ΠΑΝΕΠΙΣΤΗΜΙΟ ΠΑΤΡΩΝ
UNIVERSITY OF PATRAS

ΑΝΟΙΚΤΑ ακαδημαϊκά μαθήματα ΠΠ

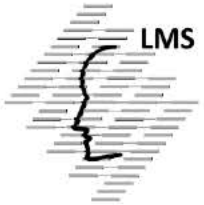
Αριθμητικός Έλεγχος Εργαλειομηχανών

Ενότητα 9: Two – Axis Programming

Δημήτρης Μούρτζης, Επίκουρος Καθηγητής
Πολυτεχνική Σχολή
Τμήμα Μηχανολόγων & Αεροναυπηγών Μηχανικών

COMPUTER NUMERICAL CONTROL OF MACHINE TOOLS

Laboratory for Manufacturing Systems and Automation
Department of Mechanical Engineering and Aeronautics
University of Patras, Greece



Dr. Dimitris Mourtzis

Assistant Professor

Patras, 2015

Objectives of section 6

- Identify the **basic parts** of a CNC program
- Describe the **word address code** format
- Write simple **two-axis** programs in word address format to perform **hole** operations
- Write simple **two-axis milling** programs using the word address format
- Write simple **two-axis** programs that combine **milling and hole operations**

Introduction

- This section is concerned with **manual** programming of CNC machinery
- For purposes of continuity the **same machine** will be used for the next several sections
- No two CNC machines program exactly alike
- However, learning to program the machine used in the examples, only **minimal effort** will be required to program other CNC machines
- Programming is done in a format called **Word Address** which is the most common machine code format used today
- **The machine programmed in this section is a vertical machining center**

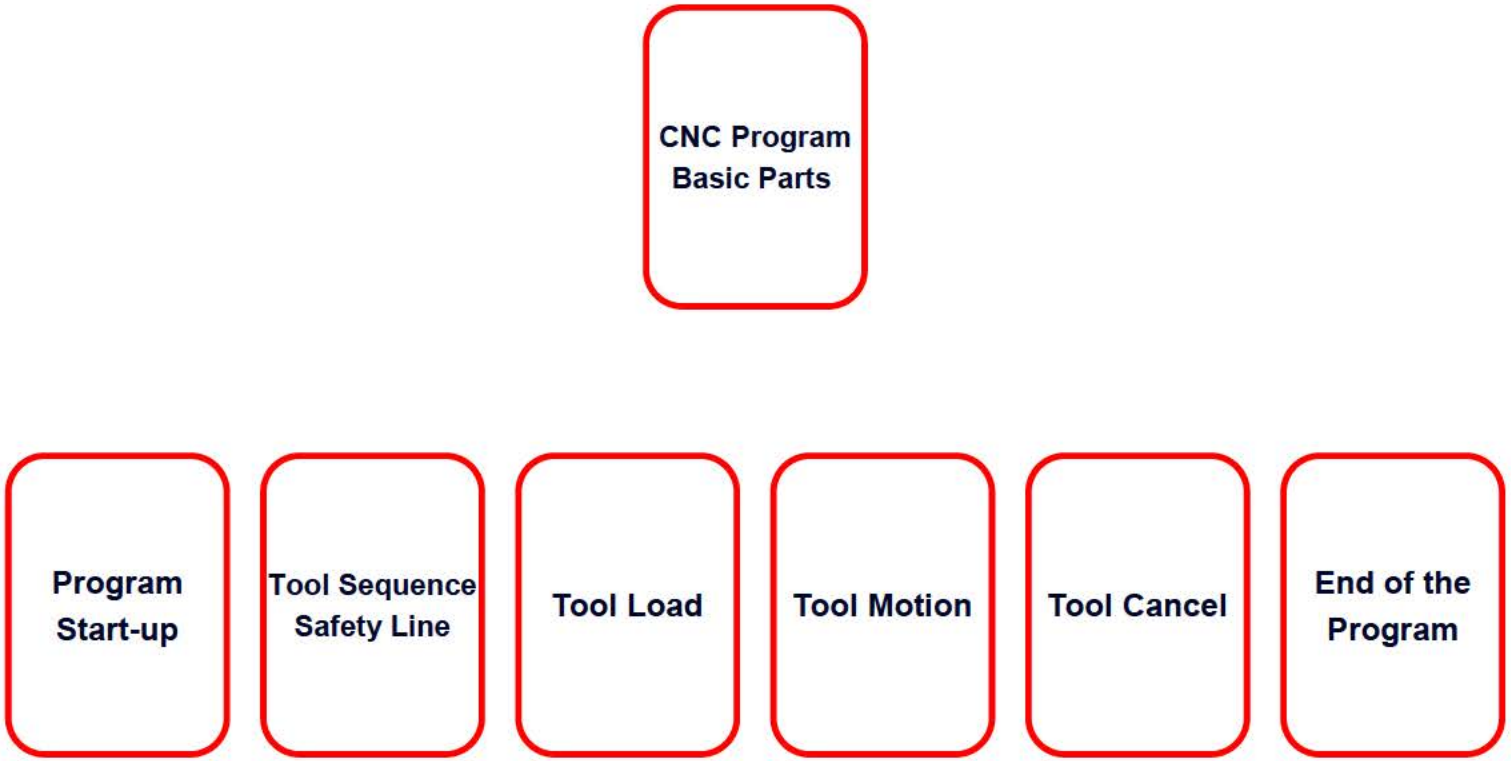
Introduction

- The machining center is equipped with a **FANUC M**achine **C**ontrol **U**nit (**MCU**)
- It is a **Continuous Path Type Machine**
- The program codes used on FANUC MCU are similar to those used on other MCUs such as **General Numeric and General Electric**
- **Two-Axis mill** programming is not so common in real world application but ***in educational level is a prerequisite for easier understanding of Three-Axis programming***

Parts of a CNC Program

Parts of CNC Program

- Regardless the MCU being programmed all CNC programs consist of the same basic parts



Parts of a CNC Program

Program Startup

- The program startup section serves to issue any commands required at the start of the tape only
- For instance, setting the program to inch mode would only be required at the beginning of the program

Tool Safety Block

- The tool sequence safety block(s) serves to issue commands to cancel for any machine modes that could have been left active if the machine operator interrupted the tool cycle
- By issuing a safety block, the programmer and operator know the state of the machine at the beginning of the tool cycle

Parts of a CNC Program

Tool Load Blocks

- The tool load section are those blocks **of a tool sequence where the tool is placed in the spindle**, either manually or by the machine's automatic tool changing mechanism
- The **tool length compensation** is turned on

Tool Motion Blocks

- The tool motion section contains the code for the ***actual cutting tool motion***
- It is where all the machining work is actually done

Parts of a CNC Program

Tool Cancel Blocks

- The tool cancel section turns off the tool length compensation and ***returns the tool to the tool change position***
- All active cycle commands should be turned off in this section and the control left in a state ready to load the next tool

End of Tape Blocks

- The end of program blocks ***issue any commands necessary after all tool motion is complete***, but before the program terminates
- Often this section consists simply of the end of program code

Word Address Format

Word Address Characteristics:

- Word Address was developed as a **tape programming** format
- Word Address is also named Variable Block Format because the program **lines (blocks)** may vary in length according to the information contained in them
- Earlier tape formats required an entry for all possible machine registers
- In these earlier formats a zero was programmed as a null input if the register values were to be unaffected
- In Word Address **the blocks need only contain necessary information**
- Although Word Address was developed as a Tape Format is used as the format for **M**anual **D**ata **I**nput (**MDI**) on many CNC machines

Word Address Format

Addresses:

The block format for word address is as follows:

N...G…X...Y...Z....I...J...K...F...H…S...T…M…

- Only the information needed on a line need be given
- Each of the letters is called an address (or word)

Word Address Format

N - The block sequence number

- An **N** number is used to **number the lines of NC code** for operator and/or programmer reference
- **N** numbers are **ignored by the controller** during program execution
- Most NC controls **allow a block to be searched** for by the sequence number for editing or viewing purposes.

G - Initiates a preparatory function

- Preparatory functions **change the control mode** of the machine
- Examples of preparatory functions are rapid / feedrate mode, drill mode, tapping mode, boring mode, and circular interpolation
- Preparatory functions are called prep functions or more commonly G Codes

Word Address Format

X: Designates an X-axis coordinate.

X also is used to enter a time interval on FANUC and FANUC style controllers

Y: Designates a Y-axis coordinate

Z: Designates a Z-axis coordinate

I: Identifies the X-axis arc vector (the X-axis center point of an arc)

J: Identifies the Y-axis arc vector (the Y-axis center point of an arc)

K: Identifies the Z-axis arc vector (the Z-axis center point of an arc)

S: Sets the spindle rpm

H: Specifies the tool length compensation register

F: Assigns a feedrate

T: Specifies the standby tool (to be used in the next tool change)

M: Initiates miscellaneous functions (M functions)

- **M functions** control **auxiliary** functions such as :
  - the turning on and off of the spindle and coolant,
  - initiating tool changes, and
  - signaling the end of a program

PREPARATORY FUNCTIONS (G CODES) USED IN MILLING

- Following is a list of preparatory functions used in CNC milling examples in this text. Other codes commonly used on General Numeric controllers are also listed.

G00-Rapid traverse positioning.
G01-Linear interpolation (feed rate movement).
G02-Circular interpolation clockwise.
G03-Circular interpolation counterclockwise.
G04-Dwell.
G10-Toollength offset value.
G17-Specifies X/Y plane.
G18-Specifies X/Z plane.
G19-Specifies Y/Z plane.
G20-Inch data input (on some systems).
G21-Metric data input (on some systems).
G22-Safety zone programming.
G23-Cross through safety zone.
G27-Reference point return check.
G28-Return to reference point.
G29-Return from reference point.
G30-Return to second reference point.
G40-Cutter diameter compensation cancel.
G41-Cutter diameter compensation left.
G42-Cutter diameter compensation right.
G43-Toollength compensation positive direction.
G44-Toollength compensation negative direction.
G45-Tool offset increase.
G46-Tool offset decrease.

PREPARATORY FUNCTIONS (G CODES) USED IN MILLING

G47-Tool offset double increase.
G48-Tool offset double decrease.
G49-Tool length compensation cancel.
G50-Scaling off.
G51-Scaling on.
G73-Peck drilling cycle.
G74-Counter tapping cycle.
G76-Fine boring cycle.
G80-Canned cycle cancel.
G81-Drilling cycle.
G82-Counter boring cycle.
G83-Peck drilling cycle.
G84-Tapping cycle.
G85-Boring cycle (feed return to reference level).
G86-Boring cycle (rapid return to reference level).
G87-Back boring cycle.
G88-Boring cycle (manual return).
G89-Boring cycle (dwell before feed return).
G90-Specifies absolute positioning.
G91-Specifies incremental positioning.
G92-Program absolute zero point.
G98-Return to initial level.
G99-Return to reference (R) level.

PREPARATORY FUNCTIONS (G CODES) USED IN TURNING

- Following is a list of preparatory functions used in CNC milling examples in this text. Other codes commonly used on FANUC controllers are also listed.

G00-Rapid traverse positioning.
G01-Linear interpolation (feedrate movement).
G02-Circular interpolation clockwise.
G03-Circular interpolation counterclockwise.
G04-Dwell.
G10-Toollength offset value setting.
G17-Specifies X/Y plane.
G18-Specifies X/Z plane.
G19-Specifies Y/Z plane.
G20-Inch data input (on some systems).
G21-Metric data input (on some systems).
G22-Stored stroke limit on.
G23-Stored stroke limit off.
G27-Reference point return check.
G28-Return to reference point.
G29-Return from reference point.
G30-Return to second reference point.
G40-Tool nose radius compensation cancel.
G41-Tool nose radius compensation left.
G42-Tool nose radius compensation right.
G50-Programming of work coordinate system.
G68-Mirror image for double turrets on.
G69-Mirror image for double turrets off.
G70-Inch programming (some systems) or finish cycle.
G71-Metric programming (some systems) or stock removal In turning code.
G72-Stock removal in facing code.
G73-Pattern repeat.

PREPARATORY FUNCTIONS (G CODES) USED IN TURNING

G74-Z axis peck drilling.
G75-Groove cutting cycle, X axis.
G76-Multipass thread cutting.
G90-Absolute positioning.
G91-Incremental positioning.
G94-Per minute feed (some systems).
G95-Per revolution feed (some systems).
G98-Per minute feed (some systems).
G99-Per revolution feed (some systems).

MISCELLANEOUS (M) FUNCTIONS USED IN MILLING AND TURNING

- Following is a list of miscellaneous functions used in the milling and turning examples in this text. Other M functions common to General Numeric and FANUC controllers are also listed.

M00-Program stop.
M01-0ptional stop.
M02-End of program (rewind tape).
M03-Spindle start clockwise.
M04-Spindle start counterclockwise.
M05-Spindle stop.
M06-Tool change.
M08-Coolant on.
M09-Coolant off.
M13-Spindle on clockwise, coolant on (on some systems).
M14-Spindle on counterclockwise, coolant on.
M17-Spindle and coolant off (on some systems).
M19-Spindle orient and stop.

M21-Mirror image X axis.
M22-Mirror image Y axis.
M23-Mirror image off.
M30-End of program, memory reset.
M41-Low range.
M42-High range.
M48-0verride cancel off.
M49-0verride cancel on.
M98-Jump to subroutine.
M99-Return from subroutine.

Summary 1/3

The important concepts presented in this section are:

- An NC or CNC program consists of six basic parts
  I. Program startup section
  II. Tool sequence safety line
  III. Tool load (or tool change) section
  IV. Tool motion sequence
  V. Tool cancel section
  VI. End of program section

- In word address format, each CNC command is called a *word.* Each word begins with an alpha address which identifies the command's function

- The address is followed by a numeric value. Some values are used to set machine modes.
- Others are used to specify positioning coordinates

Summary 2/3

- The spindle must be positioned safely out of the way at the end of the program, to allow safe loading and unloading of the workpiece
- This is accomplished in both the milling and drilling examples by sending the spindle back to its tool change location at the end of the program
- Incremental programs differ from absolute programs only in the coordinates used
- Programs in absolute and incremental positioning use the same programming logic
- In incremental positioning, it is imperative that the machine start and stop in the same location
- Failure to program for this will result in incorrect positioning for the second cycle

Summary 3/3

- To perform hole operations, it is necessary to position the spindle over the centerline of the hole
- A program stop command is used at hole locations to halt the program and enable the operator to drill the hole
- When programming coordinates for milling, an allowance must be made for the size of the cutter

Vocabulary Introduced in this section

- Addresses
- End of tape blocks
- Leading zero
- Program startup blocks
- Tool cancel blocks
- Tool load blocks
- Tool motion blocks
- Tool safety blocks
- Trailing zero
- Two-axis programming

End of Section

Funding

- This educational material has been developed in the teaching duties of the respective educator.
- The Project "**Open Academic Courses at the University of Patras**" has funded only the reformation of the educational material.
- The Project is implemented within the context of the Operational Programme "Education and Lifelong Learning" (EdLL) and is co-funded by the European Union (European Social Fund) and national resources.

Reference Note

License Note

Notes Preservation

Any reproduction or modification of this material must include:

- the Reference Note
- the License Note
- the Notes Preservation statement
- the Third Parties' Works Note (if exists)

as well as the accompanying hyperlinks.