



MICRO CNC

Manufacturing Engineering: CNC machining



CP7449

MATRIX

www.matrixsl.com

Copyright© 2017 Matrix Technology Solutions Ltd.

	Page
Introduction to Computer Aided Design & Manufacture	4
What is CAD/CAM?	4
Introduction to Computer Numerical Control	5
What are CNC Machines?	5
Parts of a CNC Control System	5
What is a Lathe?	6
What is a Milling Machine?	8
Introduction to Open & Closed Loop Systems	10
What is a Control System?	10
What is Open Loop Control?	10
What is a Closed Loop System?	11
Introduction to Part Programming & Program Efficiency	13
What is Part Programming?	13
Part Programming	13
Manual Programming	13
Conversational Programming	13
CAM Programming	14
Introduction to CNC Processes for Turning & Milling	15
CNC Turning	15
CNC Milling	15
Machine Turning	16
Cutting Tools used for Turning	16
Types of Cutting Tool	17
Typical Turning Operations	17
Machine Milling	19
Introduction to Parameters	20
What are Tooling Parameters?	20
Introduction to the Development of a CNC Part Programme	22
Writing a Part Program - What's Involved?	22
Reference Points & Coordinates	22
Machine Origin	22
Program Origin	22
Part Origin	22
Axis Designation	23
Lathe Axis Designation	23
Milling Machine Axis Designation	25

	Page
Introduction to CAD Software	25
What is CAD Software?	25
Example - Designing a Part in CAD Software	27
Sample Part - CAD Design Example	27
Introduction to CAM Software	31
Example - Making a Part with the MicroCNC Milling Machine	33
4-Axis Milling Example	33
Worksheet 1 Lathe - Stepped Cylindrical Part	42
Worksheet 2 Lathe - Tapered Cylindrical Part	43
Worksheet 3 Lathe - Complex Cylindrical Part	44
Worksheet 4 3-Axis Milling Machine - Simple Jigsaw47	
Worksheet 5 3-Axis Milling Machine - Interlocking Jigsaw 1	47
Worksheet 6 3-Axis Milling Machine - Interlocking Jigsaw 2	49
Worksheet 7 4-Axis Milling Machine - Pen Holder	50



WHAT IS CAD/CAM

Computer Aided Design (CAD) and Computer Aided Manufacturing (CAM) are an integral part of modern product development.

CAD/CAM Systems streamline the Design and Manufacturing processes using Computers to:

- Create conceptual designs from 2D sketches through to fully rendered 3D models
- To control a range of machines such as lathes and mills to produce parts

Computer Aided Design (CAD) systems are used to help people design ideas, build models and visualise prototypes. It was originally used to help people with technical drawings but it has developed into a process to help with the design of products in industry and in schools.

Since the beginning of the industrial revolution the way products have been made has developed significantly and the use of machines to replace 'Hand Made' parts has increased significantly. One of the most dramatic changes was the introduction of Computer Aided Manufacture (CAM). CAM systems use computers to control machines and assist with the manufacture of products. With the introduction of CAD/CAM systems many industries have been able to set up highly automated factories, using automated machines and robots, to manufacture parts and products to very high standards.

Some of the advantages of CAD/CAM systems include:

- Easier data storage and retrieval
- Repeatability & precision
- Flexibility
- Automation capability

Activity

Briefly describe how these advantages relate to CAD/CAM systems. Give an example for each advantage for both the CAD system and the CAM system.

Introduction to Computer Numerical Control



WHAT IS CNC?

Computer Numerical Control (CNC) is a technique of controlling the actions of a machine through the use of coded instructions.

Many manufacturing sectors use CNC Machines tools to perform a range of manufacturing operations such as:

- Turning
- Milling
- Cutting
- Routing

What are CNC Machines?

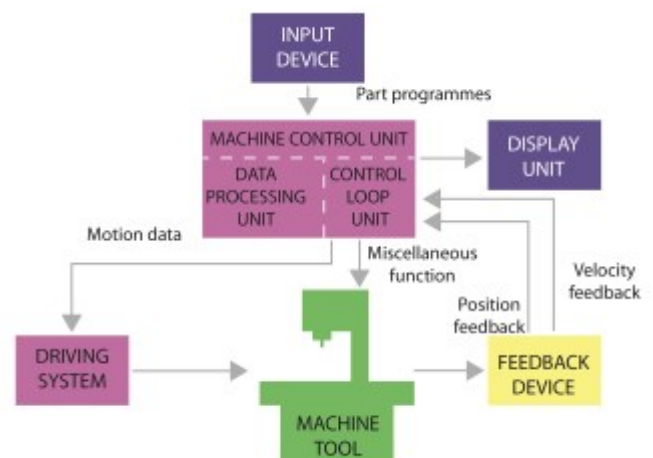
Modern CNC machines are programmable, automated tools used for machining components from a wide range of materials; they perform many different functions from turning, milling, cutting, routing, etc. They first appeared in the 1950s and use numerical control techniques to control the actions of the machine by input of instructions in the form of a code. Today CNC machines are used for many specialised applications covering virtually all manufacturing processes.

Some of these processes include:

- Lathes for all turning and cylindrical contouring operations.
- Milling Machines. Including simple 3-Axis Machines through to 6-Axis Machines. Milling machines are used for creating a wide range of complex components.
- Routers and engravers for normally light work, producing grooves and profiles.
- Laser Cutters for engraving and cutting sheet materials.

Parts of CNC Control Systems

- Input Device – such as a computer, used to enter part program.
- Machine Control Unit – functions include: read/decode part program, generate axis motion commands, receive/interpret feedback, etc.
- Drive control – controls position of tools or workpiece.
- Display – provides information about the operation being performed.
- Feedback – informs MCU on position, velocity, orientation, etc. of workpiece.
- Machine Tool – Lathe, Drill, Milling Machine etc.





WHAT IS A LATHE?

A lathe is a tool that rotates the workpiece about a central axis to perform operations such as turning, cutting, drilling, etc. with the machined component being symmetrical about the axis of rotation.

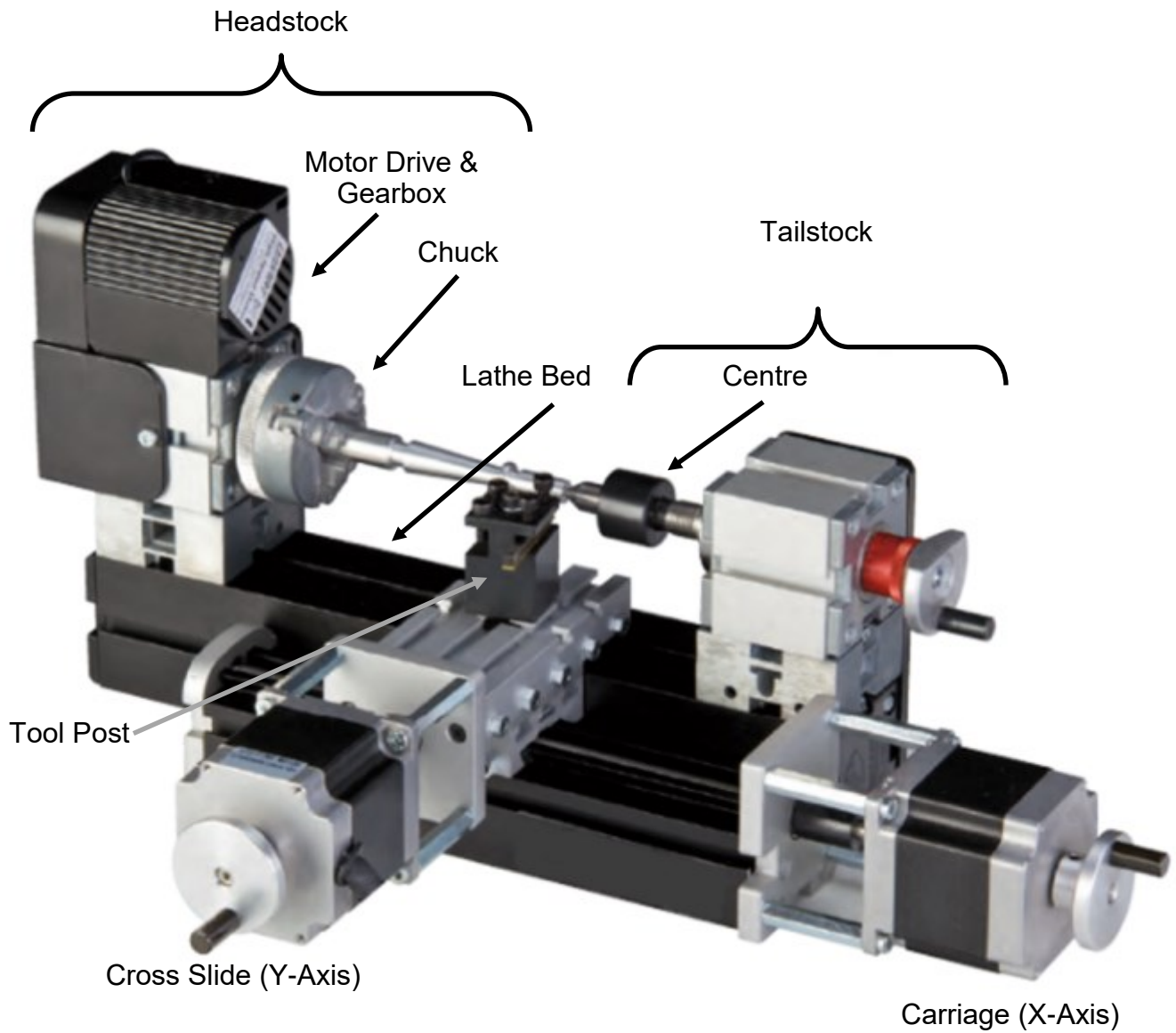
On lathes it is the cutting tool that is moved in relation to the workpiece being machined.

On a CNC lathe the position of the cutting tool is controlled by a computer.

The main parts of a lathe are:

- THE BED, which is (almost always) a horizontal beam (although CNC lathes commonly have an inclined or vertical beam for a bed to ensure that swarf, or chips, fall free of the bed).
- THE HEADSTOCK contains high-precision spinning bearings. Rotating within the bearings is a horizontal axle parallel to the bed.
- THE SPINDLE rotates within the bearing and is used to fix work-holding accessories such as the CHUCK.
- The spindle is driven by a GEAR DRIVE connected to a POWER SOURCE. In most modern lathes this power source is an integral ELECTRIC MOTOR controlled by the speed controls.
- THE TAILSTOCK can be positioned along the bed and contains a barrel, which does not rotate, but can slide in and out parallel to the axis. This holds the CENTRE, which is used to support a workpiece, or to hold drill bits for drilling axial holes in the workpiece.
- THE CARRIAGE forms the X-axis and is used to move the cutting tool along the workpiece at a right angle to the bed.
- THE CROSS SLIDE sits on top of the carriage and forms the Y-axis allowing the cutting tool to be moved in and out of the workpiece.
- THE TOOL POST can be mounted to the cross and allows the cutting tool to be positioned at any angle to the workpiece to enable different cutting processes to be performed.
- **SEE OVER THE PAGE FOR A LABELLED IMAGE**

The image below shows the Matrix Micro CNC Lathe with the main parts labelled.





WHAT IS A MILLING MACHINE?

Milling machines are tools designed to machine metal, wood, and other solid materials. Often automated, milling machines can be positioned in either a vertical or horizontal orientation.

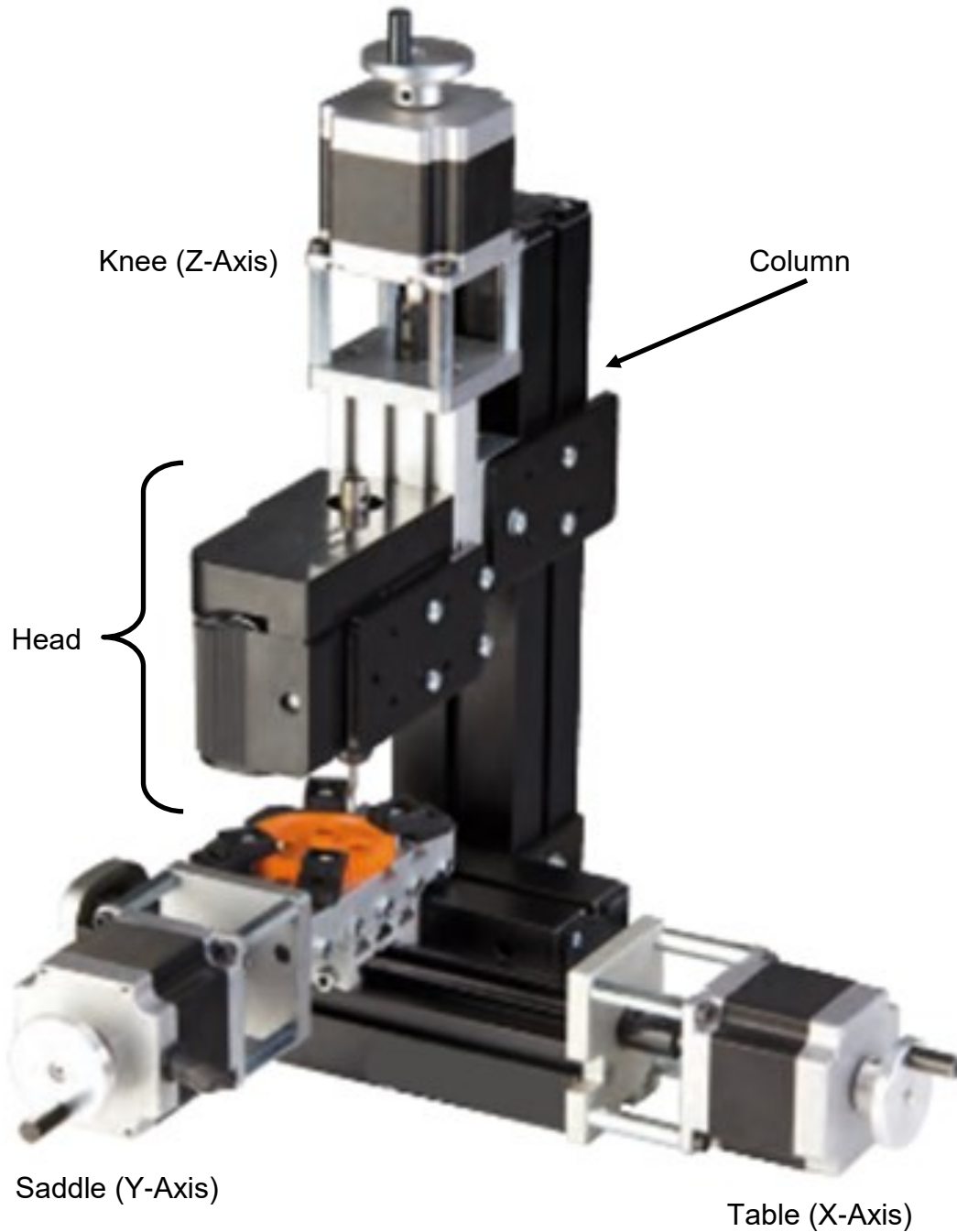
On milling machines both the workpiece and cutting tools are moved along different axes; with the cutting tool rotating to remove material from the work piece.

The main parts of a milling machine are:

- THE BASE gives support and rigidity to the machine and also acts as a reservoir for the cutting fluids.
- THE COLUMN face is a precision machined section used to support and guide the knee when it is moved vertically.
- THE KNEE is attached to the column face and may be moved vertically on the column face along the Z-axis.
- THE SADDLE is placed between the knee and the worktable and moves along the Y-axis.
- THE TABLE rests on guideways in the saddle and travels longitudinally in a horizontal plane along the X-axis. It supports the vice and the work.
- THE HEAD is the upper section of the vertical milling machine and consists of a spindle, driving motor and other controlling mechanism.
- **SEE OVER THE PAGE FOR A LABELLED IMAGE**



The image below shows the Matrix Micro CNC Vertical Milling Machine with the main parts labelled.



Introduction to Open & Closed Loop Systems



WHAT IS A CONTROL SYSTEM?

A CNC Machine Tool requires a Control System to manage the operation of motor drives, these control both the position and velocity of the machines axes.

Each axis is driven independently following the instructions from a prepared program containing coded instructions to drive motor functions, tool speed, feed rate etc.

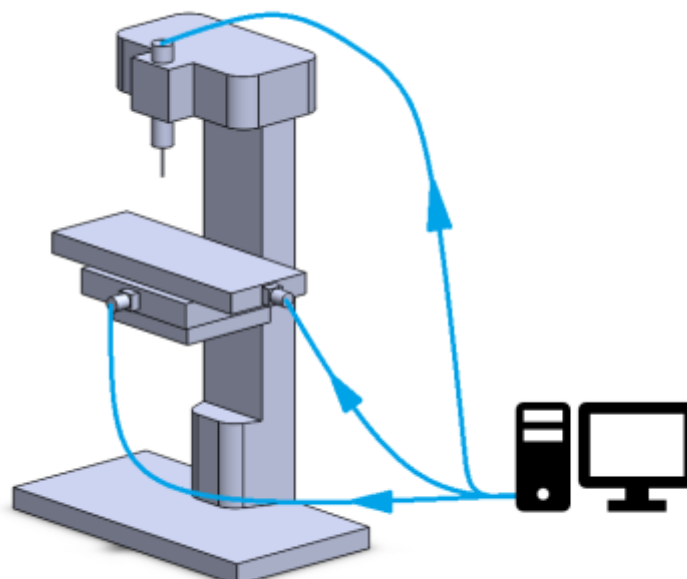
There are two ways to activate the servo drives: the open-loop system and the closed-loop system.

What is Open Loop?

Open-loop systems are set up to achieve desired results; however, there's no way of checking if the results have been achieved.

The Machine Control Unit (MCU) receives programmed instructions, generally in the form of G-code, through an input device or from a CAM program. These instructions are then converted to electrical signals to control servo motors of each axis.

The main drawback of an open-loop system is that there is no feedback system in place to check whether the program position and velocity has been achieved. If the system performance is affected by load, temperature, etc. the actual output could be different from the desired output.



Open-Loop System



WHAT IS A CONTROL SYSTEM?

A CNC Machine Tool requires a Control System to manage the operation of motor drives, these control both the position and velocity of the machines axes.

Each axis is driven independently following the instructions from a prepared program containing coded instructions to drive motor functions, tool speed, feed rate etc.

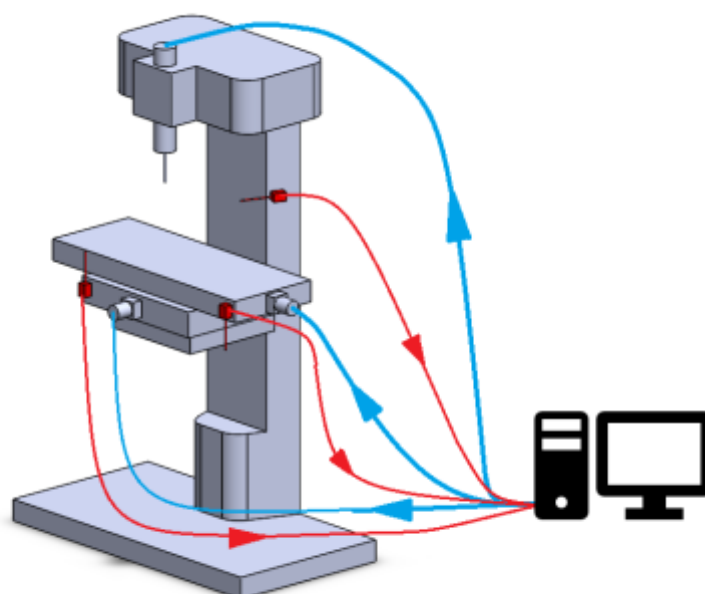
There are two ways to activate the servo drives: the open-loop system and the closed-loop system.

What is Closed Loop?

Closed-loop systems are able to correct errors in position and velocity in order to meet target results. This is achieved through feedback from sensors.

Again the MCU receives programmed instructions that are converted to electrical signals to control servo motors of each axis as with an open loop system. However, with a closed loop system the MCU receives additional information from either Analogue or Digital sensors connected to each axis. These sensors monitor the position and velocity at the output and adjust the process accordingly.

Closed-loop systems are very powerful and accurate because they are capable of monitoring operating conditions through feedback subsystems and automatically compensating for any variations in real-time. Most modern closed-loop CNC systems are able to provide very close resolution of 0.00254mm.



Closed-Loop System

Activity

Investigate how feedback is achieved through the use of linear encoders and resolvers.





WHAT IS PART PROGRAMMING?

Machining a component involves the relative movement between a cutting tool and a workpiece. In machine tools this is accomplished by either moving the tool with respect to workpiece or vice versa.

In order to define relative motion of two objects, reference directions are required to be defined. These reference directions depend on the type of machine tool and are defined by considering an imaginary coordinate system on the machine tool. A program defining motion of tool / workpiece in this coordinate system is known as a part program.

Part Programming

In general terms there are three methods of Part Programming, Manual, Conversational, and Computer Aided Manufacturing (CAM); each method has advantages and disadvantages depending on the type, and complexity, of the part being made, and the type of tooling to be used to manufacture a part.

Manual programming is usually employed when the parts to be produced are simple, or when new programs are not required very often. Manual programming is also sometimes used where there is a need for the CNC program to execute as efficiently as possible. In Manual Programming the programmer will write the program in the same language that the CNC machine will use.

Advantages

- Can be very efficient if/when correctly formatted.

Disadvantages

- Can be error prone
- Can be tedious to write programmes

Conversational programming is usually employed in busy machine shops when programs must be written while the tooling machine is in use, being set up, or during down time for repair etc. Conversational programming is beneficial where companies have lots of repeat business, small production runs, or production runs have very short cycle times.

Advantages

- Can quickly create a CNC program.
- Programs can be entered without any mathematical knowledge.
- Removes the tediousness of manual programming.

Disadvantages

- Cannot be programmed while the machine is in use.

Worksheet 1

Electronic control

CAM programming methods are usually employed when there are a variety of machines to program, there is lots of new business (many programs to create), and/or jobs are quite complex (making it difficult to manually program).

Advantages

- CNC program is generated automatically.
- Programs are generally more efficient than conversational programmes.
- Removes the tediousness of manual programming.

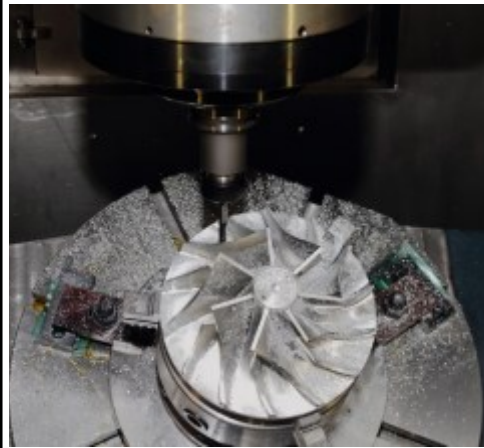
Disadvantages

- CAM systems are costly (Not as much today as a few years ago)



Introduction to

CNC Processes for Turning & Milling



CNC milling and turning machines are found in precision engineering shops across the country and are used to manufacture all kinds of components. However, a range of CNC machines are required to produce the different geometries and features required e.g. cylinders, flat surfaces, rebates, bores, and tapped holes, etc.

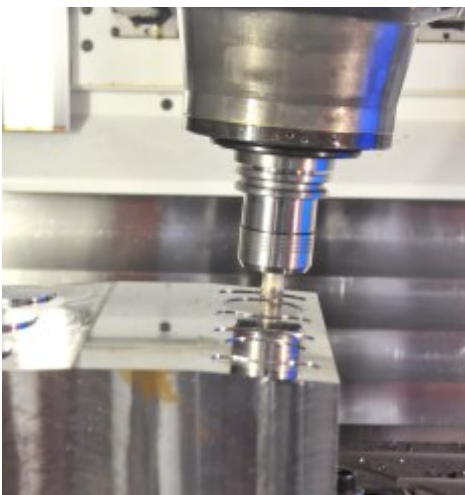
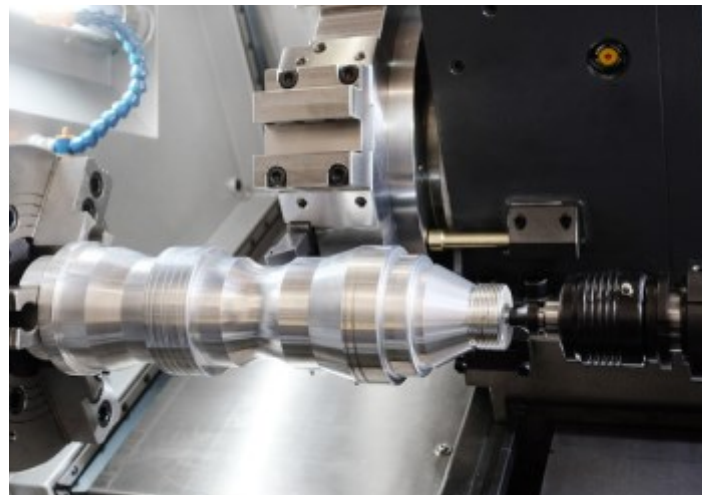
To ensure parts are produced in the most efficient way possible it is important to understand the limits and capabilities of the different types of CNC machine. This worksheet looks at the difference between two of the major processes; CNC Turning and CNC Milling.

CNC Turning

CNC Turning uses a cutting tool that is single-pointed, and that is inserted parallel to the material that is to be cut. The workpiece is rotated at varying speeds, the cutting tool being able to traverse the two axes of motion (X and Y) to produce cuts that are off exact diameters and depths.

CNC turning is used for creating cylindrical shapes such as those that you find in drive shafts or bolts. A CNC turning machine can create a perfect circular outer edge to a workpiece.

CNC turning can also be used on the inside edges of the material where it is used to create a circular, or tubular, cavity.



CNC Milling

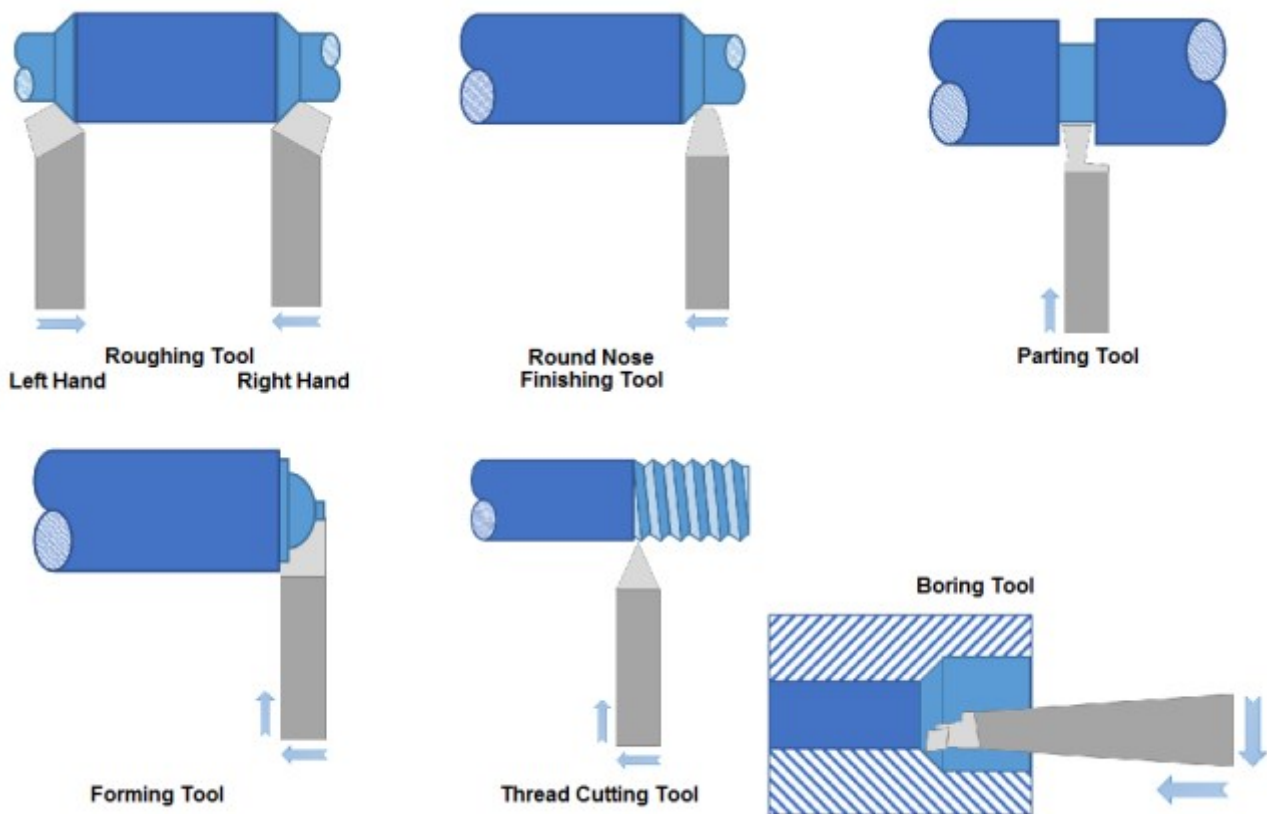
CNC Milling uses a multi-pointed cutting tool that is rotated, and moved along the workpiece, at varying speeds; the workpiece is moved along the X and Y axes, while the cutting head is moved in the Z-axis. Milling machines are therefore used to create flat components, or components with complex shape geometries.

CNC milling is used to create complicated shapes of irregular, or non-symmetrical dimensions such as engine blocks. A CNC milling machine can create perfectly flat faces, grooves, holes, recesses, and tapered features. They can work on both external and internal faces of a component, the only limitation being the positioning and access of the cutting tool to the required face to be machined.

Machine Turning

Turning is the process of generating machined components by progressively removing small amounts of material from a rotating workpiece by moving a single point cutting tool linearly along the length of the workpiece. Usually 'Turning' refers to the machining the external surface of the workpiece; whereas 'Boring' applies to machining of internal surfaces, and 'Facing' refers to cutting the end surface of the material i.e. a surface perpendicular to the rotating axis.

Cutting tools used for turning



Roughing tools are used to quickly remove material and get the stock to 'rough' dimensions before finishing to the exact size with a finishing tool.

Roughing and finishing tools can be right handed, left handed, or round nosed. A right hand tool will cut from right to left i.e. towards the headstock, a left hand tool will cut from left to right i.e. towards the tail stock, while a round nosed tool will cut in both directions.

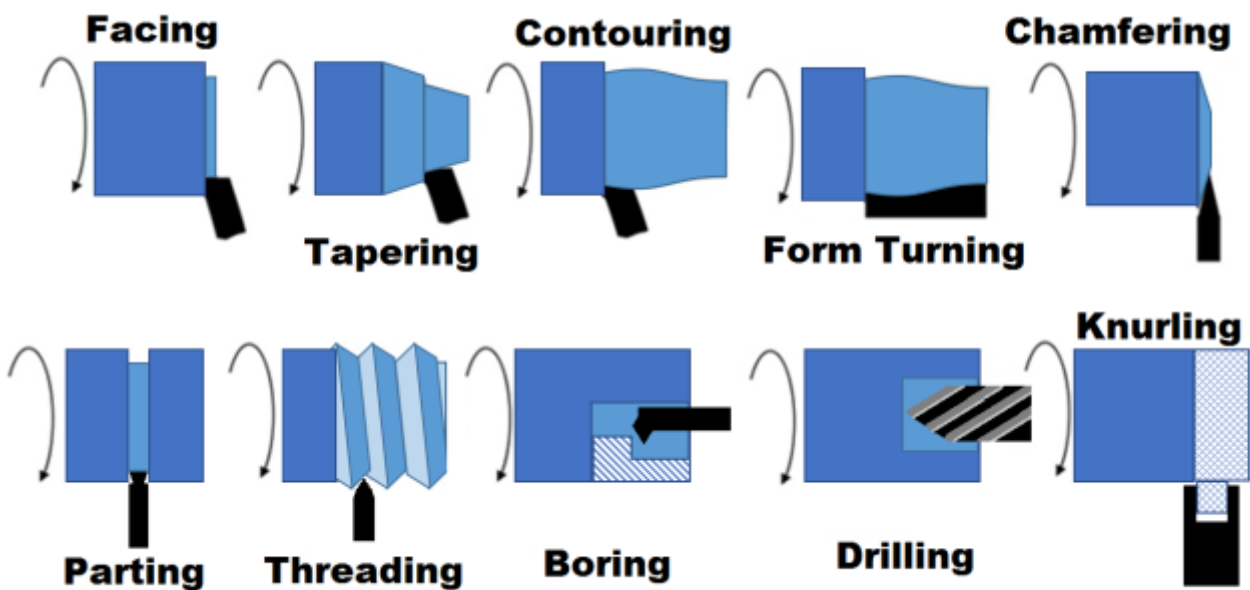
Other 'Specialist' tools, such as forming, thread cutting, and boring tools are also available to perform specific cutting operations.

Types of Cutting Tool

Cutting tools come in three main types

- Forged – made from a single piece of steel; supplied as blanks, these cutting tools are ground to shape and sharpened prior to use. Typically used on ‘soft’ non-ferrous metals.
- Carbide tipped – made from a cast steel shaft with a carbide steel tip
- Replaceable tip

Typical ‘Turning’ operations performed on a lathe



Activity

Use the resources available to you to briefly describe the action of the cutting tool movement in relation to the following operations

- Facing

- Taper Turning

- Contour Turning

- Chamfering

- Parting

- Boring

- Briefly describe the construction and use of a form cutting tool, a thread cutting tool, and a knurling tool.

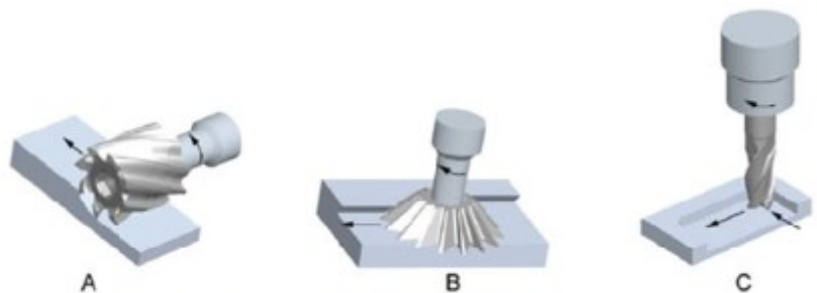
Machine Milling

Milling is a process of generating machined components by progressively removing small amounts of material, or stock, from a workpiece. The workpiece is moved at a relatively slow rate, or feed, along the X and Y axes while the milling cutter rotating at a comparatively high speed is moved along the Z-Axis.

The characteristic feature of the milling process is that each milling cutter tooth removes a small amount of material in the form of small individual chips.

Types of Milling Cutters

The three basic milling operations are shown opposite: (A) slab milling, (B) face milling and (C) end milling.

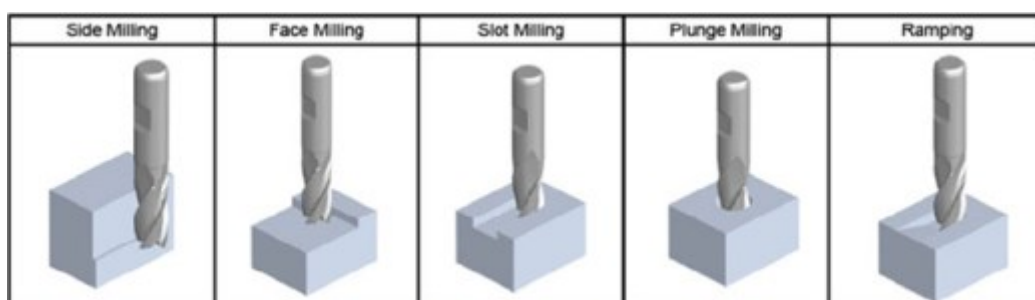


* Diagram images reference: <http://www.nestools.com/technical-information-milling.php>

In slab milling the axis of cutting tool rotates parallel to the surface of the stock material being machined. The cutting tool has a number of teeth around its circumference, these can be straight or helical cutting at an orthogonal or oblique angle respectively.

In face milling and end milling, the cutting tools are mounted in a spindle with an axis of rotation vertical in relation to the surface of the stock material being machined. In face milling the milled surface results from the action of cutting teeth located on the edge and face of the cutting tool; whereas, in end milling the cutting teeth are located on both the end face of the cutter and the periphery of the cutting tool, in addition, the cutting tool can also be tilted to machine tapered surfaces.

There are many different operations that come under the term 'end milling'. For each different operation there is a different MRR (Material Removal Rate) that increases with the engagement section of the cutter on the workpiece.



* Diagram images reference: <http://www.nestools.com/technical-information-milling.php>



WHAT ARE TOOLING PARAMETERS?

Machining involves the relatively simple process of removing material from a piece of stock material in order to create a desired part.

Removal of material is achieved through the relative movement of the work piece and tool, either by rotating the workpiece (as in a lathe) or by rotating the tool (as in a milling machine); the tools used can be single point, multiple point, or abrasive tools.

Within this simple process lie numerous parameters that affect many aspects of the machining process such as the time taken to complete a job, the quality and accuracy of the finish, the life of the tool, the cost of production, etc.

These are collectively known as Tooling Parameters.

Many of the parameters involved in these processes are interrelated and changing one value can have a huge effect on another parameter. The following list gives a brief description of the most common machine setup and tooling parameters required.

1. **Idle speed:-** Generally this is the speed of the cutting tool when the machine is not cutting but is still in a running state.
2. **Processing speed:-** This is the effective speed during specific machining operations.
3. **Default Speed:-** The default running speed of the machine unless operating under any specific processing speed.
4. **Material machinability:-** This is how easy or difficult it is to cut. The material's hardness is the main factor here; typically a soft material is easier to cut than a harder material, however, other properties of a material such as ductility, strength, and toughness also have an influence.
5. **Cutting Tool Material:-** In metal-cutting, High Speed steel and Carbide are two major tool materials widely used with ceramic tools and Cubic Boron Nitride (CBN) being used for machining very tough and hard materials. The material used determines the tool's hardness, strength, wear resistance, etc. these parameters determine how fast the tool can cut efficiently on a job.
6. **Cutting speed:-** Cutting speed is the relative speed at which the tool cuts through the stock material. It is normally expressed in meters per minute and higher cutting speeds generally allow for high rates of production. For every type of material tool there is an ideal cutting speed; these are usually supplied by the tool manufacturers in the form of guidelines.

- 7 Spindle Speed:- Spindle speed is expressed in RPM (revolutions per minute). It is derived from the cutting speed and the work diameter cut for turning on a lathe, or the cutting speed and tool diameter for milling, drilling, etc. If V is the cutting speed and D is the diameter of cutting, then Spindle speed $N = V / (\pi \times D)$
- 8 Depth of cut:- This indicates how much a cutting tool digs into the stock material (in mm) to remove material in the current pass.
- 9 Feed Rate:- The relative speed at which the tool is linearly traversed over the workpiece to remove the material. In case of rotating tools with multiple cutting teeth, such as mill cutting tool, the first parameter used is the feed rate calculated in terms of "feed per tooth," expressed in millimetres (mm/tooth). The second parameter used is then the "feed per revolution" (mm/rev). This is effectively feed per tooth multiplied by the number of teeth in the cutter. For lathe operations, the feed rate is effectively the feed per revolution that states how much a tool advances in one revolution of workpiece.
- 10 Tool Geometry:- Different cutting angles, such as rake angle, clearance angles, relief angle, approach angle, etc. determine how effectively the cutting tool digs into the component to remove material. These parameters effect the cutting speed and feed rate of tools.
- 11 Coolant:- This is used to take away the heat produced in metal-cutting, it also acts as a lubricant in cutting hard materials to reduce tool wear. Coolants can range from cutting oils, water-soluble oils, oil-water spray, etc. and again effect cutting speeds and feed rate.
- 12 Machine/ Spindle Power:- Adequate power should be available to drive the spindle and axes' motors to enable the cutting tool to efficiently remove the material. The power required for cutting is based on the material removal rate and is generally expressed in cubic centimetres per minute, which depends on work material, tool material, the cutting speed, depth of cut, and feed rate.

```

N1 G00 X-35 Z-5
N2 G01 X10 Z0 F100
N3 G01 X0 Z-30 F100
N4 G01 X10 Z0 F100
N5 G01 X0 Z-30 F100
N6 G01 X10 Z-0 F100
N7 G01 X0 Z-40 F100
N8 G00 X-35 Z-5
M30
    
```

WRITING A PART PROGRAM—WHAT’S INVOLVED?

In previous worksheets part programs were discussed as the method of defining motion of the cutting tool and the work piece in order to machine a component from a piece of stock.

This work sheet will provide an overview of the processes involved in writing a part program

Reference Point & Coordinates

- **Machine Origin**

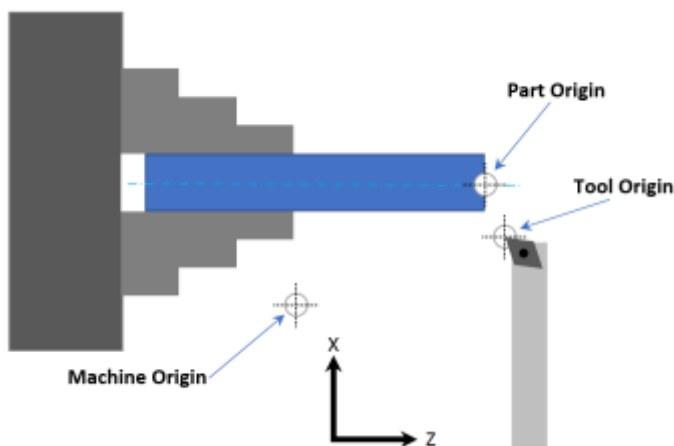
The machine origin is a fixed position set by manufacturer; on some machines this can be changed but generally it cannot be changed. Movement of the cutting tool is measured from this position. The controller always remembers cutting tool distance from the machine origin.

- **Program Origin**

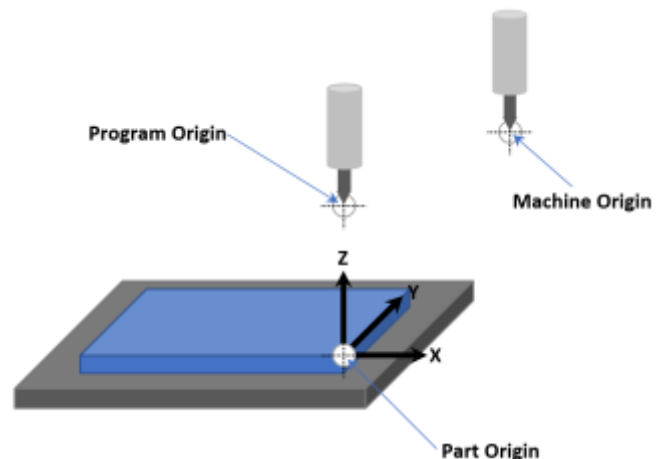
Generally referred to as home position. The program origin is the point from where the cutting tool motion starts when executing a program; the cutting tool returns to this position at the end of the cycle. This can be any point within the workspace of the cutting tool with the exception of the space occupied by the workpiece. For CNC lathes this is generally the position where tool change is carried out.

- **Part Origin**

The part origin can be set at any point within the workspace of the cutting tool and needs to be defined for each new setup. The part origin is the point at which material cutting begins, this is generally along one edge of the stock material, the part to be machined existing within this space. Establishing the part origin is also referred to as setting the datum, zero shift, work shift, or floating zero.



Lathe Origin Points (2-Axes)

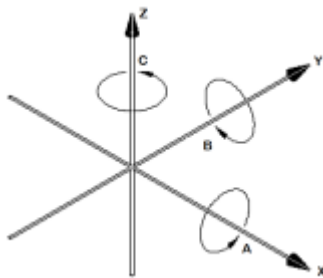


Milling Machine Origin Points (3-Axes)

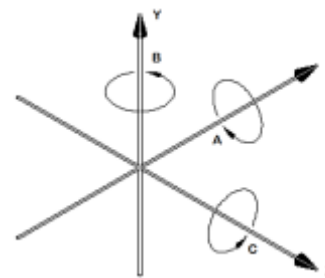
Axis Designation

A workpiece being machined potentially has six degrees of freedom with respect to a Cartesian coordinate system. Three of them are linear movements (X, Y, Z) and other three are rotary (A, B, C). Machining of simple part does not require all degrees of freedom. However, not all machining operations require all six.

Vertical Machine Coordinates

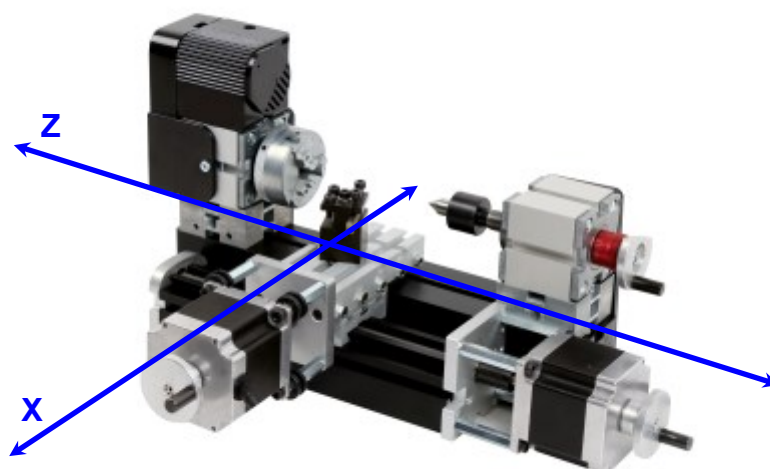


Horizontal Machine Coordinates



Lathe Axis Designation

A lathe has two degrees of freedom (X, Y). The cutting tool moves along these axes while the workpiece is rotated. As the work piece is being rotated moving the cutting tool along either axis would result in symmetrical cuts being made to the workpiece.



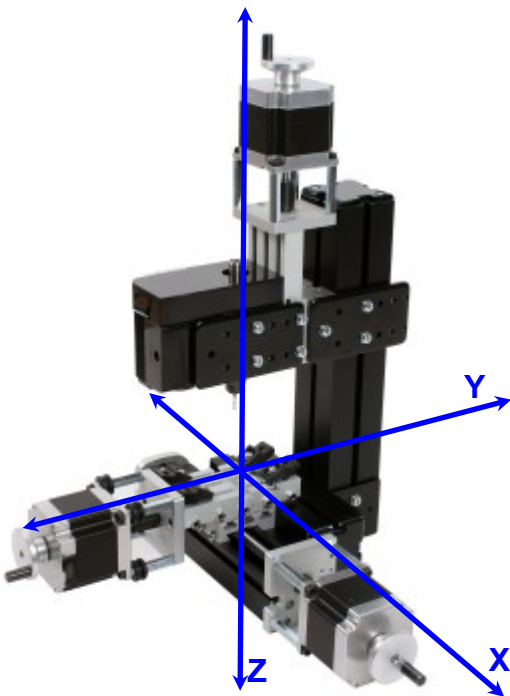
Milling Machine Axis Designation

Milling machines can have between three and six degrees of freedom; however, with an increasing number of degrees of freedom the complexity of hardware and programming increases.

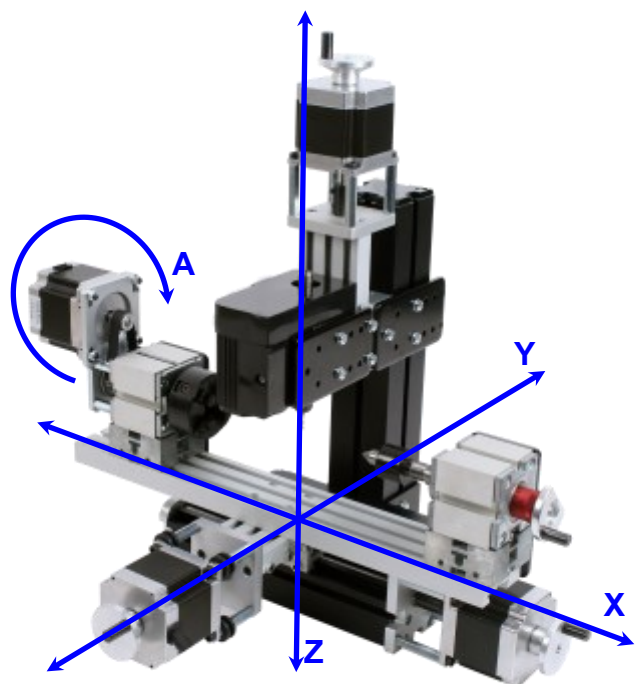
A 3-axis vertical milling machine has three degrees of freedom (X, Y, Z) as shown below. Note that the Z-axis moves vertically in relation to the workpiece; for a horizontal milling machine the Y-axis & Z-axis are reversed, this is to ensure the cutting tool is always operating along the Z-axis.

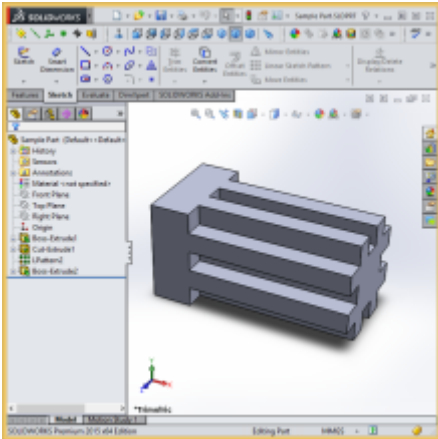
A 4-axis vertical milling machine has four degrees of freedom (X, Y, Z, A) as shown below. In this case the fourth degree of freedom 'A' allows the workpiece to be rotated about the X-axis allowing all faces to be machined.

3-Axis Vertical Milling Machine



4-Axis Vertical Milling Machine





WHAT IS CAD SOFTWARE?

Computer Aided Design (CAD) Software programs such as Solidworks, AutoDesk Inventor, etc. are sophisticated drawing packages used by designers, engineers, and architects to produce detailed precision drawings, technical illustrations, and models.

CAD software can be used to create two-dimensional (2D) drawings or three-dimensional (3D) models. Once a design has been finalised in a CAD package the drawings or models can be used with CAM Software to generate G-Code and control CNC Machines.

Some of the advantages of CAD Software include:

- Ability to produce very accurate 2D drawing or 3D model designs
- Easy conversion and manipulation of drawings and models
- 3D model components can be combined to create complex assemblies
- Automated generation of parts lists or bill of materials (BOM)

There are many different CAD software packages available but most have a similar working environment. This usually consists of a workspace where you will create your drawings and models. Workspaces will consist of a number of toolbars and menus for selecting different drawing tools, shapes, textures, views, etc. Your first steps in CAD will be to familiarise yourself with the working environment of the CAD software package you will be using.

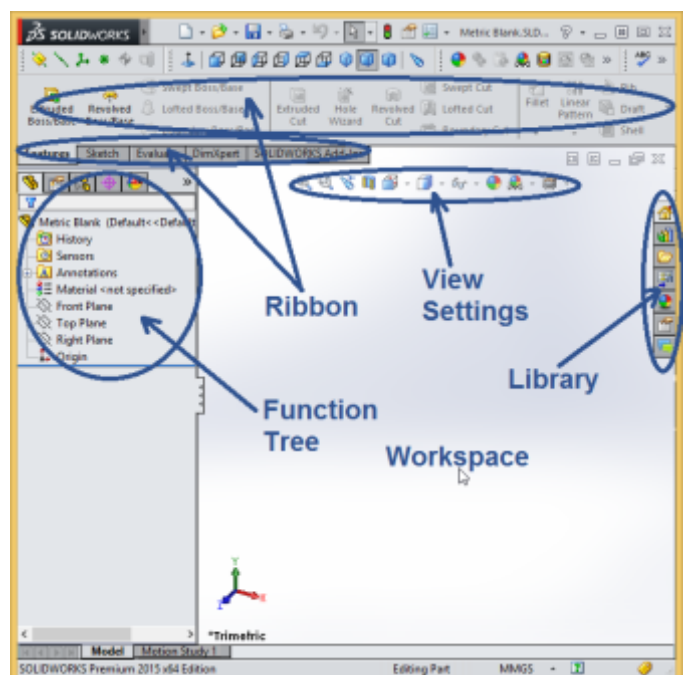
Activity - Familiarise yourself with a CAD Environment.

The image opposite shows a typical CAD environment, in this case Solidworks.

All CAD packages will have similar features although they may have different names, and the tool bars etc. may be in different locations.

These features typically include standard and customisable tool ribbons (seen at the top of the window); these include standard commands such as 'File Save', 'Open', etc. Other features will be:

- Workspaces: This is where your sketches and 3D model are created and displayed.



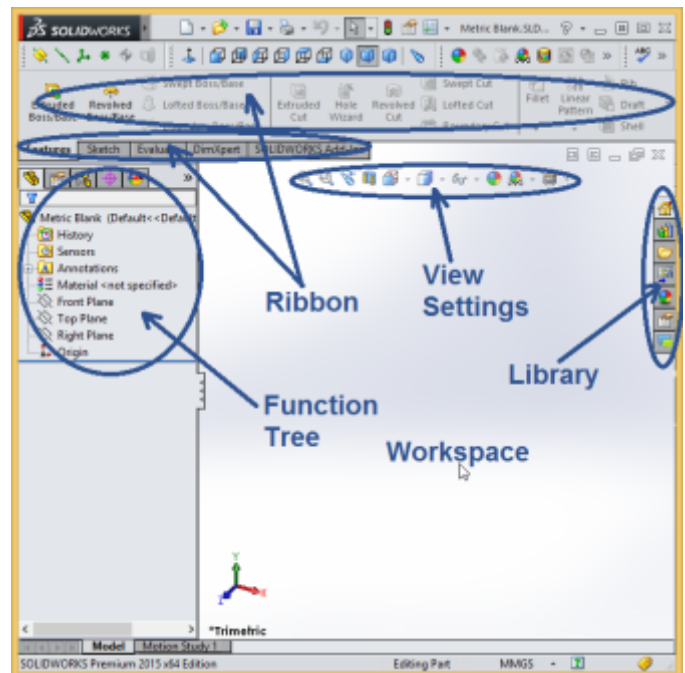
- **Workspaces:**
This is where your sketches and 3D models are created and displayed.

- **Function tree:**
This contains all your design features. At the top of the tree is the first thing drawn/created with the latest features in order show below. For an assembly, the function tree contains all the parts that form the assembly.

- **Ribbon:**
The Ribbon contains shortcuts to the most commonly used functions. It has tabs such as features, sketch, Appearance, etc. It is usually customisable so that tabs show what you most use.

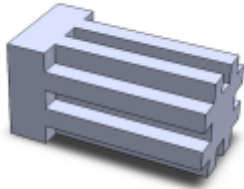
- **View Settings:**
This toolbar contains options to set different views of the model. Options typically include view orientation, view display, background scene, zoom, perspective, etc.

- **Library:**
The library usually contains quick access to pre-defined options such as design library, pallets, file explorer, properties, etc.



Example

Design a part in CAD Software



SAMPLE PART - CAD DESIGN EXAMPLE

In this walkthrough example you will be shown the procedure for designing a sample part using a typical CAD package (Solidworks for this example); it should be possible to follow this example using what ever CAD package you have available.

This design will be used later in a CAM software example.

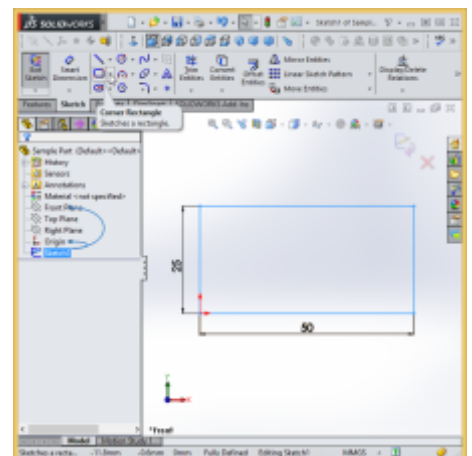
This example shows how to create a 3D model design that can be used with typical CAM packages to create real parts. A brief introduction to the steps involved in designing a part will be given and you will be introduced to some typical CAD operations.

It is important that you are familiar with the working environment of the CAD package to be used; this example only outlines the steps involved and does not show how to setup a workspace, or access menus, functions, etc.

Start by opening your CAD package and begin a new sketch. In this example the design is started by sketching a rectangle from the origin of the Z-plane.

The plane chosen is not important for this design as the part will be symmetrical around its central axis.

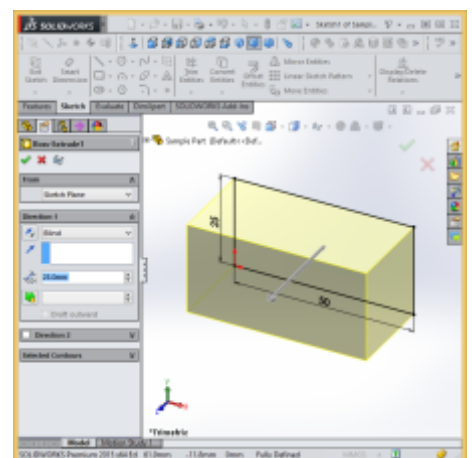
For other designs it may be necessary to choose a specific axis, relating to the machine tools orientation, in order to be able to manufacture the part correctly or produce all the features of the design.



After we have produced a sketch of the part's outline this is 'Extruded' to form a 3D model.

It is important to understand how 3D models relate to the sketches from which they are produced. If a feature is designed correctly, parameters such as length, angle, etc. can be easily modified and the 3D model updated. However, if it is not correctly defined when changes are made to a sketch, it will not be possible to rebuild the model.

In this example the 2D sketch has been extruded to a depth of 25mm; this will produce a solid 3D model with dimensions of 50 x 25 x 25 mm to match stock material size.



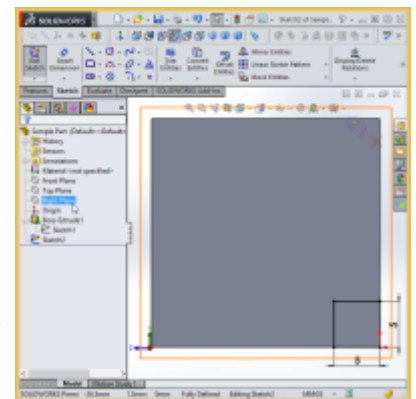
Example

Design a part in CAD Software

The next step is to create another sketch on the end face of the 3D model. This sketch will be used to create a new feature of the part.

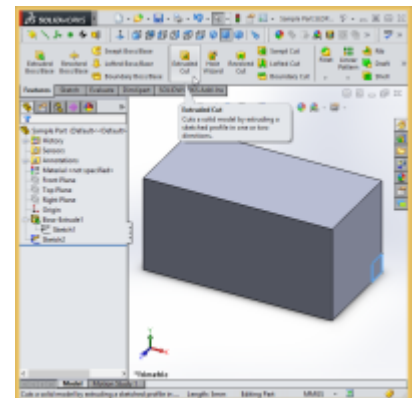
On the right hand end face sketch a 5 x 5 mm square at the origin of the part.

Later in the design if we want to change the size of the feature we can return to the sketch and alter the dimensions; these will then be reflected in the subsequent design elements of our part.

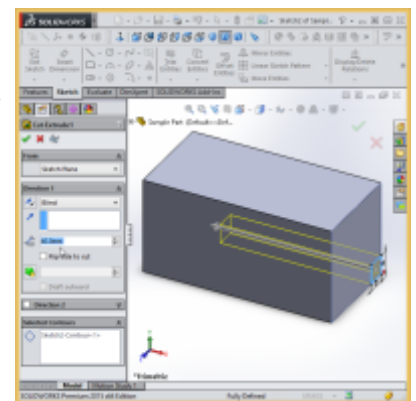


After completing the sketch we need to add an 'Extrude Cut' feature. This is effectively the reverse of an Extrusion; instead of adding 'Material' a cut will remove material from our 3D model.

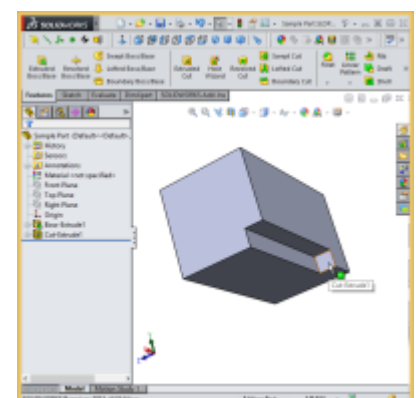
Extruded cuts can be made to go through the whole part or some defined section of the model.



In this example we only want to cut away a section of the part. Make an extruded cut by selecting the sketch from the previous step and then cut into the material to a depth of 40mm.



After completing the operation your part should look like this.

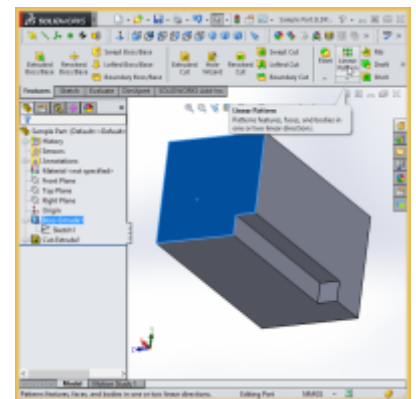


Example

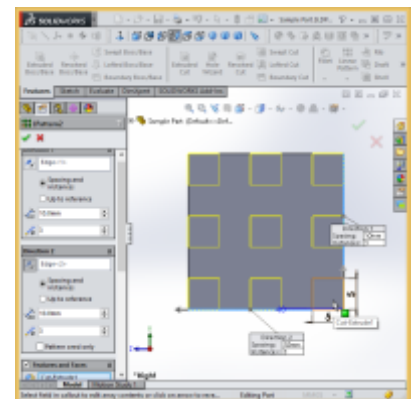
Design a part in CAD Software

The next step is to use the feature from the previous step to create a pattern around our model. A pattern is simply the duplication of the feature; these can be linear or circular depending on how you need to duplicate the feature.

For this design we are going to create a linear feature in two directions so select the extruded cut feature from the previous step and then select the 'Linear Pattern' feature option.



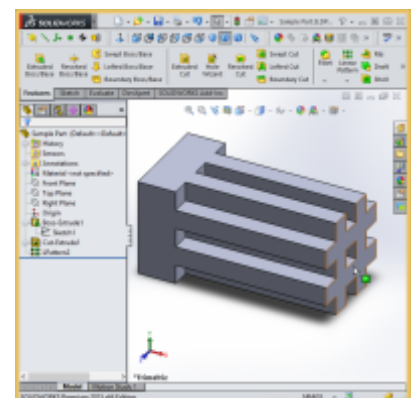
For this example the extruded cut feature will be duplicated 3 times along the Y and Z Axes with a spacing of 10mm between the centres of each feature .



After completing the linear pattern feature the model should look like the image to the right.

Since we used the linear pattern feature tool and chose to replicate the extruded cut feature 3 times in two directions we now have nine instances of the extruded cut feature.

For this design we do not want to include the square hole in the centre of the model in the machining process so we need to remove that feature from our model.

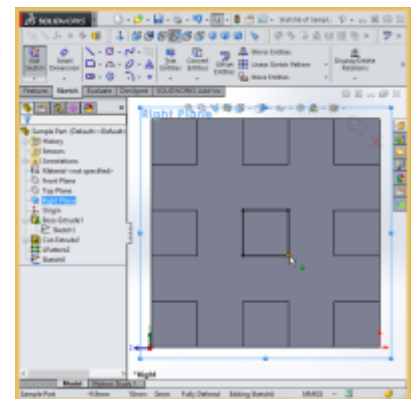


Example

Design a part in CAD Software

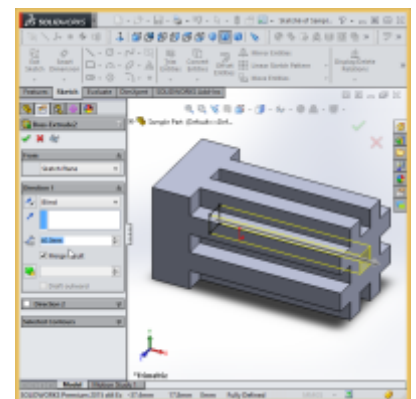
To remove the central hole we create a new sketch on the model. Note that you can create the sketch on the end face of the model or on the face of the extruded cut feature. If we choose the end face of the block we would need to extrude into the material to fill the hole. If we choose the face of the extruded cut we would then extrude out of the material.

For this design we are going to create a new sketch on the face of the extruded cut.

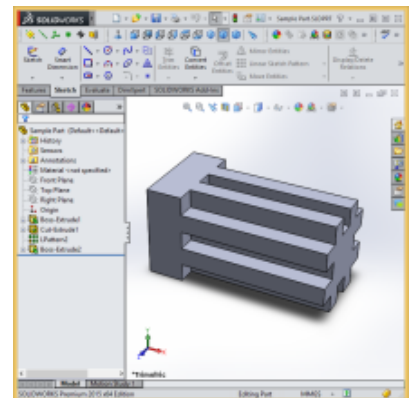


For this example the sketch is extruded out of the material from the inner face of the cut, this will fill the hole from the inside to the outer edge of the block.

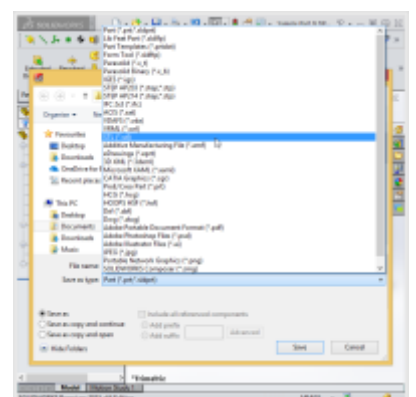
Create an extrusion of 40mm from the previous sketch.



The 3D model should now look like the image to the right. This part is now complete and is ready to be exported to a CAM package. However, most CAM packages require models to be in a specific file format.



To ensure this design can be imported into a CAM package we will save the file as a .STL file type.



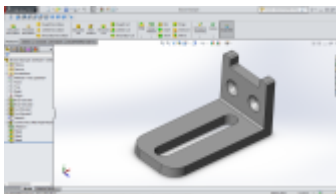


Computer Aided Manufacturing (CAM) software is used to control machine tools for the purpose of manufacturing components.

CAM software is generally used following the CAD process, where a 2D or 3D design is converted into detailed instructions (G-code) that drive computer numerical control (CNC) machine tools; although designs can also be produced in some CAM software as well.

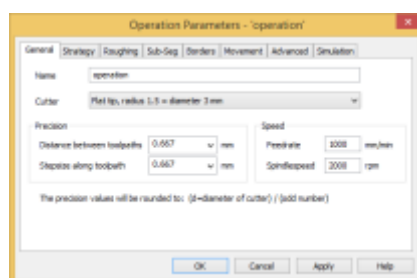
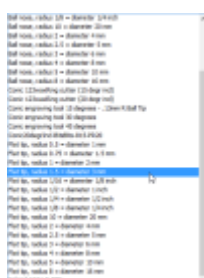
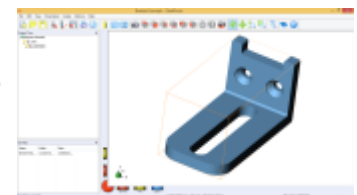
Once a component design has been finalised the part needs to be manufactured. Generally the procedure for creating the part would be along the lines of the following method:

1. Create a solid 3D model of the part to be produced. Any standard CAD format is acceptable.
2. Import the solid model into the Computer Aided Manufacturing (CAM) software.
3. Input the raw material stock size and set the part's coordinate origin.
4. Input the necessary information for each tool used in machining the part features. Typically, a tool library will exist, which is simply a database of tools and their related parameters.
5. For each part feature, select the appropriate tool from the library and set the parameters necessary for machining that feature. Typical parameters include spindle speed, depth of cut, feed rate, number of passes, tool path pattern, etc.
6. Verify the programmed tool paths by running the CAM software's virtual machining cycle.

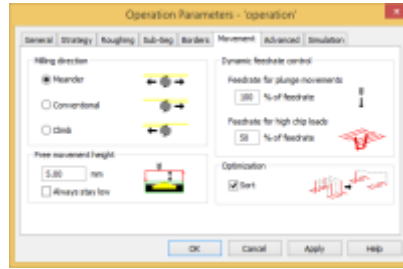
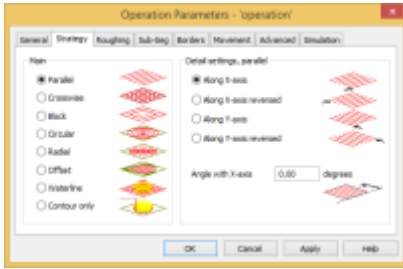


Component is designed in CAD software (e.g. Solidworks) and imported into CAM software.

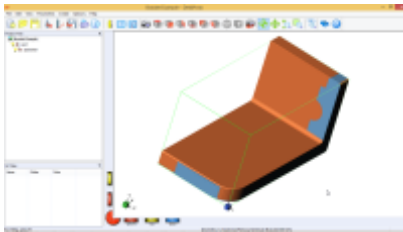
Within the CAM software the size of the raw material is set up and the coordinates are zeroed.



Cutting tools are selected and their tooling parameters setup and stored.



Typical tooling parameters include spindle speed, depth of cut, feed rate, number of passes, tool path pattern, etc.

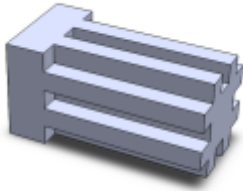


A cutting simulation is then run in the CAM software to catch any errors before cutting.

Example

Making a Part with Micro CNC Machines

4-AXIS MILLING EXAMPLE



In this walkthrough example you will be shown the procedure for making a small component using the Micro CNC 4 axis Milling Machine.

In this example the part to be made is the Sample Part designed in the CAD example, as shown in the image to the left.

This example is using DeskProto, a 3D CAM software package.

This example shows how to import a CAD design into a typical CAM software environment; while the procedures for different software packages will be similar, there will be differences in the menus and options available.

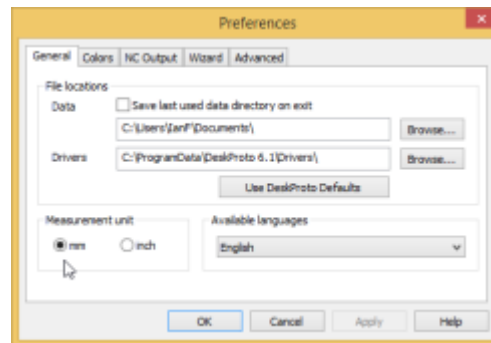
Once the design has been imported, DeskProto will be used to create tool paths and then translate these into G-code ready to be converted by the MicroCNC control system into physical movement of the cutting tool.

First Launch of DeskProto

The first time you open DeskProto you will be prompted to set up the unit dimensions. You can also change the dimension settings through the options menu in the main program. After changing/selecting 'mm' Click 'OK' to continue.



Welcome Screen



Options Menu



Start Screen

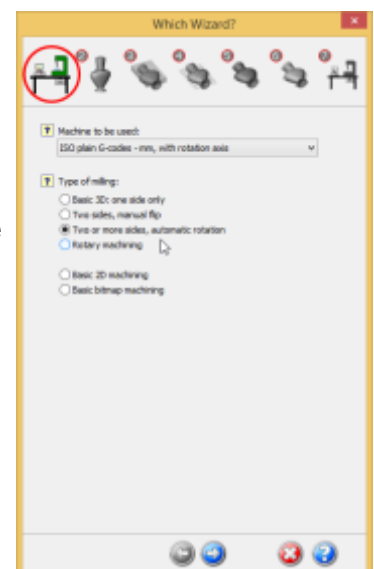
On the start Screen select 'Start Wizard'.

Which Wizard Screen

The first step is to set the type of operation required to create the part.

For this example we need to set the software to 'Two or more sides, automatic rotation'.

Once selected click the blue arrow to proceed to the next setup screen.



What to Machine Screen

This screen is used to select the required CAD file.

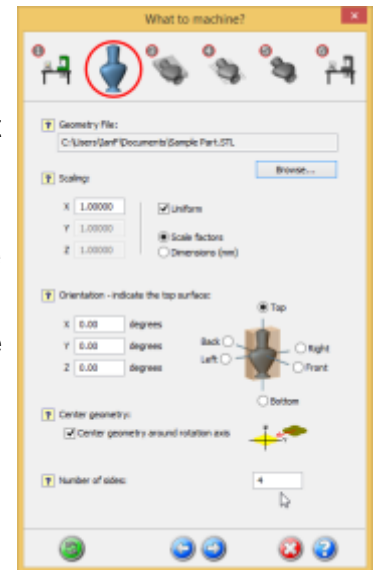
Use the browse button to open a file dialogue and select the .STL file for the part to be machined.

Set the scaling factor to 1 - Uniform.

Select the orientation to indicate which is the top face i.e. the face to be machined.

For this example the 'Number of sides' value needs to be changed to 4

Click the blue arrow to proceed to the next setup screen.



Material and Supports Screen

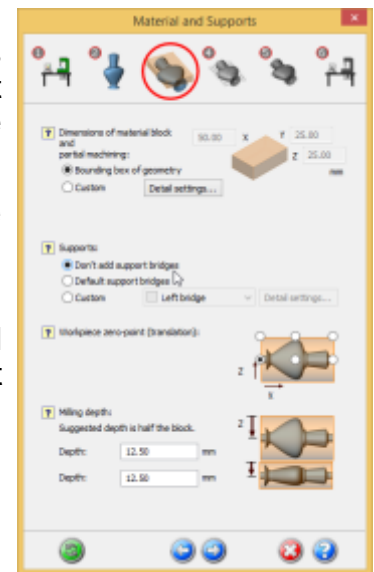
This screen is used to provide the details of the stock, supports (if required), zero (origin) point and mill depth.

When a CAD file is imported into DeskProto a bounding box is applied to the imported model. This represents the smallest space in which the part can be enclosed. This is essentially the smallest size of stock that the part could be produced from.

In many instances the stock material will be larger than the finished part so we will also need to add a zero point as a starting reference.

Select 'Bounding box of geometry'. For this part we do not need any supporting bridges so select 'Don't add support bridges'. Set the zero origin to be the base of the part and set the milling depth to 12.50mm (for this example).

Click the blue arrow to proceed to the next setup screen.



Roughing Screen

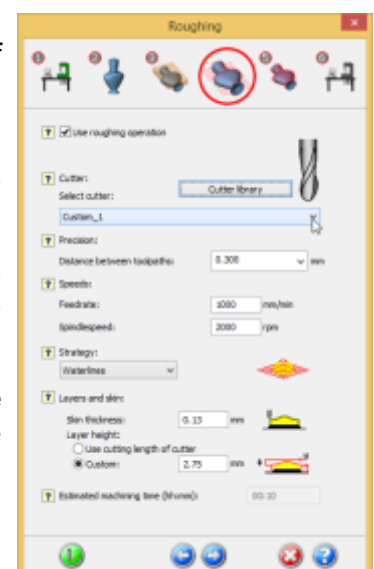
Roughing is the first stage in removal of material from a piece of stock. It is generally a fast cutting pass that is slightly larger than the completed part will be.

This screen is used to select the type and size of the cutter to be used, the distance between the tool paths and the cutting speed.

Most CAM software will also allow users to select the cutting strategy and other parameters. You will also need to enter the type and length of the cutter.

Select 'Use roughing operation', set the cutter to the type you are using; for this example we want the cutting strategy to be 'Waterlines'

Click the blue arrow to proceed to the next setup screen.



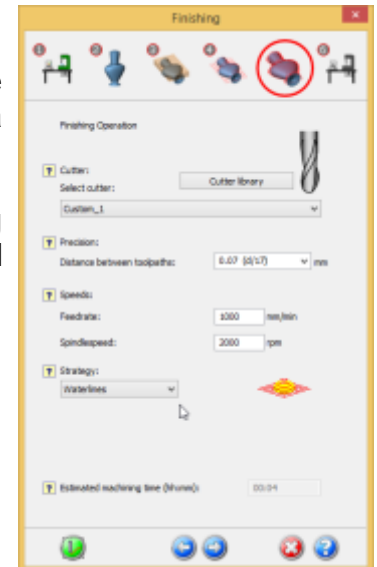
Finishing Screen

The finishing operation is the final pass and the one where the part will be 'Finished' to the correct dimensions. This will be a fine, slow pass of the cutting tool using small steps.

Select the type of cutting tool you are using, if the type of cutting tool is not in the drop down options select 'Cutter Library' and enter the details of your cutter.

Again select the 'Waterlines' as the cutting strategy.

Click the blue arrow to proceed to the next setup screen.



Send to Machine Screen

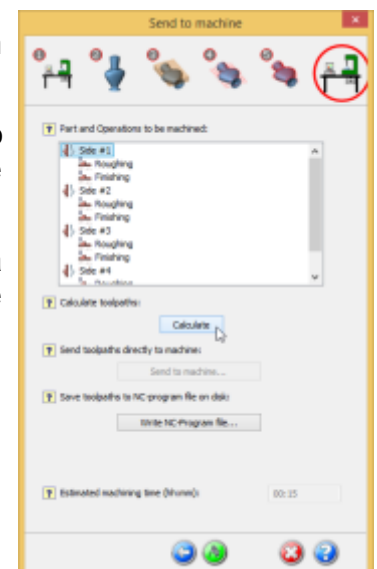
The final screen is a confirmation screen and provides an overview of the cutting program the wizard has created.

If you are happy with the cutting program select 'Calculate' to calculate the cutting tool paths. However for this example we need to set some more parameters.

If the cutting program is acceptable it could be sent directly to a CNC machine if one is connected to the PC, otherwise it can be saved as an NC-program.

Select 'Calculate' to generate a cutting program.

Click the Green checker flag button to exit the setup wizard.



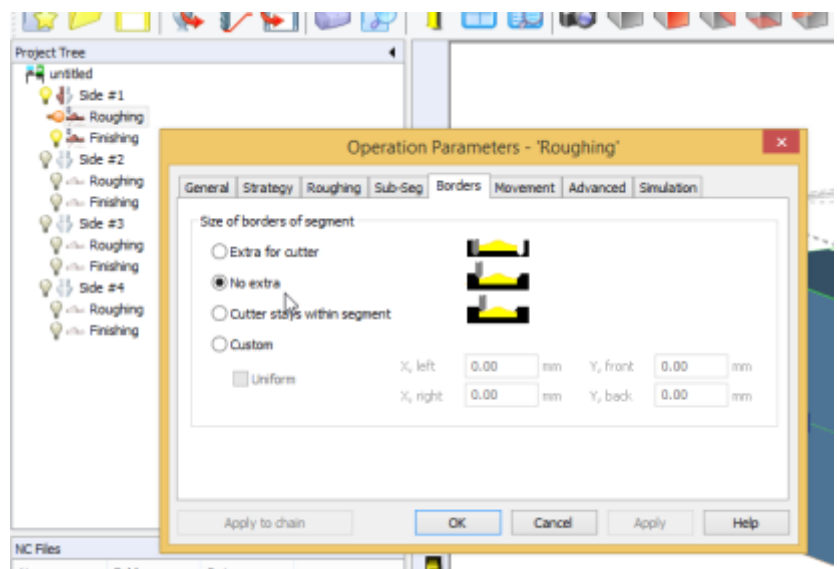
Additional Parameters

For this example we need to change the settings for the borders around the part. As we are machining a block of material there are no outside borders.

From the 'Project Tree' double click on 'Side #1' 'Roughing' to open the Operational Parameters window.

Select the 'Borders' tab and select 'No Extra' apply and exit.

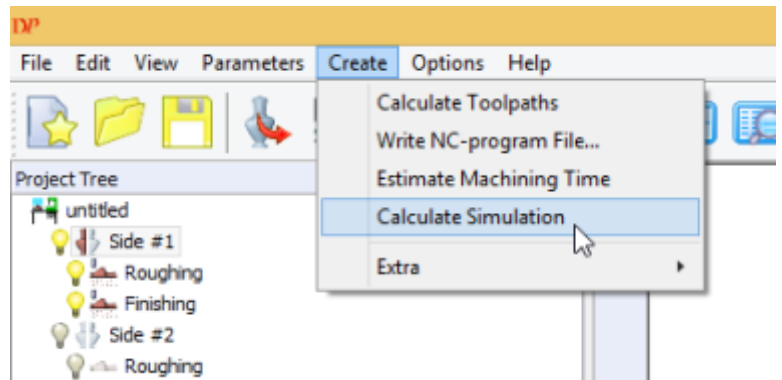
Repeat the above for 'finishing' and for the other sides.



Simulation

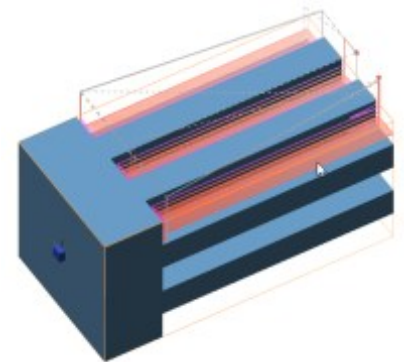
Before saving the G-Code it is possible to view a simulation of the cutting tool paths.

This can be done as a check to ensure the cutting operation is as expected.



After creating the simulation the tool paths are shown around the part as shown opposite.

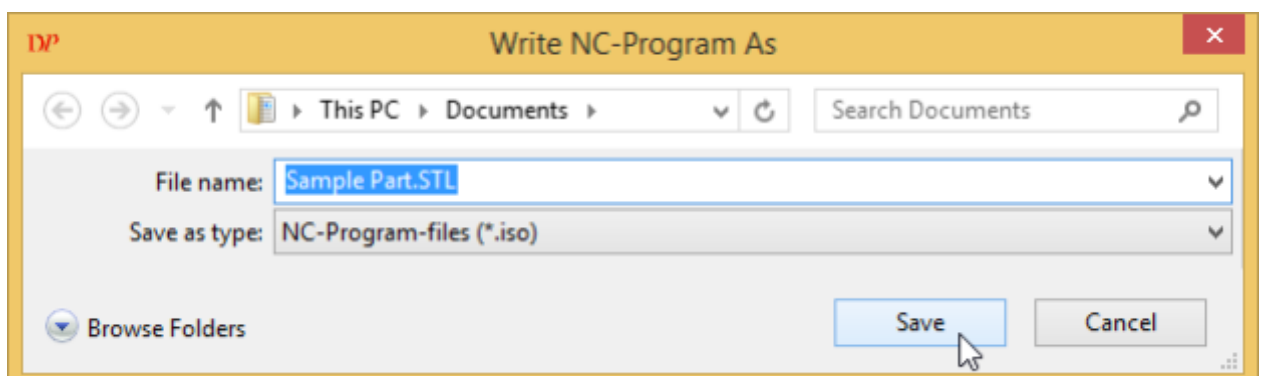
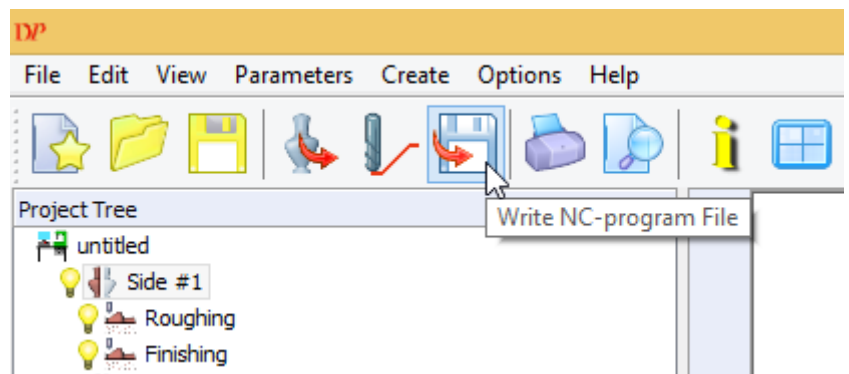
Run a simulation for each side to ensure the correct parameters have been set.



Saving the GCode File

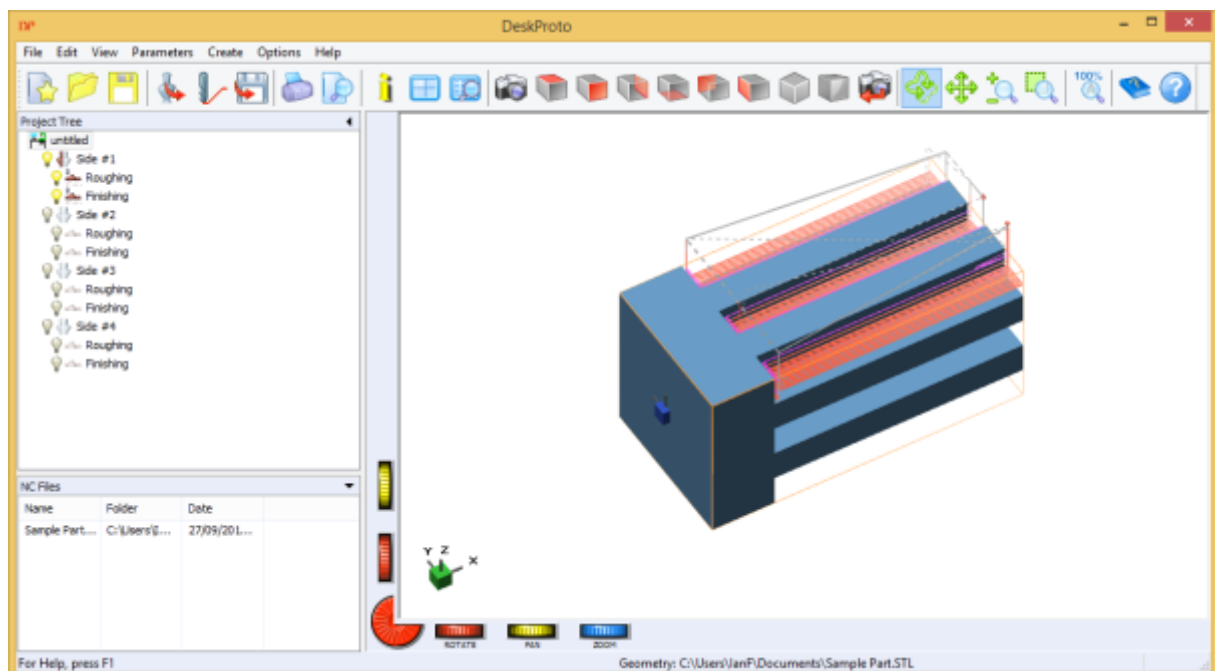
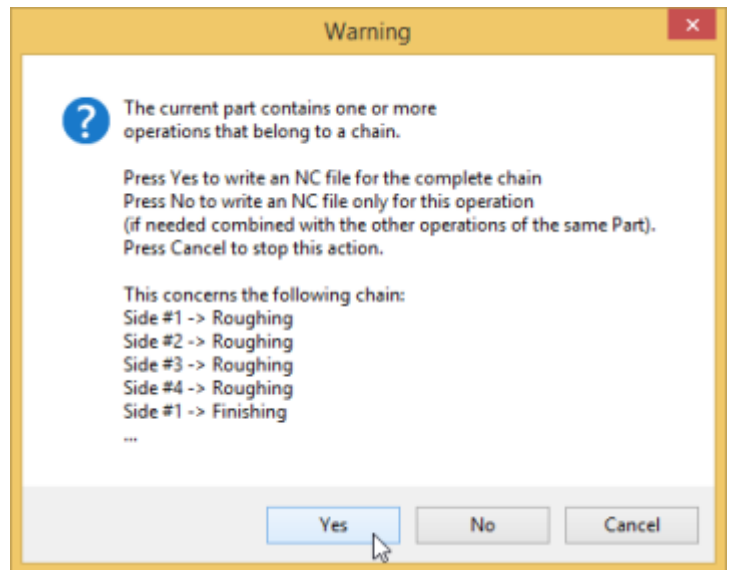
After checking all the cutting parameters are correct the cutting program needs to be saved as G-code. This is done by saving the file as a .ngc

Note: The file name needs to be 8 characters or less



The G-code file can be for the entire operation or for a single operation chain. This depends on whether you need to change the tool to move between operations.

A warning will let you know if you are saving multiple operations as a single G-code file.



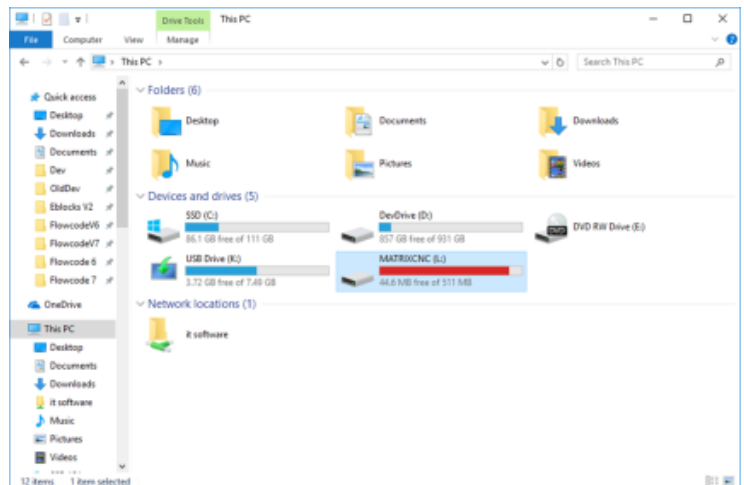
After exiting the setup wizard the CAD model is shown with the programmed toolpaths displayed.

Copying files to the CNC machine

Once DeskProto has been used to generate the G-Code the saved .NGC file needs to be copied to the Micro CNC Controller as follows:

Connect USB cable from PC to Micro CNC Controller.

A new drive will appear on your local computer folder.

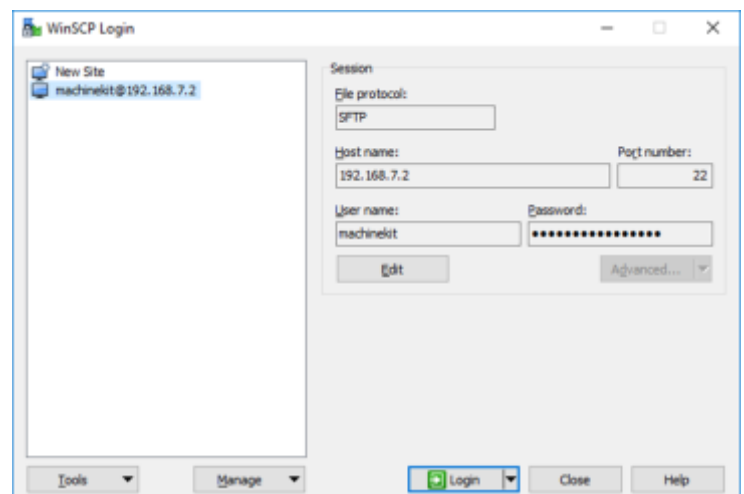


Browse to the following directory.

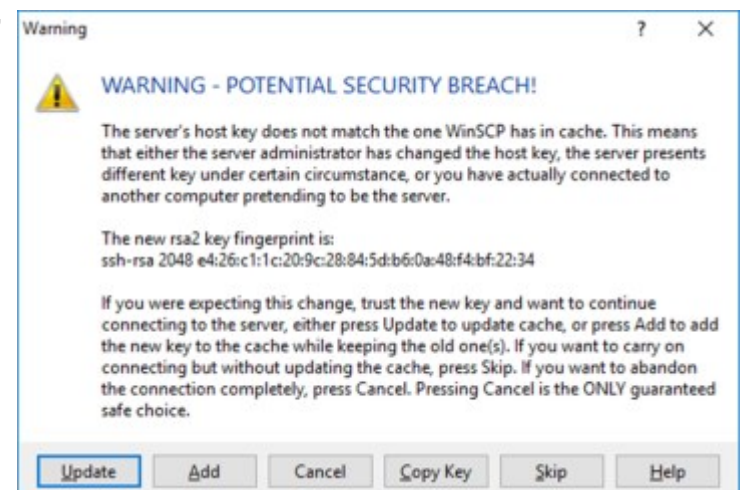
MatrixCNC /

Double click the file
Launch_File_Transfer.bat

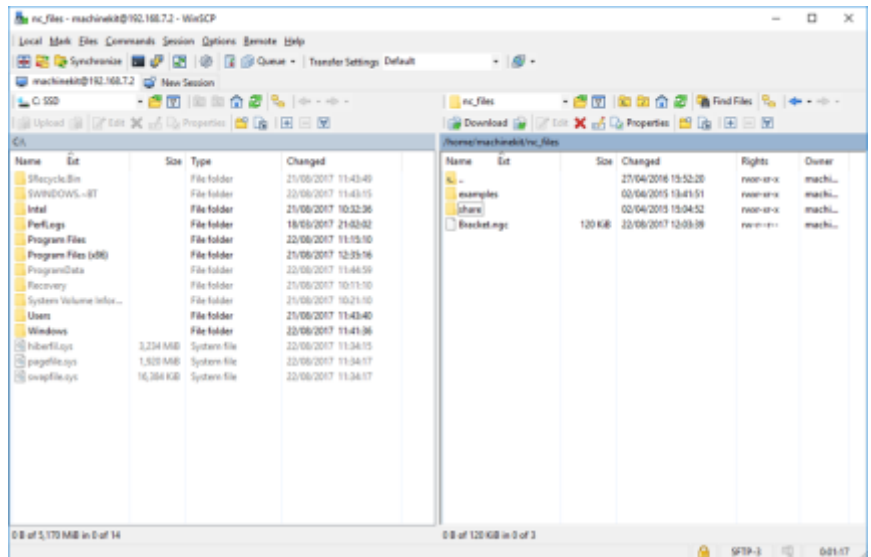
Click Login



If you get a Warning message click 'Add' and then login again.



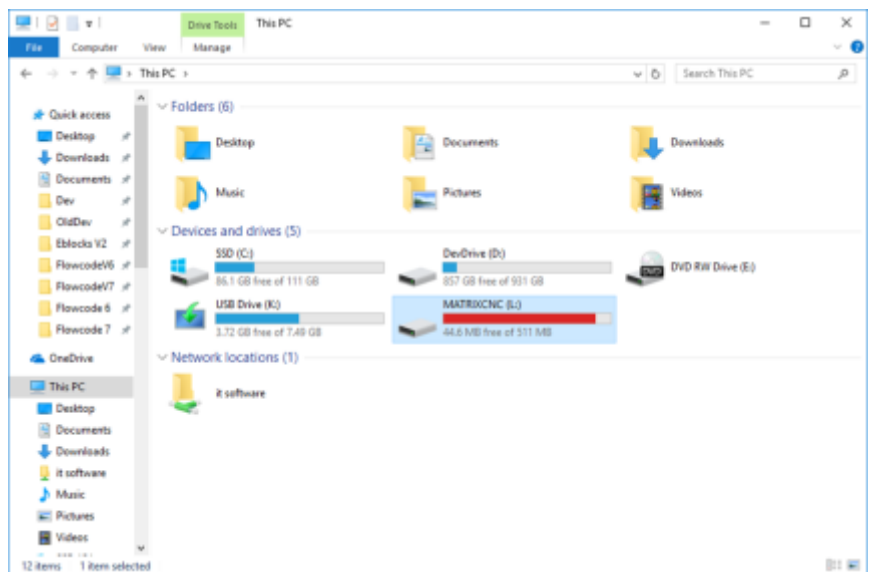
Copy your g code .ngc files to the /home/machinekit/nc_files folder.



Performing A Cutting Operation

Connect USB cable from PC to CNC Machine.

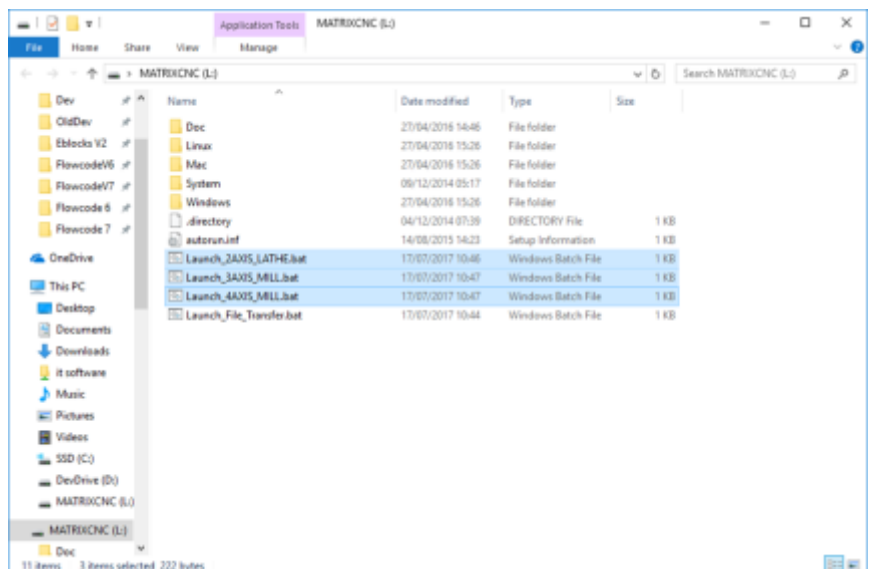
A new drive will appear on your local computer folder.



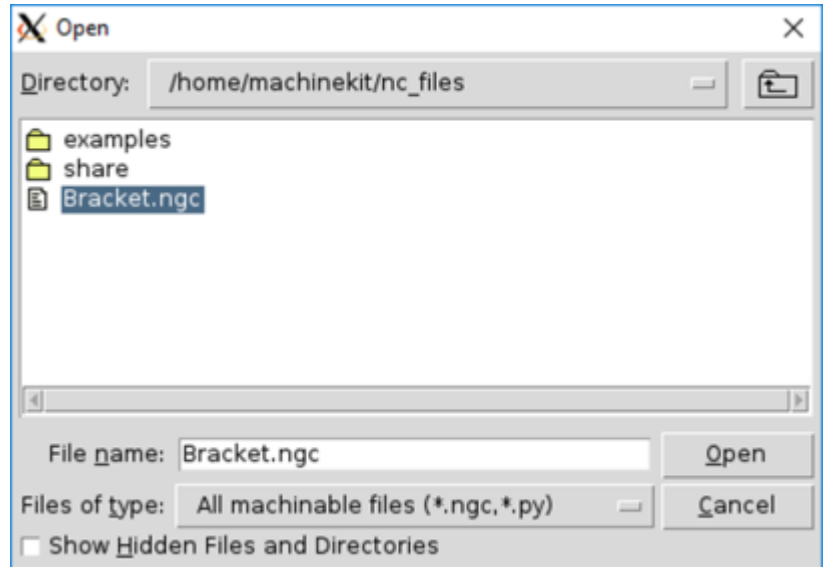
Browse to the following directory.

MatrixCNC /

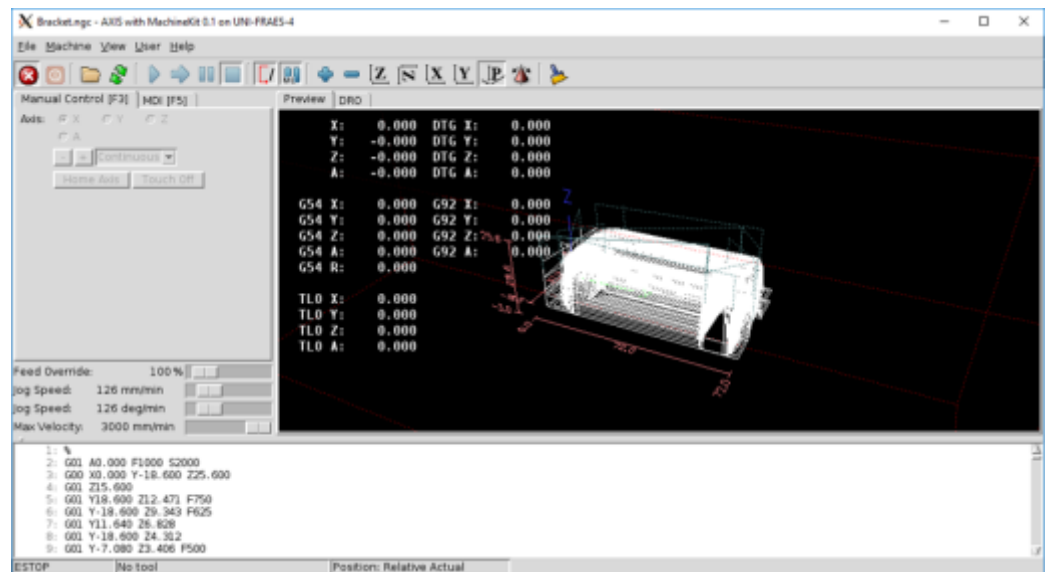
Double click the file that corresponds to your machine setup.



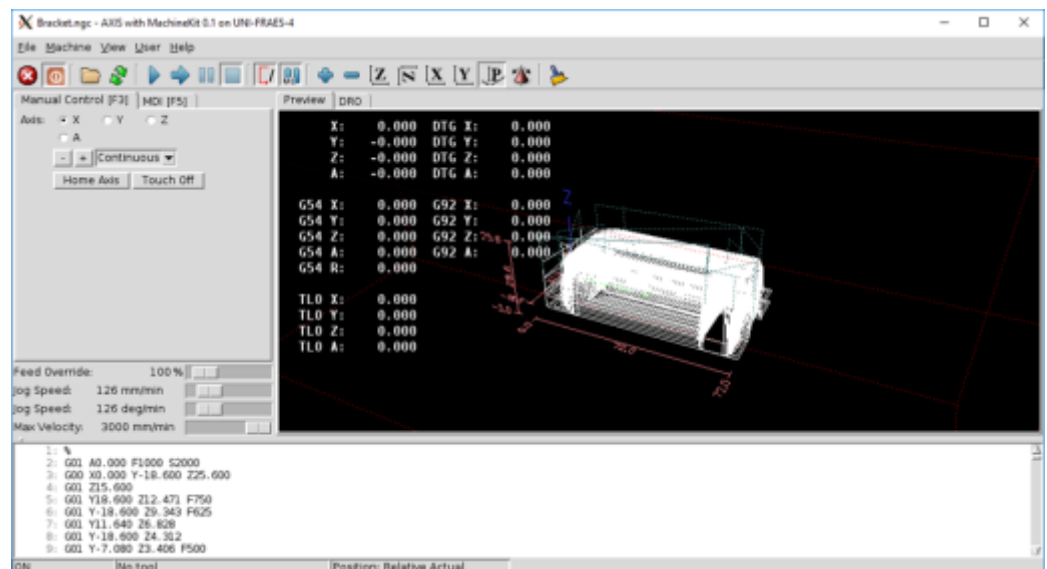
Click the Open icon to open your G code file.



Click the Toggle Emergency Stop button (top left) to bring the machine online.



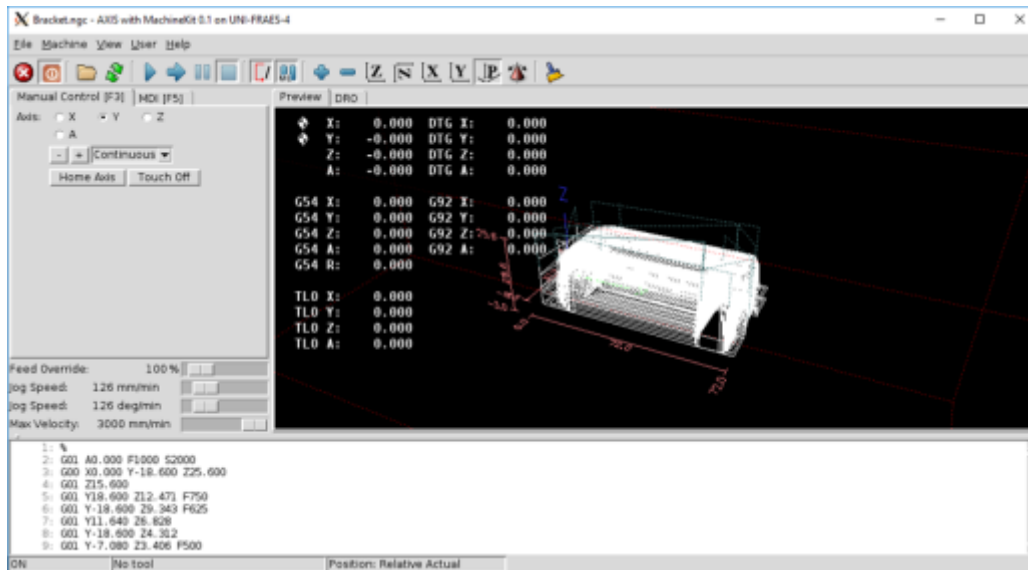
Click the machine power button (2nd from Left) to power up the machine ready for operations.



Select your axis, (X, Y, Z, A) and use the + / - buttons to move the axis.

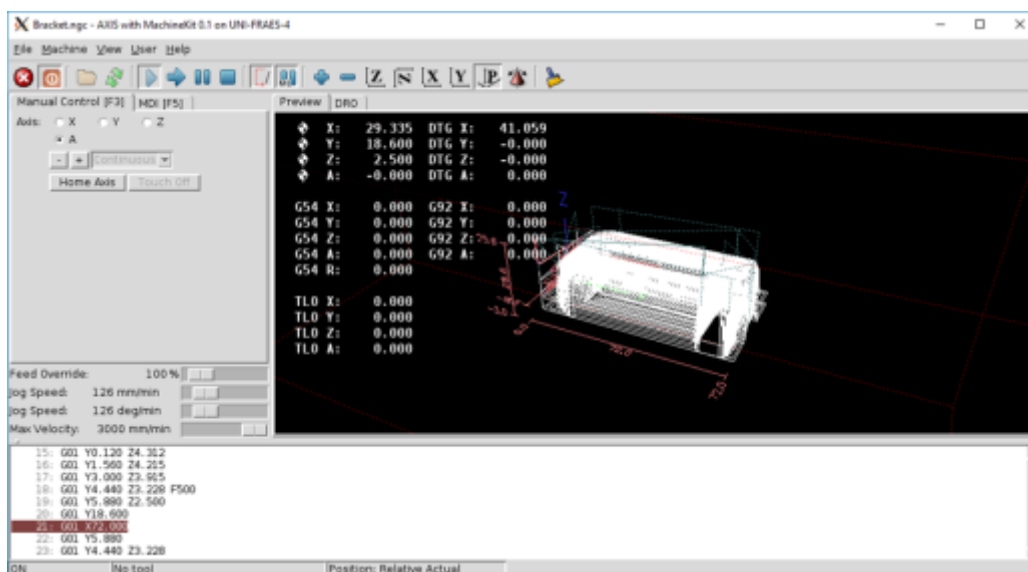
Select each axis in turn and click the Home Axis button to confirm the axis has been initialised in the start 0,0,0,0 position.

An icon will appear next to the axis that have been homed, shown are X and Y.



Once all axes are homed start the operation by clicking the Play button.

The screen will animate and show the current position of the end cutter as well as the G code.



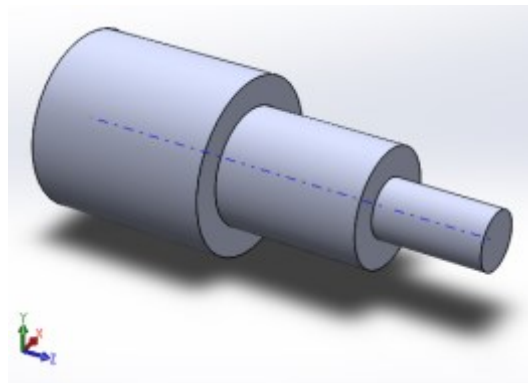
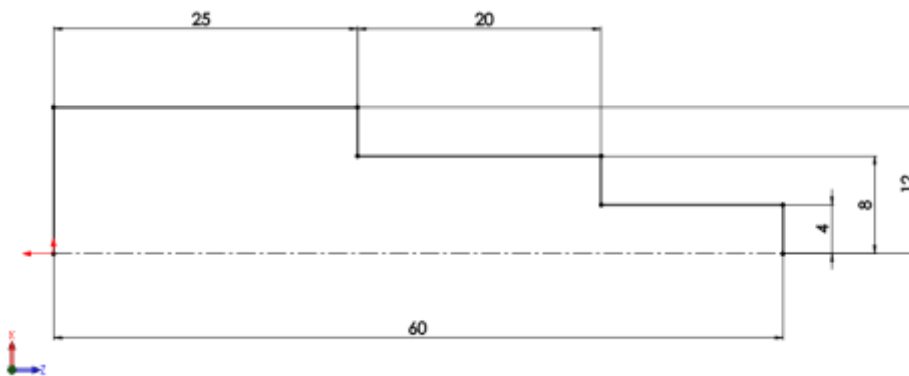
Worksheet 1

Lathe - Stepped Cylindrical Part

Lathe Task 1

This task involves producing a simple part using the MicroCNC Lathe. You should use the following 2D Drawing and 3D model to:

1. Produce the part manually - this will involve marking out the stock material and manually operating the X, and Y-axes independently.
2. Manually write G-code to your CAM software.
3. Re-create the drawing and model using your CAD package and export this to your CAM system.



;Worksheet1 - Lathe - Manual Written G-Code

;Assuming 0,0 Origin at end (Tailstock) of 28mm diameter stock, 14mm from central pivot

;Absolute Coordinates

```
G21 ;mm Units
G90 ;Absolute Coordinates
G1 X-2 F2 ;Move X axis down 2mm into material speed 2mm/s
G1 Z-60 F2 ;Move along stock piece cutting 12mm radius
G1 Z0 F10 ;Move back to Z0
G1 X-6 F2 ;Move X axis down another 4mm into material
G1 Z-35 F2 ;Move along stock piece cutting 8mm radius
G1 Z0 F10 ;Move back to Z0
G1 X-10 F2 ;Move X axis down another 4mm into material
G1 Z-15 F2 ;Move along stock piece cutting 4mm radius
G1 X0 F10 ;Move X axis back to start position
G1 Z0 F10 ;Move Z axis back to start position
```

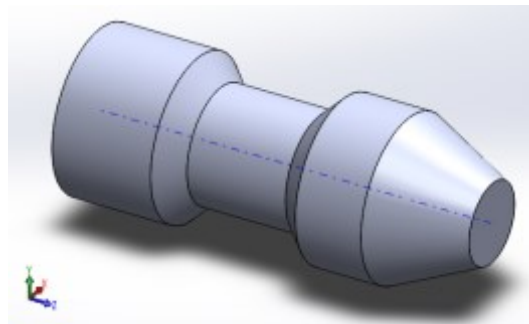
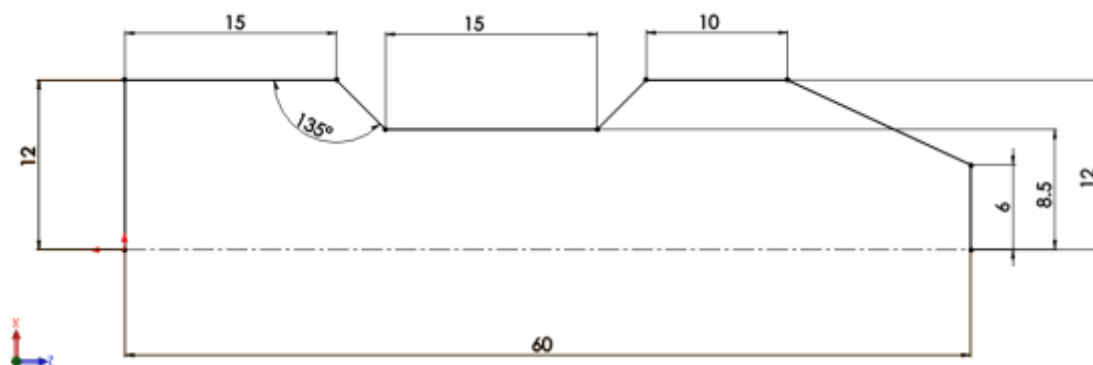
Worksheet 2

Lathe - Tapered Cylindrical Part

Lathe Task 2

This task involves producing a relatively simple part using the MicroCNC Lathe. You should use the following 2D Drawing and 3D model to:

1. Produce the part manually - this will involve marking out the stock material and manually operating the X, and Y-axes both independently and simultaneously .
2. Manually write G-code to your CAM software.
3. Re-create the drawing and model using your CAD package and export this to your CAM system.



;Worksheet2 - Lathe - Manual Written G-Code

;Assuming 0,0 Origin at end (Tailstock) of 28mm diameter stock, 14mm from central pivot

;Absolute Coordinates

```
G21 ;mm Units
G90 ;Absolute Coordinates
G1 X-8 F2 ;Move X axis 8mm into stock, at speed 2mm/s (6mm radius)
G1 Z-13 X-2 F2 ;Move X axis diagonally out of the stock (6mm to 12mm radius)
G1 Z-23 F2 ;Move along stock cutting 12mm radius
G1 Z-26.5 X-5.5 F2 ;Move X axis diagonally into stock 3.5mm (8.5mm radius)
G1 Z-41.5 F2 ;Move along stock cutting 8.5mm radius
G1 Z-45 X-2 F2 ;Move X axis diagonally out of stock 3.5mm (12mm radius)
G1 Z-60 F2 ;Move along stock cutting 12mm radius
G1 X0 F10 ;Move X axis back to start position
G1 Z0 F10 ;Move Z axis back to start position
```

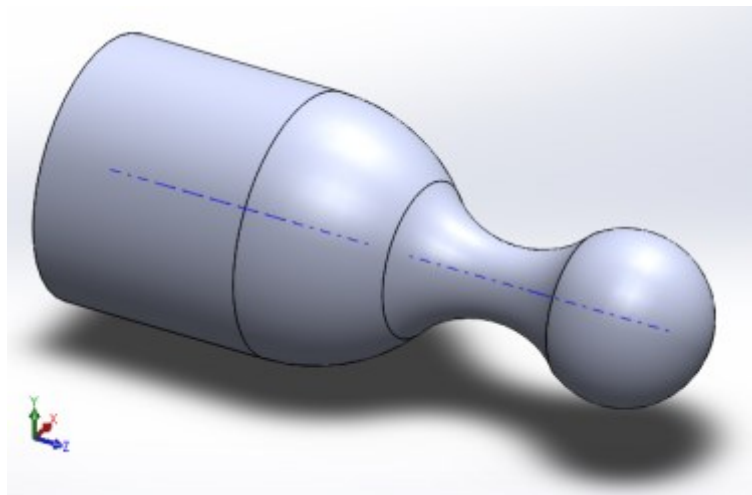
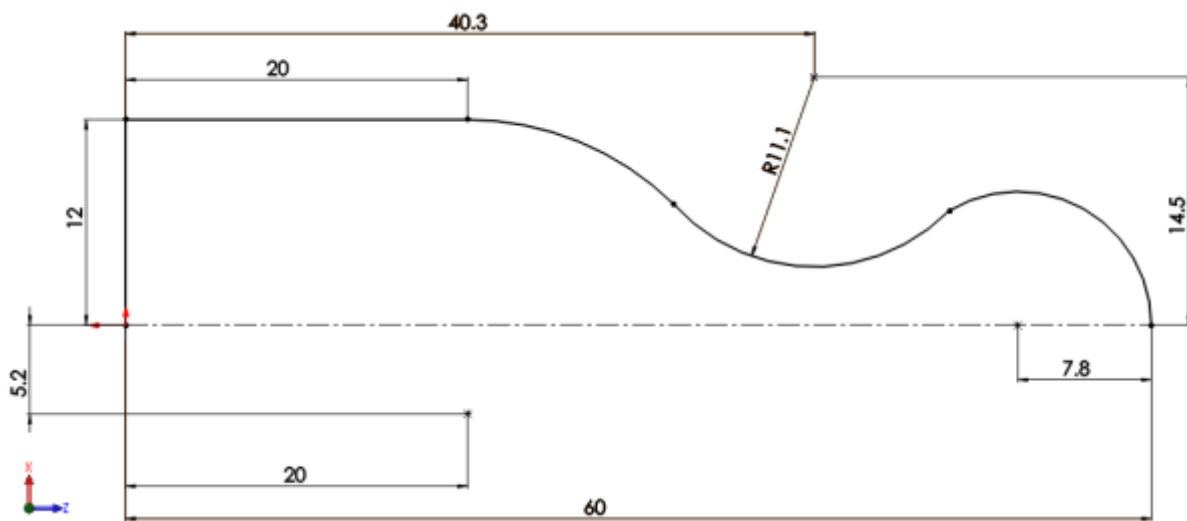
Worksheet 3

Lathe - Complex Cylindrical Part

Lathe Task 3

This task involves producing a part with complex geometry using the MicroCNC Lathe. You should use the following 2D Drawing and 3D model to:

1. Produce the part manually - this will involve marking out the stock material and manually operating the X, and Y-axes both independently and simultaneously.
2. Re-create the drawing and model using your CAD package and export this to your CAM system.



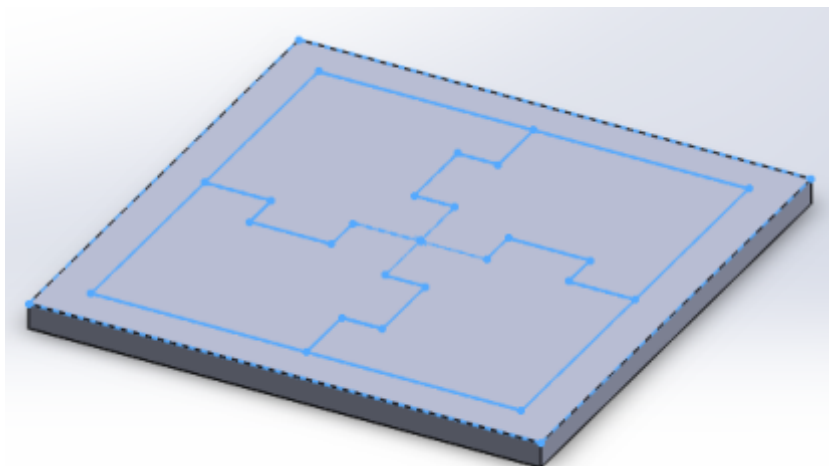
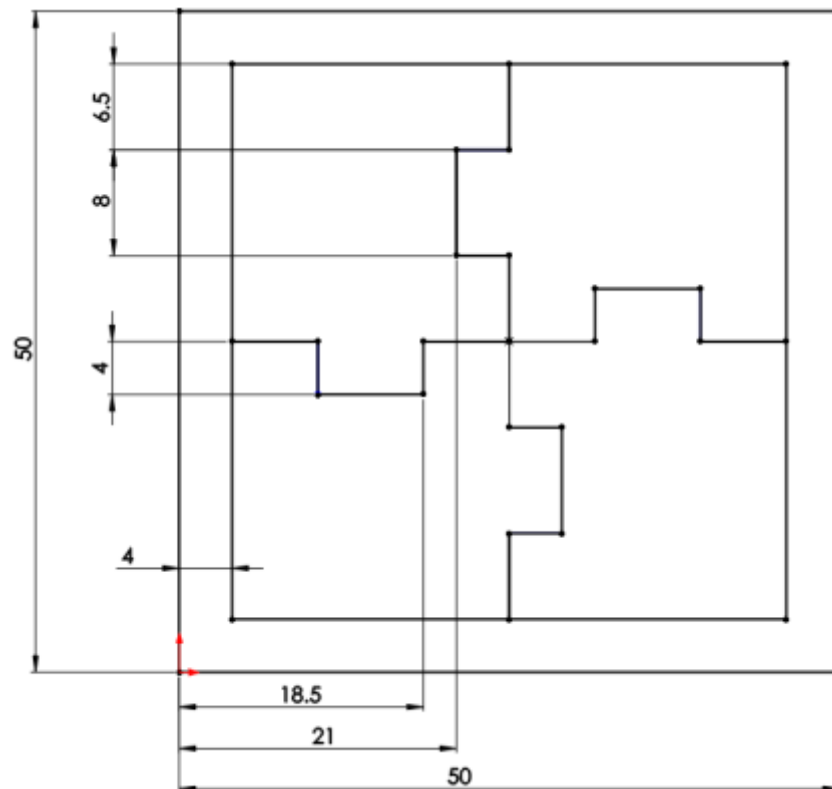
Worksheet 4

3-Axis Milling - Simple Jigsaw

3-Axis Milling Machine Task 1

This task involves producing a simple jigsaw using the MicroCNC 3-axis Milling Machine. You should use the following 2D Drawing and 3D model to:

1. Produce the part manually - this will involve marking out the stock material and manually operating the X, and Y-axes independently; you will also control the depth of the cutting tool by manually operating the Z-axis.
2. Manually write G-code to your CAM software.
3. Re-create the drawing and model using your CAD package and export this to your CAM system.



Worksheet 4 (cont.)

3-Axis Milling - Simple Jigsaw



;Worksheet4 - 3-Axis - Manual Written G-Code
;Assuming 0,0,0 start origin at bottom left corner of stock, Z should be positioned so the mill bit is as close as possible to the top of the stock but not touching
;Absolute Coordinates

G21 ;mm Units
G90 ;Absolute Coordinates
G0 Z1 F10 ;Move Z axis up to 1mm to perform a rapid movement
G0 X4 Y25 F10 ;Move to absolute 4, 25 rapid movement

G1 Z-3 F2 ;Move Z axis down 3mm into material - Begin inner cut - Horizontal
G1 X10.5 F2 ;Move X+ by 6.5mm
G1 Y21 F2 ;Move Y- by 4mm
G1 X18.5 F2 ;Move X+ by 8mm
G1 Y25 F2 ;Move Y+ by 4mm
G1 X31.5 F2 ;Move X+ by 13mm (2 x 6.5mm)
G1 Y29 F2 ;Move Y+ by 4mm
G1 X39.5 F2 ;Move X+ by 8mm
G1 Y25 F2 ;Move Y- by 4mm
G1 X46 F2 ;Move X+ by 6.5mm
G0 Z1 F2 ;Move Z axis up to 1mm to perform a rapid movement

G0 X25 Y4 F10 ;Move to absolute 25, 4 rapid movement

G1 Z-3 F2 ;Move Z axis down 3mm into material - Continue inner cut - Vertical
G1 Y10.5 F2 ;Move Y+ by 6.5mm
G1 X29 F2 ;Move X+ by 4mm
G1 Y39.5 F2 ;Move Y+ by 8mm
G1 X25 F2 ;Move X- by 4mm
G1 Y31.5 F2 ;Move Y+ by 13mm (2 x 6.5mm)
G1 X21 F2 ;Move X- by 4mm
G1 Y18.5 F2 ;Move Y+ by 8mm
G1 X25 F2 ;Move X+ by 4mm
G1 Y46 F2 ;Move Y+ by 6.5mm
G0 Z1 F2 ;Move Z axis up to 1mm to perform a rapid movement

G0 X4 Y4 F10 ;Move to absolute 4, 4 rapid movement

G1 Z-3 F2 ;Move Z axis down 3mm into material - Outer perimeter cut - Double
check your clamps are not in the way!
G1 X46 F2 ;Move X+ by 42mm
G1 Y46 F2 ;Move Y+ by 42mm
G1 X4 F2 ;Move X- by 42mm
G1 Y4 F2 ;Move Y- by 42mm
G0 Z1 F2 ;Move Z axis up to 1mm to perform a rapid movement

G0 X0 Y0 F10 ;Move to absolute 0, 0 rapid movement
G0 Z0 F10 ;Move to absolute 0,0,0 rapid movement

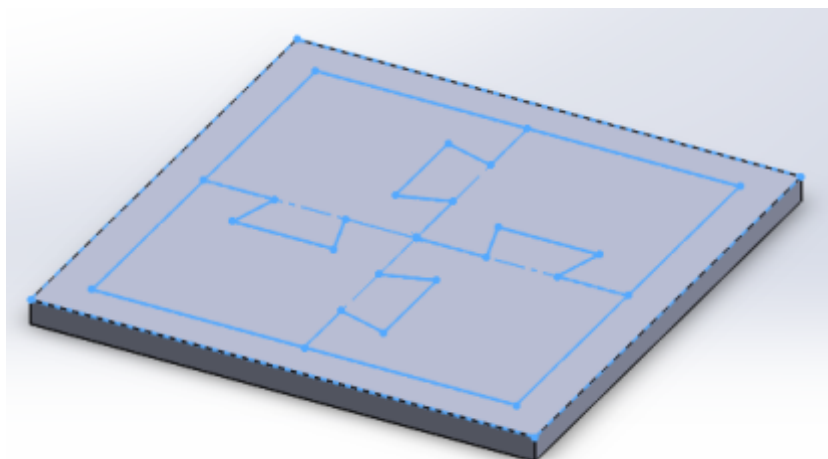
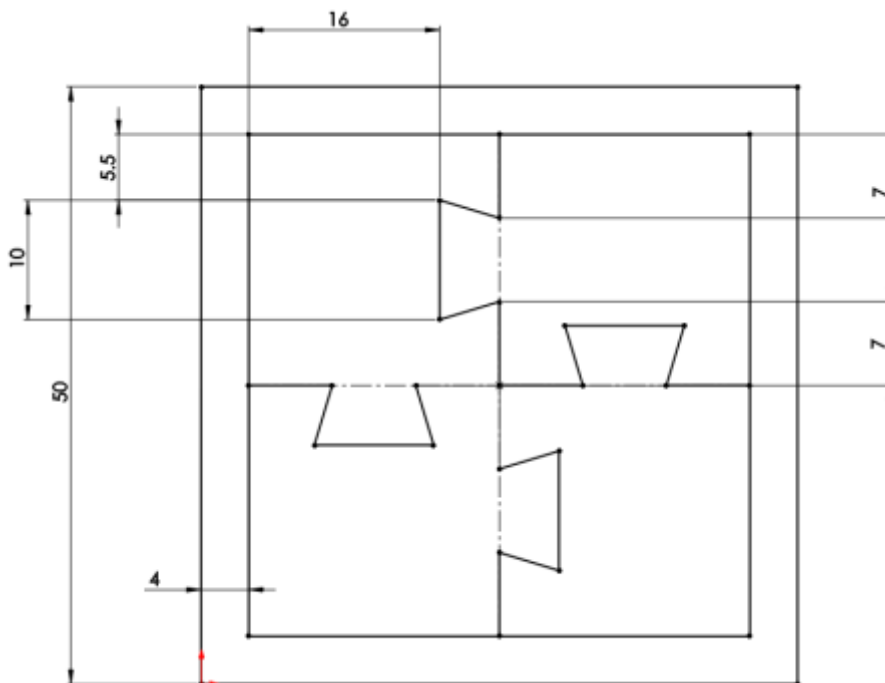
Worksheet 5

3-Axis Milling - Interlocking Jigsaw 1

3-Axis Milling Machine Task 2

This task involves producing a simple interlocking jigsaw using the MicroCNC 3-axis Milling Machine. You should use the following 2D Drawing and 3D model to:

1. Produce the part manually - this will involve marking out the stock material and manually operating the X, and Y-axes both independently and simultaneously; you will also control the depth of the cutting tool by manually operating the Z-axis.
2. Manually write G-code to your CAM software.
3. Re-create the drawing and model using your CAD package and export this to your CAM system.



Worksheet 5 (cont.)

3-Axis Milling - Interlocking Jigsaw 1



;Worksheet5 - 3-Axis - Manual Written G-Code

;Assuming 0,0,0 start origin at bottom left corner of stock, Z should be positioned so the mill bit is as close as possible to the top of the stock but not touching

;Absolute Coordinates

G21 ;mm Units
G90 ;Absolute Coordinates
G0 Z1 F10 ;Move Z axis up to 1mm to perform a rapid movement
G0 X4 Y25 F10 ;Move to absolute 4, 25 rapid movement

G1 Z-3 F2 ;Move Z axis down 3mm into material - Begin inner cut - Horizontal
G1 X11 F2 ;Move X+ by 7mm
G1 X9.5 Y20 F2 ;Move X- by 1.5mm Y- by 5mm
G1 X19.5 F2 ;Move X+ by 10mm
G1 X18 Y25 F2 ;Move X- by 1.5mm Y+ by 5mm
G1 X32 F2 ;Move X+ by 14mm
G1 X30.5 Y30 F2 ;Move X- by 1.5mm Y+ by 5mm
G1 X40.5 F2 ;Move X+ by 10mm
G1 X39 Y25 F2 ;Move X- by 1.5mm Y- by 5mm
G1 X46 F2 ;Move X+ by 7mm
G0 Z1 F2 ;Move Z axis up to 1mm to perform a rapid movement

G0 X25 Y4 F10 ;Move to absolute 25, 4 rapid movement

G1 Z-3 F2 ;Move Z axis down 3mm into material - Continue inner cut - Vertical
G1 Y11 F2 ;Move Y+ by 7mm
G1 X30 Y9.5 F2 ;Move X+ by 5mm Y- by 1.5mm
G1 Y19.5 F2 ;Move Y+ by 10mm
G1 X25 Y18 F2 ;Move X- by 5mm Y- by 1.5mm
G1 Y32 F2 ;Move Y+ by 14mm
G1 X20 Y30.5 F2 ;Move X- by 5mm Y- by 1.5mm
G1 Y40.5 F2 ;Move Y+ by 10mm
G1 X25 Y39 F2 ;Move X+ by 5mm Y- by 1.5mm
G1 Y46 F2 ;Move Y+ by 7mm
G0 Z1 F2 ;Move Z axis up to 1mm to perform a rapid movement

G0 X4 Y4 F10 ;Move to absolute 4, 4 rapid movement

G1 Z-3 F2 ;Move Z axis down 3mm into material - Outer perimeter cut - Double
check your clamps are not in the way!

G1 X46 F2 ;Move X+ by 42mm
G1 Y46 F2 ;Move Y+ by 42mm
G1 X4 F2 ;Move X- by 42mm
G1 Y4 F2 ;Move Y- by 42mm
G0 Z1 F2 ;Move Z axis up to 1mm to perform a rapid movement

G0 X0 Y0 F10 ;Move to absolute 0, 0 rapid movement
G0 Z0 F10 ;Move to absolute 0,0,0 rapid movement

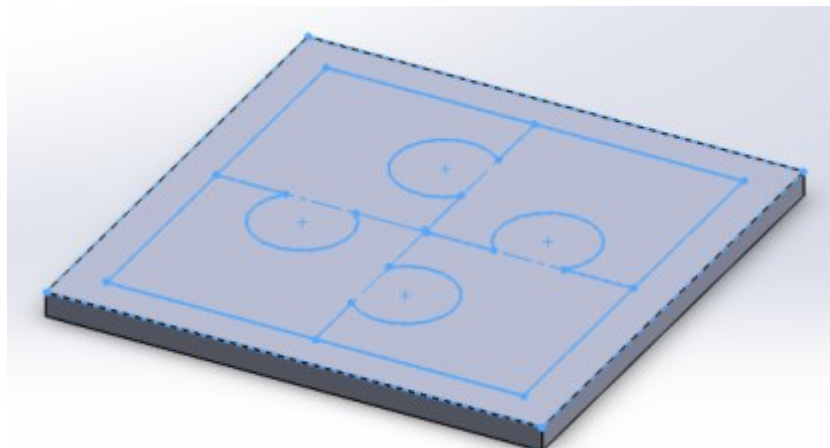
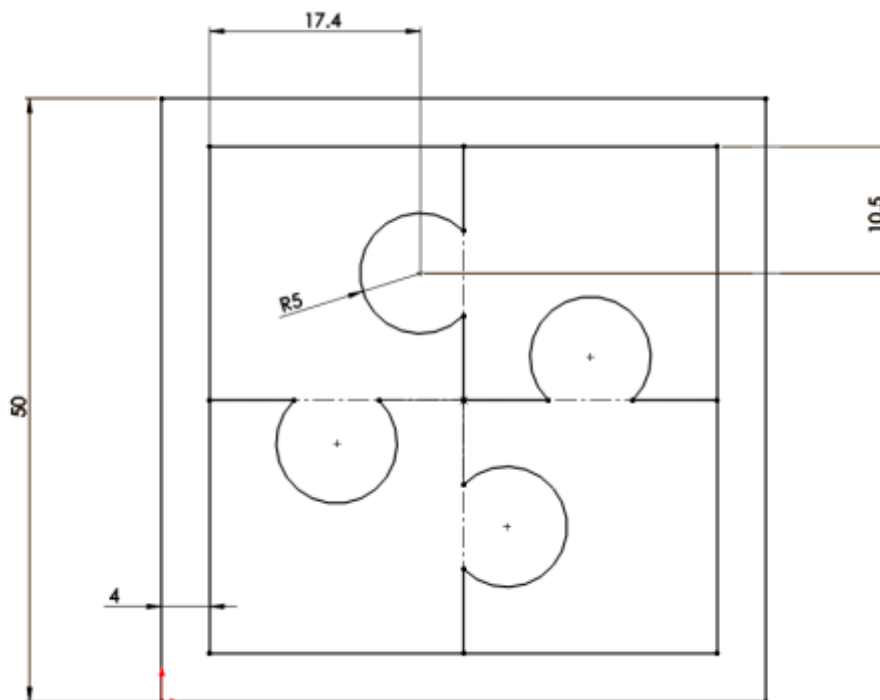
Worksheet 6

3-Axis Milling - Interlocking Jigsaw 2

3-Axis Milling Machine Task 3

This task involves producing a Complex interlocking jigsaw using the MicroCNC 3-axis Milling Machine. You should use the following 2D Drawing and 3D model to:

1. Produce the part manually - this will involve marking out the stock material and manually operating the X, and Y-axes both independently and simultaneously; you will also control the depth of the cutting tool by manually operating the Z-axis.
2. Re-create the drawing and model using your CAD package and export this to your CAM system.



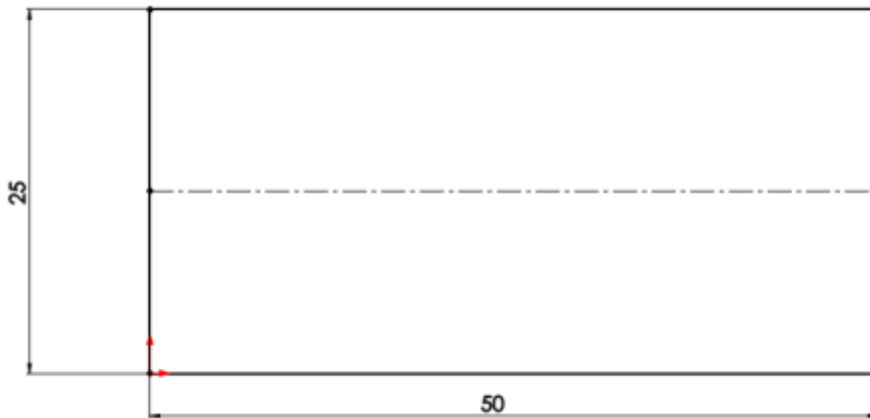
Worksheet 7

4-Axis Milling - Pen Holder

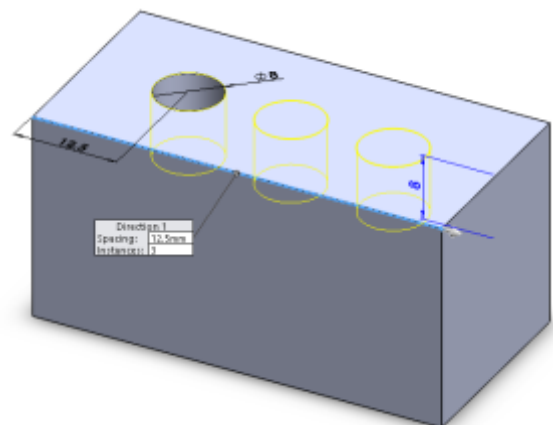
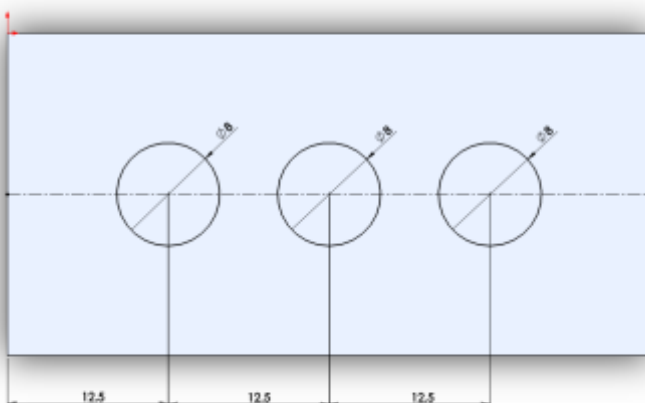
4-Axis Milling Machine Task 1

This task involves producing a small pen holder using the MicroCNC 4-axis Milling Machine. For this task you machine two faces of a 50 x 25 x 25 mm block. You should use the following 2D Drawings and 3D model to:

1. Produce the part manually - this will involve marking out the stock material and manually operating the X, and Y-axes both independently and simultaneously; you will also control the depth of the cutting tool by manually operating the Z-axis.
2. Manually write G-code to your CAM software.
3. Re-create the drawing and model using your CAD package and export this to your CAM system.



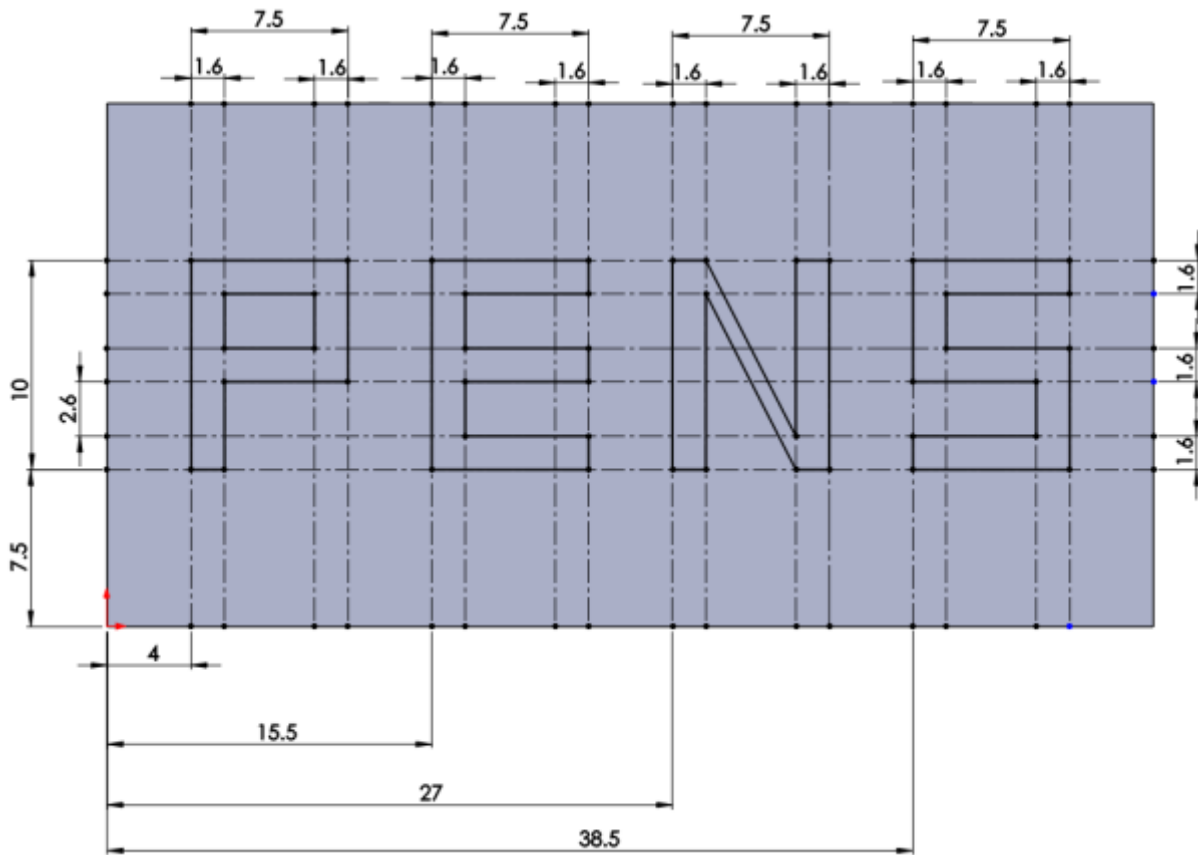
Block Dimensions



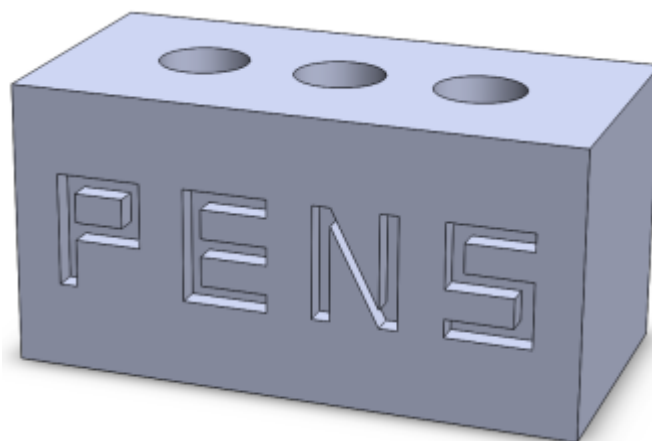
Hole Dimensions and Extrusion

Worksheet 7

4-Axis Milling - Pen Holder



Lettering Dimensions



Lettering Extrusion

Worksheet 7

4-Axis Milling - Pen Holder

;Worksheet7 - 4-Axis - Manual Written G-Code
;Assuming 0,0,0 start origin at bottom left corner of stock, Z should be positioned so the mill bit is as close as possible to the top of the stock but not touching
;Ensure 10mm of cutting mill is available
;Absolute Coordinates

G21 ;mm Units
G90 ;Absolute Coordinates
G0 Z1 F10 ;Move Z axis up to 1mm to perform a rapid movement
G0 X4 Y7.5 F10 ;Move to absolute 4, 7.5 rapid movement

G1 Z-2 F2 ;Move Z axis down 2mm into material - P cut
G1 Y17.5 F2 ;Move Y+ by 10mm
G1 X11.5 F2 ;Move X+ by 7.5mm
G1 Y12.5 F2 ;Move Y- by 5mm
G1 X4 F2 ;Move X- by 7.5mm
G0 Z1 F2 ;Move Z axis up to 1mm to perform a rapid movement

G0 X23 Y7.5 F10 ;Move to absolute 23, 7.5 rapid movement

G1 Z-2 F2 ;Move Z axis down 2mm into material - E cut
G1 X15.5 F2 ;Move X- by 7.5mm
G1 Y12.5 F2 ;Move Y+ by 5mm
G1 X23 F2 ;Move X+ by 7.5mm
G1 X15.5 F2 ;Move X- by 7.5mm
G1 Y17.5 F2 ;Move Y+ by 5mm
G1 X23 F2 ;Move X+ by 7.5mm
G0 Z1 F2 ;Move Z axis up to 1mm to perform a rapid movement

G0 X27 Y7.5 F10 ;Move to absolute 27, 7.5 rapid movement

G1 Z-2 F2 ;Move Z axis down 2mm into material - N cut
G1 Y17.5 F2 ;Move Y+ by 10mm
G1 X34.5 Y7.5 F2 ;Move X+ by 7.5mm Y- by 10mm
G1 Y17.5 F2 ;Move Y+ by 10mm
G0 Z1 F2 ;Move Z axis up to 1mm to perform a rapid movement

G0 X38.5 Y7.5 F10 ;Move to absolute 38.5, 7.5 rapid movement

G1 Z-2 F2 ;Move Z axis down 2mm into material - S cut
G1 X46 F2 ;Move X+ by 7.5mm
G1 Y12.5 F2 ;Move Y+ by 5mm
G1 X38.5 F2 ;Move X- by 7.5mm
G1 Y17.5 F2 ;Move Y+ by 5mm
G1 X46 F2 ;Move X+ by 7.5mm
G0 Z1 F2 ;Move Z axis up to 1mm to perform a rapid movement

G0 X55 Y0 F10 ;Move Cutting tool out of the way of rotation
G0 A90 F10 ;Move A axis 90 degrees rapid movement
G0 X6 Y8.5 F10 ;Move to absolute 6, 8.5 rapid movement

Worksheet 7

4-Axis Milling - Pen Holder

```
G1 Z-5 F2           ;Move Z axis down 5mm into material - Hole 1
G1 Y16.5 F2        ;Move Y+ by 8mm
G1 X7 F2           ;Move X+ by 1mm
G1 Y8.5 F2         ;Move Y- by 8mm
G1 X8 F2           ;Move X+ by 1mm
G1 Y16.5 F2        ;Move Y+ by 8mm
G1 X9 F2           ;Move X+ by 1mm
G1 Y8.5 F2         ;Move Y- by 8mm
G1 X10 F2          ;Move X+ by 1mm
G1 Y16.5 F2        ;Move Y+ by 8mm
G1 X11 F2          ;Move X+ by 1mm
G1 Y8.5 F2         ;Move Y- by 8mm
G1 X12 F2          ;Move X+ by 1mm
G1 Y16.5 F2        ;Move Y+ by 8mm
G1 X13 F2          ;Move X+ by 1mm
G1 Y8.5 F2         ;Move Y- by 8mm
G1 X14 F2          ;Move X+ by 1mm
G1 Y16.5 F2        ;Move Y+ by 8mm
G1 X6 Y8.5 F10     ;Move to absolute 6, 8.5
G1 Z-10 F2         ;Move Z axis down 10mm into material - Hole 1
G1 Y16.5 F2        ;Move Y+ by 8mm
G1 X7 F2           ;Move X+ by 1mm
G1 Y8.5 F2         ;Move Y- by 8mm
G1 X8 F2           ;Move X+ by 1mm
G1 Y16.5 F2        ;Move Y+ by 8mm
G1 X9 F2           ;Move X+ by 1mm
G1 Y8.5 F2         ;Move Y- by 8mm
G1 X10 F2          ;Move X+ by 1mm
G1 Y16.5 F2        ;Move Y+ by 8mm
G1 X11 F2          ;Move X+ by 1mm
G1 Y8.5 F2         ;Move Y- by 8mm
G1 X12 F2          ;Move X+ by 1mm
G1 Y16.5 F2        ;Move Y+ by 8mm
G1 X13 F2          ;Move X+ by 1mm
G1 Y8.5 F2         ;Move Y- by 8mm
G1 X14 F2          ;Move X+ by 1mm
G1 Y16.5 F2        ;Move Y+ by 8mm
G0 Z1 F2           ;Move Z axis up to 1mm to perform a rapid movement
```

Worksheet 7

4-Axis Milling - Pen Holder

```
G0 X21 Y8.5 F10      ;Move to absolute 21, 8.5 rapid movement

G1 Z-5 F2            ;Move Z axis down 5mm into material - Hole 2
G1 Y16.5 F2          ;Move Y+ by 8mm
G1 X22 F2            ;Move X+ by 1mm
G1 Y8.5 F2           ;Move Y- by 8mm
G1 X23 F2            ;Move X+ by 1mm
G1 Y16.5 F2          ;Move Y+ by 8mm
G1 X24 F2            ;Move X+ by 1mm
G1 Y8.5 F2           ;Move Y- by 8mm
G1 X25 F2            ;Move X+ by 1mm
G1 Y16.5 F2          ;Move Y+ by 8mm
G1 X26 F2            ;Move X+ by 1mm
G1 Y8.5 F2           ;Move Y- by 8mm
G1 X27 F2            ;Move X+ by 1mm
G1 Y16.5 F2          ;Move Y+ by 8mm
G1 X28 F2            ;Move X+ by 1mm
G1 Y8.5 F2           ;Move Y- by 8mm
G1 X29 F2            ;Move X+ by 1mm
G1 Y16.5 F2          ;Move Y+ by 8mm
G1 X21 Y8.5 F10      ;Move to absolute 6, 8.5
G1 Z-10 F2           ;Move Z axis down 10mm into material - Hole 2
G1 Y16.5 F2          ;Move Y+ by 8mm
G1 X22 F2            ;Move X+ by 1mm
G1 Y8.5 F2           ;Move Y- by 8mm
G1 X23 F2            ;Move X+ by 1mm
G1 Y16.5 F2          ;Move Y+ by 8mm
G1 X24 F2            ;Move X+ by 1mm
G1 Y8.5 F2           ;Move Y- by 8mm
G1 X25 F2            ;Move X+ by 1mm
G1 Y16.5 F2          ;Move Y+ by 8mm
G1 X26 F2            ;Move X+ by 1mm
G1 Y8.5 F2           ;Move Y- by 8mm
G1 X27 F2            ;Move X+ by 1mm
G1 Y16.5 F2          ;Move Y+ by 8mm
G1 X28 F2            ;Move X+ by 1mm
G1 Y8.5 F2           ;Move Y- by 8mm
G1 X29 F2            ;Move X+ by 1mm
G1 Y16.5 F2          ;Move Y+ by 8mm

G0 Z1 F2             ;Move Z axis up to 1mm to perform a rapid movement
```

Worksheet 7

4-Axis Milling - Pen Holder

```
G0 X36 Y8.5 F10      ;Move to absolute 36, 8.5 rapid movement

G1 Z-5 F2           ;Move Z axis down 5mm into material - Hole 3
G1 Y16.5 F2         ;Move Y+ by 8mm
G1 X37 F2           ;Move X+ by 1mm
G1 Y8.5 F2          ;Move Y- by 8mm
G1 X38 F2           ;Move X+ by 1mm
G1 Y16.5 F2         ;Move Y+ by 8mm
G1 X39 F2           ;Move X+ by 1mm
G1 Y8.5 F2          ;Move Y- by 8mm
G1 X40 F2           ;Move X+ by 1mm
G1 Y16.5 F2         ;Move Y+ by 8mm
G1 X41 F2           ;Move X+ by 1mm
G1 Y8.5 F2          ;Move Y- by 8mm
G1 X42 F2           ;Move X+ by 1mm
G1 Y16.5 F2         ;Move Y+ by 8mm
G1 X43 F2           ;Move X+ by 1mm
G1 Y8.5 F2          ;Move Y- by 8mm
G1 X44 F2           ;Move X+ by 1mm
G1 Y16.5 F2         ;Move Y+ by 8mm
G1 X36 Y8.5 F10     ;Move to absolute 6, 8.5
G1 Z-10 F2          ;Move Z axis down 10mm into material - Hole 3
G1 Y16.5 F2         ;Move Y+ by 8mm
G1 X37 F2           ;Move X+ by 1mm
G1 Y8.5 F2          ;Move Y- by 8mm
G1 X38 F2           ;Move X+ by 1mm
G1 Y16.5 F2         ;Move Y+ by 8mm
G1 X39 F2           ;Move X+ by 1mm
G1 Y8.5 F2          ;Move Y- by 8mm
G1 X40 F2           ;Move X+ by 1mm
G1 Y16.5 F2         ;Move Y+ by 8mm
G1 X41 F2           ;Move X+ by 1mm
G1 Y8.5 F2          ;Move Y- by 8mm
G1 X42 F2           ;Move X+ by 1mm
G1 Y16.5 F2         ;Move Y+ by 8mm
G1 X43 F2           ;Move X+ by 1mm
G1 Y8.5 F2          ;Move Y- by 8mm
G1 X44 F2           ;Move X+ by 1mm
G1 Y16.5 F2         ;Move Y+ by 8mm
G0 Z1 F2            ;Move Z axis up to 1mm to perform a rapid movement

G0 X0 Y0 F10        ;Move to absolute 0, 0 rapid movement
G0 Z0 F10           ;Move to absolute 0,0,0 rapid movement
```

About This Document

Version Control Statement



About this document:

Code: CP7449

Developed for the whole Micro CNC product range

<http://www.matrixsl.com/microcnc/>

Date	Release notes	Release version
August 2017	First version released	CP7449-01
November 2017	New pages addition	CP7449-02
23 08 2023	Reformatted to new style	