

Trade of Toolmaking	
Module 6:	Introduction to CNC
Unit 2:	Part Programming
	Phase 2

Published by



Table of Contents

Document Release History	3
Unit Objective.....	4
Introduction	4
1.0 Converting A Conventional Working Drawing Into A Co-Ordinate Drawing	5
1.1 Location of machine datum and select workpiece datum	5
1.2 Absolute (Cartesian) And Incremental Dimensioning	5
1.3 Use And Applications Of Polar Co-Ordinates	6
2.0 Planning The Sequences Of Machining Operations Using Centre Line Programming	6
2.1 Plan The Sequence Of Machining Operations Using Centre Line.....	6
3.0 Preparing A Part Program For A Simple Milled Or Turned Component	7
3.1 Architecture Of Structure Of A Cnc Program I.E. Metric, Absolute Or Incremental Programming, Safety Line, Tool Call And Setup, Tool Path Generation, End Of Program	7
3.2 Preparation Of A Part Program Incorporating The Use Of Part Programming Sheets, Co- Ordinate Sheets, Operation Sheets	7
3.3 ISO Letter Address (Machine Code)	8
3.4 Word Address Format Used In ISO Programming	8
3.5 ISO Part Program Codes: 'G' Codes And 'M' Codes.....	8
3.6 Tool Clearance Plane And Tool Change Positions	9
3.7 Determination Of Co-Ordinates Using Trigonometry: Sine, Cos And Tan Rules	9
3.8 Calculation Of Spindle Speeds And Feeds (Feed/Rev And Feed/Min)	10
3.9 Modal And Non Modal Commands	14
3.10 Trailing Zero Suppression/Leading Zero Suppression	14
Summary	15
Suggested Exercises	16
Questions	17
Answers.....	18
Recommended Additional Resources.....	19
Reference Books	19

Document Release History

Date	Version	Comments
25/09/2014	2.0	SOLAS transfer

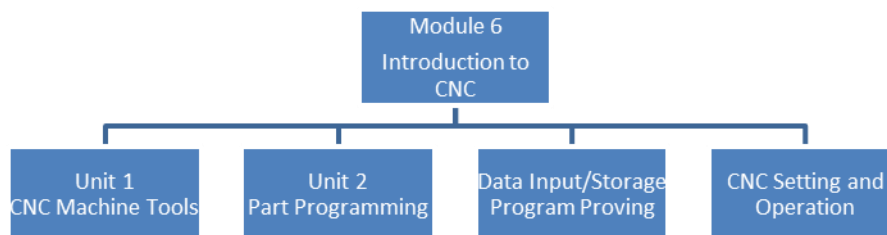
Unit Objective

On completion of this unit you will be able to convert conventional drawings to co-ordinate drawings and prepare CNC part programs.

Introduction

Module six of this course covers CNC machining. This is the second unit in module six and explains how to convert conventional drawings to co-ordinate drawings, which can then be entered into a table showing the X and Y values. The dimension values can then be taken directly from the tabulation and entered into the CNC program.

It is important to plan the sequence of operations prior to writing the part program. A zero point is chosen on the drawing and all co-ordinates are referenced from this point, using the rectangular (Cartesian) co-ordinate system. This unit also explains how the program is structured and how the machine control unit controls the machine in response to the coded commands which make up the part program. Various commands are used to controlling the movements of the cutter, coolant on and off, spindle clockwise and counter clockwise, tool change etc.



By the end of this unit you will be able to:

- Convert a conventional working drawing into a co-ordinate drawing and tabulate the co-ordinate values.
- Plan the sequence of machining operations using centre line programming.
- Prepare a part program for a simple milled or turned component.

1.0 Converting A Conventional Working Drawing Into A Co-Ordinate Drawing

Key Learning Points

Location of machine datum and select workpiece datum. Absolute (Cartesian) and incremental dimensioning. Use and applications of polar co-ordinates.

1.1 Location of machine datum and select workpiece datum

A conventional drawing will need to be converted into a co-ordinate drawing prior to writing the CNC program. This can be done by first choosing the a datum point on the workpiece and then writing in the 'x' and 'y' co-ordinates at each corner, radius centre, hole position, etc., of the workpiece on the drawing. With co-ordinate drawings all features are dimensioned from either a horizontal or a vertical datum edge. The dimension values can then be entered directly into the CNC program. The x and y values can also be entered into a table on the drawing.

The *datum* is a reference edge from which measurements are made. Prior to milling or drilling the component of the jig, the plates are marked out using the same datum edges as those specified on the drawing. All features are marked out and measured from one end of the plate. Normally there are two datum edges, which are at right angles to each other.

Ref: Timings, R.L. 1998, *Manufacturing technology*, vol. 1, 3rd edn, Pearson Education Limited, chapter 5, *Numerical control part programming*, sec. 5.14, *Part programming*, p. 197.
ISBN-13: 9780582356931

1.2 Absolute (Cartesian) And Incremental Dimensioning

The Cartesian or Rectangular Co-ordinate System forms the basis of numerical control (NC) measurement. Using rectangular coordinates, any specific point in space can be described in mathematical terms along two axes, X and Y, which are perpendicular to each other. The X axis is the horizontal line and the Y axis is vertical line and where the two lines intersect is called the origin (0, 0), where X is zero and Y is zero.

When absolute dimensioning is used on a drawing, it means that all features are dimensioned from either the vertical datum edge or the horizontal datum edge. For programming this is an advantage, as dimension values can then be taken directly from the drawing and entered into the program.

For incremental dimensioning, e.g. for a series of holes, each of the holes is dimensioned from the previous hole. If you are using preparing a program using absolute co-ordinates, then the positions have to be calculated before entering the values into the program. Another problem with incremental dimensioning is, as each hole has its own positional tolerance, there can be a build up of tolerances and therefore is less accurate than absolute dimensioning.

Ref: Simmons, Colin H & Maguire, Dennis E 2004, *Manual of engineering drawing*, 2nd edn, Elsevier Science & Technology, chapter 14, *Dimensioning principles*, p. 100.
ISBN-13: 9780750651202

1.3 Use And Applications Of Polar Co-Ordinates

Polar co-ordinate is used where a feature on a drawing is described by a length and an angle measured from a particular point. This can be easily written into the CNC program. It is also of use when programming a number of holes positioned at various angles around a Pitch Circle Diameter (PCD).

2.0 Planning The Sequences Of Machining Operations Using Centre Line Programming

Key Learning Points

Plan the sequence of machining operations using centre line.

2.1 Plan The Sequence Of Machining Operations Using Centre Line

The sequence of operations should be planned out prior to writing the part program. Study the drawing and decide on a zero point (or reference point or origin). The ideal location for the zero point is on the bottom left corner. Using the rectangular co-ordinate system as explained above, this point becomes (0,0), so all other X and Y co-ordinates will have plus signs, which can make it easier to calculate when programming. Features such as slots are normally dimensioned on the drawing to the centre of the slot, therefore the X and Y co-ordinates can be taken directly from the drawing dimensions. The slot closest to the zero point is normally chosen as start point for machining the first slot. For a *straight slot*, linear interpolation (G01) is used, where the X and Y co-ordinates for the start and finish points are marked up on the drawing. For a *circular slot*, circular interpolation (G02, clockwise or G03, counter clockwise) is used where the X and Y co-ordinates for the start and finish points are marked up on the drawing. Also I and J are used, which specified the arc centres the X and Y directions.

3.0 Preparing A Part Program For A Simple Milled Or Turned Component

Key Learning Points

Architecture of structure of a CNC program i.e. metric, absolute or incremental programming, safety line, tool call and setup, tool path generation, end of program. Preparation of a part program incorporating the use of part programming sheets, co-ordinate sheets, operation sheets. ISO letter address (machine code). Word address format used in ISO programming. ISO Part program codes: 'G' codes and 'M' codes. Tool clearance plane and tool change positions. Determination of co-ordinates using trigonometry: Sine, Cos and Tan rules. Calculation of spindle speeds and feeds (Feed/Rev and Feed/Min). Modal and non modal commands. Trailing zero suppression/leading zero suppression.

3.1 Architecture Of Structure Of A Cnc Program I.E. Metric, Absolute Or Incremental Programming, Safety Line, Tool Call And Setup, Tool Path Generation, End Of Program

The machine control unit controls the machine in response to the coded commands which make up the part program. These commands are identified by a capital letter which is called an *address*. The commands also contain numbers which follow the letters. The combination of the letter address and the number is known as a *word*. Each line of a program is called a block, which may contain a number of words.

Before writing a part program, a decision has to be made whether to use absolute (G90) or incremental (G91) dimensioning, to use inches (G70) or millimetres (G71). Tool call up is denoted by the letter 'T' followed by the number of the tool e.g. T1.

Ref: Timings, R.L. 1998, *Manufacturing technology*, vol. 1, 3rd edn, Pearson Education Limited, chapter 5, *Numerical control part programming*, sec. 5.7, *Program terminology*, p. 183.
ISBN-13: 9780582356931

3.2 Preparation Of A Part Program Incorporating The Use Of Part Programming Sheets, Co-Ordinate Sheets, Operation Sheets

Program Sheets – The part program can be written into pre-printed part program sheets. These sheets provide a neat and orderly way of setting out the program. They also provide permanent documentation of the job to be machined.

Co-ordinate sheets – These sheets are used as a supplement to the program sheet. All relevant co-ordinates are laid out on this sheet. A separate co-ordinate point will be specified for each point where the cutter needs to change direction. The sheets are useful as an aid for proving the program and for locating and editing errors when they occur.

Operation Sheet – This sheet is intended as an aid to the operator. It itemises each operation in sequence and identifies the tools required for each operation.

3.3 ISO Letter Address (Machine Code)

This is the most widely used system, where each word starts with a letter character called an address. There are two types of words, Dimensional Words and Management Words.

Dimensional Words are words relating to dimensions, i.e. X, Y, Z, which are used to define the machine axes, and I, J, K, refer circles and arcs of circles.

Management Words are words that are not related to dimensions, are as follows: N, G, F, S, T, M.

N: Block sequence number address. Blocks are often inserted in steps of 5 to allow for blocks to be inserted if needed.

G: Used to prepare or inform the machine controller of the functions required for the next operation.

F: Feed rate address.

S: Spindle speed address.

T: Tool number address.

M: This is a miscellaneous command and used for such commands as spindle start and stop and coolant on and off.

3.4 Word Address Format Used In ISO Programming

This refers to the format in which the words must take:

N4: The character N followed by 4 digits, e.g. N0001 is the first block in the program.

G2: Preparatory function 'G' followed by 2 digits.

F4: The character F followed by 4 digits.

S4: The character S followed by 4 digits.

T2: The character T followed by 2 digits.

M2: The character N followed by 2 digits.

Ref: Timings, R.L. 1998, *Manufacturing technology*, vol. 1, 3rd edn, Pearson Education Limited, chapter 5, *Numerical control part programming*, sec. 5.7, *Program terminology*, p. 183.
ISBN-13: 9780582356931

3.5 ISO Part Program Codes: 'G' Codes And 'M' Codes

G Code: These are preparatory functions used to change the mode of movement of the machine, such as rapid slide movement (G00), circular interpolation clockwise (G02), controlled feed rate (G01), absolute (G90) or incremental movements (G91) etc.

M Code: This is a miscellaneous command and used for such commands as spindle on clockwise (M03) and stop (M05) and coolant on (M08) and off (M09).

Ref: Timings, R.L. 1998, *Manufacturing technology*, vol. 1, 3rd edn, Pearson Education Limited, chapter 5, *Numerical control part programming*, table 5.1, *Preparatory functions*, p. 185, table 5.2, *Miscellaneous functions*, p. 186.

ISBN-13: 9780582356931

3.6 Tool Clearance Plane And Tool Change Positions

Tools of various lengths can be setup using a common datum. Tool 1 (T01) is setup first by touching it off the work surface and then setting the Z axis to zero. All other tools are set by touching them off the work surface, using the Tool 1 zero setting as a reference. The differences between Tool 1 and other tools are stored in the computer, where the Z axis offset is applied to each tool to compensate for the differences in length when compared to Tool 1.

Tool call up is denoted by the letter 'T' followed by the number of the tool e.g. T01. The G code G06 is a preparatory function used for the command to change the tool.

Ref: Timings, R.L. 1998, *Manufacturing technology*, vol. 1, 3rd edn, Pearson Education Limited, chapter 5, *Numerical control part programming*, sec. 5.10, *Tool length offset*, p. 190.

ISBN-13: 9780582356931

3.7 Determination Of Co-Ordinates Using Trigonometry: Sine, Cos And Tan Rules

Trigonometry deals with the ratio between the sides of a right angled triangle and provides a method of calculating unknown sides and angles. Three important trigonometrical ratios are sine, cosine and tangent, usually written as Sin, Cos and Tan, where:

$$\text{Sin} = \frac{\text{Opposite}}{\text{Hypotenuse}}$$

$$\text{Cos} = \frac{\text{Adjacent}}{\text{Hypotenuse}}$$

$$\text{Tan} = \frac{\text{Opposite}}{\text{Adjacent}}$$

If a feature on a drawing, such as a drilled hole, is dimensioned at an angle of 20° and is 30mm from the origin, then the X and Y co-ordinates can be calculated using trigonometry. Draw a right angled triangle and label the longest side (hypotenuse) 30mm, label the horizontal line X (adjacent) and the vertical line Y (opposite). The X and Y co-ordinates can be calculated by using the formulae:

$$\sin 20^\circ = \frac{Y}{30}$$

$$Y = \sin 20^\circ \times 30 = 10.26\text{mm}$$

$$\cos 20^\circ = \frac{X}{30}$$

$$X = \cos 20^\circ \times 30 = 28.19\text{mm}$$

Ref: Bird, John 2005, *Basic engineering mathematics*, 4th edn, Elsevier Science & Technology, chapter 19, *Introduction to trigonometry*, p. 142.

ISBN-13: 9780750665759

3.8 Calculation Of Spindle Speeds And Feeds (Feed/Rev And Feed/Min)

Calculation of spindle speeds for the milling machine:

The correct spindle speed needs to be set for each cutter. This is calculated by entering the cutting speed and the cutter diameter into the RPM formula, where the *Cutting Speed* is expressed in meters per minute. Charts are available that recommend the correct cutting speed for a particular material, e.g. for mild steel, a typical cutting speed of 30 meters/min is used and for tool steel the cutting speed is 20 metres/min. Therefore the spindle speed will be lower for the tool steel when compared to that of mild steel.

Recommended cutting speed in metres per minute:

Material	High Speed Steel (metres/min)	Carbide Cutter (metres/min)
Tool Steel	21 - 33	68 – 158
Mild Steel	33 – 42	98 – 188
Cast Iron	21 – 30	68 – 98
Brass	42 – 75	120 – 180
Aluminium	60 – 105	180 - 350
Plastic	60 - 450	unlimited

To find the correct RPM (revs per minute) setting of the spindle the following formula should be used;

$$\text{RPM} = \frac{\text{Cutting Speeds in metres per minute} \times 1000}{\text{Circumference of cutter in millimetres}}$$

$$= \frac{S \times 1000}{\pi \times D}$$

For example, for a mild steel part a cutting speed of 30 m/min is chosen from the above table. If it is to be machined with a 20mm High Speed Steel Cutter, the RPM is calculated as follows:

$$\text{RPM} = \frac{30 \times 1000}{3.14 \times 20} = 478$$

Feed Rate

Feed rate is the speed at which the workpiece is moved relative to the cutter.

0.05 is chosen from the cutter manufacturers tables. The feed rate is calculated as follows:

$$\begin{aligned}\text{Feed Rate} &= \text{feed/tooth} \times \text{No. of cutting teeth} \times \text{RPM} \\ &= 0.05 \times 4 \times 478 \\ &= 96 \text{ mm/min}\end{aligned}$$

Time to Cut

If the workpiece is 100mm long, then the time to cut is calculated as follows:

$$\begin{aligned}\text{Time to cut} &= \frac{\text{length of cut in mm}}{\text{Feed rate in mm/min}} \\ &= \frac{100}{96} \\ &= 1.04 \text{ minutes} \\ &= 62.5 \text{ seconds}\end{aligned}$$

Calculation of spindle speeds and feeds on the Lathe:

Cutting speed is expressed in metres per minute, which refers to the distance in metres that a tool may travel across the material being cut. The spindle speed is calculated by entering the cutting speed and the cutter diameter into the RPM formula, where the *Cutting Speed* is expressed in meters per minute. Charts are available that recommend the correct cutting speed for a particular material, e.g. for mild steel, a typical cutting speed of 30 meters/min is used and for tool steel the cutting speed is 20 metres/min. Therefore the spindle speed will be lower for the tool steel when compared to that of mild steel.

Recommended cutting speed in metres per minute:

Material	High Speed Steel (metres/min)	Carbide Cutter (metres/min)
Tool Steel	21 - 33	68 – 158
Mild Steel	33 – 42	98 – 188
Cast Iron	21 – 30	68 – 98
Brass	42 – 75	120 – 180
Aluminium	60 – 105	180 - 350
Plastic	60 - 450	unlimited

To find the correct RPM (revs per minute) setting of the spindle the following formula should be used;

$$\begin{aligned}\text{RPM} &= \frac{\text{Cutting Speeds in metres per minute} \times 1000}{\pi \times D}\end{aligned}$$

For example, for a mild steel part a cutting speed of 30 m/min is chosen from the above table. If a Ø50mm bar is to be machined the RPM is calculated as follows:

$$\text{RPM} = \frac{30 \times 1000}{3.14 \times 50} = 190$$

Feed

Feed is the rate at which the cutting tool moves along or across the work. The feed is adjustable and can be slow (fine feed) or fast (coarse feed). For each single revolution of the workpiece of the workpiece, the tool will move along the work by a certain distance. This is referred to as the feed per revolution.

For a conventional lathe if a feed rate of 0.25mm is chosen, this means that the tool will move 0.25mm for one revolution, which would be a typical feed used on a conventional lathe.

For a CNC machine the feed is calculated by using the following formula:

$$\text{Feed} = \frac{\sqrt{8r_e \times R_t}}{1000}$$

where

r_e = Toolnose radius

R_t = Profile depth μm

If the toolnose radius (r_e) is 0.4mm and the Profile depth (R_t) of 15 μm is chosen from a data chart, then the feed is calculated as follows:

$$\begin{aligned}\text{Feed} &= \frac{\sqrt{8 \times 0.4 \times 15}}{1000} \\ &= 0.219 \mu\text{m/rev}\end{aligned}$$

3.9 Modal And Non Modal Commands

Some functions are modal, which means that the command remains in effect until cancelled or superseded by a command of the same time.

3.10 Trailing Zero Suppression/Leading Zero Suppression

Modern systems allow a decimal point to be inserted and use the format X4,3, which means that the X dimension can have up to 4 digits before the decimal point and 3 digits after. For older systems, many of which are still in use, did not use a decimal point but used *leading* and *trailing zeros* to indicate the position of the decimal point.

Leading Zero Suppression – This is the removal or suppression of any zeros on the left of the number when written out in the seven digit format, with 4 digits before the decimal point and 3 after, e.g. the dimension 145.25mm written as 145250.

Trailing Zero Suppression – With this system any zeros to the right of the decimal point are removed or suppressed, e.g. the dimension 145.25mm becomes 014525 for a control which accepts trailing zeros.

Summary

Converting a conventional working drawing into a co-ordinate drawing: A conventional drawing will need to be converted into a co-ordinate drawing prior to writing the CNC program. This can be done by first choosing the a datum point on the workpiece and then writing in the 'x' and 'y' co-ordinates at each corner, radius centre, hole position, etc., of the workpiece on the drawing. With co-ordinate drawings all features are dimensioned from either a horizontal or a vertical datum edge. The dimension values can then be entered directly into the CNC program.

The *datum* is a reference edge from which measurements are made. Prior to milling or drilling the component of the jig, the plates are marked out using the same datum edges as those specified on the drawing. All features are marked out and measured from one end of the plate. Normally there are two datum edges, which are at right angles to each other

Planning the sequences of machining operations using centre line programming:

The sequence of operations should be planned out prior to writing the part program. Study the drawing and decide on a zero point (or reference point or origin). The ideal location for the zero point is on the bottom left corner. Using the rectangular co-ordinate system, this point becomes (0,0), so all other X and Y co-ordinates will have plus signs, which can make it easier to calculate when programming. Features such as slots are normally dimensioned on the drawing to the centre of the slot, therefore the X and Y co-ordinates can be taken directly from the drawing dimensions.

Preparing a part program for a simple milled or turned component:

The machine control unit controls the machine in response to the coded commands which make up the part program. These commands are identified by a capital letter which is called an *address*. The commands also contain numbers which follow the letters. The combination of the letter address and the number is known as a *word*. Each line of a program is called a block, which may contain a number of words.

Before writing a part program, a decision has to be made whether to use absolute (G90) or incremental (G91) dimensioning, to use inches (G70) or millimetres (G71). Tool call up is denoted by the letter 'T' followed by the number of the tool e.g. T1.

Suggested Exercises

1. Using a drawing from your course work, convert it into a co-ordinate drawing, using the bottom left hand corner as the datum point.
2. Write down the six *Dimensional Words* used in CNC programming.
3. Write down the *Management Words* and explain what they mean.
4. What are the meanings of the following G Codes:
G00, G01, G02, G03 and G05.
5. What letter is used for the tool call up and what preparatory function is used for the command to change the tool.

Questions

1. What is the difference between absolute dimensioning and incremental dimensioning and which one benefits CNC programming?
2. What is the definition of a datum?
3. In the Cartesian or Rectangular Co-ordinate System, which axis is horizontal and which axis is vertical?
4. What is each line in a CNC program called?
5. What are 'G' codes?

Answers

1. When absolute dimensioning is used on a drawing, it means that all features are dimensioned from either the vertical datum edge or the horizontal datum edge. For incremental dimensioning, e.g. for a series of holes, each of the holes is dimensioned from the previous hole.
For programming, absolute dimensioning is an advantage, as dimension values can then be taken directly from the drawing and entered into the program.
2. The datum is a reference edge from which measurements are made.
3. The horizontal axis is the X axis and the vertical axis is the Y axis.
4. Each line in a CNC program is called a Block.
5. 'G' codes are preparatory functions used to change the mode of movement of the machine, such as rapid slide movement (G00) and controlled feed rate (G01).

Recommended Additional Resources

Reference Books

Timings, R.L. 1998, *Manufacturing technology*, vol. 1, 3rd edn, Pearson Education Limited.

ISBN-13: 9780582356931

Simmons, Colin H & Maguire, Dennis E 2004, *Manual of engineering drawing*, 2nd edn, Elsevier Science & Technology.

ISBN-13: 9780750651202

Bird, John 2005, *Basic engineering mathematics*, 4th edn, Elsevier Science & Technology.

ISBN-13: 9780750665759