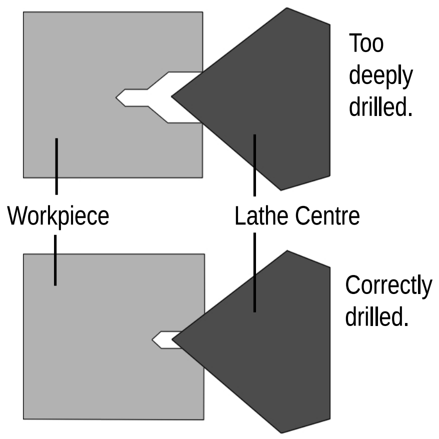
Drills

The Centre Drill.

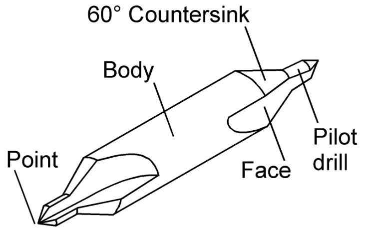
The centre drill is used in drilling and countersinking centre holes in one operation. Countersinks are set at 60° and a wide variety of drill sizes are available according the the operation required.

When drilling operations are performed, centre drills are used to provide pilot holes, to help guide the main drill bit.

When using lathe centres, the hollow created at the bottom of the countersinking hole leaves clearance for the centre point and provides a reservoir for lubricant.



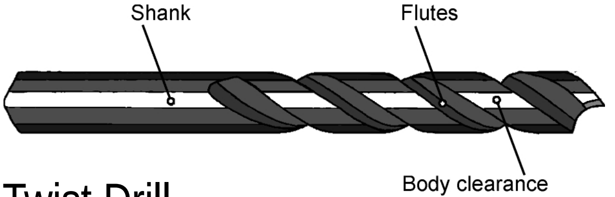
Centre Drilling for use with a Lathe Centre.



Centre Drill.

The Twist Drill.

Twist drills, available in a range of diameters, are used for boring circular holes into the workpiece.



Twist Drill.

Tool Geometry

Cutting tools are categorised depending on the number of cutting edges they have:

Single Point Cutting Tools.

These type of tools have a single effective cutting edge and remove excess material from the workpiece along the cutting edge. They are used in lathes and boring bar operations. Cutting tools using specially ground tips are usually preset to a specific set of dimensions and are mounted in toolholders. The tool assembly is then placed in a toolpost or turret, which is usually indexable.

Multi-point Cutting Tools.

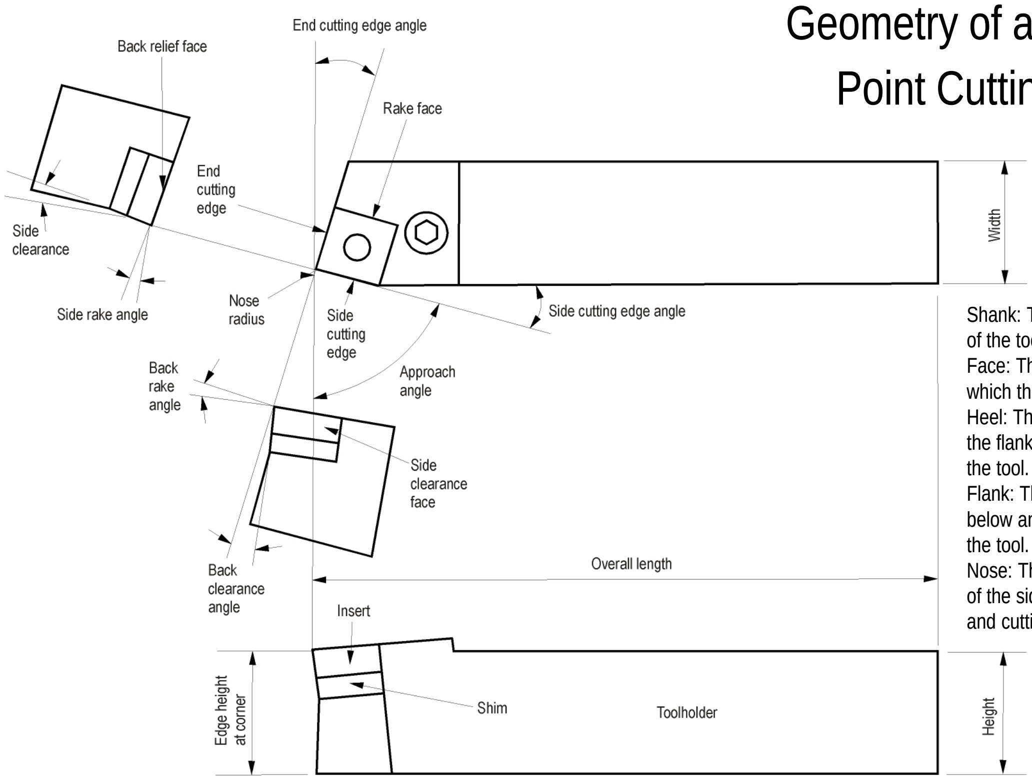
These type of tools have more than one cutting edge to remove excess material from the workpiece. They are used for milling, drilling, reaming, countersinking and counter boring operations on machining centres, milling machines and drills.

Special Purpose Tools.

These types of tools are especially designed for specific operations, such as taps and dies for internal and external thread cutting and grinding tools.

All cutting tools have certain angles and clearances to make them cut more efficiently. The various angles ground on a tool bit are called *basic tool angles*, often referred to by the term *tool geometry*. These angles and shapes can also be defined by a sequence of statements called *tool signature*. The geometry of a typical single point cutting tool is shown below:

Geometry of a Single Point Cutting Tool.



Shank: The main body of the tool.
Face: The surface on which the chip impinges.
Heel: The intersection of the flank and the base of the tool.
Flank: The surface below and adjacent to the tool.
Nose: The intersection of the side cutting edge and cutting edge.

Angle Systems

Back Rake Angle.

This is the angle that measures the degree of slope of the face of the tool from the nose towards the rear. A downward slope towards the nose is a negative back rake. A downward slope from the nose is a positive back rake. If there is no slope, the back rake angle is zero.

Side Rake Angle.

This is the angle that measures the degree of slope of the face of the tool from the cutting edge. A downward slope towards the cutting edge is a negative back rake. A downward slope from the cutting edge is a positive back rake. If there is no slope, the side rake angle is zero.

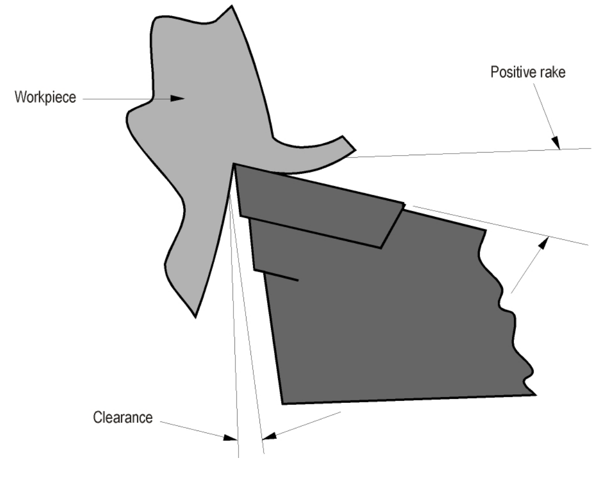
Positive Rake Angle System.

The face of the tool slopes away from the cutting edges and slopes towards the back of the tool, forming an acute angle with the workpiece. Cutting efficiency is optimum, since the tool penetrates the workpiece easier, shearing the material away, but htis also results in a much more fragile cutting edge. Material is moved away from the machined workpiece. Postive rake tools are indexable on one side only.

Use a Positive Rake Tool:

- To machine work hardened materials.
- When using machines of low power output.
- For turning long shafts of small diameters.
- When the workpiece lacks rigidity and strength.
- When machining below recommended cutting speeds.

Positive Rake Angle System.



Angle Systems

Negative Rake Angle System.

The face of the tool slopes away from the cutting edge and slopes upwards towards the back or side of the tool, forming an obtuse angle with the workpiece (the tool is not pointed against the movement of the work piece, but in the same direction). The cutting edge is stronger than positive rake systems and both top and bottom cutting edges can be used. This is especially important when taking interrupted cuts using carbide cutting tools, since most of the load occurs at the rear of the cutting edge, where there is more strength. Tool wear is reduced and heavier depths of cut may be performed, though material tends to move towards the machined workpiece.

Use a Negative Rake Tool:

- To machine high-strength alloys, using a high power output machine.
- When using intermittent cuts and heavy feedrates.

Negative Rake Angle System.

Side Cutting Edge Angle.

This is the angle between the side cutting edge and the edge of the toolholder shank, often referred to as the *lead angle*. The side cutting edge angle is usually set at around 15 degrees, although angles up to 40 degrees can be used. An increase in tool angle will give:

- An increase in the life of the tool.
- Better surface finishes.
- Quicker dissipation of heat.
- A wider surface on which the cutting force may be distributed.
- A decrease in the size of the chips removed from the material.
- Larger cutting speeds to be used.

Angle Systems

End Cutting Edge Angle.

This is the angle between the plane of the end cutting edge and a line perpendicular with the toolholder shank. The end cutting edge angle, usually set at 8 to 15 degrees, prevents the cutting edge of the tool from rubbing against the side of the workpiece. In conjunction with the feedrate used, this angle will affect the surface finish of any machining.

Side Clearance Angle.

This is the angle between the plane of the side cutting edge and a line perpendicular with the toolholder shank. The side clearance angle prevents the cutting edge of the tool from rubbing against the side of the workpiece. Larger feedrates require larger side clearance angles.

Nose Radius.

The nose (tip) of a tool is given a radius to strengthen the tool point, giving a longer tool life. Heat build-up is reduced and better surface finishes are obtained through the use of a tool nose radius.

A larger tool nose radius (1.5mm+) permits use of greater depths of cut and faster feedrates. Larger tool nose radii can be used if both the toolholder shank and the workpiece are rigid. This must be balanced against chatter which may occur when using a large tool nose radius, since the length of contact between the cutting edge and the workpiece is higher.

A small tool nose radius (0.4mm) is recommended when the workpiece cannot be securely held, lacks rigidity or has tubular construction. If the depth of cut is smaller than the tool nose radius, the surface finish can be poor quality.

Turning tools using indexable carbide inserts incorporate a standard nose radius of 0.2mm, 0.4mm, 0.8mm and 1.2mm, whilst heavy duty inserts are provided with a large nose radius of 1.2mm or 1.6mm.

Depth of cut (mm)	Tool Nose Radius (mm)
3 or less	0.5-0.75
3-10	1.0
12-20	1.5-2.0
20-30	2.0-3.0

Tool Signature.

Tools are specified using a standard abbreviated system, known as the *tool signature*. The data states the effective angles of the tool, normal to the cutting edge. All values are true, as long as the toolholder shank is mounted at right angles to the workpiece axis. An example signature of a single point cutting tool is shown in the table on the right.

Back rake angle	0
Side rake angle	7
End relief angle	7
Side relief angle	8
End cutting angle	15
Side cutting angle	15
Nose radius (mm)	0.8

Tool Signature

Table showing recommended tool geometry for a single point cutting tool for different types of work materials.

Work material	Hardness HB	Tool material	Back rake angle degrees	Side rake angle degrees	End clearance angle degrees	Side clearance angle degrees	Side & end cutting edge angle degrees
Free	85	HSS	10	12	5	5	15
machining	to	Brazed carbides	0	6	5	5	15
steels	225	Throwaway carbides	-5	-5	5	5	15
Mild	225	HSS	8	10	5	5	15
steels	to	Brazed carbides	0	6	5	5	15
	325	Throwaway carbides	-5	-5	5	5	15
Medium carbon		325	HSS	0	10	5	5 15
steels, alloy	to	Brazed carbides	0	6	5	5	15
steels	425	Throwaway carbides	-5	-5	5	5	15
Tool	45 HRC	HSS	0	10	5	5	15
steels	to	Brazed carbides	-5	-5	5	5	15
	58 HRC	Throwaway carbides	-5	-5	5	5	15
Stainless	135	HSS	0	10	5	5	15
steels	to	Brazed carbides	0	6	5	5	15
	275	Throwaway carbides	-5	-5	5	5	15
Cast iron,	100	HSS	5	10	5	5	15
grey, ductile,	to	Brazed carbides	0	6	5	5	15
malleable	200	Throwaway carbides	-5	-5	5	5	15
Cast iron,	200	HSS	5	8	5	5	15
grey, ductile,	to	Brazed carbides	0	6	5	5	15
malleable	300	Throwaway carbides	-5	-5	5	5	15
Cast iron,	300	HSS	5	5	5	5	15
grey, ductile,	to	Brazed carbides	-5	5	5	5	15
malleable	400	Throwaway carbides	-5	-5	5	5	15
Aluminium	30	HSS	20	15	12	10	5
alloys	to	Brazed carbides	3	15	5	5	15
	150	Throwaway carbides	0	5	5	5	15
Copper	40	HSS	5	10	8	8	5
alloys	to	Brazed carbides	0	8	5	5	15
	200	Throwaway carbides	0	5	5	5	15

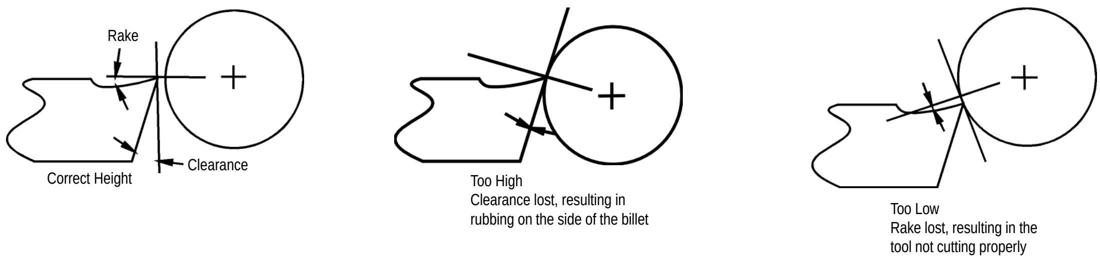
Tool Fault and Remedies Chart

Tool Wear.	Characterisations.	Remedies.
Flank wear.	Excessive amounts of flank wear characterises the end of the tool life.	Select more wear-resistant grade. Reduce cutting speed.
Notch wear.	The tool tip is in danger of breaking. Characterised by local wear in the area of the primary cutting edge, where it contacts the workpiece surface. Caused by hard surfaces and work-hardened burrs.	Strengthen cutting edge. Select smaller approach angle (45°). Reduce feedrate.
Crater wear.	Characterised by wear on the rake face.	Use coated hardmetal grades. Select positive insert geometry. Reduce cutting speed. Reduce chip cross section.
Edge chipping.	The tool tip is in danger of breaking. Characterised by minor chipping along the cutting edge, accompanied by flank wear. Edge chipping occurring outside the cutting area indicates excessive chip removal.	Select tougher grade. Reduce feedrate on the start of the cut operation. Vary the feedrate. Change the chip breaker geometry. Change the approach angle.
Insert breakage.	Characterised by damage to both the tool and the workpiece. Often caused by notches or excessive wear.	Select tougher grade. Use stronger insert with a larger corner radius. Select chip breaker geometry for heavy chip removal sections. Reduce feedrate and depth of cut.
Built-up edges.	Characterised on the rake face by the work material welding with the cutting material, especially when attempting to cut difficult to machine materials. Can break off, but often causes damage to the cutting edge. Results in poor surface finishes on machined workpieces.	Increase cutting speed. Use coated hardmetals (cermets). Select positive cutting edge geometry. Use cutting fluid.
Plastic deformation.	The tool tip is in danger of breaking. Characterised by overloading of the cutting edge, along with high machining temperatures.	Reduce cutting speed. Use lower feedrates. Use greater wear-resistant hardmetal grades.
Thermal cracks.	The tool tip is in danger of breaking. Characterised by small cracks running across the cutting edge, caused by thermal shock loads during interrupted cutting operations.	Use grade with greater resistance to thermal shock. Check use of cutting fluid (cutting fluid not recommended for interrupted cuts).
Chip control.	The size of chip removed is affected by the type of work material, feedrate and depth of cut. Short chips can cause vibrations and overloading of the cutting edge. The tool tip is in danger of breaking. Long chips coil around the tool and workpiece.	Avoid small depths of cut below 1x radius, except when finishing. If chips are too long, select chip breaker geometry for small chips, or increase feedrate. If chips are too short, select chip breaker geometry for long chips, or reduce feedrate.
Surface finish.	The surface finish is affected by the configuration and condition of the cutting tool point, the cutting conditions and the rigidity of the machining setup.	Increase cutting speed. Increase radius. Use cermet grade tools when cutting steel. Avoid vibrations. Use appropriate cutting fluid. Vary feedrate slightly. Change approach angle. Select different chip breaker geometry. Check rigidity of tool and holding system.
Shape and dimensional accuracy.	Both shape and dimensional accuracy are affected by the condition of the machining tool setup.	Select grade with better wear resistance. Check cutting parameters. Check rigidity of tool and holding systems. Reduce cutting forces.
Vibrations and instability.	Vibrations usually occur in thin walled tubes and non-rigid setups. Characterised by unbalanced and excessive cutting forces.	Increase tool approach angle. Use positive geometries. Use small radii. Reduce chip break cross sections.
Burring.	Burring often unavoidable when machining steel. Use chamfering methods wherever possible.	Select inserts with a positive geometry. Use sharp cutting edge insert (ie, cermet). Reduce approach angle.

Tool Setting

Tool Setting.

No matter what configuration or type of tool used, it must be postioned at the correct height, in the toolholder and toolpost, for efficeient cutting. Ideally, the tool should be positioned as far back into the toolpost as possible. Too great an overhang can cause vibration and chatter, leading to poor surface finishes.



Tool Setting Heights.

Further information on Tool Setting can be found in the ‘Cutting Parameters’ section.

Selection of Cutting Parameters

The lathe cutting parameters determined by the operator are the *feedrate*, *depth of cut* and *spindle speed*. All three must co-ordinated to take into account a number of factors:

1) *To achieve short cycle times.*

   Increasing the feedrate, depth of cut or spindle speed parameters will remove more material chips per unit time, reducing the time required to complete the CNC program. This must be balanced against accompanying high tool wear which will counter any time saved, due to more frequent tool tip and toolholder changes.

2) *To achieve lower machined part costs.*

   Any increase in cutting paramteers leading to a reduction in cycle times will also reduce overall labour and machining costs. Tool costs will rise due to greater wear. A balance must be chosen so that the wear-related tool costs are not excessive, possibly through the selection and use of an appropriate coolant.

3) *To achieve high machined workpiece quality.*

   The choices for flexibility of cutting parameters become much more limited, due to the specialised settings for obtaining high quality surface finishes and dimensional accuracy. The choice of parameters will depend on the type of tool tip being used, the load capabilities of the machine and the vibration characteristics of the tool, workpiece and machine.

Type of Material.

The type of material being machined during any cutting operation will effect most of the other factors involved in the process. The harder a material, the greater the load will be on the cutting tool. Therefore, the hardness of a material must be considered when selecting the geometry of the tool, the spindle speed, the feed rate and the cutting depth. If the load on the tool is too great, it may flex or even break during a cut and damage the workpiece. Therefore, the properties of the material being cut must be considered in order to choose the appropriate tool. The material used will alos impose restrictions on the types of coolant available as certain materials are not compatible with certain types of coolant.

Selection of Cutting Parameters

Machining Operations.

The selection of the correct spindle speed, feed rate, cutting depth and cutting tool for each machining operation is important since incorrect settings can lead to chattering, poor finishes and tool breakage. There are three basic types of machining operation:

1) *Roughing* - The roughing cut is used to remove largest amount of material from the billet safely and begin to shape the billet into the size and shape required. Roughing cuts use low spindle speeds at high feed rates, coupled with deep cuts to remove a large amount of material quickly. With such large cuts, the cutting tool is place under great load. Therefore special roughing tools should be used as they provide a higher strength and rigidity and stop chattering while also reducing the chance of the tool breaking.
2) *Finishing* - A finishing cut is the final cut on a billet. It is an accurate cut which takes a work piece into the tolerance required and leaves it correctly shaped and textured to allow for the extra finishing cut. A finishing cut usually leaves a depth of around two to four millimeters (2 - 4 mm).
3) *Extra Finishing* - This is a very light cut used to leave a smooth surface finish on the cut material. The surface finish depends on the depth of cut also speed and feed, the depth of cut is the distance between the tip of the cutting tool and the amount set on the dial to be removed. With an extra finishing cut, the depth should be no more than one millimeter (1 mm).

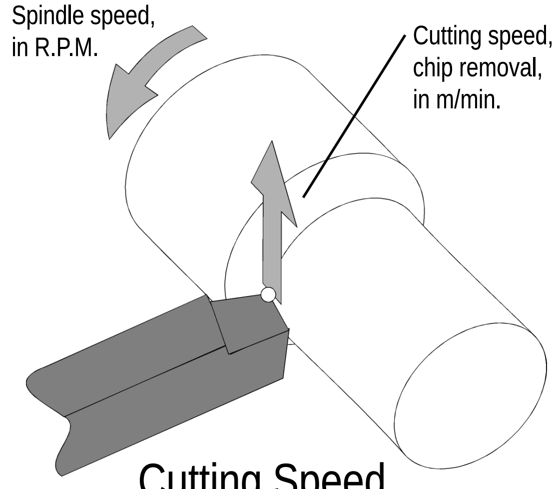
Cutting Speed.

Each material has an ideal speed at which it may be cut, according to the quality of surface finish required and the properties of the workpiece material.

Whilst machining, this cutting speed value must remain constant. The cutting speed is the equivalent of the workpiece surface speed at the point being machined (see diagram opposite).

A small spindle speed and small diameter being turned results in a low cutting speed.

A large spindle speed and large diameter being turned results in a large cutting speed.



Cutting Speed.

Ideal cutting speeds (m/min) for different types of material are shown in the table below:

Work Material	Tool Material	Cutting Speed (m/min)			
		Depth of cut (mm)			
		5-10	2-5	0.5-2	0.1-0.5
		Feedrate (mm/rev)			
		0.4	0.25-0.5	0.2-0.3	0.05-0.2
Free machining steels	HSS	20-40	40-70	40-110	50-120
	Carbide	90-150	120-180	150-250	200-500
Mild steels	HSS	25-35	30-50	30-60	40-80
	Carbide	60-120	80-150	120-200	150-450
Medium carbon steels	HSS	15-25	25-45	25-50	30-70
	Carbide	50-110	60-120	90-150	120-300
Alloy steels	HSS	10-15	15-25	15-35	20-45
	Carbide	30-65	40-80	60-100	80-180
Tool steels steels	HSS	15-20	20-25	20-30	30-60
	Carbide	50-110	60-120	90-150	120-300
Stainless steels	HSS	15-20	15-25	15-30	20-50
	Carbide	40-60	40-70	50-80	50-90
Cast iron, grey, ductile, malleable	HSS	20-25	25-30	35-45	40-60
	Carbide	60-90	70-100	80-110	80-120
Aluminium alloys	HSS	40-70	70-100	90-120	100-200
	Carbide	60-150	80-180	90-450	150-600
Copper alloys	HSS	40-60	60-100	90-120	100-200
	Carbide	50-110	60-150	90-180	120-310
Magnesium alloys	HSS	40-70	70-100	90-120	100-200
	Carbide	60-150	80-180	90-450	150-600

Billet Diameter and Spindle Speed

Billet Diameter.

The billet diameter dictates the spindle speed of an operation, the larger the diameter the slower the spindle speed. To measure spindle diameter use a vernier caliper or micrometer as shown on pages 29 and 30.

Spindle Speed.

The spindle speed is the rate at which the machine spindle revolves, determined by the type of material being cut and the diameter of the workpiece being cut.

In CNC programs, spindle speeds are usually signified by the code letter “S”.

Spindle speeds are measured in R.P.M. (Revolutions Per Minute).

The following formula is used to determine the required spindle speed:

$$N = \frac{1ØØØ \times V}{p \times d}$$

where N=Spindle Speed (R.P.M.)

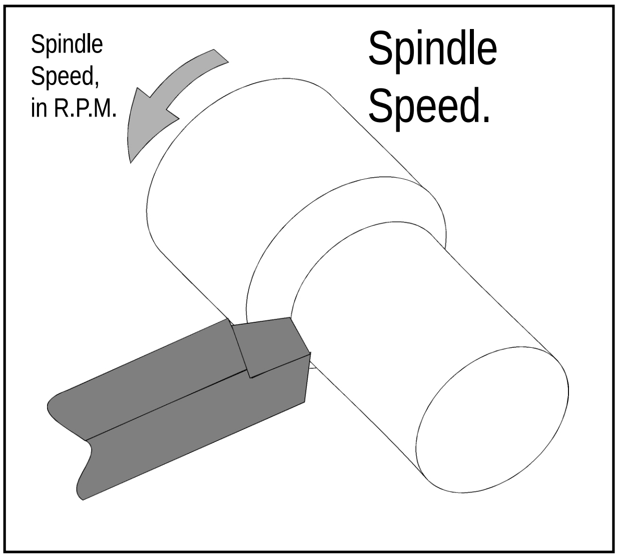
V=Cutting speed (m/min from the table on the previous page).

d=Diameter being turned (mm).

p=3.142.

For example, the spindle speed required for turning a 3Ømm mild steel bar, using a 2mm depth of cut, a feedrate of Ø.25mm/rev and a carbide tipped tool would be:

$$N = \frac{1ØØØ \times 8Ø}{3.142 \times 3Ø} = 85Ø \text{ R.P.M.}$$



The following M codes are used for controlling the Spindle:

MØ3 - Spindle Forward.

MØ4 - Spindle Reverse.

MØ5 - Spindle Stop.

M13 - Spindle on and coolant on.

M14 - Spindle Reverse and coolant on.

Feedrate

Feedrate.

The feedrate is the rate at which the cutting tool advances in the machining direction. In lathe work, distinction is made between longitudinal feed, when the tool travels in a direction parallel to the workpiece axis (Z) and cross feed, where the tool travels in a direction perpendicular to the workpiece (X).

The feedrate determines the machining speed. For this reason, feedrates are chosen according to the available cutting force and the surface finish required. By using coolants, higher feedrates and improved surface finishes can be obtained.

Maximum feedrate is limited by the following factors:

- Cutting edge strength
- Workpiece rigidity
- Surface finish required
- Tool chip size and space

In CNC programs, feedrates are usually signified by the code letter “F”.

Feedrates are measured in mm/min (Millimetres Per Minute) or mm/rev (Millimetres Per Revolution).

Feedrate (mm/min)=Feedrate (mm/rev) x Spindle Speed (rev/min).

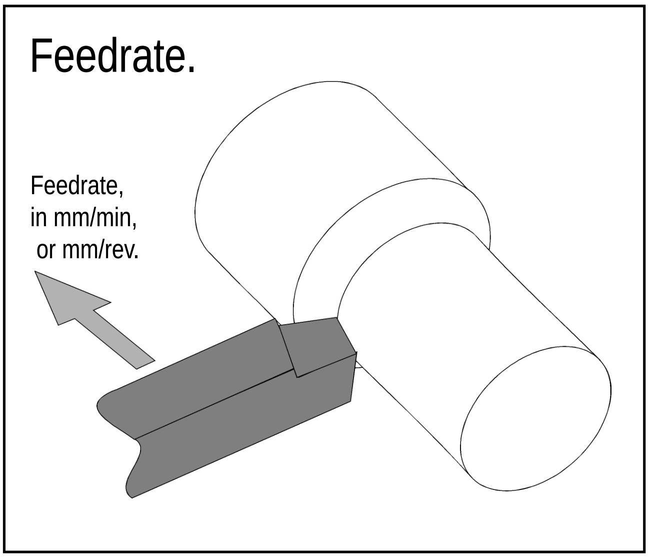
The following G codes are used for controlling the Feedrate:

G98 - Feedrate per minute.

G99 - Feedrate per revolution.

G20 - Imperial Data Input (Inches).

G21 - Metric Data Input (Millimetres).



Depth of Cut and Tool Setting Angles

Depth of Cut.

The depth of cut determines the rate of chip removal and the accuracy of any cutting operations. The greater the cutting depth, the greater the amount of material removed but this is achieved at the cost of accuracy.

When roughing, the depth of cut is set high (around 4mm), although this is dependant on the maximum rate that material can be removed from the workpiece.

When finishing, the depth of cut is set low (around 0.5mm), since the finishing cut must be very accurate, with a good surface finish.

Tool Setting Angles.

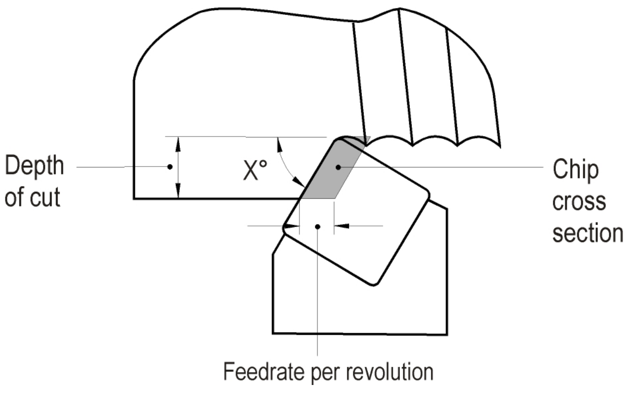
The choice of feedrate and depth of cut determines the size of the chip cross section.

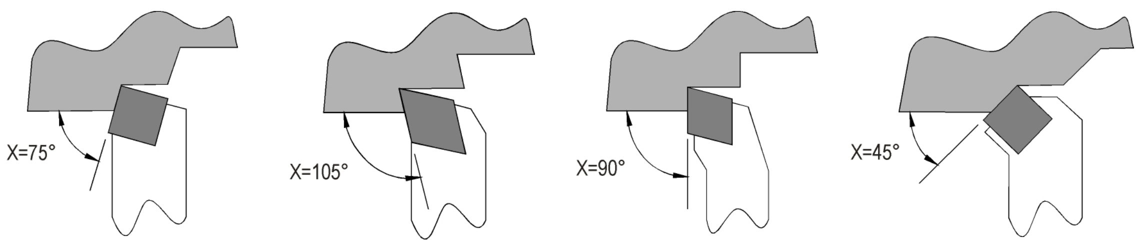
The following formulae are used to determine the size of the chip cross section and the amount of material removed during a machining operation:

Chip cross section=Feedrate per revolution x Depth of cut.

Stock removal volume=Chip cross section x Cutting speed.

Chip Cross Section.





Different Setting Angles.

Cutting Fluids and Coolants

Whilst it is possible to carry out some machining operations dry, wet machining using the appropriate cutting fluid (coolant) offers many advantages, including extended tool life, potential for better surface finishes and higher cutting depths and speeds.

A liquid coolant is used to prevent heat build up while the cutting tool is in use on a workpiece. The friction and cutting force between a tool and workpiece can cause extremely high temperatures. If left untreated, this would raise both the workpiece and tool to such temperatures that they would become malleable, making accurate cutting impossible. Therefore, a coolant is sprayed onto the tool and workpiece to absorb the heat and then evaporate, effectively keeping the tool and workpiece cool.

The coolant chosen should be selected primarily on the basis of the material being cut, as not all coolants are compatible with all materials. Therefore, choosing the right coolant can be critical to getting a properly cut component.

The other factor to be considered when choosing coolants is the operation that you wish to carry out, as some coolants are purpose made for specific operations or operation characteristics such as high feed/speed rates, deep cuts etc.

Commonly used coolants:

- *Soluble Oil* - these are inexpensive mineral oils containing emulsifying agents which enable them to be mixed with water and to remain stable. Use when machining copper.
- *Straight Mineral Oil* - these are low viscosity oils suitable only for light machining without any heavy tool loading. Use when machining Aluminium, brass.
- *Lard Oil* - these are mixtures of both mineral and fatty (lard) oils which are generally used over a much larger range of operations than straight mineral oils. Use when machining Aluminium, brass.
- *Sulphur Based Oil* - these contain sulphur content, allowing them to retain their lubricating properties under severe conditions. Use when machining Steel.
- *Dry* - Use when machining Cast iron.

CNC Machine Safety

General Safety Rules.

Due to the great cutting forces that CNC lathes are capable of producing and the increased feed and speed rates over conventional machines, great care must be taken at all times. Any fault in the program or the set up of the machine may have disastrous effects both to the machine and the operator.

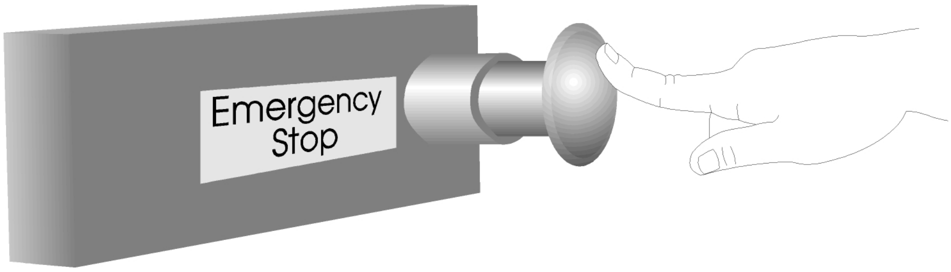
The seven safety rules listed below should be followed at all times when using CNC machines:

- **Make sure you know how to stop the machine in case of an emergency.**
- **Make sure that all machine guards are in position at all times.**
- **If you suspect something is going wrong, STOP the machine immediately.**
- **Isolate (switch off and unplug) the machine before making any adjustments.**
- **Do not attempt to use the machine until you are sure you can use it correctly.**
- **Programs should be tested prior to machining by using a dry run/computer simulation.**
- **Keep hands away from moving parts.**

The Emergency Stop Button.

All CNC machines are fitted with an large easy to reach when emergency stop button, usually red in coluour, incase the machine should do something unexpected. Pressing the button will immediately stop all slide, tool and spindle movements. Following the use of an emergency stop button, the CNC machine will need to be homed (datumed). This is because most of the information used by the controller, such as the tool position relative to the machines working enevelope, is lost when power is cut to the slides.

Make sure you know where the emergency stop button is on the machine you will be using!



CNC Machine Safety

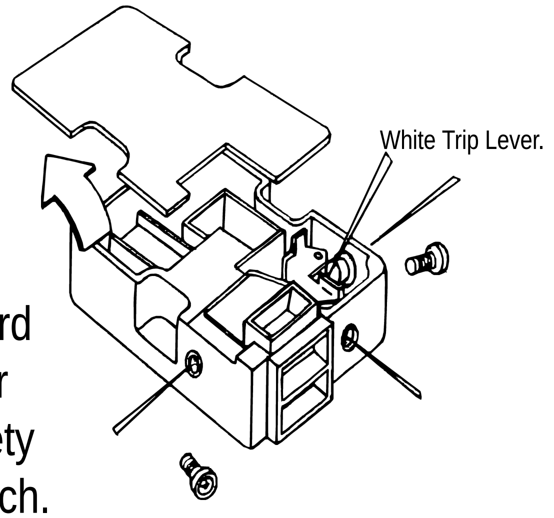
Axis Limit Switches.

Limit switches can be fitted to both CNC lathe slides to prevent overtravel. They are used when the toolpost has overtravelled and activated the limit switch, preventing the slide from fouling the machine casing and assemblies. To reset a limit switch, depress the ‘Axis Limit Override’ button and simultaneously press the appropriate axis key to move the toolpost away from the limit switch and back onto its regular section of slideway, then home each axis individually.

Guard Door Safety Switch.

CNC lathes supplied with CE type approval include a guard door safety switch, preventing entrance to the working area when machining operations are taking place, or the 24 volt circuit has failed. When the guard is opened, the CNC machine slides can only be moved in ‘Jog Step Incremental Mode’ - a safety feature that only allows an axis to be moved a set distance, each time its axis movement button is pressed. Continuous and rapid movements can only be performed when the guard switch is closed.

When complete power failure has occurred, entrance to the working area can only be obtained by manually tripping the safety switch, as shown in the diagram below.



Guard Door Safety Switch.

Safety Posters

Always wear eye protection when it is provided.

Make sure that you are dressed safely. No loose clothes or undone laces.

Keep long hair tied back or in a hat.

Safety Posters

Do not run in the workshop. Think of other peoples' safety too.

Keep the workshop and machines clean and tidy.

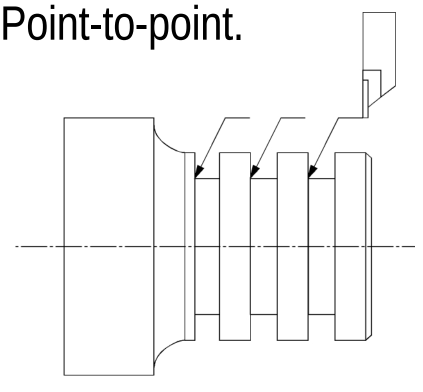
If in doubt, ask!

Positional Control

Movement of the CNC lathes slides to predetermined positions may be done in three ways:

Point-to-point.

This is the programming of instructions which will move the slide or slides to the next position required at a preprogrammed *Rapid Traverse* rate. Point-to-point control is sometimes referred to as the *Positioning System*. One or more axis may be involved, but the movements are not co-ordinated with each other so care must be taken to avoid collision with clamping arrangements or the workpiece.



Point-to-point.

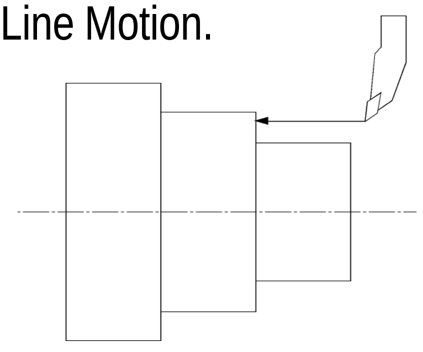
Note:

No cutting should be performed using point-to-point positioning.

Denford machines use code GØØ for point-to-point positioning.

Line Motion.

Line Motion is also referred to as *Linear Interpolation.* This refers to the programming of the next slide position required and also a feedrate to be used. More than two slides can be programmed to move at any one time under line motion (to produce taper effects). Line motion is used when the tool is to cut in a straight line.



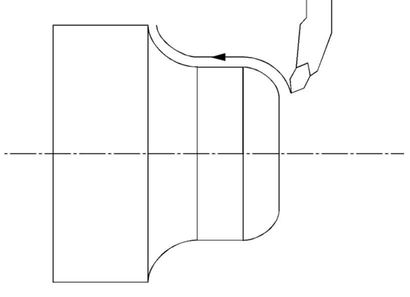
Line Motion.

Note:

Denford machines use code GØ1 for line motion.

Contouring.

This involves programming that is similar to line motion in that the next position is specified and a feedrate given. Angular and curved movement can be achieved by this method of positioning.



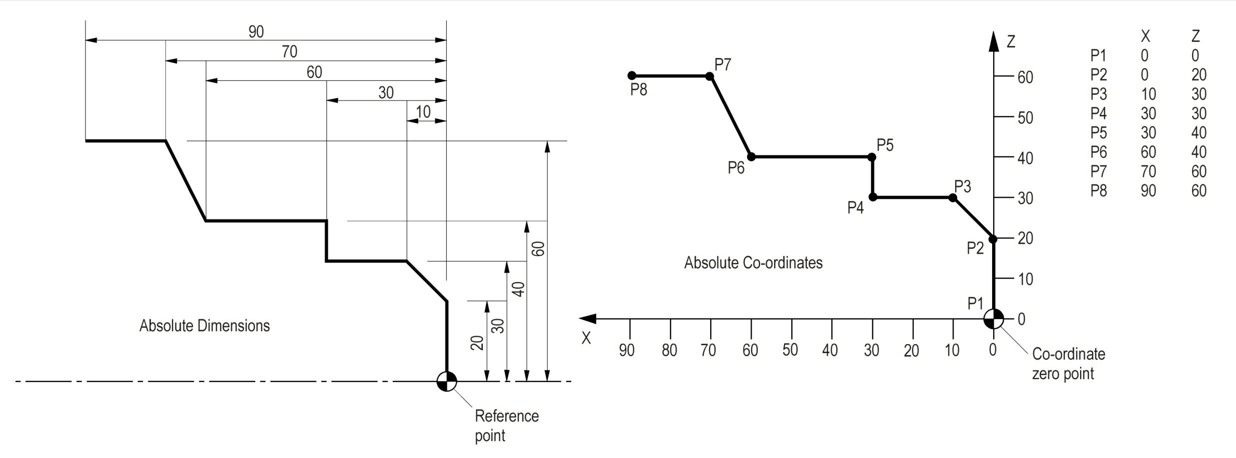
Contouring.

Note:

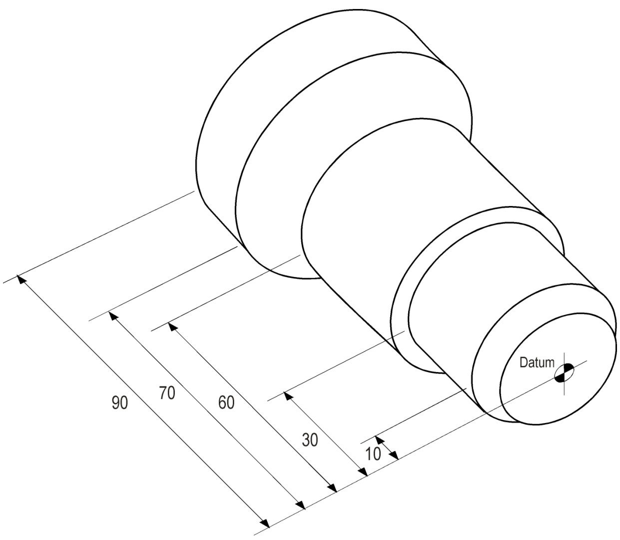
Denford machines use codes GØ2 and GØ3 for circular interpolation.

Co-ordinate Dimensioning - Absolute

Data in an *absolute dimension system* always refers to a fixed reference datum point, as shown in the example below. The dimension lines run parallel to the co-ordinate axes and always start at the reference point. Absolute dimensions are also referred to as *reference dimensions*. When plotting *absolute co-ordinates*, this point has the function of the co-ordinate zero point.



Absolute Dimensioning Example.

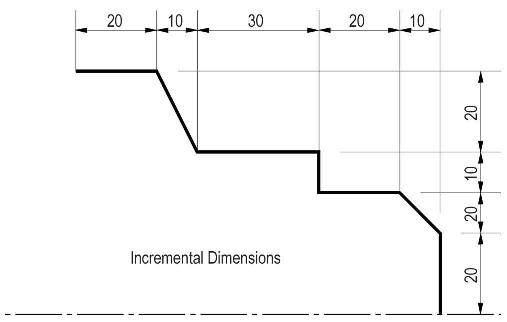


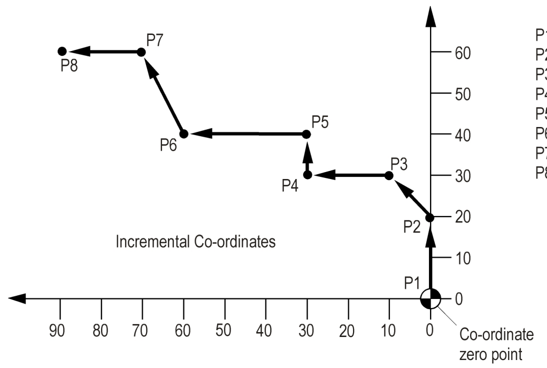
Advantages:

- There is no build-up or accumulation of errors between dimensions.
- In cases of interruptions that force the operator to pause or stop the CNC machine, the cutting tool automatically returns to the previous position, so the machining can proceed from the same block where it was interrupted.
- The dimensional data and positions in the part program can be easily changed, if required.

Co-ordinate Dimensioning - Incremental

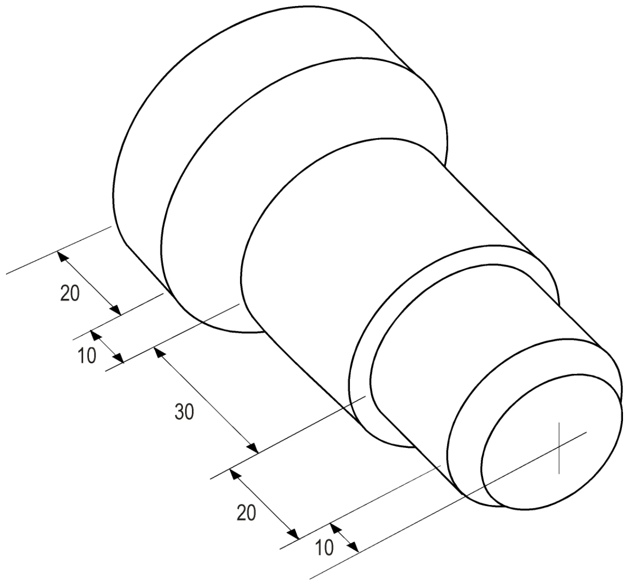
When using an *incremental dimension system*, every measurement refers to a previously dimensioned position. An incremental dimension is the distance between adjacent points. These distances are converted into incremental co-ordinates by accepting the last dimension point as the co-ordinate origin for the new point. Incremental dimensions are also referred to as *relative dimensions* or *chain dimensions*.





	X	Z
P1	0	0
P2	0	20
P3	10	30
P4	30	30
P5	30	40
P6	60	40
P7	70	60
P8	90	60

Incremental Dimensioning Example.



Advantages:

- Incremental positioning is advantageous when certain contours require repeating several times. The associated program sections can be used immediately without the need for a co-ordinate shift.

Disadvantages:

- If an error is made in one dimension this will mean that any subsequent positions will be incorrect.

Data Format

A Part Program may be defined as 'all the required data necessary in order to machine a component to the required specifications.'

The data or part program must follow the manufacturers format and is in a series of *blocks*. A Block is one line of data within the program. In each block there are a number of *words* and each word is made up of a number of *characters*.

Each operation requires a separate block. A typical block is shown below:

N G X Z F S T M

Each of these letters are known as *addresses*. After each address comes a number. The address and number together form a *word*
eg. GØØ (This is the program word defining Rapid Traverse).

Where:

N	refers to the Number of the block.
G	refers to the G code or Preparatory function.
X	refers to the distance travelled by the slide tool in the X direction.
Z	refers to the distance travelled by the slide tool in the Z direction.
F	refers to the Feedrate.
M	refers to the Miscellaneous Function.
S	refers to the Spindle speed.
T	refers to the Tooling management.

N - Block, or Program Line, Number.

Each new block of information must be allocated a number.
N means the Number of the block, or program line.

N ØØ1Ø
N ØØ2Ø
N ØØ3Ø
etc....

It is recommended that the blocks are programmed in increments of 1Ø. This allows for any future editing by leaving the opportunity to insert extra blocks where required. Alternatively, unrequired blocks may be removed.

Data Format

G - G code or Preparatory Function.

A G Code or Preparatory function is a command to the control unit to perform some specific cutting tasks or functions. These tasks or functions are used for a number of purposes:

GØØ Rapid Traverse
GØ1 Point to Point positioning
GØ2 Clockwise Circular Interpolation
GØ3 Anti-clockwise Circular Interpolation
G7Ø Imperial Units
G71 Metric Units
G9Ø Absolute Dimensions
G91 Incremental Dimensions

Some of the G codes have to be cancelled after use. They are known as modal.

For example,

GØ1 - Linear Motion (Programmed feed rate)
G42 - Cutter compensation.

Note:

G4Ø is the code to cancel cutter compensation.

Others will only operate in the block in which they are placed, these are known as non-modal.

For example,

GØ4 - Dwell

X and Z Co-ordinate Positional Data.

These letters refer to movement along the designated axes,

For example,

X 3ØØ could mean a slide or tool movement of 3ØØ millimetres in the X positive direction and, Z - 3ØØ could mean a slide or tool movement in the Z negative direction.

Data Format

F - Feedrate.

The feedrate is the speed at which the cutting tool is fed into the workpiece. It can be entered into the computer either as millimetres per minute (mm/min) or millimetres per revolution (mm/rev).

To change from mm/rev to mm/min requires prior G code changes

For example,

G94 or G95.

It is important that the programmer uses the correct feed rates for the operation to be carried out as an incorrect feed rate may result in damage to the cutting tool or workpiece. Information on the various feedrates and cutting speeds can be obtained from a *cutting tool* manufacturers brochure.

M - M code or Miscellaneous Function.

M codes or Miscellaneous functions, are commands to the control unit to perform a specific non-cutting task. M codes perform tasks other than those related to slide movement. Some examples are given below :

MØØ	Program stop.
MØ2	End of program.
MØ3	Spindle on clockwise.
MØ4	Spindle on Anti-clockwise.
MØ8	Coolant On.
MØ9	Coolant Off.

As with G codes, some of the M codes are modal and so must be cancelled after use.

S - Spindle Speed.

The speed of the spindle is usually programmed at the start of the program and when a new cutting tool is used.

The speed of the spindle can be programmed in the following ways:

- Revolutions per minute.
- Cutting Speeds in metres/minute.
- Constant Cutting Speeds in metres/minute.
- A chosen number from the manufacturers tables.

For most control systems today the spindle speed in revolutions per minute is programmed.

T - Tooling Arrangement.

This refers to the number of the tool required to be used for a particular operation.

Each tool is given a number and placed in a waiting area before being called upon. The tools must be pre-set in the machine prior to storage.

Zero Suppresssion

Any data that is fed into a CNC computer must conform to the manufacturer's classification or the computer will not understand it.

A typical example would be:

N4   G2   X4/3   Z4/3 .... etc

To explain this take X4/3:

The first digit (in this case 4) refers to the maximum number of digits that can be allowed before the decimal point.

The second digit (in this case 3) refers to the maximum number of digits that can be allowed after the decimal point.

For example, X 1689.913.

If there is only one figure after the address character for example N4

This means that the largest number of digits before a decimal point and none after it.

It is very important that any data entered into the machines computer is done so in a way that the computer understands.

Two methods are described below:

- *Fixed Block* - Each block of information must be entered into the computer even though some of the words have not changed from the previous block.
- *Variable Block* **-** The data can be entered in any order without having to repeat unchanged data.

Zero Suppression.

It may be that the data classification of a block is 3/2. As we know, this refers to the maximum and minimum number of digits before and after the decimal point.

But what if the programmer has to program a figure of 16.1 into this block?

It can be seen that there are only two digits prior to the decimal point and only one after it.

Some systems will require the programmer to input leading and trailing zeros

For example, Ø16.1Ø

Other systems will just require the original digits to be fed into the computer

For example, 16.1

The leading zero and the trailing zero are not required. We call this omitting of the zeros, *Zero Suppression*.

Program Proving

All CNC part programs should be carefully checked before they are set in motion on a machine tool. The checking, or verification, of a part program is to establish that no errors are present in the program which may cause damage to the machine, the workpiece and more importantly, the operator.

Methods of Checking and Verifying:

Visual Inspection.

The actual program is run, with no workpiece present in the chuck. Through purely visual inspection, all the programmed movements in all axes are checked, together with any tool offset and cutter compensation features. This method represents the least form of verification and should not be relied on entirely.

Graphical Simulation.

A graphical simulation package emulates the CNC lathe by using computer graphics to plot out any tool movement and cutting of the workpiece on the VDU screen. Any errors in the program will be observed and highlighted prior to the program being entered into the machine. The programmer can alter or 'Edit' their program as the simulation takes place.

Dry Run.

This method of program proving is done on the machine tool but the workpiece or billet to be cut is not installed. As the tool or cutter moves in air the operator looks to see that the cutting tool will not collide with proposed clamping arrangements or other projections within the set-up. Feedrate override facilities can be used to slow down the speed at which the program runs.

Single Step Execution.

Using single step mode, the part program can be run block by block with the operator in full control. All CNC machines have this facility and it should be used as part of a safe setting up and verification procedure.

All the above methods of verifying part programs are to check that the correct cutting conditions apply. If any doubt occurs on the part of the operator they must take action to prevent the part program running.

In a situation where there is likely to be damage to the machine or workpiece the operator *must* stop the machine. It is very important that the operator is familiar with the emergency stop procedure and the correct start up routine as the tool or machine slides may move to a pre-programmed position, when the start button is pressed.

Example Program

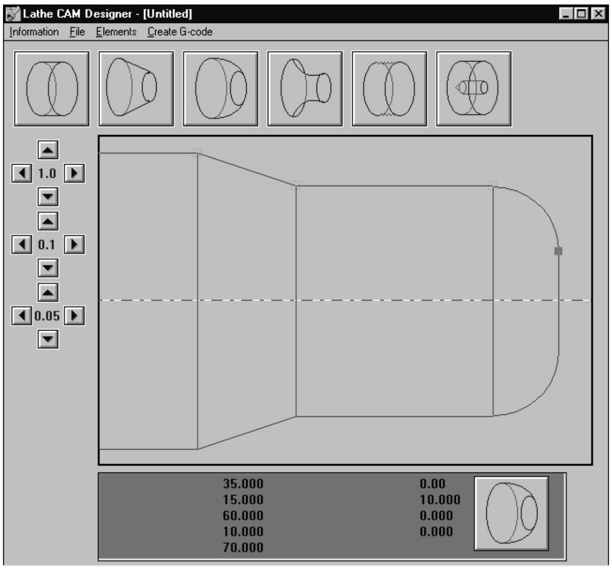
```
(Lathe CAM Designer - test.LCD)
(25/8/1999)
(Novaturn (metric))
(Post fanucl:1.2 23 May 1994)
N1G21
[BILLET X50 Z80
N2G98
N3G28U0W0
N4M6T0101
N5M3S1082
N6G0X50Z2
N7X52
N8X50
N9G1Z-69.75F140
N10X52
N11G0Z2
N12X47
N13M3S1151
N14G1Z-69.75
N15X49
N16G0Z2
N17X44
N18M3S1230
N19G1Z-53.14
N20X46
N21G0Z2
N22X41
N23M3S1320
N24G1Z-48.64
N25X43
N26G0Z2
N27X38
N28M3S1424
N29G1Z-44.14
N30X40
N31G0Z2
N32X35
N33M3S1546
N34G1Z-9.64
N35X37
N36G0Z2
N37X32
N38M3S1691
N39G1Z-4.36
N40X34
N41G0Z2
N42X29
N43M3S1866
N44G1Z-2.5
N45X31
N46G0Z2
N47X26
N48M3S2081
N49G1Z-1.3
N50X28
N51G0Z2
N52X23
N53M3S2353
N54G1Z-0.46
N55X25
N56G0Z2
N57X20
N58M3S2706
N59G1Z0.05
N60X22
N61G0Z2
N62X17
N63M3S3183
N64G1Z0.32
N65X19
N66X52
N67G0Z2
N68G1Z0
N69X15
N70M3S1546
N71G3X35Z-10K-10
N72G1Z-40
N73M3S1203
N74X45Z-55
N75Z-70
N76X52
N77G0Z2
N78M5
N79G28U0W0
N80M30
```

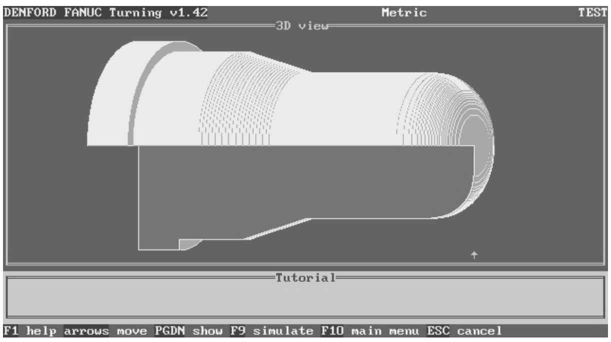
Program - "Test.fnc".

This program machines an aluminium billet, 75mm long (excluding a 20mm length section required for the chuck) with a 50mm diameter.

Below left: Denfords Lathe CAM Designer Package was used to design the test part. The G code program was generated using Lathe CAM's post processor.

Below right: 3d view of the test part generated using the Denford CNC turning machine controlling software.





Example Program

This section looks at the program "Test.fnc", in more detail, explaining what each particular line means....

(Lathe CAM Designer - test.LCD)
(25/8/1999)
(Novaturn (metric))
(Post fanucl:1.2 23 May 1994)

N1G21 **Define Working Parameters.** The N word designates the program line number. G21 defines the units being used as metric, ie, millimetres.

[BILLET X50 Z80 **Define Working Parameters.** A square bracket indicates a Denford directive, used only with Denford machines. The BILLET directive is used for simulation package graphics, indicating that the billet is 80mm long with a diameter of 50mm.

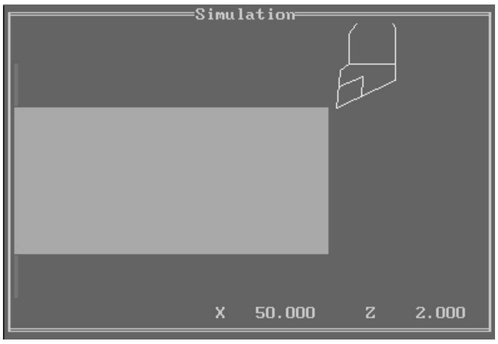
N2G98 **Define Working Parameters.** G98 commands a per minute feedrate, in this case, millimetres per minute will be used.

N3G28U0W0 **Program Start-up.** G28 commands a reference point return, homing the machine slides. The U and W co-ordinates can be used to specify an intermediate point that the toolpost will move to first, but since the values are both zero, no intermediate point is used.

N4M6T0101 **Program Start-up.** M6 commands an automatic tool change to tool number 0101.

N5M3S1082 **Program Start-up.** M3 switches the spindle on clockwise. The S word defines the value of the spindle speed, 1082RPM.

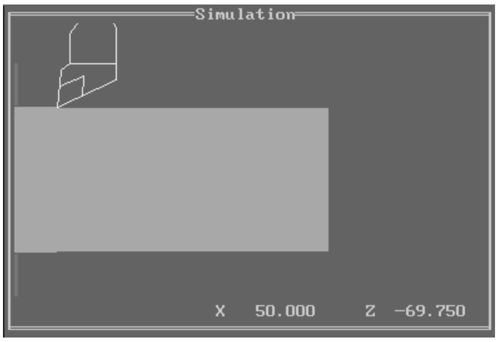
N6G0X50Z2 **Program Start-up.** G0 commands a rapid positioning traverse to the co-ordinates, X=50, Z=2 (just in front of the face end circumference of the billet) - see screenshot below.



N7X52 **Program Start-up.** Continue G0 to co-ordinate X=52 (moving at right angles away from the spindle centreline).

N8X50 **Roughing Cycle 1.** Continue G0 to co-ordinate X=50 (moving at right angles towards the spindle centreline).

N9G1Z-69.75F140 **Roughing Cycle 1.** G1 commands a linear interpolation, straight line cutting to co-ordinate Z=-69.75 (cutting a straight line parallel to the spindle centreline, towards the headstock). The F word defines a feedrate of 140mm/minute - see screenshot below.



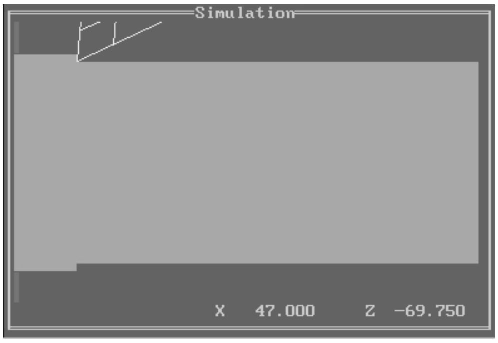
N10X52 **Roughing Cycle 1.** Continue G1 to co-ordinate X=52 (moving at right angles away from the spindle centreline).

N11G0Z2 **Roughing Cycle 1.** G0, rapid traverse, to co-ordinate Z=2 (moving in a straight line parallel to the spindle centreline, back to just in front of the face end of the billet).

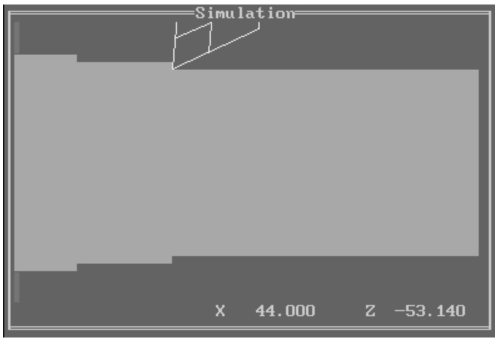
N12X47 **Roughing Cycle 2.** Continue G0 to co-ordinate X=47 (moving at right angles towards the spindle centreline).

Example Program

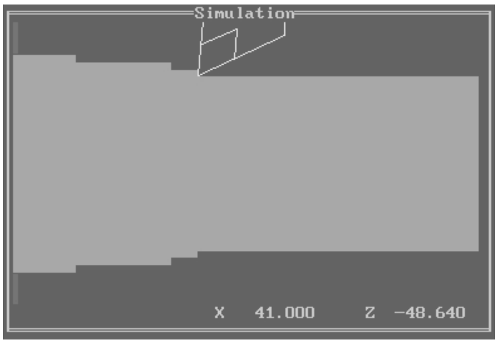
N13M3S1151	**Roughing Cycle 2.** M3 switches the spindle on clockwise, with a speed of 1151RPM.
N14G1Z-69.75	**Roughing Cycle 2.** G1, linear interpolation, to co-ordinate Z=-69.75 (cutting a straight line parallel to the spindle centreline, towards the headstock) - see screenshot below.



N15X49	**Roughing Cycle 2.** Continue G1 to co-ordinate X=49 (moving at right angles away from the spindle centreline).
N16G0Z2	**Roughing Cycle 2.** G0, rapid traverse, to co-ordinate Z=2 (moving in a straight line parallel to the spindle centreline, back to just in front of the face end of the billet).
N17X44	**Roughing Cycle 3.** Continue G0 to co-ordinate X=44 (moving at right angles towards the spindle centreline).
N18M3S1230	**Roughing Cycle 3.** M3 switches the spindle on clockwise, with a speed of 1230RPM.
N19G1Z-53.14	**Roughing Cycle 3.** G1, linear interpolation, to co-ordinate Z=-53.14 (cutting a straight line parallel to the spindle centreline, towards the headstock) - see screenshot below.

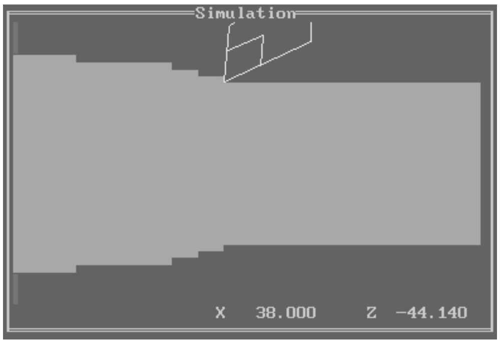


N20X46	**Roughing Cycle 3.** Continue G1 to co-ordinate X=46 (moving at right angles away from the spindle centreline).
N21G0Z2	**Roughing Cycle 3.** G0, rapid traverse, to co-ordinate Z=2 (moving in a straight line parallel to the spindle centreline, back to just in front of the face end of the billet).
N22X41	**Roughing Cycle 4.** Continue G0 to co-ordinate X=41 (moving at right angles towards the spindle centreline).
N23M3S1320	**Roughing Cycle 4.** M3 switches the spindle on clockwise, with a speed of 1320RPM.
N24G1Z-48.64	**Roughing Cycle 4.** G1, linear interpolation, to co-ordinate Z=-48.64 (cutting a straight line parallel to the spindle centreline, towards the headstock) - see screenshot below.

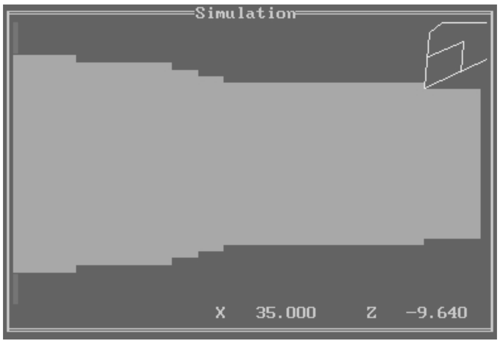


Example Program

N25X43	**Roughing Cycle 4.** Continue G1 to co-ordinate X=43 (moving at right angles away from the spindle centreline).
N26G0Z2	**Roughing Cycle 4.** G0, rapid traverse, to co-ordinate Z=2 (moving in a straight line parallel to the spindle centreline, back to just in front of the face end of the billet).
N27X38	**Roughing Cycle 5.** Continue G0 to co-ordinate X=38 (moving at right angles towards the spindle centreline).
N28M3S1424	**Roughing Cycle 5.** M3 switches the spindle on clockwise, with a speed of 1424RPM.
N29G1Z-44.14	**Roughing Cycle 5.** G1, linear interpolation, to co-ordinate Z=-44.14 (cutting a straight line parallel to the spindle centreline, towards the headstock) - see screenshot below.



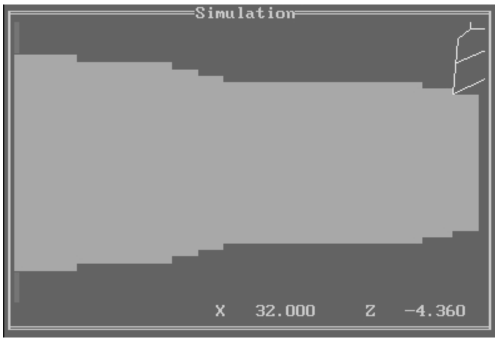
N30X40	**Roughing Cycle 5.** Continue G1 to co-ordinate X=40 (moving at right angles away from the spindle centreline).
N31G0Z2	**Roughing Cycle 5.** G0, rapid traverse, to co-ordinate Z=2 (moving in a straight line parallel to the spindle centreline, back to just in front of the face end of the billet).
N32X35	**Roughing Cycle 6.** Continue G0 to co-ordinate X=35 (moving at right angles towards the spindle centreline).
N33M3S1546	**Roughing Cycle 6.** M3 switches the spindle on clockwise, with a speed of 1546RPM.
N34G1Z-9.64	**Roughing Cycle 6.** G1, linear interpolation, to co-ordinate Z=-9.64 (cutting a straight line parallel to the spindle centreline, towards the headstock) - see screenshot below.



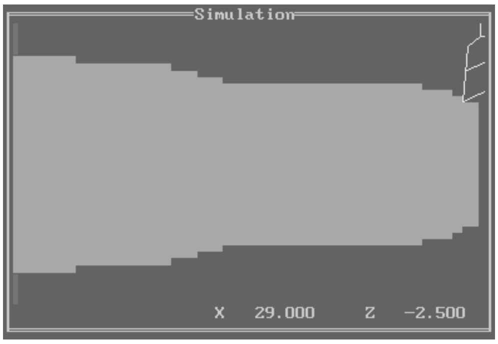
N35X37	**Roughing Cycle 6.** Continue G1 to co-ordinate X=37 (moving at right angles away from the spindle centreline).
N36G0Z2	**Roughing Cycle 6.** G0, rapid traverse, to co-ordinate Z=2 (moving in a straight line parallel to the spindle centreline, back to just in front of the face end of the billet).
N37X32	**Roughing Cycle 7.** Continue G0 to co-ordinate X=32 (moving at right angles towards the spindle centreline).
N38M3S1691	**Roughing Cycle 7.** M3 switches the spindle on clockwise, with a speed of 1691RPM.

Example Program

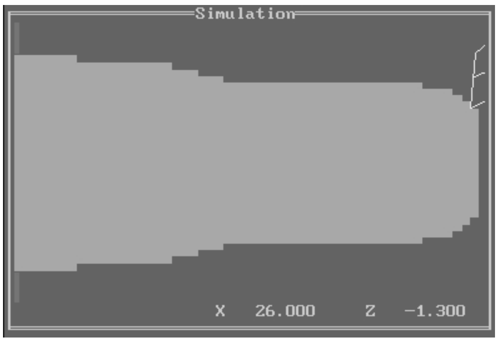
N39G1Z-4.36	**Roughing Cycle 7.** G1, linear interpolation, to co-ordinate Z=-4.36 (cutting a straight line parallel to the spindle centreline, towards the headstock) - see screenshot below.



N40X34	**Roughing Cycle 7.** Continue G1 to co-ordinate X=34 (moving at right angles away from the spindle centreline).
N41G0Z2	**Roughing Cycle 7.** G0, rapid traverse, to co-ordinate Z=2 (moving in a straight line parallel to the spindle centreline, back to just in front of the face end of the billet).
N42X29	**Roughing Cycle 8.** Continue G0 to co-ordinate X=29 (moving at right angles towards the spindle centreline).
N43M3S1866	**Roughing Cycle 8.** M3 switches the spindle on clockwise, with a speed of 1866RPM.
N44G1Z-2.5	**Roughing Cycle 8.** G1, linear interpolation, to co-ordinate Z=-2.5 (cutting a straight line parallel to the spindle centreline, towards the headstock) - see screenshot below.

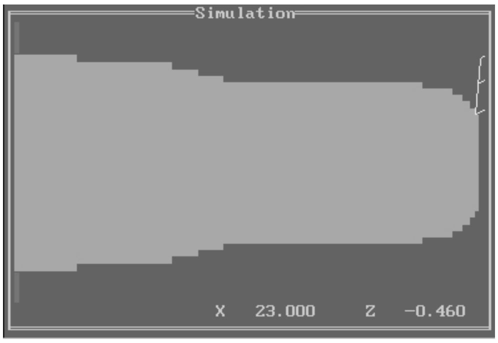


N45X31	**Roughing Cycle 8.** Continue G1 to co-ordinate X=31 (moving at right angles away from the spindle centreline).
N46G0Z2	**Roughing Cycle 8.** G0, rapid traverse, to co-ordinate Z=2 (moving in a straight line parallel to the spindle centreline, back to just in front of the face end of the billet).
N47X26	**Roughing Cycle 9.** Continue G0 to co-ordinate X=26 (moving at right angles towards the spindle centreline).
N48M3S2081	**Roughing Cycle 9.** M3 switches the spindle on clockwise, with a speed of 2081RPM.
N49G1Z-1.3	**Roughing Cycle 9.** G1, linear interpolation, to co-ordinate Z=-1.3 (cutting a straight line parallel to the spindle centreline, towards the headstock) - see screenshot below.

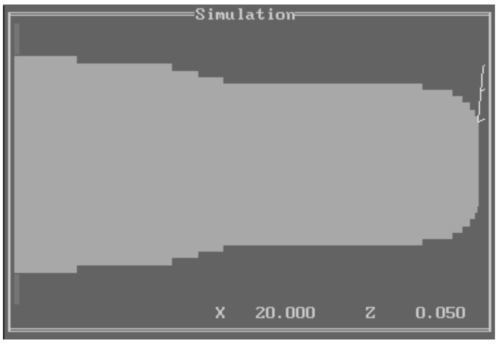


Example Program

N50X28	**Roughing Cycle 9.** Continue G1 to co-ordinate X=28 (moving at right angles away from the spindle centreline).
N51G0Z2	**Roughing Cycle 9.** G0, rapid traverse, to co-ordinate Z=2 (moving in a straight line parallel to the spindle centreline, back to just in front of the face end of the billet).
N52X23	**Roughing Cycle 10.** Continue G0 to co-ordinate X=23 (moving at right angles towards the spindle centreline).
N53M3S2353	**Roughing Cycle 10.** M3 switches the spindle on clockwise, with a speed of 2353RPM.
N54G1Z-0.46	**Roughing Cycle 10.** G1, linear interpolation, to co-ordinate Z=-0.46 (cutting a straight line parallel to the spindle centreline, towards the headstock) - see screenshot below.



N55X25	**Roughing Cycle 10.** Continue G1 to co-ordinate X=25 (moving at right angles away from the spindle centreline).
N56G0Z2	**Roughing Cycle 10.** G0, rapid traverse, to co-ordinate Z=2 (moving in a straight line parallel to the spindle centreline, back to just in front of the face end of the billet).
N57X20	**Roughing Cycle 11.** Continue G0 to co-ordinate X=20 (moving at right angles towards the spindle centreline).
N58M3S2706	**Roughing Cycle 11.** M3 switches the spindle on clockwise, with a speed of 2706RPM.
N59G1Z0.05	**Roughing Cycle 11.** G1, linear interpolation, to co-ordinate Z=-0.05 (cutting a straight line parallel to the spindle centreline, towards the headstock) - see screenshot below.

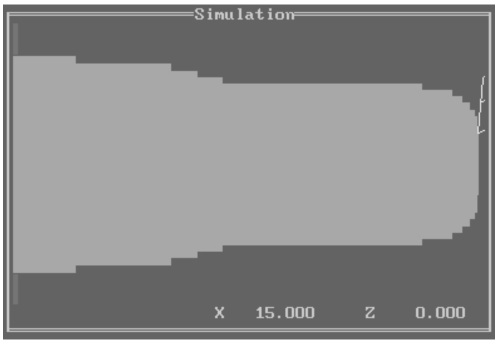


N60X22	**Roughing Cycle 11.** Continue G1 to co-ordinate X=22 (moving at right angles away from the spindle centreline).
N61G0Z2	**Roughing Cycle 11.** G0, rapid traverse, to co-ordinate Z=2 (moving in a straight line parallel to the spindle centreline, back to just in front of the face end of the billet).
N62X17	**Roughing Cycle 12.** Continue G0 to co-ordinate X=17 (moving at right angles towards the spindle centreline).
N63M3S3183	**Roughing Cycle 12.** M3 switches the spindle on clockwise, with a speed of 3183RPM.
N64G1Z0.32	**Roughing Cycle 12.** G1, linear interpolation, to co-ordinate Z=0.32 (cutting a straight line parallel to the spindle centreline, away from the headstock).
N65X19	**Roughing Cycle 12.** Continue G1 to co-ordinate X=19 (cutting at right angles away from the spindle centreline).
N66X52	**Roughing Cycle 12.** Continue G1 to co-ordinate X=52 (cutting at right angles away from the spindle centreline).
N67G0Z2	**Finishing Cut.** G0, rapid traverse, to co-ordinate Z=2 (moving back to just in front of the face end of the billet).

Example Program

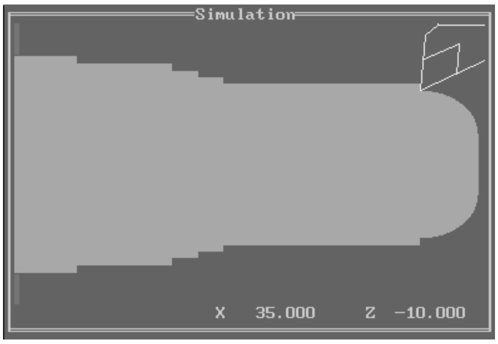
N68G1Z0 **Finishing Cut.** G1, linear interpolation, to co-ordinate Z=0 (cutting a straight line parallel to the spindle centreline, towards the headstock).

N69X15 **Finishing Cut.** Continue G1 to co-ordinate X=52 (cutting at right angles away from the spindle centreline) - see screenshot below.

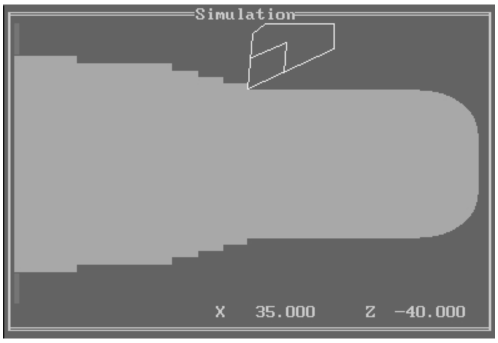


N70M3S1546 **Finishing Cut.** M3 switches the spindle on clockwise, with a speed of 1546RPM.

N71G3X35Z-10K-10 **Finishing Cut.** G3 commands a counterclockwise circular interpolation, arc line cutting, to the co-ordinate X=35, Z=-10, with K=-10. The K word defines the distance from the start point to the arc centre. The tool starts from near the spindle centreline, at the face end of the billet and moves both towards the headstock end and away from the spindle centreline, to precisely cut the arc required - see screenshot below.



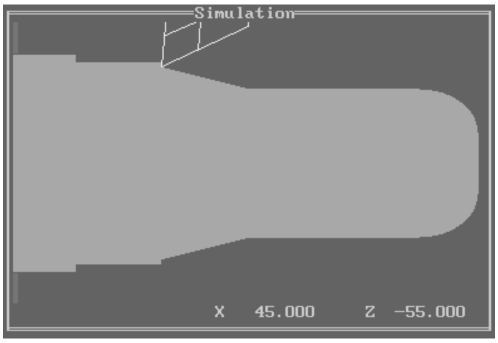
N72G1Z-40 **Finishing Cut.** G1, linear interpolation, to co-ordinate Z=-40 (cutting a straight line parallel to the spindle centreline, towards the headstock) - see screenshot below.



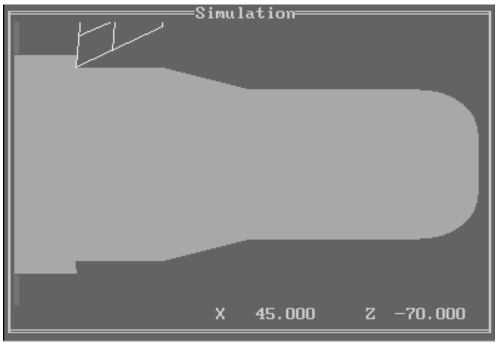
N73M3S1203 M3 switches the spindle on clockwise, with a speed of 1203RPM.

Example Program

N74X45Z-55 **Finishing Cut.** Continue G1 to co-ordinate X=45, Z=-55 (cutting a taper outwards in the direction of the headstock) - see screenshot below.



N75Z-70 **Finishing Cut.** Continue G1 to co-ordinate Z=-70 (cutting a straight line parallel to the spindle centreline, towards the headstock) - see screenshot below.



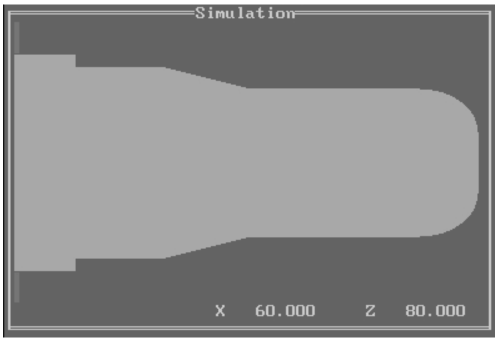
N76X52 **Finishing Cut.** Continue G1 to co-ordinate X=52 (cutting at right angles away from the spindle centreline).

N77G0Z2 **Finishing Cut.** G0, rapid traverse, to co-ordinate Z=2 (moving in a straight line parallel to the spindle centreline, back to just in front of the face end of the billet).

N78M5 **Program End.** M5 commands the spindle to stop rotating.

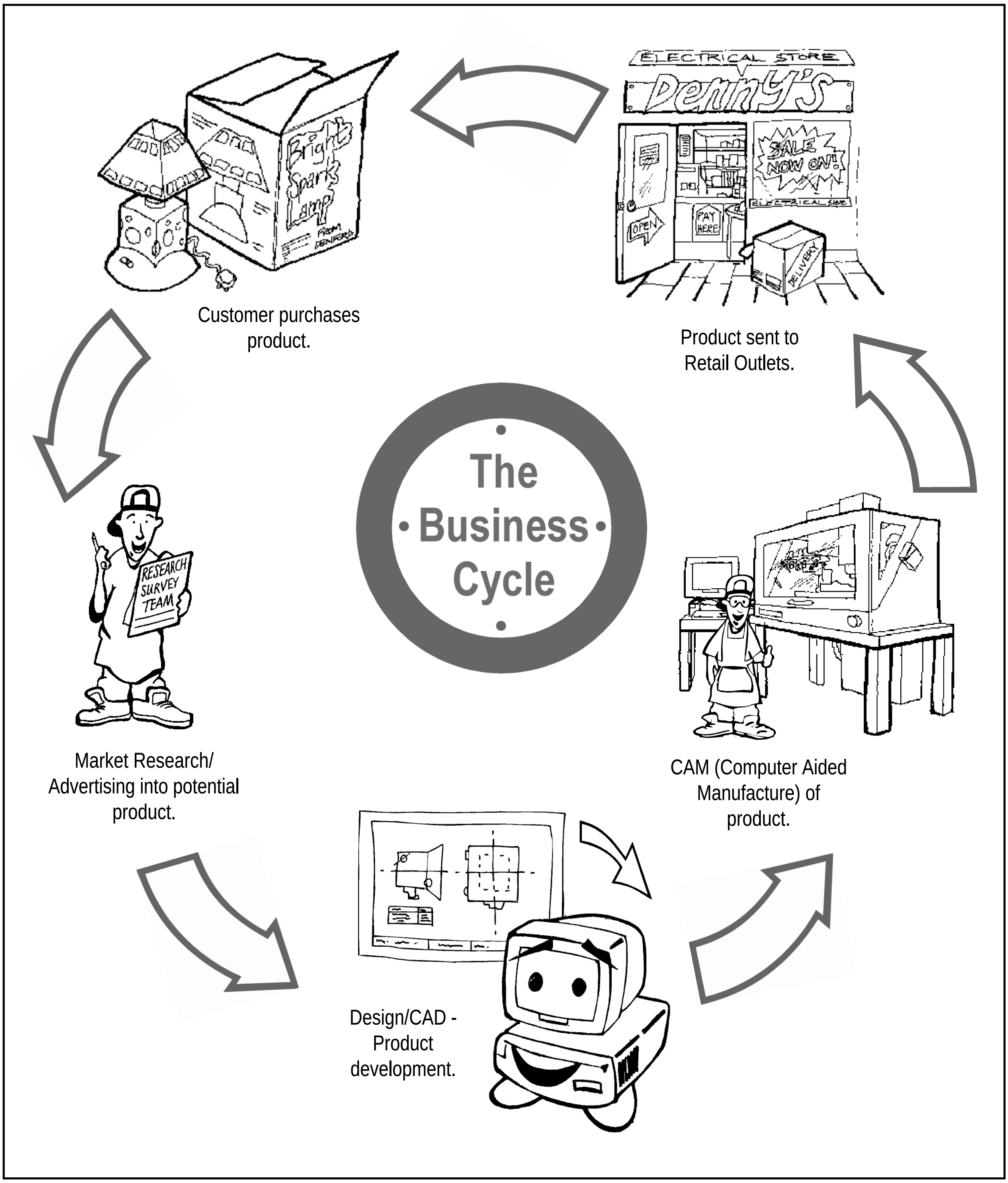
N79G28U0W0 **Program End.** G28 commands a reference point return, homing the machine slides. The U and W co-ordinates can be used to specify an intermediate point that the toolpost will move to first, but since the values are both zero, no intermediate point is used.

N80M30 **Program End.** M30 indicates the end of the program and rewinds control back to the start of the program - see screenshot below.



The Business Cycle

CNC machines and CAD/CAM can play an important role in the success of a business, by aiding the companies efficiency, flexibility and quality....



Why use CNC Machines?

CNC machines are used to great effect by industry, particularly in areas of production where repeat tasks or high accuracy is required. In many situations, they can offer great advantages over human operated machines...

The advantages of CNC machines:

- CNC machines, once programmed, will perform a repeat task until instructed to stop. Each component produced will be exactly the same size and shape. This saves money on designing the jigs and fixtures - units which hold the material and help check it's being machined to the correct size.
- The ability to store a part program in the memory. With some CNC machines it may be possible to store more than one program in the control unit memory at any one time.
- Part programs may be edited relatively easily after entry, either at the machine tool itself (called on line) or away from the machine tool (called off line).
- Repetitive machining operations can be done by smaller sub-programs, therefore making part programs shorter.
- Cutter compensation and offsets are made much easier.
- Most modern CNC machines are equipped with program proving software. This software makes it possible to produce the component shape in the form of a graphical display on a VDU, plotter or printer. Any errors in the part program can therefore be identified prior to machining.
- Most CNC control units are able to communicate with other computer-based systems, for example CAD/CAM (Computer Aided Design and Manufacture).
- Waste material can be reduced, since a CNC machine is much less likely to make an error than a manual machine. CNC machines can run 24 hours a day, if necessary, with no signs of fatigue, unlike machines requiring a human operator.
- Companies can estimate the manufacturing costs for CNC production much more accurately, compared to a production line with manual machines.
- Consistently high quality of parts produced, including parts involving complicated contours.
- Tool inventory low and hence tool storage costs lower than manual machines.
- Applicable to a wider range of production methods, from one-off through to mass production.
- Easier to change between units of measurement, Metric to Imperial and vice versa.

The disadvantages of CNC machines:

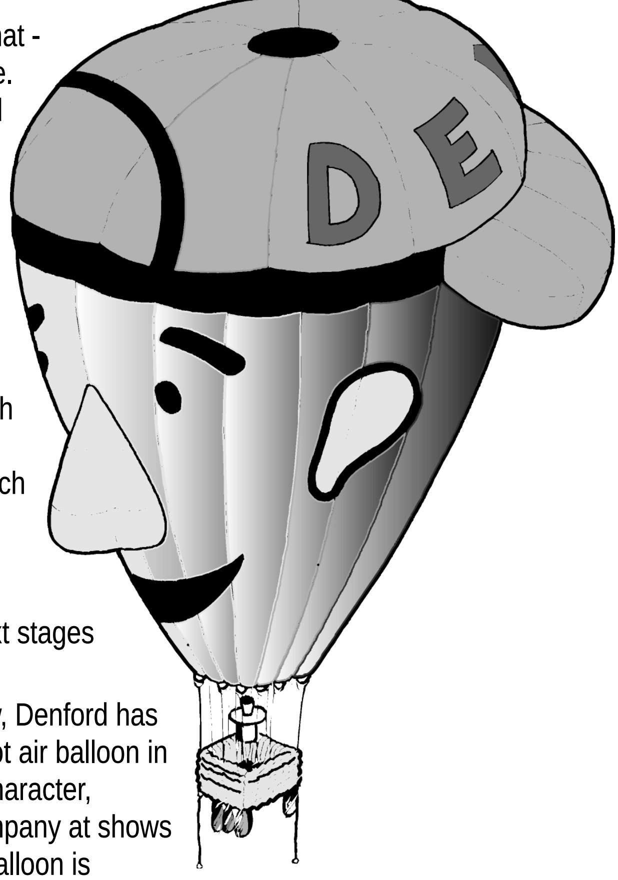
- The cost of buying and installing the machines can be quite high, compared to human operated machines.
- Maintenance of CNC machines requires a high order of skill and trained personnel.
- The company needs to train both operators *and* programmers of the installed system.
- If very large numbers of *identical* components need to be made (high quantity mass production) it is probably more cost effective for the company to install a specially designed automatic machine.

Production Sizes

One factor which can influence the decision to use CNC machines is the number of identical components that are to be produced on the machines. There are basically three methods used depending on the scale of production:

a) One-off production.

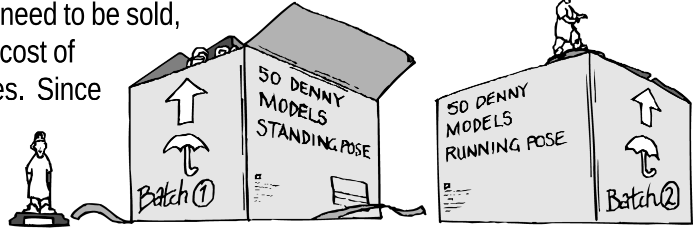
One-off simply means just that - only one component is made. Most products manufactured as one-offs are expensive to buy, since each component has to be individually designed, made, checked and fitted. Some one-off products, such as a suspension bridge, have engineering safety as a much higher priority than the production costs. Others such as "prototypes" are manufactured purely to test public reaction on a product, before committing to the next stages of manufacture.



In the example shown below, Denford has decided to manufacture a hot air balloon in the shape of their cartoon character, “Denny”, to promote the company at shows and exhibitions. Only one balloon is required and safety is an important factor, so the most economical method for production would be one-off.

b) Batch production.

Quite often, a company will need to make a small number of identical products, say 40 special benches for a new theme park. Using one-off production, the benches would be individually designed and made. Therefore, they would be very expensive to produce. If mass production was used, lots of benches would need to be sold, just to cover the initial cost of setting up the machines. Since only 40 are required, this set-up cost would be too great.

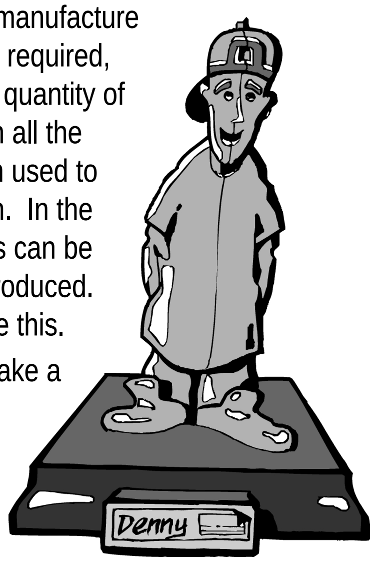


Production Sizes

b) Batch Production.

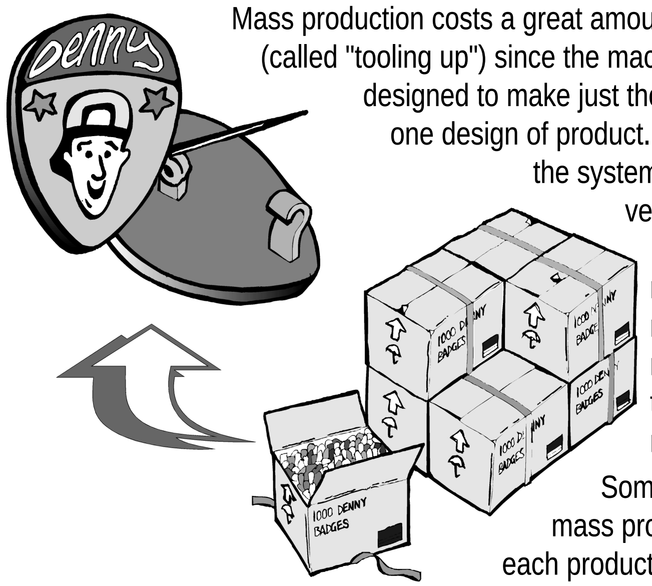
continued....

In cases such as these, it makes better sense to manufacture every component needed to build all the products required, rather like a miniature production line. This small quantity of products, made one after is called a batch. When all the products have been made, the machines are then used to build completely different items for another design. In the future, if more products are needed, the machines can be reset and another batch of components can be produced. CNC machines are ideally suited to a situation like this.



In the example shown, Denford has decided to make a limited edition model of their cartoon character, "Denny", in two different poses, to sell to its customers. There will be 50 models of each pose, so the most economical method would be batch production.

c) Mass production.



Mass production costs a great amount of money to install (called "tooling up") since the machines are specially designed to make just the components needed for one design of product. However, once running, the system makes the products very cheaply. It is only cost effective if a very high number of products are needed, say 2500 or more products, since these initial tooling costs need to be overcome.

Some products which are mass produced are very complex, each product itself being made of thousands of different components.

Probably, the most well known example of this system is the car production line, where computers, robots and automatic machines are used to set-up a system which can run virtually "human" free.

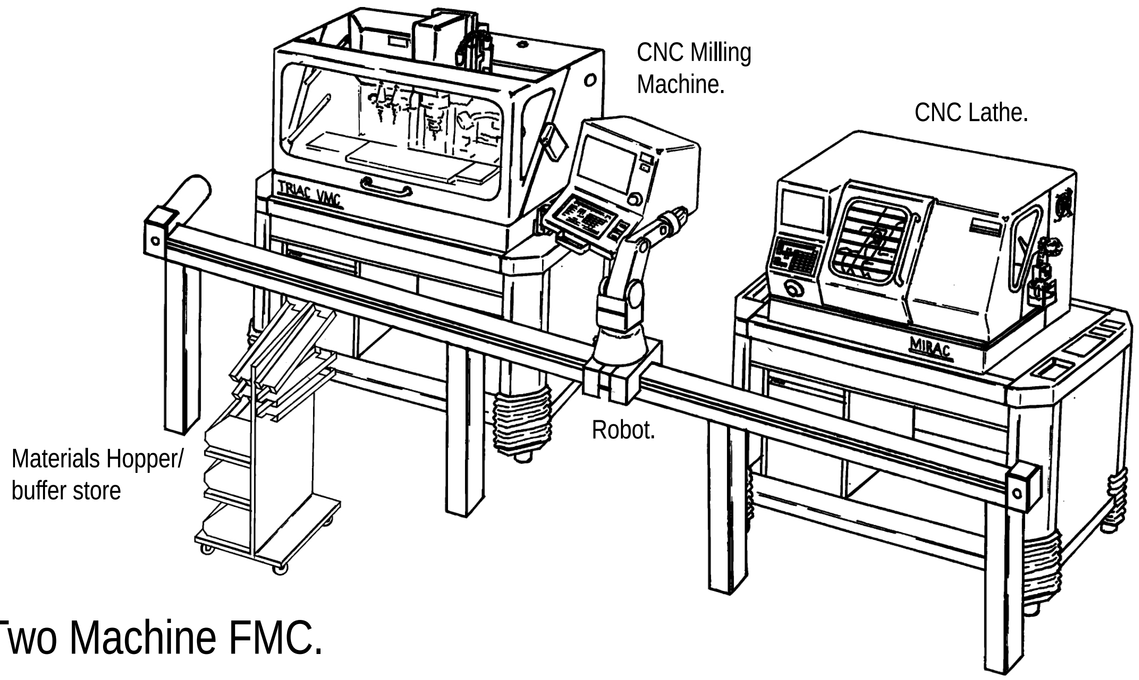
In the example shown, Denford have discovered that their cartoon character, "Denny", seems more popular than ever, so the company decides to launch a special promotional "Denny" badge. 150,000 badges will be made, so it will be cost effective to use mass production.

Flexible Manufacturing Cells (FMC)

Many companies cannot justify spending such a high amount of capital (money) on long, fully automated production lines, since the machines are usually limited to making one design only. In cases such as these, the company installs a number of flexible manufacturing cells (FMC). Each FMC may contain a number of machines, programmed to carry out a particular task on the particular component being produced. A number of these FMCs are then linked together in a certain sequence (order) to form a production line to make all the different components for the design.

The FMC illustrated below comprises of one CNC milling machine and one CNC lathe, a hopper where the raw materials and finished parts are stored (this is sometimes called a buffer store), and a robot moving on a slide, for taking the parts to the different areas of the cell.

Using a system such as this allows the company more flexibility to expand, simply by adding more FMCs into the production line, when required. When a particular job is finished, any number of FMCs can be reprogrammed and recombined into a 'new' production line for a different design.

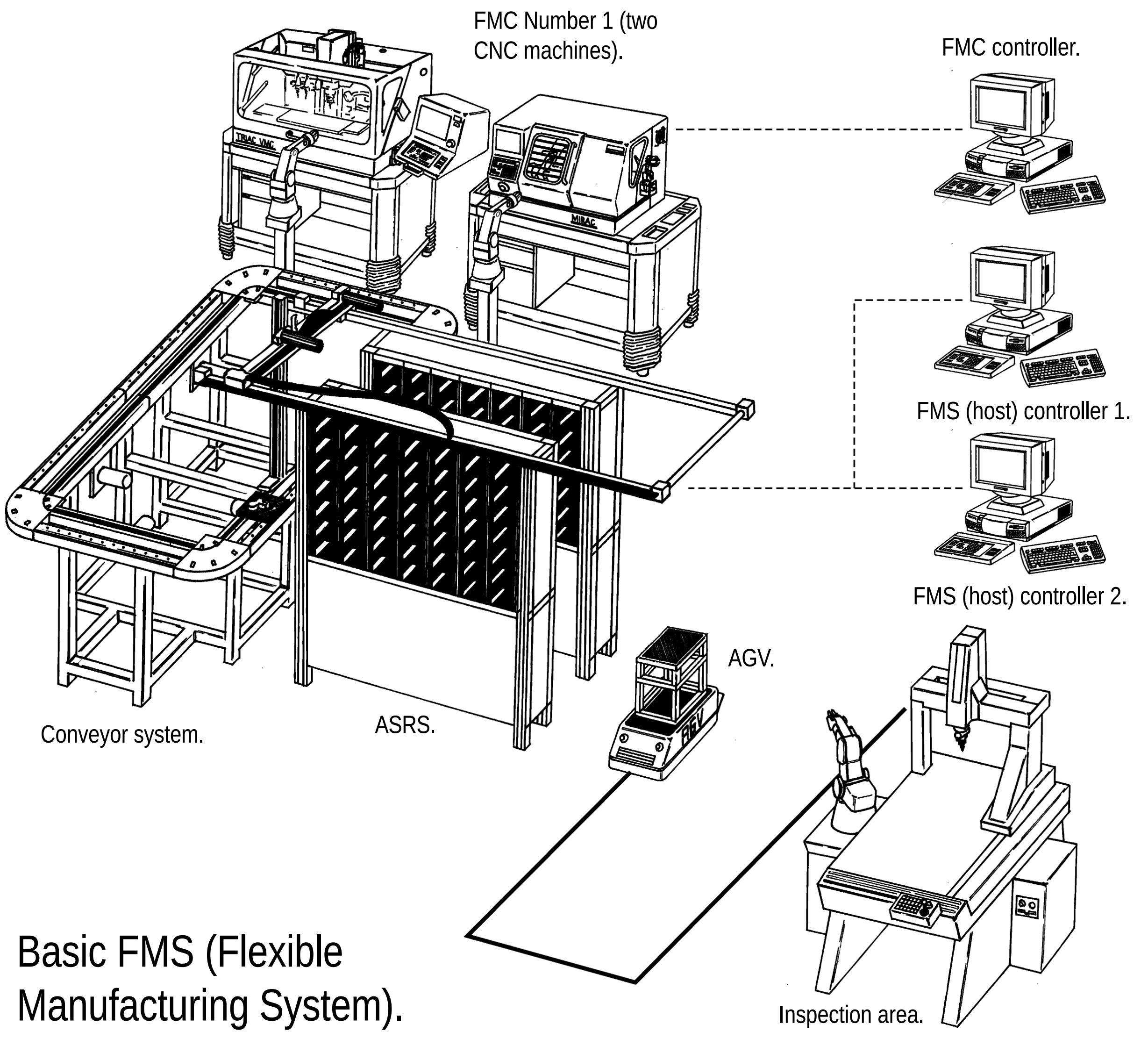


Two Machine FMC.

Flexible Manufacturing Systems (FMS)

A number of flexible manufacturing cells can be combined with other pieces of equipment to form a production line, called a flexible manufacturing system, or FMS.

The system is very flexible, since it builds or reduces in size according to how many FMCs or other pieces of equipment are needed. The FMS shown would be ideal for a small company, since only one FMC is shown (it comprises of one CNC milling machine and one CNC lathe), although further FMCs could be added as the company expands.



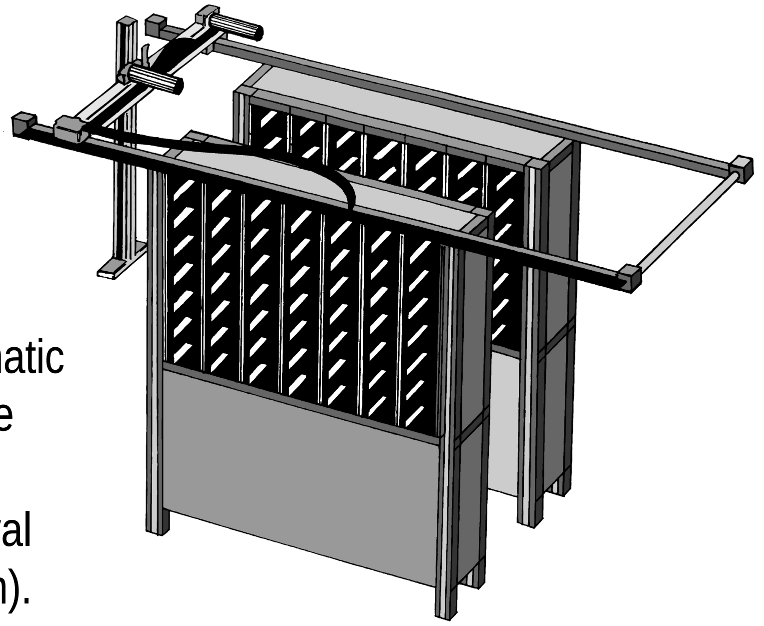
Basic FMS (Flexible Manufacturing System).

What does each part of the FMS do?

The flexible manufacturing system (FMS) shown on the last page contains many other pieces of equipment, as well as the flexible manufacturing cells (FMCs). All these different units work together to form the production, or assembly line. To keep track of what each particular unit is doing, a number of computers, called Host Controllers, are used to monitor the FMS to ensure everything runs efficiently. These are in addition to the Cell Controller computers for each FMC.

A component which needs working upon is lifted out of one of the storage bays on the Automatic Storage and Retrieval System (called the ASRS) and onto a conveyor system. Each component usually sits on its own platform, called a pallet.

The conveyor moves the pallet to the correct FMC, where a robot lifts the component onto one of the CNC machines. The same robot may move the component a number of times between all the CNC machines in its FMC, until all its work has been completed. The robot will then move the component back onto the conveyor, which in turn moves it to the next FMC unit, if fitted, for the next stages of work.



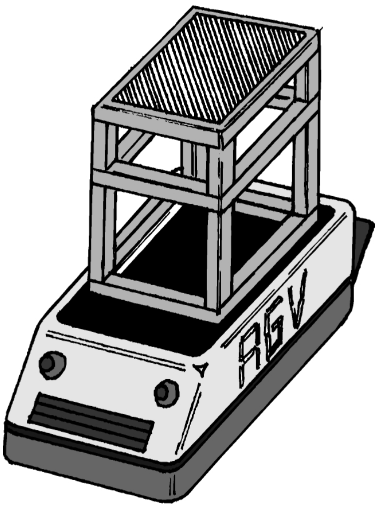
ASRS (Automatic Storage and Retrieval System).

What does each part of the FMS do?

Once the component has travelled through every FMC required, it is taken back to the ASRS and loaded onto an Automatically Guided Vehicle, or AGV. This is used to transport the component over longer distances, perhaps even into a different building. The AGV works by following a set route, which is marked out on the factory floor by lines or wires. Many companies use AGVs to transport large components between storage areas, cleaning areas, FMCs and (as shown on the main FMS diagram) inspection areas.

The most famous example of this is the FIAT car assembly plant, where each car moves around the factory being assembled on its own AGV. In the early 1980's when its FMS was introduced, the company advertised its cars by showing them moving round the factory on their AGVs.

AGV (Automatic Guided Vehicle).



Robots

Robots play an essential role in the success of flexible manufacturing systems. Although robots can come in any shape or size, the most recognisable form of robot is one programmed to imitate the action of a human arm.

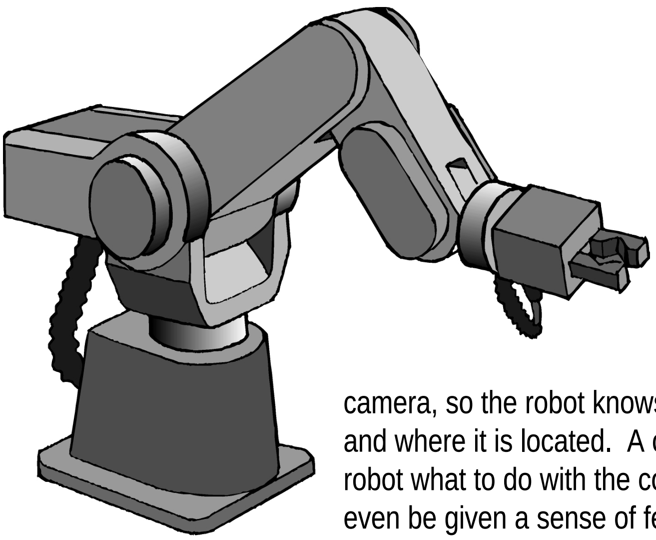
Robots are quite common sights in many industries, such as car production lines. They are used to assemble vehicles, spray paint, weld and unload CNC machines.

Robot advantages:

a) Robots are excellent for highly repetitive, dangerous and hazardous operations.

b) Robots can usually increase the rate of productivity (speed at which products are made) by upto 50% in tasks such as loading and unloading of machines and inspection of parts.

c) Robots work consistently (the same way each time) to position components very accurately, for long periods of time, without stopping.



Robots can also be combined with other systems to provide fully flexible packages. Installing a device similar to a tv camera will allow a robot to see. A vision program is used to interpret the information from the camera, so the robot knows what the component is and where it is located. A control program tells the robot what to do with the component. Robots can even be given a sense of feel through the use of fibre optic cables beaming infrared light.

Robots

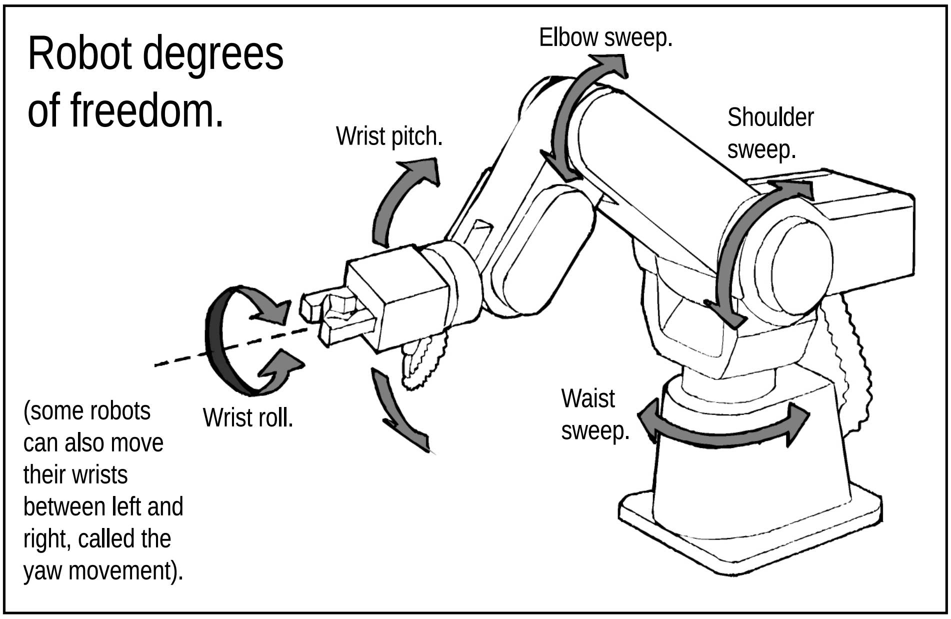
Robots give the impression of mechanical human arms since they offer the same range of movement as the human arm itself, right down to intricate wrist movements. The amount of different movements on a robot are called its "*degrees of freedom*", for example, the robot illustrated has 5 degrees of movement. At the end of the robot wrist, specially designed and shaped grippers are used to hold the objects being moved.

Robots will perform mostly any task but need to be programmed first. Programming a robot can be performed by a variety of methods such as:

a) physically limiting the areas of movement of the robot using stops and switches,

b) moving the robot through a series of movements (which are recorded) either by physically moving it each step, or using a remote hand-held controller,

c) programming the robot with a computer (this is commonly used with CAD/CAM systems).



Glossary

ABSOLUTE .... In absolute programming, zero is the point from which all other dimensions are described.

ADC................................ Analogue-to-digital converter. A device that converts an analogue value to its digital equivalent.

ADDRESS .... An alphabetic character used to define the start of a program word.

AGV................................ Automatic guided vehicle - an unmanned vehicle which moves workpieces around the different sections of a factory production line.

AMPLIFIER .... Also referred to as an amp. A device for increasing the power level of an electrical signal. Incorporated in closed loop servo motor systems.

ANALOGUE .... In numerical control, the term analogue implies a continuously variable quantity.

ARC................................ A portion of a circle.

ASRS .... Automatic storage and retrieval system - a large unit used for keeping workpieces not being worked upon.

ATC .... Automatic tool changer - if fitted to a lathe, cutting tools do not need to be changed manually.

AUTOMATIC .... Automatic Cycle - A mode of control system that continuously runs a cycle or stored program until a program stop or end of program word is read by the controller.

AUXILIARY .... Auxiliary Function - The function of the CNC machine (ie, F, S, T, M codes etc.) other than co-ordinate based commands.

AXIS (AXES) .... The planes of movement for the cutting tool, referred to as X (transverse, at 90° to the Z axis) and Z (longitudinal, parallel with the chuck spindle centreline). Combinations of both axes allow precise co-ordinates to be described.

BATCH .... A factory production method where a small number (ie,50) of identical products are made on machines before using the machines for making a different design of product.

BILLET .... The technical name for the workpiece (the material being cut by the lathe).

BLOCK .... A line of data within a program.

BUFFER STORE .... An area of a factory production line where workpieces are held temporarily, before being sent to the next stage of production.

CAD................................ Computer aided design.

CAM .... Computer aided manufacture.

CANNED CYCLE .... A fixed sequence of operations that are permanently stored in the control system. The fixed cycles can be called and used by a single command in the part program.

CHARACTER .... A number, letter or symbol as entered into a CNC program.

CHIP............................... The pieces of workpiece material, cut away from the billet.

CHUCK .... The area where the billet is clamped.

CIRCULAR .... Circular Interpolation - G code term for a programmed arc movement.

CLOSED LOOP .... A controlling system which does not have any feedback information.

CNC .... Computer Numerical control.

Glossary

COMPARATOR .... A device used in an open loop system which compares two signals and balances any differences between them.

COOLANT .... Liquid used for lubricating and cooling tool tips, allowing higher spindle speeds to be used.

CO-ORDINATES .... Positions or relationships of points. Co-ordinates are usually described using two numbers referring to the (X and Z) axes, e.g. the co-ordinate (23,35) means X axis = +23 units and Z axis = +35 units.

CUTTER .... The actual "cutting" tool which machines the workpiece. Cutters are fixed into "tool holders" on an ATC, identified by the machine as different numbers. Odd numbers are single point cutting tools (eg, roughing tools) and even numbers are multi-point cutting tools (eg, twist drills).

CUTTING SPEED .... The equivalent of the workpiece surface speed at the point being machined.

CYCLE .... A sequence of events or commands.

DAC .... Digital-to-analogue converter. A device that converts an digital value to its analogue equivalent.

DATA .... An order or sequence of numbers and information understood by a computer.

DATUM .... The co-ordinate (point) from which a series of measurements are taken.

DESKTOP TUTOR .... The input control keypad for the machine. Keypad overlays are interchangeable according to the type of control method required.

DIRECTORY .... An area of a disk containing the names and locations of the files it currently holds.

DISK .... A computer information storage device, examples, C: (drive) is usually the computers hard (internal) disk and A: (drive) is usually the floppy (portable 3.5" diskette) disk.

DNC .... Direct Numerical Control referring to the operation of a number of different machines from a single relatively large computer.

DRIVE .... The controller unit for a disk system.

DRY RUN .... A test run of a CNC program to see if it contains any mistakes. The workpiece cannot be cut on a dry run since no parts of the machine tool will move.

DWELL .... A programmed time delay.

ELECTROMAGNET .... A number of wires wrapped round an iron bar. When an electric current is passed through these wires, a magnetic field is generated.

ELEMENTS .... Denford Lathe CAM Designer software elements are different shapes that are put together to make a design.

END OF BLOCK SIGNAL EOB - A symbol or indicator that defines the end of a block of data. The 'pc' equivalent of the 'return' key.

ERROR .... The deviation of an attained value from a desired value.

FEEDBACK .... Information about an object or system being controlled. It allows the potential for any mistakes to be corrected.

FEEDRATE .... The rate (speed), in mm/min or mm/rev, at which the cutting tool advances along the workpiece.

Glossary

FILE................................ An arrangement of instructions or information, usually referring to work or control settings.

FLUTE............................ The grooved channels spiralling up the sides of a multi-point cutter which allow waste material to be removed as the workpiece is cut.

FMC .... Flexible manufacturing cell - a number of CNC machines linked together by a robot and under the control of a computer (the cell controller).

FMS................................ Flexible manufacturing system - a number of FMCs and other pieces of equipment linked together to form a production line, all controlled by computers (called host controllers).

G CODE .... The programming language relating to commands about slide movement of the lathe. G-Codes are the series of letters and numbers that make up the language used by CNC machinery.

HARDWARE .... Equipment such as the machine tool, the controller, or the computer.

HEADSTOCK .... End area of the lathe containing the spindle, drive motor and chuck.

HOME .... Command signal instructing the datuming of machine axes.

IDEAL SPEED................ A number, measured in mm/min, relating to the most efficient speed one blade can cut a specific type of material in a straight line.

INCH CONTROL .... See Jog Control.

INCREMENTAL .... Incremental programming uses co-ordinate movements that are related from the previous programmed position. Signs are used to indicate the direction of movement.

INDEXABLE .... Tool tip that can be reversed until all its cutting edges are worn, then thrown away and replaced by another.

ISO .... International Standards Organisation.

JIGS & FIXTURES .... Devices used (or especially designed and made) to hold workpieces so the correct areas can be accurately and easily reached and cut.

JOG CONTROL .... Jog (or Inch) Control - Manual positioning of the machine using small fixed length movements per axis key press (called Jog Steps) or user defined length movements through holding the axis key down (called Continuous).

M CODE .... The programming language relating to commands about specific (non slide movement) tasks on the lathe.

MACHINE BED .... Area of the lathe running parallel to the spindle.

MACHINE CODE .... The code obeyed by a computer or microprocessor system with no need for further translation.

MACHINE DATUM .... The Machine Datum (or reference) point is the co-ordinate set automatically by the machine so it can relate the position of its cutting tool to both its slides (axes). The machine must be instructed to search for this point when it is first switched or the power supply is interrupted during a machining session. When set, it is the point from which the machine takes all its measurements (displayed on screen).

MASS .... A factory production method where a very large number (ie,100,000) of identical products are made, usually on purpose build fully automatic machines.

Glossary

MCU .... Machine Control Unit - the computer hardware and software used to control all the operations of the CNC machine.

MDI................................. Manual Data Input - entering data into a CNC computer by hand.

MODAL .... Modal codes entered into the controller by a CNC program are retained until changed by a code from the same Modal group or cancelled.

NC .... Numerical control - a system which allows numerical data (from a computer) to be processed into functions and movement commands.

OFF-LINE .... Working away from the machine tool, for example, entering program data or program proving on a computer not linked to the machine tool.

ONE-OFF .... A production method where only one product or design is made (sometimes called a prototype).

OPEN LOOP .... A controlling system which contains feedback information.

PART .... The designed component manufactured, or being manufactured, from the billet.

PC .... Personal computer.

PERIPHERY .... The outside edge of an object.

PROGRAM..................... A systematic arrangements of instructions or information to suit a piece of equipment.

PULSE .... A pattern of variation of a quantity, such as voltage or current.

QUILL............................. The moveable front nose of the tailstock which can extend or retract to hold parts.

RAKE .... Elevational angle of the tool tip.

RAM .... Random access memory. Data can be temporarily written to this memory.

RAPID TRAVERSE .... Fast movement of the cutting tool through the two machine axes when not cutting any material.

REFERENCE POINTS... The machine has two reference points (hidden microswitches) used in setting the limits of movement for its slides (axes).

REPEATABILITY............ The accuracy of a system to attain identical positions or values.

ROM .... Read only memory. Data cannot be written to the device, only read (eg, CD-ROM).

ROTOR .... The part of a motor (with fixed magnets) that spins.

RPM .... Revolutions per minute (rev/min) - a measure of spindle speed.

SADDLE......................... The toolpost carrier fitted to the transverse slide.

SERVO........................... An continuously variable electric motor incorporating facility for feedback information.

SIGNATURE .... Set of numbers describing a tool tips shape and angles

SIMULATION .... A command within the lathe controlling software which allows the work performed by a program to be viewed as a plan or 3d view.

SLIDES .... Another name for the two machine axes - see axis.

SOFTWARE................... Programs, tool lists, sequence of instructions etc.....

SPINDLE........................ The longitudinal shaft connecting the chuck with the drive motor.

Glossary

SPINDLE SPEED........... The rate at which the spindle, its attached chuck and the workpiece rotates.

STATOR .... The part of a motor (with a series of electromagnets) that remains still.

STEPPER MOTOR .... An electric motor which rotates in equally spaced units (called steps) each time a voltage signal is sent.

TACHOMETER .... A speed measuring instrument generally used to determine the revolutions per minute.

TAILSTOCK .... Area where drills are held and long billets are supported, at the opposite end of the machine to the headstock.

TOOL OFFSET .... When machining, allowances must be made for the different length and size of tools being used. Each tool used on a part requires its individual tool offsets to be registered.

TOOL PATH .... Tool Path shows the route followed by the CNC lathe when cutting the design.

TOOLPOST..................... The holder for the various cutting tool profiles. Most automatic toolposts are located above/behind the spindle centreline. Most manual toolposts are located below/in front of the spindle centreline.

TORQUE .... Radial twisting force.

TRANSDUCER .... A device used to monitor the amount of rotations taken by an object (ie, the number of times a motor turns round). This allows the precise positions reached by objects (such as the slides) to be described.

TRAVERSE .... Movement of the cutting tool through the two machine axes between cutting settings.

WORD .... Composed from an address letter and a number to describe a CNC setting, operation or function.

WORK (WORKPIECE)... The actual material being turned. The work is sometimes referred to as the 'billet'.

WORK AREA .... The Work Area represents the surfaces available for machining.

ZERO OFFSET .... Sometimes referred to as zero shift. A means of shifting the co-ordinate zero point from a fixed known zero point.

Notes