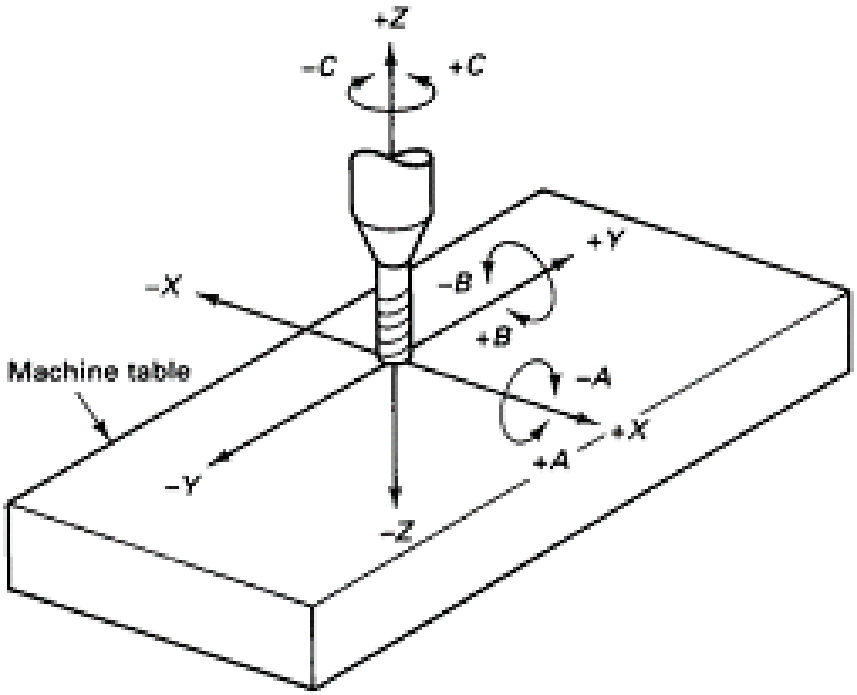
Machining Level IV

Based on October, 2023 Curriculum Version II



Module Title: Multiple Spindle and/or Multiple Axis NC/CNC Machining Centre

Module code: IND MAC4 M05 1023

Nominal duration: 80 Hours

Prepared by Ministry of Labor and Skills

October, 2023

Addis Ababa, Ethiopia

Acknowledgment

Ministry of Labor and Skills and Ministry of Health wish to extend thanks and appreciation to the many representatives of TVET instructors and respective industry experts who donated their time and expertise to the development of this Teaching, Training and Learning Materials (TTLM).

Acronyms

2D	Two Dimension
3D	Three Dimension
CAD	Computer-Aided Design
CAM	Computer- Aided Manufacturing
in.	Inch /Inches
M	Metre/s
Mm	Millimetre/s
Rpm	Revolution per minute

Table of Contents

Acknowledgment I

Acronyms II

Table of Contents III

Introduction to the module 1

Module Unit 1

Learning objectives of the Module 1

Module Learning Instructions 1

Unit one: NC/CNC machine program elements 2

1.1. Types of CNC Machine 2

1.2. Multi-axis NC machine 16

1.3. Selection of appropriate program elements for machines 21

1.4. Self-check - 1 23

Unit Two: Write NC/CNC machine program and operation sheet 24

2.1. Use of engineering drawings to define machine function and tool path geometry 24

2.2. Coordinate of tool path, machine function and points on the part to be produced 26

2.3. Plan and sequence of machining operations to be programmed 28

2.4. Selection of advanced operations in NC/CNC machine 33

2.5. *Standard* program *code* writing format 39

2.6. Operation sheet 45

2.7. Self-check - 2 46

Unit Three: Trial program 47

3.1. Hazards and control measures associated with NC/CNC machines 47

3.2. Manual mode Machines operation to test and prove the program 51

3.3. Editing the NC/CNC programing 54

3.4. Self-check - 3 58

3.5. Operation sheet – 3.1 59

3.6. LAP Test 59

Reference 60

Developer profile 61

Introduction to the module

This module defines the skills and knowledge required to identify numerically controlled (NC) and computer numerically controlled (CNC) machine program elements, write and trial a machine program for multiple spindle and multiple axis NC and CNC machining centres, and prepare an operation sheet. The program may use common M and G codes and include the programming of advanced operations, using canned cycles and sub-routines. Programs are trialled and edited as necessary to adjust operation of centre. Multi-axis linkage machining center is that numerical control system controls multiple axes to move coordinately and synchronously according to the requirements of functional relation.

Module Unit

This module covers the units:

- NC/CNC machine program elements
- Write NC/CNC machine program and operation sheet
- Trial program

Learning objectives of the Module

At the end of this session, the students will able to:

- Identify NC/CNC machine program elements
- Write NC/CNC machine program and operation sheet
- Trial program

Module Learning Instructions

For effective use this modules trainees are expected to follow the following instruction:

1. Read the information written in each unit
2. Accomplish the Self-checks at the end of each unit
3. Perform Operation Sheets which were provided at the end of units
4. Do the “LAP test” giver at the end of each unit and
5. Read the identified reference book for Examples and exercise

Unit one: NC/CNC machine program elements

This learning unit is developed to provide the trainees the necessary information regarding the following content coverage and topics:

- Types of NC/CNC machine
- Selection of ***appropriate program*** elements for machines

This unit will also assist you to attain the learning outcomes stated in the cover page. Specifically, upon completion of this learning guide, you will be able to:

- Select ***appropriate program*** elements for machine controller

1.1. Types of CNC Machine

Machines such as lathe, milling, drilling is done manually and have more responsibility of humans to perform and have a higher average of accuracy and precision in taking the time. So to resolve this problem, the machine manufactured CNC which is known as Computer Numerically controlled.

Computer numerical control (CNC) machines are widely used in the production industries. They have achieved great popularity in these recent years.

CNC machine defined as a machining system that is managed by a computer, they enable manufacturers to automate many manufacturing operations that would otherwise have to be done manually. As a result, CNC-powered machines are used to increase productivity, decrease waste, and even eliminate the risk of human error. CNC machines can produce a number of products by using actuators, drives, and software programs.

CNC Machines can be classified based on the motion type, on the control loops, on the number of axes and on the power supply. A list of the common CNC machining operations and the types of CNC machines is outlined next. There are many custom built CNC's which fit outside of these common types.

Types of CNC Machine

Following are the types of CNC machine:

1. Lathe CNC machine
2. Milling CNC machine
3. Drilling CNC machine
4. Grinding CNC machine
5. Laser cutting CNC machine
6. Plasma cutting CNC machine
7. Electric discharge CNC machine
8. Router CNC machine
9. CNC machine with automatic tool changes
10. 3-D printer
11. 5-axis CNC machine
12. Pick and place machine

1. CNC Lathe Machine

Lathe CNC machines are defined by their capability to turn materials during operation. They have less number of axes than CNC milling machines, making them shorter and more compact.



Figure1.1 : Lathe CNC machine

CNC lathe machines consist of a lathe at the center that manages and transfers material programmatically to the computer. At the present time, it is widely used as a lathe due to its

fast and accurate function. Once the initial setup is done, a semi-skilled worker can operate it easily. This type of lathe is also used for mass production such as capstan and turret. But there is no programmed fed system.

2. CNC Milling Machine

It is one of the most common types of CNC machine, that have built-in tools for drilling and cutting. The materials are located inside a milling CNC machine, after which the computer will lead the tools to drill or cut them.

Most of the CNC milling machines are available in 3 to 6-axis configurations. This machine is used to produce gears like spur gear and is also used to drill the workpiece bore and make slots by inserting part program into the system.

A semi-skilled worker can operate it easily. It is also used for mass production such as a capstan and turret. But there is no programmed fed system. The parts made by this machine are very precise in dimensional tolerance.



Figure: Milling CNC machine

3. CNC Drilling Machine

The CNC drilling machine is typically applied for mass production. Drilling machines, however, often have a multi-function machining center that is occasionally mingled and

sometimes twisted. The greatest sink time for CNC drilling is with tool changes, so for speed, the variation of hole diameter must be reduced.

The fastest machine size for drilling holes consists of several spindles in the turret with drills of different diameters pre-mounted for drilling. This type of CNC machine can perform reaming, counterboring, and tapping holes.



Figure: Drilling CNC machine

4. CNC Grinding Machine

It is a precision performance tool that uses a rotating wheel to cut metal away from the metal. Typically, CNC grinding machines are used for camshafts, ball bearings, transmission shafts, and other working pieces, which require an accurate and correct finish. Numerous pieces made using a CNC grinding machine are cylindrical. A grinding machine also use to make other types of workpieces. In CNC grinding machines “CNC” holds for computerized numerical control.



Figure: Grinding CNC machine

5. CNC Laser cutting machine

The Laser-cutting CNC machines are designed to cut through hard materials, although they use a laser to perform this task instead of a plasma torch. Lasers offer a high degree of accuracy, but they are not as effective as plasma torches.

Laser-cutting CNC machines commonly use one of these three types of lasers that is CO2, neodymium (Nd), or yttrium-aluminum-garnet (Nd: YAG).



Figure: Laser cutting CNC machine

6. CNC Plasma Cutting Machine

This machine similar to milling CNC machines, plasma-cutting CNC machines are also used to cut materials. But, they differ from their milling counterparts by doing this operation applying a plasma torch.

A plasma cutting machine is defined as it is a method that cuts by electrically conductive materials using an accelerated stream of hot plasma. These types of CNC machine feature a high-powered torch that's capable to cut through rough materials like metal.



Figure: Plasma cutting CNC machine

7. CNC Electrical Discharge Machine

These are also known as a spark CNC machine, is a special type of CNC machine that uses electric sparks to manage materials to the desired shape.

Electrical discharge has a transient effect, meaning that it can degrade the elements from which it is presented. The electric discharge CNC machines take advantage of this by designing controlled sparks to reshape materials.

The materials are located between the top and bottom of the electrode, after which the computer decides how much electrical discharge the electrodes produce.



Figure: Electric discharge CNC machine

8. CNC Router Machine

The CNC router machine is works likewise to other CNC machines such as lathe or milling. The principal difference is that all of this carpenter works manually such as interior, door carving, and exterior decoration, wood panels, signboards, wood frames, molding, musical instruments, furniture, and more.

According to the sketch, you can design if possible you can create in the system then execute it on a CNC router machine. This provides a better surface finish. It would be a very good machine in terms of door design and more.

9. CNC Machine with Automatic Tool Changes

Automatic tool changer CNC machine is applied to develop the tool carrying capacity and production rate of the machine. And also used to improve the machine's capacity to work with a number of tools. This type of CNC machine changes the tool very quickly and easily handles and change worn out or broken tools.

The most important thing about this CNC machine is that it reduces non-productive time. The Automatic tool changer CNC machine is a perfect step towards complete automation.

10. 3-D Printer

It is a CNC machine in which elements are printed layer-by-layer. The design and drawing are created by the CAD and CAM process and then the 3D printer used to reflect that design. The 3D printer CNC machine used for the printing of buildings and premises.



Figure: 3-D printer

11. 5-Axis CNC Machine

The 5-Axis CNC machine is a machine with a total of 5 axes. Originally, it has 3 axes (X, Y, and Z), the cutting operation of any tool has done in 3 directions, but 2 additional axes were added (A, B) which means there are 5 axes. This machine is used to produce sculptures.



Figure: 5-axis CNC machine

12. Pick and Place Machine

This is another type of CNC machine. It is used in the industry. The pick and place CNC machine has many nozzles that lift electrical components for electronic equipment and place them in the correct location. They are generally used in the manufacture of cell phones, computers, and tablets, etc.



Figure: Pick and place machine

1.1.1. Classification of CNC machines based on the motion type

A. Point-to-Point Systems

1. It is used in some CNC machines such as drilling, boring and tapping machines…etc.
2. The control equipment for use with them are known as point-to-point control equipment.
3. Feed rates need not to be programmed.
4. In theses machine tools, each axis is driven separately.

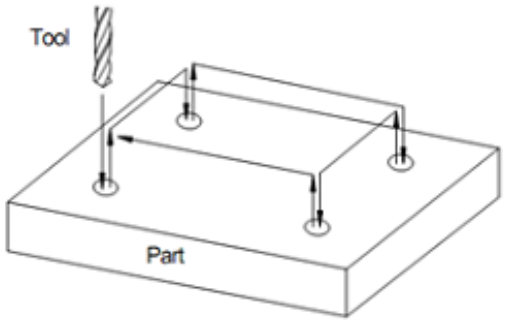


Figure: Point-to-Point Systems

B. Contouring Systems (Continuous Path Systems)

- It is used in CNC machine tools such as milling machines.
- These machines require simultaneous control of axes.
- Contouring machines can also be used as point-to-point machines, but it will be uneconomical to use them unless the work piece also requires having a contouring operation to be performed on it.

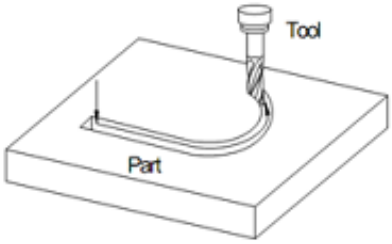


Figure: Continuous Path Systems

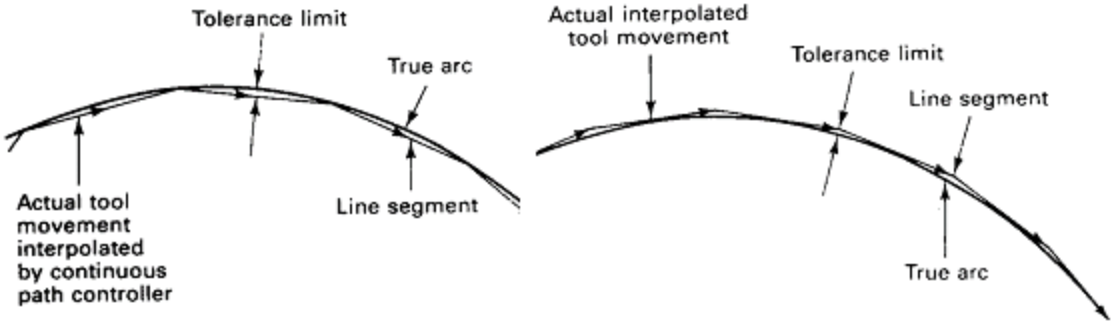


Figure: Interpolation used for continuous path movement

1.1.2. Classification of CNC machines based on the control loops

A. Open Loop Systems

- Programmed instructions are fed into the controller through an input device.
- These instructions are then converted to electrical pulses (signals) by the controller and sent to the servo amplifier to energize the servo motors.
- The primary drawback of the open-loop system is that there is no feedback system to check whether the program position and velocity has been achieved.
- The open -loop system is generally used in point-to-point systems where the accuracy requirements are not critical.
- Very few continuous-path systems utilize open-loop control.

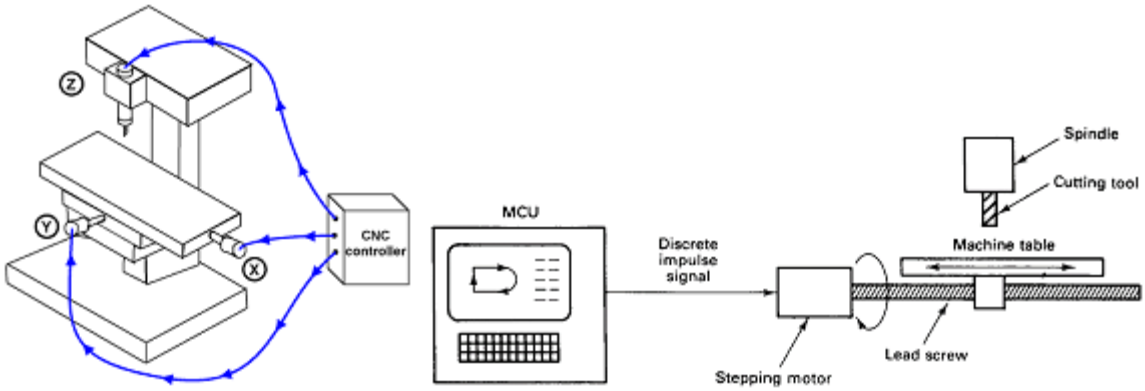


Figure: Configaration for an Open Loop Systems

B. Closed Loop Systems

- The closed-loop system has a feedback subsystem to monitor the actual output and correct any discrepancy from the programmed input.
- These systems use position and velocity feed back.
- The feedback system could be either analog or digital.
- The analog systems measure the variation of physical variables such as position and velocity in terms of voltage levels.
- Digital systems monitor output variations by means of electrical pulses.
- 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.

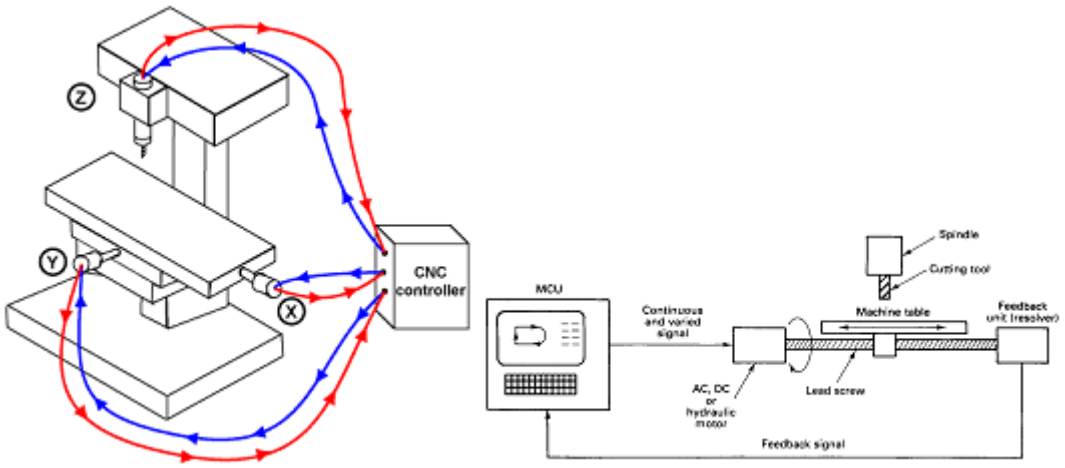


Figure: Configaration for Closed Loop Systems

1.1.3. Classification of CNC machines based on the power supply

- Mechanical power unit refers to a device which transforms some form of energy to mechanical power which may be used for driving slides, saddles or gantries forming a part of machine tool.
- The input power may be of electrical, hydraulic or pneumatic.



Figure: AC Servo Motor for CNC Machine Tool

1.1.4. Classification of CNC machines based on the number of axes

CNC machines will have movement that is either linear, or straight line, in motion or rotational. Typically the X, Y and Z axis are used for linear movements, whereas A, B or C will correspond to rotational axis, with A rotating about the X axis, B about the Y axis, and C about the Z axis.

CNC machine designs will vary greatly. On some styles, the part will move or rotate while the spindle and tool remains stationary, in others the part is stationary and the spindle and tool moves, and a third type is a mixture of either the part and spindle moving.

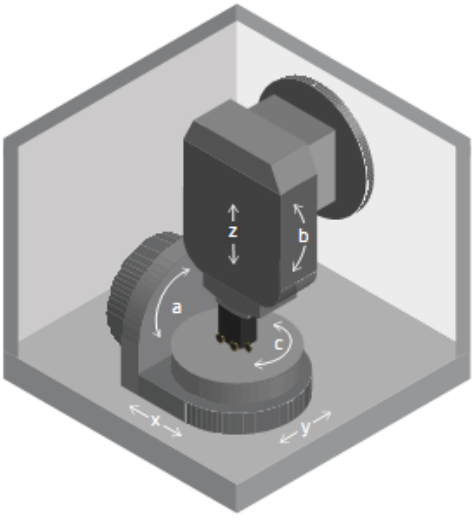


Figure: CNC machine movement

A. Two and three axes CNC Machines

- CNC lathes will be coming under 2 axes machines. There will be two axes along which motion takes place.
- The saddle will be moving longitudinally on the bed (Z-axis) and the cross slide moves transversely on the saddle (along X-axis).
- Machine axis defined as spindle movement
- In 3-axes machines, there will be one more axis, perpendicular to the above two axes.
- By the simultaneous control of all the 3 axes, complex surfaces can be machined.

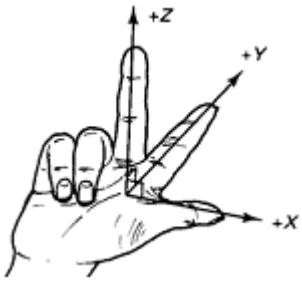
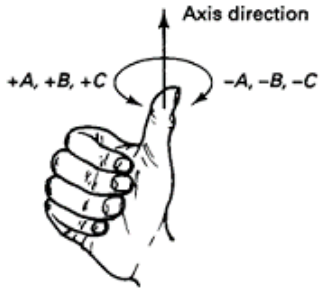
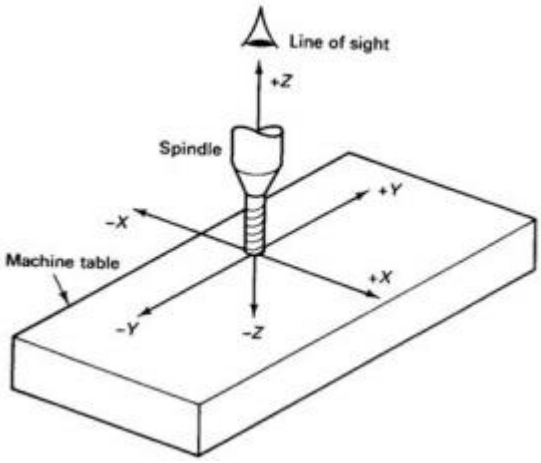
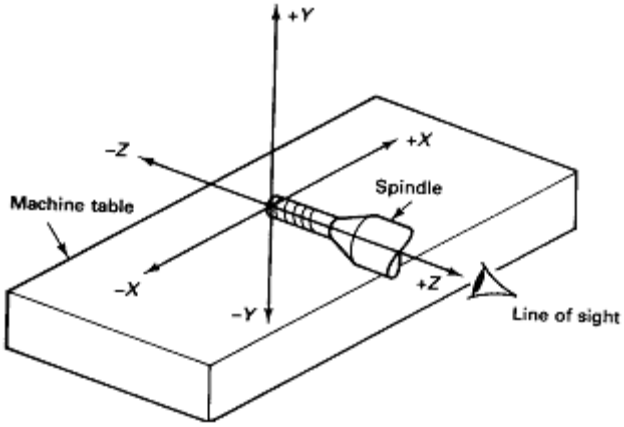
<table>
<tr><td>
</td><td>
</td></tr>
<tr><td>(a) The right –hand rule for linear motion</td><td>(b) The right–hand rule for rotary motion</td></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td>(c) Machine axis for a three–axis vertical CNC machine</td><td>(d) Machine axis for a three–axis horizontal CNC machine</td></tr>
</table>

Figure: Machine axis defined as spindle movement

3-Axis Machining are the most common CNC machines produced commercially. They can move in the X, Y and Z axes simultaneously. The most common size of the 3 axis machine is one with a work table of 40x20 inches. Work performed on 3-axis machine can be simple prismatic parts, all the way to complex molds, medical or aerospace parts.

B. 3+1 or 3+2 Axis Machining

This type of machining, and CNC machine, combines rotational movements with linear movements. In this type of machining, the part will first rotate into position, and then a typical 3-axis machining operation will commence after rotation. The rotation and linear movement does not happen simultaneously.

If the machine supports rotation around just one axis, then it is considered 3+1 machining, for 3 linear axis and one rotational axis. If the machine can rotate about two axes, it is referred to as a 3+2 axis machining.

CNC machines can be purpose built as a 3+1 or 3+2 axis mill, but more commonly, they are created by adding a rotating component to an existing 3-axis mill. The latter method will reduce the work envelop size, so that should be taken into consideration when outfitting a mill.

One major advantage to 3+1 and 3+2 machining is in reducing individual setups when milling a part. Some parts may require milling on different sides, and on a 3-axis machine, the machinist would remove and reposition the part between operations. This can sometimes take a long time, where it could rotate into position automatically with 3+1 or 3+2 functionality.

Note that setting up and programming 3+1 and 3+2 axis machines manually, versus programming with a Computer Aided Manufacturing (CAM) system is often difficult, time consuming and prone to errors.

C. 4-Axis Simultaneous

The difference in 3+1 and 4-axis simultaneous is that the part can rotate on one axis at the same time it is milling in 3-axis. This type of machining is necessary when part geometry wraps around a part, common in the aerospace and other industries.

While all 4-axis simultaneous milling machines can perform 3+1 axis milling, not all 3+1 axis mills can do 4-axis simultaneous. One needs to make sure of their machine’s capabilities before trying to run a program it may not support.

D. 5-Axis Machining

This type of machining allows for the most flexibility in milling the part at different tool orientations. Here, the part can rotate in two separate axes, usually A&C or B&C, at the same time the 3 linear axes are moving. Machining in simultaneous 5-axis is popular in parts where complex shapes, and small clearances, are the norm; such as impellers, turbines, intake ports and more.

True simultaneous 5-axis machines can run in all the previous modes; 3-axis, 3+1 axis, 3+2 axis, 4-axis simultaneous plus 5-axis simultaneous. While all 5-axis simultaneous machines can perform 3+2 axis machining, not all 3+2 axis machines can do 5-axis simultaneous. It is a function of the machine's design and the CNC controller installed. These mills are typically more expensive than the other mill types, for the same size, but offer the most flexibility when setting up your jobs.

1.2. Multi-axis NC machine

Machine tool is the symbol of a country's manufacturing level. Modern NC machining is developing towards higher speed, precision, intelligence, flexibility, automation and reliability. Multi-axis NC machine tools reflect the above points. Multi-axis linkage machining center is that numerical control system controls multiple axes to move coordinately and synchronously according to the requirements of functional relation. According to the number of axes, numerical control machines are divided into two-axis linkage, two and a half axes linkage, triaxial linkage, four-axis linkage, five- axis linkage, etc. As the design complexity of workpieces is higher and higher, the use of triaxial linkage and four-axis linkage can't meet the demand of many complex special-shaped products. Machining efficiency and machining precision are improved through multi axes that more than five linkage machining center, but they are more strict with professionals and technology, and also increase the processing cost, narrow the adaptations.

In order to design and develop the multi-axis machining center for complex parts, the principle of the typical five-axis machining center is set to be the foundation. Its processing results showed that the machining quality and efficiency have improved distinctly, which means that five-axis linkage machining center has the strong flexibility and superiority in

machining processing. The principle of five-axis linkage machining center is typical to multi-axis linkage machining center and its adaptability is strong enough.

Multi-axis linkage CNC machine tools are the concentrated expression of variety process of numerical control technology, and they should meet all kinds of complicated surface parts processing. With the number of axes for linkage increasing, however, the cost will be higher, the structure will be more complicated, and the technician requirements will be more stringent, the software programming will be more difficult.

With the progress of science and technology, product parts machining processes are more and more difficult, and the machining accuracy is becoming higher and higher. The application of multi-axis linkage machining center is the main way to solve these problems. As a major means to solve special-shaped complex, which contains complex curved surface and crank shaft and so on, five-axis linkage machining center is very representative in multi-axis linkage machining center.

1.2.1. The Principle of Multi-Axis Linkage Machining Center

Multi-axis NC machining mainly include three aspects: (1) physical process simulation of machining and product quality forecasting, (2) dynamic response of process system and stability analysis, (3) efficient NC machining process planning. Five-axis linkage machining center is typical at all aspects of multi-axis linkage machining center, because it represents the highest state of the machine tool manufacturing industry at present. Detailed introductions of five-axis linkage machining center's principles are as follows.

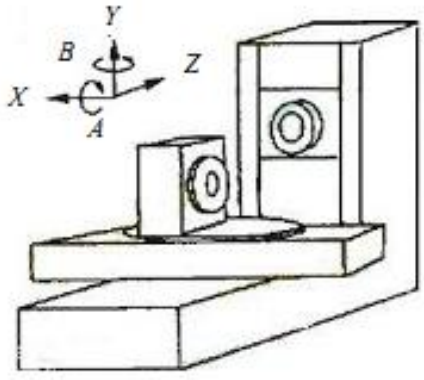


Figure 1 The principle of horizontal five-axis linkage machining center

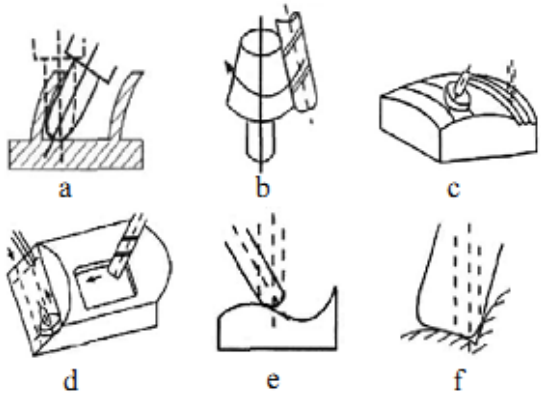


Figure 2 The characteristics of five-axis linkage machining

The ***characteristics of five-axis linkage machining center*** are shown in Figure 2: (a) It can avoid cutter interference, and have wide adaptability and process complex parts which is difficult for triaxial linkage machining center. (b) For ruled surface parts, it can use flank milling to processing by a formation shape in good quality and high efficiency. (c) As for general multiple-layered surfaces, especially for relatively large flat surfaces, it is available to use large diameter milling cutter to approach the surface. In this way, the feeding of the cutter is fewer; the residual height of the workpiece is smaller. It improves the machining efficiency and surface quality greatly. (d) The benefit of five-axis machining center is having the ability to machine complex shapes in a single set-up. It is helpful to improve the positional precision of the surface and to gain high efficiency. (e) When five-axis machining center works, the cutter can be in the most efficient cutting state to the workpiece's surface relatively. (f) Under certain machining conditions, it's good to use large size of cutting tool to avoid the interference, and the cutters are in good condition of rigidity at the same time, this method improves the machining efficiency and precision.

1.2.2. The Structure of the Machining Center

Most five-axis linkage machining centers are the structure of 3 + 2. There are three straight axes and two of the three ratate as the rotation axes. According to the three straight axes and the combination of two of three rotation axes, five-axis linkage machining center has three types, they are X, Y, Z, A, B (Figure 1); X, Y, Z, A, C; X, Y, Z, B, C. By the form of the combination of the two axes of rotation, there are three kinds of forms: double rotating table type, turntable oscillating type and double pendulum type. Because of physical reasons, the size of the specifications of the machine tool and the machining range of objects are decided by the three kinds of structure forms.

1. Head 2. The tail 3. Head frame base 4. End bracket base 5. Z1 axis drag plate	6. Z2 axis drag plate 7. Z axis base 8. X2 axis drag plate 9. Bed 10. X1 shaft dragging plate	11. Y dragging plate 12. B axis 13. Stent 14. Milling spindle 15. Knife library 16. Knife library supporter

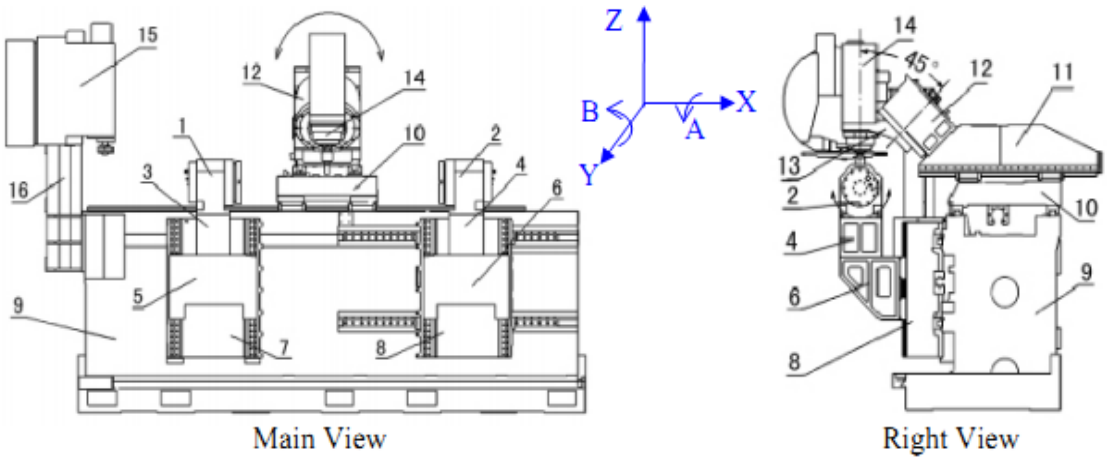


Figure 3 The structure of five axis machining center

Importance of higher axes machining:

- Reduced cycle time by machining complex components using a single setup.
- In addition to time savings, improved accuracy can also be achieved as positioning errors between setups are eliminated.
- Improved surface finish and tool life by tilting the tool to maintain optimum tool to part contact all the times.
- Improved access to under cuts and deep pockets. By tilting the tool, the tool can be made normal to the work surface and the errors may be reduced as the major component of cutting force will be along the tool axis.
- Higher axes machining has been widely used for machining sculptures surfaces in aerospace and automobile industry.

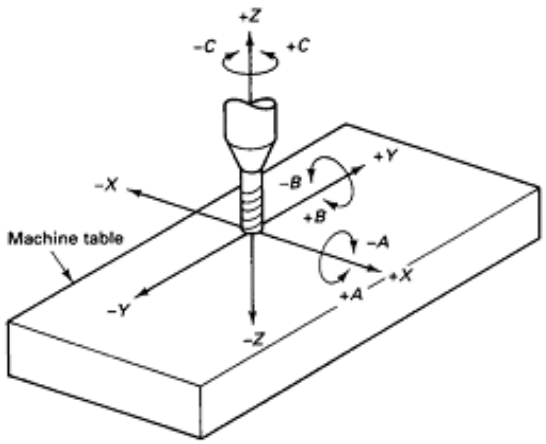


Figure: Machine axis for a six–axis vertical CNC machine

1.2.3. The Application of Multi-Axis Linkage Machining Center

Multi-axis linkage machining center can solve spatial arbitrary curved surface along the surface normal, spatial twisted curved surface machining interference and turning parts processing. The application of multi-axis linkage machining center is very widely, covering ship manufacturing, aerospace and other industrial fields. In terms of turbine blade, because of the complex curved surface, certain bending and distortion degree, Machining blade is difficult but also important.

The five-axis linkage machining center can solve the problems of impact between the blade and the cutter, and ensure the process surface without overcut and dead angles when the machine processing the blade.

A five-axis linkage machining center of turbine blade is shown in Figure 3. This five-axis linkage machining center is equipped with professional programming software. The machining center is composed of straight axes and rotating shafts. Straight axes are the X, Y, Z. Rotating shafts are the blade rotating shaft A and the tool swinging shaft B.

In this machining center, the routes of the Y axis and the Z axis are shorter than usual; the X axis is longer in order to adjust the length of the blade. The purpose of the X2 drag plate is to adjust the distance between the Z1 and Z2 to adapt to the length of the blade, it’s not moving at the time of processing; Z1 drag plate and Z2 drag plate clamp blade and move in a linear motion with the same direction and the same rate at the same time; A shaft and vice-A shaft which combined as one as the rotating shaft, rotate at the same time the same direction and the same rate, as a result, the pulling force is produced, the machining precision is improved and the distortion is decreased; In order to maintain the standard of processing range and avoid the influence of torque on the rigidity of the B axis, the angle between the B axis and the horizontal is 45°, the rotation range of the B axis is ±50°.

Through the case analysis, the five-axis linkage machining center has complex structure and needs professional programming software badly. From the point of the result, the five-axis linkage machining center leads to better quality and efficiency than four-axis linkage and three-axis linkage machining center. From this point of view, the more the machine axes are,

the more complicated the parts is, and the higher the machining precision is. Along with the development of technology, five-axis linkage machining center is no longer the most advanced multi-axis linkage machining center. More and more multi-axis linkage machining center need to be developed to adapt to wider range of requirements.

1.3. Selection of appropriate program elements for machines

The CNC System of Machining Center is the brain of multi-axis machining center, and also the main element to decide functionality and performance of the machining center. Numerical control system mainly includes three aspects: (1) fine interpolation of multi-axis NC machining, (2) interaction of equipment and process, (3) closed loop control of NC machining process.

Essentially speaking, based on the input of the numerical control procedures, five-axis linkage machining center CNC system interpolates the ideal trajectory, and then outputs the orders to the executive parts of the machining center to machining workpieces that needed. The basic function of the NC system includes 5 modules: the input, the trajectory interpolation, the servo control, the feedback control and the automatic tool change. It's an enclosed control system with position control as well. To ensure that the operating components are on schedule, the feedback control system tracks the performance of the implementation of the measurement in real time, and puts the feedback of the possibly error signals to the control system, then combines those possibly error signals with the next input signals as the output signals to executive parts.

The high-grade CNC system should meet the following basic elements: (1) the multi-channel and multi-axis linkage; (2) the functions of space tool compensation and optimization; (3) full or double closed-loop control ability; (4) interpolation period < 0.5 ms; (5) predictive control > 1000 procedures section; (6) data operation and switching unit < l nm.

Elements of Part program

- Preparatory functions: which unit, which interpolator, absolute or incremental programming, which circular interpolation plane, cutter compensation, etc.,
- Coordinates: 3 translational and 3 rotational axis

- Machining parameters: feed, and speed
- Tool control: next tool number, tool change
- Cycle functions: canned cycle, drill cycle, ream cycle, bore cycle, mill cycle
- Miscellaneous functions: coolant on/off, spindle on/off, programme rewind, spindle rotation direction, etc.

1.4. Self-check - 1

Directions: Attempt to answer all the questions listed below. Illustrations may be necessary to aid some explanations/answers. Note: Be precise and scientific in writing.

Part I - Instructions: For the following questions say **True** or **False.**

1. 1
2. 2

Part II: Multiple Choice - Instructions: For each of the following question **choose the best correct answer** and **circle the letter** of your choice from the given alternatives.

1. 1

   a. A b. B

   c. C d. D

2. 2

Part III: Marching - Instructions: For the following questions match from **"B" to "A" column**

No.	"A" Column	"B" Column
_____ 1.		B.
_____ 2.		C.
_____ 3.		D.

Part IV: Short answer – Instructions answer all the questions listed below.

1. 1
2. 2

Unit Two: Write NC/CNC machine program and operation sheet

This learning unit is developed to provide the trainees the necessary information regarding the following content coverage and topics:

- Use of Engineering drawings to define machine function and tool path geometry
- Coordinate calculation of tool path or machine functions and relevant points on the part or product to be produced
- Plan and sequence of machining operations to be programmed
- Selection of advanced operations using canned cycles and sub-routines in the particular NC/CNC machine
- ***Standard*** program ***code*** writing format
- Procedures for producing standard ***operation sheets*** of specific NC/CNC machine operation

This unit will also assist you to attain the learning outcomes stated in the cover page. Specifically, upon completion of this learning guide, you will be able to:

- Use engineering drawings to define machine function and tool path geometry
- Calculate coordinates required for tool path or machine functions.
- Select and apply advanced operations using canned cycles and sub-routines as required
- Write the program in ***standard code*** format with standard operating procedures
- Produce ***operation sheets*** to the specification and standard operating procedures

2.1. Use of engineering drawings to define machine function and tool path geometry

Before starting to program a job, it is important to become familiar with the part to be produced. From the engineering drawings, the programmer should be capable of planning the machining sequences required to produce the part. Visual concepts must be put into a written manuscript as the first step in developing a part program. It is the part program that will be sent to the machine control unit by the computer, tape, diskette, or other input media.

The programmer must first establish a reference point for aligning the workpiece and the machine tool for programming purposes. The manuscript must include this along with the types of cutting tools and work-holding devices required, and where they are to be located.

The first step in producing a CNC program is to take the information from the print and produce a program manuscript.

Operator's Manual - shall provide a complete description of the function, operating range and capacity of the machine, including drawings, diagrams, etc., of the location and operation of all indicating, operating, and control devices and instructions on use of all control devices. This manual shall keynote safety hazards and cautions.

The documentation required for CNC machine operation in terms of

a. the main forms of documentation required
    i. working diagrams /drawings
    ii. operation sheet
    iii. tool and work holding data sheets showing
        A. tools and mounting devices
        B. data required for pre-setting
        C. special tool specifications
        D. types of work holding devices
    iv. speed and feed data
    v. part programs showing data for
        A. position
        B. sequence
        C. tooling
        D. speeds and feeds
b. the essential differences between working drawings for conventional machinery and those required for CNC machine operation
    i. specification of datum
    ii. coordinate dimensioning (absolute, incremental).

Describe the factors to be considered when safe toolpath planning

a. ensure toolpath avoids collision with workholding or locating devices

b. care with offset toolpaths

c. collision free zones

d. use of manual tool change position

e. safety of operator, equipment and work.

2.2. Coordinate of tool path, machine function and points on the part to be produced

Coordinate calculation of tool path or machine functions and relevant points on the part or product to be produced

Dimensioning Guidelines

The system of rectangular coordinates is very important to the successful operation of CNC machines. Certain guidelines should be observed when dimensioning parts for CNC machining. The following guidelines will insure that the dimensioning language means exactly the same thing to the design engineer, the technician, the programmer, and the machine operator.

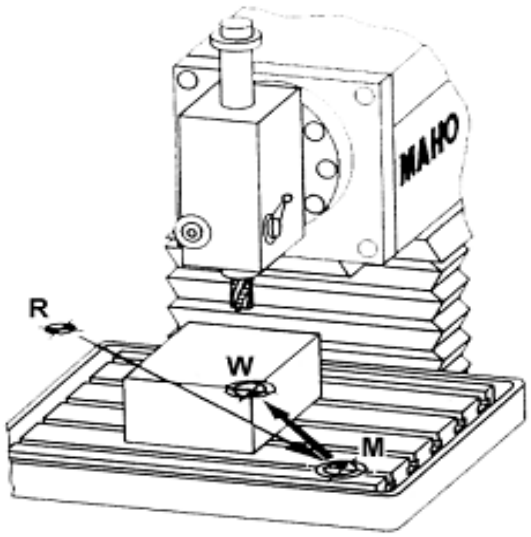
1. Define part surfaces from three perpendicular reference planes.
2. Establish reference planes along part surfaces which are parallel to the machine axes.
3. Dimension from a specific point on the part surface.
4. Dimension the part clearly so that its shape can be understood without making mathematical calculations or guesses.
5. Define the part so that a computer numerical control cutter path can be easily programmed.

Machine Zero Point

The machine zero point can be set by three methods - by the operator, manually by a programmed absolute zero shift, or by work coordinates, to suit the holding fixture or the part to be machined.

Absolute Zero Shift - The absolute zero shift can change the position of the coordinate system by a command in the CNC program.

Manual Setting - The operator can use the MCU controls to locate the spindle over the desired part zero and then set the X and Y coordinate registers on the console to zero.



R = Reference point (maximum travel of machine)

M = Machine zero point (X0,Y0,Z0) of machine coordinate system.

W = Part zero point workpiece coordinate system.

Figure 18 The relationship between the part zero and the machine system of coordinates. (Deckel Maho, Inc.)

7. Calculate cutter path coordinates for
    a. intersections
    b. polar centres
    c. arcs
    d. blind points
    e. cutter approach/exit.

8. Describe cutter compensation
    a. state the use of cutter compensation
    b. relate cutter compensation to
        i. tool diameter
        ii. tool length
        iii. tool nose radius.

9. Define the terminology relating to speeds and feeds and perform relevant calculations

a. define the terms

i. rpm

ii. cps

iii. feed/rev

iv. mm/min

b. calculate appropriate feeds and speeds for work and tool materials

c. state the need for modifying feed rates whilst profiling internal and external arcs.

4. Describe applications of feeds, speeds and depth of cut

a. convert surface metre per minute cutting speed to rpm of workpiece on a lathe using calculator and formula

b. convert surface metre per minute cutting speed to rpm on a milling cutter using calculator and formula

c. convert feed rate from metres/minute, on a lathe, to mm/revolution using calculator and formula

d. convert feed rate from mm/tooth, for a multi-tooth milling cutter, to metres/minute using calculator and formula

e. state, for tools with differing rake angles and applied cutting speeds, which would be used for

i. mild steel

ii. aluminium

f. identify cutting speed and feed rate in an operations sheet.

2.3. Plan and sequence of machining operations to be programmed

Process planning is a task comprising a broad range of activities to design and develop an appropriate manufacturing process for producing a part. Interpretation of the part design, selection of manufacturing processes, definition of operations, operation sequences, machining datums, geometrical dimensions and tolerances are some common activities associated with the task. Process planning is also “the link between product design and

manufacturing" with the supplementary commission to support design of competitive products.

Process planning is of a complex and dynamic nature, often managed by a skilled person with few, or no, explicit methods to solve the task. The work is heuristic and the result is depending on personal experiences and decisions. Since decades, there have been plenty of attempts to develop systems for computer-aided process planning (CAPP). CAPP is still awaiting its breakthrough and one reason is the gap between the functionality of the CAPP systems and the industrial process planning practice.

Process planning process

To clarify the general conception of process planning it is appropriate to make some more statements and explain the desired goal for the process planner.

Process planning is work that includes all needed activities to define a process plan. A process plan is a specification that contains information aimed to facilitate production of a certain part in a definite manufacturing system. In practice, two tacit boundary conditions narrow this definition; the process plan must be defined in a way that ensures that the produced part fulfils all requirements stated in the design specification as well as minimising the production cost.

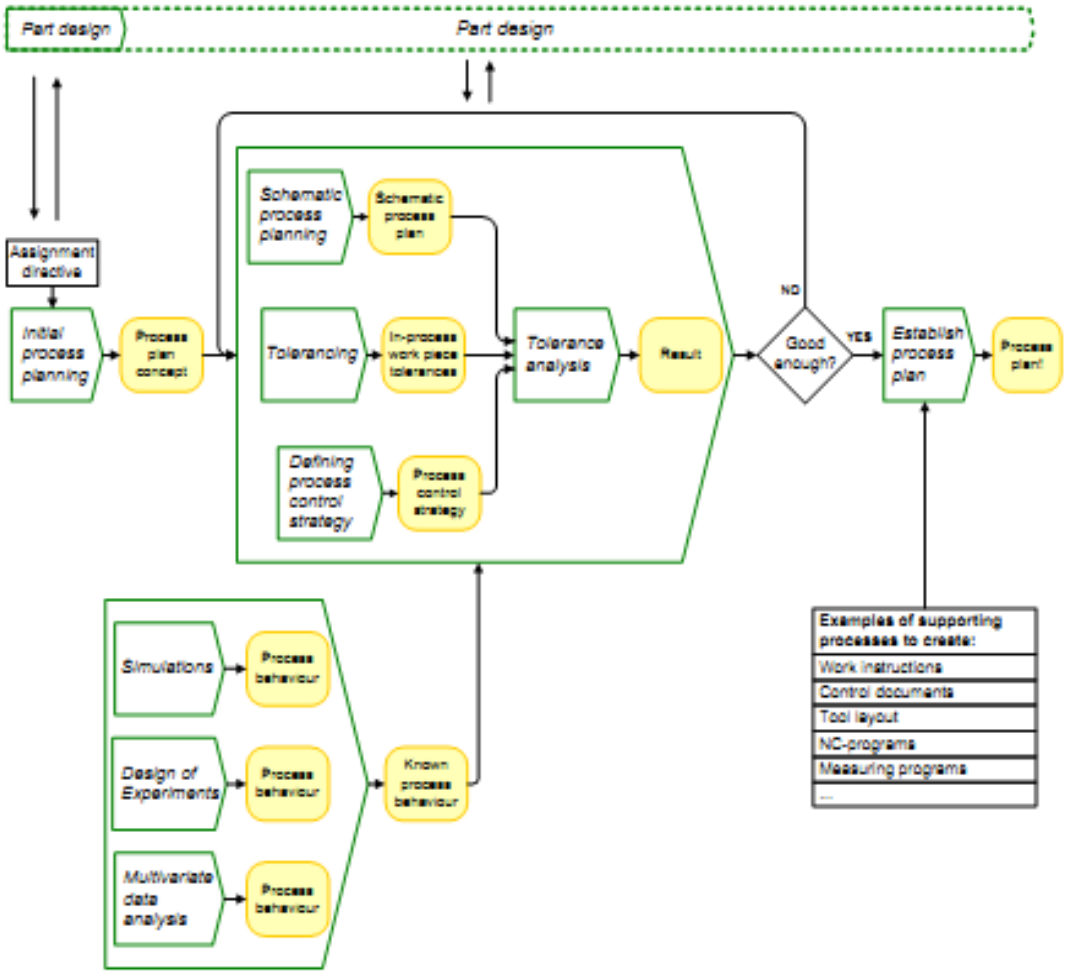


Figure 2: The PPP flow chart an approach to describe the process planning process (PPP).

According to Figure 2, the main flow starts with the activity “initial process planning” on receiving an assignment directive. The result is a process plan concept where the main directions of the following work have been set. This is used together with knowledge about the behaviour of relevant manufacturing processes as input to the three following process planning activities.

These three activities are interdependent and are performed in an iterative way starting with schematic process planning. Schematic process planning includes, for example, interpretation of design requirements, definition of production and operation sequence, machine tools, cutting and work holding tools.

Tolerancing is to define in-process workpiece (IPW) tolerances in conjunction with the schematic process plan, process behaviour and design specification of the part.

The process control strategy is based on both the schematic process plan and the required tolerances, but also on the behaviour of all included processes.

The defined process plan is then analysed regarding tolerances and the expected outcome of the process. The results are tested against acceptance criteria to decide whether the defined manufacturing process can be approved or not. If the manufacturing process has all the necessary qualities to succeed, the process plan can be established and all needed documents, programs, working instructions, etc. created.

To understand part programming in general, you must prove the ability to

4. Describe techniques used for part program preparation in terms of
    a. application and efficient use of
        i. canned cycles
        ii. subroutines
    b. efficient use of
        i. polar coordinates
        ii. area clearance: internal and external profiles
        iii. datum shifts and the use of work offsets
        iv. scaling
        vi. nested lops and macros
        v. rotation
        vi. mirror imaging
    c. the generation of sub-routines to supplement existing canned cycles
    d. the use of variables (parametrics) for machining families of parts
    e. consideration of programming requirements for fourth axis work
    f. tool nose radius compensation (TNRC) for complex contours; review of cutter diameter compensation.

To demonstrate an understanding of program planning in general, you must prove the ability to

3. State the factors that should be considered when planning a part program
    a. make maximum use of one datum setting
    b. grouping of similar operations
    c. use of roughing and finishing operations
    d. use of inspection breaks
    e. checking of drawings
    f. identify strategic dimensional checks.

To understand program planning in general, you must prove the ability to

3. Describe design considerations which assist CNC manufacture in terms of
    a. simple design practices, having taken account of the essential functional features of the component
        i. avoidance of feature which create machining problems, eg square corners on milled components; back turning on CNC lathe
        ii. avoidance of unnecessary tolerancing
        iii. provision of suitable workholding feature, eg where turned work needs to be reversed in chuck, where castings need to be located and clamped, etc
        iv. ensuring that dimensioning has been described from an appropriate datum
        v. awareness of available tooling
        vi. design to facilitate inspection.

4. Describe the interrelated considerations which influence planning decisions in terms of
    a. work material - intrinsic machinability; variations due to form of supply
    b. tooling
        i. tool grades and geometry to ISO; interpretation of tooling manufacturers' specifications; consideration of specialised tooling for plastics, difficult materials, for example alloy steels, machining characteristics of materials
        ii. fixture design and layout for single and multiple setups; layouts to ensure collision free operation; use of safe zones

iii. tooling requirements for variable batch manufacture; positioning of tools in magazine; tool library, its inherent limitations; balancing tool usage to optimise tool life

c. operating sequencing - principles of operation sequencing to minimise number of workholding and tool changes

d. speeds and feeds - optimisation of cutting conditions (taking account of component configuration); comparative material removal rates and power consumption for roughing and surface finishing operations; relationship of speed and feed rate, depth of cut and tool life

e. in process gauging - feedback to the control system affecting toolset, tool breakage.

2.4. Selection of advanced operations in NC/CNC machine

Selection of advanced operations using canned cycles and sub-routines in the particular NC/CNC machine

2.4.1. Canned cycle

A canned cycle is a way of conveniently performing repetitive CNC machine operations. Canned cycles automate certain machining functions such as drilling, boring, threading, pocketing, etc. Canned cycles are so called because they allow a concise way to program a machine to produce a feature of a part. A canned cycle is also known as a fixed cycle. A canned cycle is usually permanently stored as a pre-program in the machine's controller and cannot be altered by the user.

9. Describe the role of assigned canned cycles

a. state what is meant by an assigned canned cycle

b. state the role of assigned canned cycles

c. describe the detailed operation of assigned canned cycles.

10. Describe techniques for improving the efficiency of part programming

a. state the advantages and disadvantages of unassigned (user defined) cycles

b. describe the use of loops in terms of

i. loop calling blocks

ii. loop terminating blocks

iii. effects of loop on program sequence

iv. nested loops, nesting limit

c. describe the use of sub-routines in terms of

i. labelling of sub-routines: distinction between level and call statement

ii. effect of calling a sub-routine on program sequence

iii. specific position of sub-routines within program (for some systems).

Programming format

The operation of a CNC machine tool is typically controlled by a "part program" written a language known as G-code. Canned cycles are similar in concept to functions in a traditional computer language, and can be compared also to G-code macros. The format for a canned cycle consists of a series of parameters specified with a letter and a numerical value. The letter is referred to as an "address". (This use of the term "address" may be unfamiliar to programmers of conventional computers. It arises because in early and primitive machine controllers, the binary representation of the letter formed a physical address at which the controller would store the value following.)

N.. G.. G.. X.. Y.. R.. P.. Q.. I.. J.. Z.. F.. H.. S.. L.. A.. B.. C.. D..

These addresses and values tell the machine where and how to move. The syntax of a canned cycle may vary depending on the brand of the control. In general, the following "words" will be in a canned cycle "block".

- N= Block number
- G98 or G99= Tool retract to R-plane or prior position
- G73, G74, G76, G81-89= The function to perform, for example, G84 specifies a right-hand tapping cycle.

- X= Position of hole or pocket in X axis
- Y= Position of hole or pocket in Y axis
- R= Z axis start position, also known as the retract plane or "R-plane".
- P= Dwell time (in milliseconds, where applicable)
- Q= Depth of each peck (G73, G83) or amount of shift for boring (G76, G87)
- I= Shift amount in X direction
- J= Shift amount in Y direction
- Z= Shift amount in Z direction (Negative because cutting is done in negative Z direction)
- F= Feed rate
- H= Feed rate for finishing cut
- S= Spindle speed
- L= Number of cycle repetitions
- M= Miscellaneous functions

A, B, C and D are used for Rectangular pocket machining.

- A= Machining allowance
- B= Step over
- C= Step depth
- D= Additional depth of cut for first pass

G80 is used for cancelling the currently selected canned cycle as G-codes for canned cycles are modal.

If the machine control supports it, the user may create their own custom canned cycles. As there are numbers not already used for G-codes, new canned cycle programs can be stored at these vacant locations. This may be done on the popular Fanuc control with a technique referred to as "macro programming", after the Fanuc Macro-B language. (The term "Macro programming" in this sense is distinctly different from its more common use to refer to the action of programming a macro in G-code.)

These are examples used on a mill. Some of them have different functions on a lathe.

G73	High speed peck drilling cycle
G74	Left-hand tapping cycle
G76	Precision boring cycle
G80	Cancel any fixed cycle
G81	Drilling cycle
G82	Drilling cycle with dwell
G83	Peck drilling cycle
G84	Right-hand tapping cycle
G85	Boring cycle
G86	Boring cycle
G87	Back boring cycle
G88	Boring cycle
G89	Boring cycle

Advantages

The conciseness of canned cycles allows for quicker and easier development of programs at the machine.

As canned cycles reduce the number of blocks in a program, the storage space occupied by the program is less and the programmer escapes the tedium of writing the same instructions again and again. This reduces the potential for errors, and locating any errors that do exist is easier in a shorter program.

Job setup is also facilitated by canned cycles. Some canned cycles exist which are designed for use by machine tool operators for simple job set-up and measuring tasks.....

2.4.2. CNC Simulation Software Guide

CNC Guide simulates the FANUC CNC for part program creation, G-code testing and optimization all via a PC. Users can learn how to operate FANUC CNCs in a virtual environment without wasting valuable machining time. Machine Tool Builders and OEMs can use CNC Guide to create digital twins of their machines and test the logic for faster proof of concept.



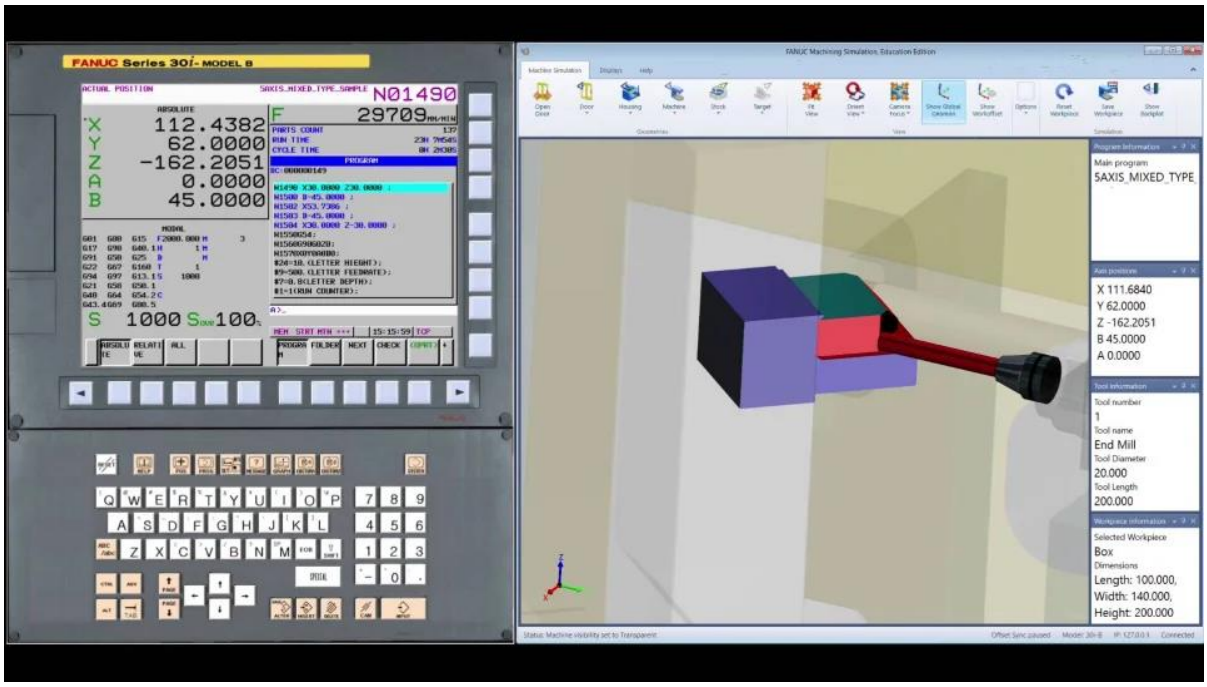


Figure: CNC Simulation Software

CNC Simulator is a powerful tool that allows CNC programs to simulate on computer before running them on an actual machine.

This application supports 2-4 axis machines to test and optimize the code, including milling machines, turning machines, laser cutters, plasma cutters, water jet cutters, 3D printers, plotters, and more. By testing and developing CNC code on a computer, you can save time, money, and resources, as well as ensure the safety of the operators.

Benefits of CNC Guide

CNC Guide teaches the programmer how to use performance-enhancing control features like cycle time estimate. CNC Guide can be used as a simplified CAD/CAM package in tandem with our conversational programming software,

2.5. *Standard* program *code* writing format

Instruction Codes

A part program consists the following words: N, G, X, Y, Z, I, J, K, F, S, T, R, M

N - Sequence number prefix (to identify the block)

G - Preparatory functions (to prepare the controller) – Example: G00 - rapid transverse move

X, Y, Z - prefixes for x, y, and z-axes (position & motion)

I, J, K – Interpolation parameters

F - Feed rate (to specify the feed)

S - Spindle speed (to specify the seed)

T - Tool selection (for ATC only)

R – Arc radius (to specify the tool radius in drill cycles)

M - Miscellaneous functions (to specify auxiliary functions) – Example: M08 = turn cutting fluid on

Summary of the Codes in Use

The codes in use during this module are given below. Note that some codes do the same thing. These codes are intended to be compatible. When the computer simulation of a CNC machine is set up, some of the modal codes are already in effect. Those codes are indicated here by an asterisk (*).

G00	Rapid traverse to position X, Z
G01	Linear feed to position X,Z
G02	Clockwise circular interpolation at feedrate
G03	Anti-clockwise circular interpolation at feedrate
G04	Dwell (with code letter X for time of dwell
G20	Diameter programming
G21*	Radius programming
G33	Thread cutting cycle
G40*	Cancel tool nose radius compensation

G41	Tool nose radius compensation left
G42	Tool nose radius compensation right
G50	Define datum point (axis pre-set)
G70	All dimensions are in inches
G71*	All dimensions are in millimetres
G74	Turning cycle
G75	Facing cycle
G76	Multi-pass thread cutting cycle
G80*	Cancel canned cycles
G90*	All X and Y dimensions are absolute
G91	All X and Z dimensions are incremental or relative
G92	Define datum point (axis pre-set)
G94	Feed in inches or millimetres per minute
G95	Feed in inches or millimetres per revolution
G96	Constant surface speed
G97	Revolutions per minute

M00	Program stop
M01	Optional program stop
M02	Machine and program stop
M03	Spindle on forward
M04	Spindle on in reverse
M05*	Spindle off
M07	Coolant on
M08	Coolant on
M09*	Coolant off
M30	Machine and program stop and tape rewind

The other codes follow. For this unit, most of these codes can take any decimal number, for example, 1234.5678, or a number with no decimal point, which will be divided by 1000 to

make it equivalent to a number with a decimal point. Form example, if we type in X40, then the computer will read this as X0.04 and if we had actually meant to type in X40,m then we would be in for a surprise. Codes which take any decimal number are marked `f' for free format. Those codes with a fixed format are indicated.

Af	Angle for G76 cycle
Df	Radius step for G76 cycle
Ef	Feed rate for thread cutting
Ff	Feed rate
If	Arc centre X coordinate or supplementary X value
Kf	Arc centre Z coordinate or supplementary Z value
N	Line number, N must be followed by four digits
Sf	Spindle speed or constant surface speed
T	Tool selection, T may be followed by either two or four digits
Uf	Incremental X value
Wf	Incremental Z value
Xf	Distance across lathe axis
Zf	Distance across lathe axis

Program

- single point-to-point drill holes
- profile a rectangle
- drill two holes using a canned cycle
- use incremental programming to machine two holes with a canned cycle as 4 but with deep-hole canned cycle
- circular interpolation to profile a circular pocket
- circular interpolation to profile radii on four corners of the rectangle as 6 or 7, but using cutter diameter compensation
- profile deep external features using a loop for several passes.

Coding used for manual part programming will depend on the machine control system. ISO letter address still appears to be the most widely used, however, and you will be expected to be familiar with particular features of the system when required. Coding used during practical work will naturally depend on local systems.

Standard coding system

You will be expected to be familiar with the following function codes when presented.

G 00 Rapid movement... point-to-point
G 01 Linear interpolation
G 02 Clockwise circular interpolation
G 03 Counter clockwise circular interpolation
G 04 Dwell

G 40 Cancel cutter diameter compensation
G 41 Cutter diameter compensation left
G 42 Cutter diameter compensation right

G 70 Inch
G 71 Metric

G 90 Absolute
G 91 Incremental
G 92 Preset position
G 94 Feed setting as mm/min
G 95 Feed in mm per rev
G 96 Constant surface speed
G 97 Direct rpm programming

M 00 Program stop
M 01 Optional stop
M 02 End program
M 03 Spindle on negative clockwise
M 04 Spindle on positive counter clockwise

M 05 Spindle off

M 06 Tool change

M 07 Mist coolant on

M 08 Flood coolant on

M 09 Coolant off

M 30 End of tape

4. Describe the basic principles of part programming in ISO letter address in terms of the requirement for
    a. N: operation sequence number
    b. G: preparatory function for
        i. rapid/linear movement
        ii. inch/metric units
        iii. absolute/incremental dimensioning
        iv. simple explanation of the use of assigned canned cycles
    c. X, Y, Z: coordinate positional data
    d. F, S: feed and speed data
    e. T: tooling details
    f. M: miscellaneous functions
        i. tool change
        ii. end of program, etc.
        iii. radius/tool length offsets
        iv. datum offsets.

To understand of part programming in general, you must prove the ability to

4. State the meaning of the following terms related to part programs and their relationship
    a. block of data
    b. block identifier
    c. letter address
    d. modal and non-modal codes.

5. State function codes which are incompatible within a block.

6. Describe common pre-set data formats

a. give examples of common formats

b. recognise shorthand notations

i. use of leading and/or trailing zero suppression

ii. decimal point within a word

c. state and recognise typical faults which arise from input of incorrectly formatted information.

2.7 Describe the absolute and incremental modes of programming

a. describe the main principles of

i. absolute programming

ii. incremental programming

b. give typical uses of the programming modes listed in a).

2.8 Describe the principles of part programming in ISO letter address

a. state the requirements for

i. N: operation sequence number

ii. G: preparatory functions

iii. X, Y, Z, etc: coordinate positioning data (I, J for circular interpolation)

iv. F, S: feed and speed data

v. T: tooling details

vi. M: miscellaneous functions

b. state the function codes which are incompatible within a block.

14. Describe the importance of verifying part programs during pre-operational checks in terms of

a. protection of operator

b. protection of tooling

c. protection of work.

2.6. Operation sheet

Procedures for producing standard operation sheets of specific NC/CNC machine operation

Process Sheet

					Work Name or Number					
					Material					
					Date			**Programmer**		
					Machine					
	N	**G**	**X/Y/U**	**Y/Z/V**	**Z/I/J/W**	**Z/J/K**	**F**	**S**	**T**	**M**

Planning Sheet

		Work Name or Number				
		Material				
		Date			**Planner**	
Operation	**Tool Off**		**Cutting Speed**	**Spindle Speed**	**Feedrate**	
	No.	**Set**	**/min**	**rev/min**	**/rev**	**/min**

2.7. Self-check - 2

Directions: Attempt to answer all the questions listed below. Illustrations may be necessary to aid some explanations/answers. Note: Be precise and scientific in writing.

Part I - Instructions: For the following questions say **True** or **False.**

1. 1
2. 2

Part II: Multiple Choice - Instructions: For each of the following question **choose the best correct answer** and **circle the letter** of your choice from the given alternatives.

1. 1

   a. A b. B

   c. C d. D

2. 2

   a. A b. C

   c. B d. D e. E

Part III: Marching - Instructions: For the following questions match from **"B" to "A" column**

No.	"A" Column	"B" Column
_____ 1.		A.
_____ 2.		B.
_____ 3.		C.

Part IV: Marching – **Instructions:** Answer all the questions listed below. Use the Answer sheet provided in the next page:

1. 1
2. 2
3.

Unit Three: Trial program

This learning unit is developed to provide the trainees the necessary information regarding the following content coverage and topics:

- Safe work practices and procedures
- Hazards and control measures associated with NC/CNC machines
- Manual mode Machines operation to test and prove the program
- Editing the NC/CNC programing via the machine controller to adjust the operation
- ***Component*** conform checking to the specification

This unit will also assist you to attain the learning outcomes stated in the cover page. Specifically, upon completion of this learning guide, you will be able to:

- Operate the machine in manual mode to test and prove the program
- Edit the program to adjust the operation of the NC/CNC machine to standard
- Check the ***componen***ts to conform to all specification

3.1. Hazards and control measures associated with NC/CNC machines

Safety devices

Safety devices: Covers, guards, or other safety devices shall be provided for all parts of the machine that present safety hazards. The safety devices shall not interfere with the operation of the machine. The safety devices shall prevent unintentional contact with the guarded part, and shall be removable to facilitate inspection, maintenance, and repair of the parts. All machine parts, components, mechanisms, and assemblies furnished on the machine, whether or not specifically required herein, shall comply with all of the Standards promulgated under OSHA that are applicable to the machine itself.

The MCU shall include electrical and safety interlocks as required for the protection of the system, machine, and the operator. The console shall be sealed to protect the contained components from dust or other environmental matter that could adversely affect performance of the MCU.

Safety

a. The manufacturer is requested to make recommendations for safe manual or mechanical handling systems for moving parts into or out of the equipment and for holding of parts where applicable.
b. Mechanisms, devices, machine structure, motors, gear-boxes, etc., which will be handled during erection and removed occasionally for maintenance or replacement shall be equipped with safe lifting attach points (i.e., lift lugs, lift rings).
c. Essential safety operating instructions shall identify safety and health hazards associated with the equipment and the procedures and practices necessary for safe operations. These instructions shall be permanently placarded on the equipment.
d. Weight in pounds of machine assembly or machine components in excess of 1000 pounds shall be marked in visible location for assembly maintenance handling.
e. Placards shall be provided to warn operator or maintenance personnel of injury hazards, such as intermittent motion, stored energy systems, or restricted clearances.
f. Covers, guards, or other safety devices shall be provided for all parts of the machine that present safety hazards. The safety devices shall prevent unintentional contact with the guarded part, and shall be removable to facilitate inspection, maintenance and repair of the parts. All machine parts, components, mechanisms, and assemblies furnished on the machine, whether or not specifically required herein, shall comply with all of the standards promulgated under OSHA that are applicable to the machine itself.
g. If the robot is stopped at a command position and loses electrical power, the tool point shall move no more than three inches, for any load not exceeding the rated load.
h. The vendor will furnish the time (seconds), the amount of travel and overshoot (degrees at the gripper) for the extended arm to stop from maximum swing speed under emergency shutdown.
i. All electrical units shall include electrical and safety interlocks as required for the protection of the system, machine, and the operator.

An awareness of following routine maintenance requirements should be included as a feature of practical work, though it does not form part of the assessment.

1. Carry out daily/weekly lubrication that may be required
2. Check - air supply pressure
   i. Air filters
   ii. Level of lubrication oil
   iii. Filters
   iv. Coolant supply
   v. Swarf clearance on pneumatic system.
3. Keep machine clean and tidy.

To demonstrate an understanding of safety procedures, candidates must prove ability to

1. State the problems associated with setting and operating a CNC machine via a console
   a. reduction in operator sense of feel
   b. lack of direct manual control
   c. lack of programming knowledge of operator.

2. Describe guarding and interlocking devices used on CNC machines
   a. types of guarding and their uses
      i. totally enclosed
      ii. interlocked
   b. effects of guarding on work/tool holding and unloading
      i. difficulty of access
      ii. restriction due to design, i.e. position of access
   c. care required during unguarded program proving.

3. State importance of program proving for prevention of damage to equipment and work
   a. to ensure collision free tool paths at the program proving stage
   b. prevention of damage to tooling workholding devices, and work.

4. State problems associated with the removal of waste from the working zone of CNC machines
    a. volume of waste to be removed
    b. hazardous nature of waste
        i. heat
        ii. sharp edges
    c. type of waste.

5. State the hazards associated with manual and automatic work and tool changing
    a. sharp cutters
    b. hazards associated with inadequate clearance
    c. hazards associated with incorrect location
    d. hazards associated with incorrect sequencing.

6. Describe procedures for restarting a CNC machine after emergency shutdown in terms of
    a. problems that may be encountered
        i. machine may try to return to home position by shortest route regardless of obstructions
        ii. functions may have been wiped from machine memory
    b. procedures to be followed
        i. move tool to clear position
        ii. reset and check datum (as necessary)
        iii. check `state' of program for tool length and diameter offsets as well as correct new program midstart position (if possible) (select suitable start position in program).

NC/CNC Advanced Part Programming

Guidelines for practical work

Introduction

The project aims to draw together the various aspects of the scheme to show clearly how the content relates to industrial practice, and will provide a common measure of your ability to undertake realistic tasks.

It has been suggested in the body of the scheme that a number of the learning objectives are likely to be accomplished during the project work and this may influence the way that teaching time is planned. As a rough guide, however, it is suggested that tutors allocate some 50% of the course to project work, and in any case not less than 30 hours.

General project requirements

The project comprises the preparation of two fully documented part programs for

a. a "relatively complex" component, using a manual programming method
b. a "relatively simple" component, using a computer-aided programming method

both of which must be proven and cut on typical industrial CNC machinery.

Project tasks

Annexed to this appendix is a range of specimen assignments for centres to use. The selection indicates what is implied by relatively complex and relatively simple, and aims to present a variety of planning and programming problems.

They are offered as a guide only, however, as it is envisaged that tutors will want to involve your employers in the choice or formulation of project work. Care should be taken, however, to ensure that alternative tasks are of an equivalent complexity, and where any doubt exists reference should be made to City and Guilds before proceeding.

3.2. Manual mode Machines operation to test and prove the program

The machine functions shall be controllable directly by manually-operated control devices, semi-automatically by manual data input devices, and automatically from parts programs in data memory, and remotely from a computer.

Manually operated systems for ways and remote areas shall have control handles mounted in accessible locations.

Manual mode Machines operation

The MCU shall receive commands from a program stored in a self-contained memory unit or from a program manually entered from a keyboard on the machine controller as manual data input (MDI). The memory shall accept programs from memory storage, magnetic tape, and MDI inputs. It shall be possible to save programs for future use by writing the memory to external storage such as a magnetic tape cassette recorder, paper tape punch, and bubble memory cartridge reader through appropriate interfaces.

To demonstrate an understanding of part production in general, you must prove ability to

4. State how a programme can be optimised during proving
    a. adjustments of feeds and speeds for maximum metal removal rate, surface finish and size consideration
    b. reduction and unnecessary tool path move
    c. adjustment of tool compensation for size and corner dimension
    d. rearrangement of machining sequence.

5. Describe how a program may be edited to enable optimisation to be achieved in terms of
    a. on line using MDI
    b. off line using text editor.

6. Explain how tool length offsets are incorporated into a program and on the machine tool
    a. use of specific G codes to call up tool length and diameter within a program
    b. use of the tool table in a control system.

To understand CNC machining principles in general, you must prove ability to

1. Describe the basic principles of numerical control

a. outline the use of binary notation for information encoding

b. distinguish between open- and closed-loop control

c. state the applications of

i. point to point for

A. drilling

B. boring

C. punching

ii. continuous path control for profile cutting

d. state the application of servo-stepping motion

e. state the basic role of linear and rotary position transducers

f. define the relationship of CNC machine axes

i. primary

ii. secondary

g. state the purpose of a machine datum

h. describe the principles of coordinate dimensioning in terms of

i. the importance of coordinate planning and calculation in the preparation of working drawings

ii. the difference between absolute and incremental co-ordinate dimensioning methods

iii. applications of absolute and incremental co-ordinate dimensioning.

Write Part Programs Manually

To provide evidence of competence in the production of part programs, the candidate must

Practical task

001 write a part program

002 transpose to input media

003 prove and edit part program

Produce a Component by Use of a Part Program

To provide evidence of competence in producing components, the candidate must

Practical task

001 load program into machine

002 produce component

003 optimise part program

3.3. Editing the NC/CNC programing

Editing the NC/CNC programing via the machine controller to adjust the operation

NC Editor Component

NC Editor is a powerful and easy to use tool for editing, simulating, back plotting and verifying G-code for CNC machining.

The intuitive interface provides instant 3D simulation while editing the program to detect potential collisions and errors at a glance and to instantly verify the changes to the code.

Integrated Job Setup for fast and easy tool creation and built-in support for multiple CNC machines further accelerate and simplify the editing and verification process to make the NC Editor a complete NC verification system that optimizes the quality and efficiency of machining processes.

Key Benefits

- NC G-code based simulation
- Backplot and material removal mode
- Error and collision detection
- Reduce setup time by using the integrated Job Setup
- Interactive toolpath visualization
- Standalone application

General Features

- Full machine simulation
- Milling: 2-axis, 3-axis
- Multi-Axis milling: 4-axis, 5-axis
- Turning / lathe
- Toolpath statistics
- Program transfer
- Canned cycle support

Editing Capabilities

- Easy editing of NC program
- Update feed / speed
- Alter parameter values using simple maths
- Remove blank lines / spaces
- Re-number block numbers
- Adjustable fontsize

Display and View

- Colors to highlight G-code functions
- Group by tool change
- Jump to comment
- Step through code, including canned cycles and subroutines
- Synchronized display in backplot / simulation

Simulation and Error Detection

- Collision detection between machine parts
- Collision detection with tool – flute, shaft, arbor and holder
- Clash detection – tool, rapid moves
- Detect wrong programming formats

Job Setup and Controller

- Handles all necessary tool data including adapters and holders
- Stock, fixture and target 3D models
- Customizable controller files

To demonstrate a sufficient command of the technology to cope with minor changes, you must prove the ability to

1. Use the basic trigonometrical ratios
    a. sine
    b. cosine
    c. tangent.

2. Use the sine and cosine rule
    a. sine rule
        A/ sin A = B/ sin B = C/Sin C
    b. cosine rule
        a2 = b2 + c2 - 2bc Cos A

6. Describe the application of computer control to the inspection process in terms of
    a. application of computer control to in-process and post-process inspection
    b. benefits
        i. integrated in-process feedback
        ii. rapid information generation for shape and statistical analysis
    c. types of equipment and applications
        i. contact types
        ii. calipers, probes
        iii. non-contact types
        iv. vision systems

d. interfacing with adaptive control.

3.4. Self-check - 3

Directions: Attempt to answer all the questions listed below. Illustrations may be necessary to aid some explanations/answers. Note: Be precise and scientific in writing. Use the Answer sheet provided in the next page.

Part I - Instructions: For the following questions say **True** or **False.**

1. 1
2. 2

Part II: Multiple Choice - Instructions: For each of the following question **choose the best correct answer** and **circle the letter** of your choice from the given alternatives.

1. 1

   a. A b. B

   c. C d. D

2. 2

   a. A

   b. B

   c. C

   d. D

Part III: Marching - Instructions: For the following questions match from **"B" to "A" column**

No.	"A" Column	"B" Column
_____ 1.		A.
_____ 2.		B.
_____ 3.		C.

Part IV: Short answer – Instructions answer all the questions listed below.

1. 1
2. 2

3.5. Operation sheet – 3.1

Operation Title:

Instruction:

Purpose:

Required tools and equipment:

1. Computer
2. Software

Precautions:

Procedures:

Step 1.

Step 2.

Step 3.

Figure

Quality criteria: The output must be identical with the given figure

3.6. LAP Test

Practical Demonstration

Name: _____ Date: _____

Time started: _____ Time finished: _____

Instruction I: Given necessary templates, tools and materials you are required to perform the following tasks within 10 hours.

Task 1:

Task 2:

Task 3:

Figure

Reference

1. 1
2. 2
3.

Developer profile

No.	Name	Quali-fication (Level)	Field of Study	Organization/ Institution	Mobile number	E-mail
1	Mohammed Bedri	M.Sc. (A)	Manufacturing Technology	Robe Poly Technic College	0913175634	
2	Mohammed Abdela	M.Sc. (A)	Manufacturing Technology	Dire Dawa Ethio Italy Polytechnic College	0907755560	Mahammedabdella49@gmail.com
3						
4						