ΠΑΝΕΠΙΣΤΗΜΙΟ ΠΑΤΡΩΝ
UNIVERSITY OF PATRAS

ΑΝΟΙΚΤΑ ακαδημαϊκά μαθήματα ΠΠ

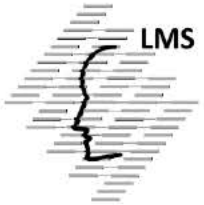
Αριθμητικός Έλεγχος Εργαλειομηχανών

Ενότητα 7: Three – 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 7

- Write simple programs to perform hole operations using **three machine axes**
- Explain what a **canned cycle** is
- Explain the difference between **initial level** and **reference level** on CNC machinery
- Explain the difference between a **modal** and **non-modal** command
- Write simple programs to perform **milling** operations using **three machine axes**
- Write simple programs involving a **machine indexer**

Parts of a CNC Program

Three-axis Programming

- Three-axis programming is used for a program sequence in which ***all three machine axes are used at the same time***

Two-and-half axis programming

- Use all three axes BUT Primarily position a location using X and Y axis
- Use Z axis to perform a drilling or milling operation
- Is the most common CNC milling programming
- ***90% of the CNC*** machining center programming
- It is the ***practical limit*** for manual programming
- Mathematical calculations for 3-axis are ***very time consuming***
- 3-axis, 4-axis and 5-axis programming are performed ***using CAD / CAM systems***
- Tool length offset is used
- Operator enters the tool lengths into the appropriate tool length offset registers in the CNC controller
- Tool length compensation ***adjust Z-axis zero point*** to account for the differences in the lengths of the various cutting tools used in the program

A Programming Task Using Three Axes

Several new word address commands

G28 - Return to reference point command

G28 is used in conjunction with other commands **to cause the spindle to position at the machine's coordinate system origin**
This point is referred to as ***home zero*** in most CNC shops
If coordinates are specified on the G28 line, the spindle will first move to the coordinates, then to home zero
In this manner the spindle may be moved to a known safe position before moving to home zero

G44 - Calls up a tool length offset register

A G44 accomplishes a Z-zero shift toward the workpiece

H - Used to assign a tool register

H01 would assign the information stored in tool length register #1
H02 would assign the information stored in tool length register #2

G49 - This is the tool length offset cancel code

A Programming Task Using Three Axes

Several new word address commands

G81 - This is the canned drill cycle

When a **G81** is issued:
The spindle rapids to the (X,Y) coordinates specified on the drill cycle line
The Z axis then rapids to the specified feed engagement point
Feeds to the final drill depth
Then rapids out of the hole to either the rapid or initial level

G80 - This is the canned cycle cancel code

When a G80 is issued, the active canned cycle code is turned off

R - This address stands for the canned cycle reference level

The *reference level* is the spot where the programmer desires the canned cycle to start feeding into the workpiece
The reference level is also called the *rapid* or *gage* level

G92 - Absolute zero set command

This command tells the control to reset the part coordinate system origin - Coordinates must be specified on the G92 block - The coordinates tell the machine where to set the origin, *relative to the current spindle position*

A Programming Task Using Three Axes

- **G99/G98**

G98 is the **return to initial level** command

G99 is the return to **rapid (reference) level** command

When a **canned cycle is active**, the **spindle may be directed to return to the rapid level** when it exits a hole with a **G99**

If the programmer desires the spindle to **return to the original starting point Z height**, the **G98** command is issued

G99 results in the faster cycle.

G98 is particularly useful for jumping over clamps and other obstructions while in a cycle

M01 - Program optional stop code

M01 functions as an **M00** with one exception: it is only effective if the optional stop switch on the machine control is turned on

When this switch, called an opstop switch, is off, the **M01** is ignored by the control

M03 - is the code for **turning the spindle on in the clockwise direction**

M05 - Turns the **spindle off**

A Programming Task Using Three Axes

M06 - Tool change code
When M06 is issued, the machine's automatic tool changer sequence will be initiated

M08 - Turns the flood coolant on

M09 - Turns the coolant off

T - Selects the tool to be put in the spindle by the tool changer

F - Assigns feedrates, as in two-axis programming

S - Designates the spindle speed

Modal / Non-Modal Commands

Modal Commands

- Codes that are active for more than one line in which they are issued
- Rapid transverse, Feedrate moves and canned cycle codes are examples of modal commands

Non-Modal command

- Is the one that is active only in the program block in which it is issued
- M00: Program Stop is an example of a Non-Modal command

Canned Cycles

- Are routines (e.g. G81) built into the control to perform standard operations
- Drilling, boring and taping are common operations
- The programmer can ***call a canned cycle instead of repetitive programming***

Modal Commands

- Most G codes put the machine in a "permanent" status, which remains in effect until it is changed or canceled by another G command
- Those are the **modal commands**

G00	Rapid Transverse
G01	Linear Interpolation
G02	Circular Interpolation, CW
G03	Circular Interpolation, CCW
G17	XY Plane
G18	XZ Plane
G19	YZ Plane
G20/G70	Inch units
G21/G71	Metric Units
G40	Cutter compensation cancel
G41	Cutter compensation left
G42	Cutter compensation right
G43	Tool length compensation (plus)

G43	Tool length compensation (plus)
G44	Tool length compensation (minus)
G49	Tool length compensation cancel
G80	Cancel canned cycles
G81	Drilling cycle
G82	Counter boring cycle
G83	Deep hole drilling cycle
G90	Absolute positioning
G91	Incremental positioning

Canned Cycles

A canned Cycle for Tapping: ***G84G99X1.Y.375Z-.753R.1F10***

G84 - G-code to turn on the tapping cycle
The spindle will feed into the work-piece with the spindle rotating clockwise, stop at the programmed Z axis coordinate, reverse the spindle, then feed back out of the workpiece until it reaches the programmed feed engagement point
G99 - Specifies that the spindle should return to the reference level (the feed engagement point) when retracting out of the hole
X/Y coordinates - Indicate the location where the cycle is to begin. The spindle will first position here at rapid traverse before moving the Z - axis.
Z coordinates - Tells the control how deep to feed the Z - axis. It is the actual Z coordinate to which the spindle is to move
R - Specifies the Z coordinate where the spindle is to begin feeding
Until the spindle reaches this coordinate, it will move in rapid traverse.
F - Sets the feedrate for the Z - axis feedrate moves

A complete cycle would look like following:

```
G84 G99 X1.Y.375Z-.753R.1F10
X1.5
Y1.375
X1.
G80
```

3-axis Programming Example

- An example of a 3 axis programming will follow

- The part will be milled in the milling machine of the picture

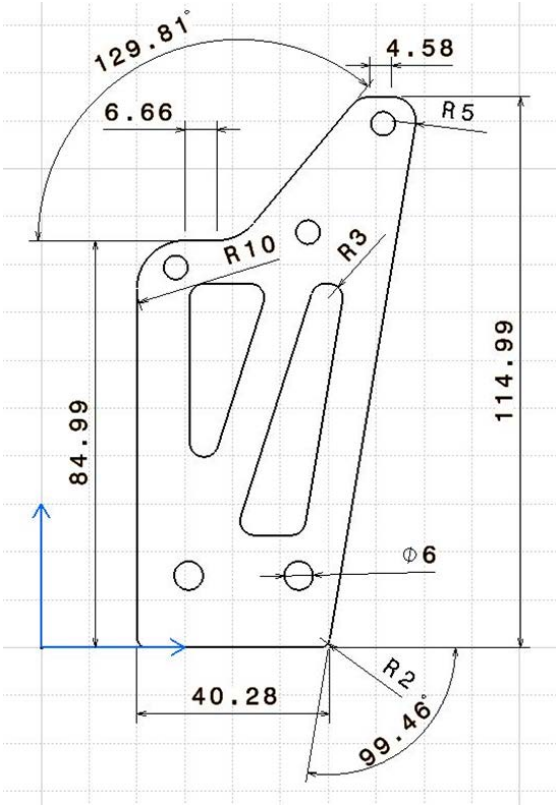


FIGURE 1 Draft Design of Part to be Milled

FIGURE 2 xyz-Drilling Machine

3-axis Programming Example

Part to be milled

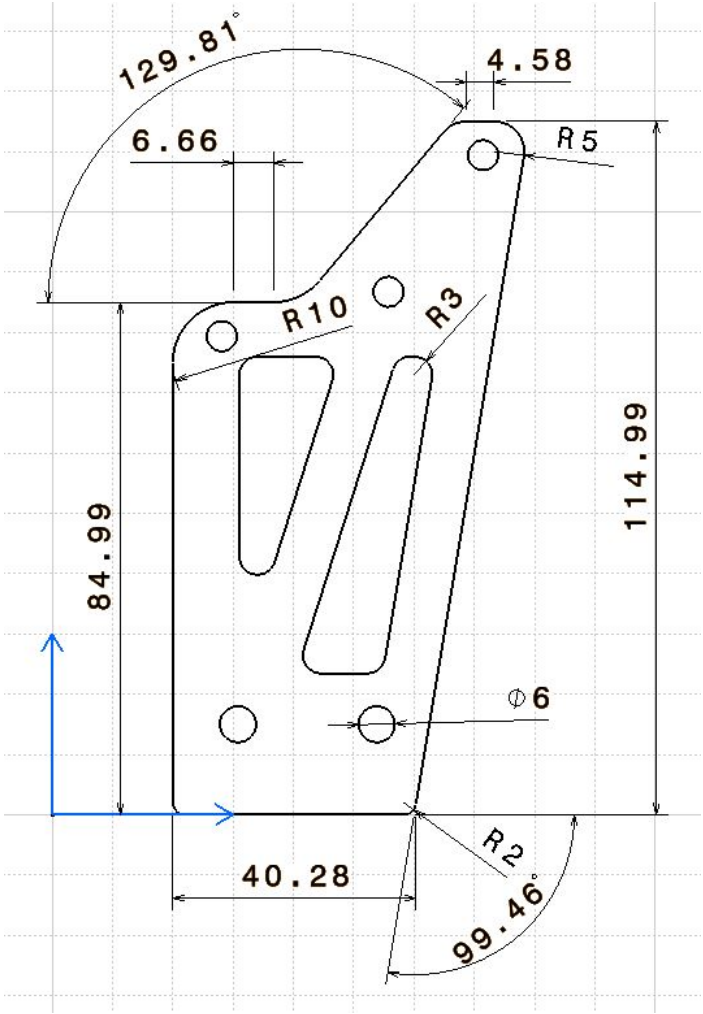


FIGURE 3 Draft Design of Part to be Milled

FIGURE 4 Rendered Design of Part

3-axis Programming Example

G code for internal pockets

```
%
O1000
N1 G49 G64 G17 G21 G80 G0 G90 G40 G99
( IMSPPCC_MILL PPTABLE 06-13-2003 )
( T4 END MILL D 8 )
N2 T0004 M6
N3 X68.635 Y80.631 S70 M3
N4 G43 Z9. H1
N5 G1 G94 Z-1. F30.
N6 X72.244 Y92.
N7 X65.025
N8 Y69.262
N9 X68.635 Y80.631
N10 G0 Z.5
N11 G1 Z-2.
N12 X72.244 Y92.
N13 X65.025
N14 Y69.262
N15 X68.635 Y80.631
N16 G0 Z-.5
N17 G1 Z-3.
N18 X72.244 Y92.
N19 X65.025
N20 Y69.262
N21 X68.635 Y80.631
N22 G0 Z-1.5
N23 G1 Z-4.
N24 X72.244 Y92.
N25 X65.025
N26 Y69.262
N27 X68.635 Y80.631
N28 G0 Z-2.5
N29 G1 Z-5.
N30 X72.244 Y92.
N31 X65.025
N32 Y69.262
N33 X68.635 Y80.631
N34 G0 Z-3.5
N35 G1 Z-6.
N36 X72.244 Y92.
N37 X65.025
N38 Y69.262
N39 X68.635 Y80.631
N40 G0 Z-4.5
N41 G1 Z-7.
N42 X72.244 Y92.
N43 X65.025
N44 Y69.262
N45 X68.635 Y80.631
N46 G0 Z-5.5
N47 G1 Z-8.
N48 X72.244 Y92.
N49 X65.025
N50 Y69.262
N51 X68.635 Y80.631
N52 Z2. F1000.
N53 X81.682 Y65.545 Z9. F30.
N54 Z-1.
```

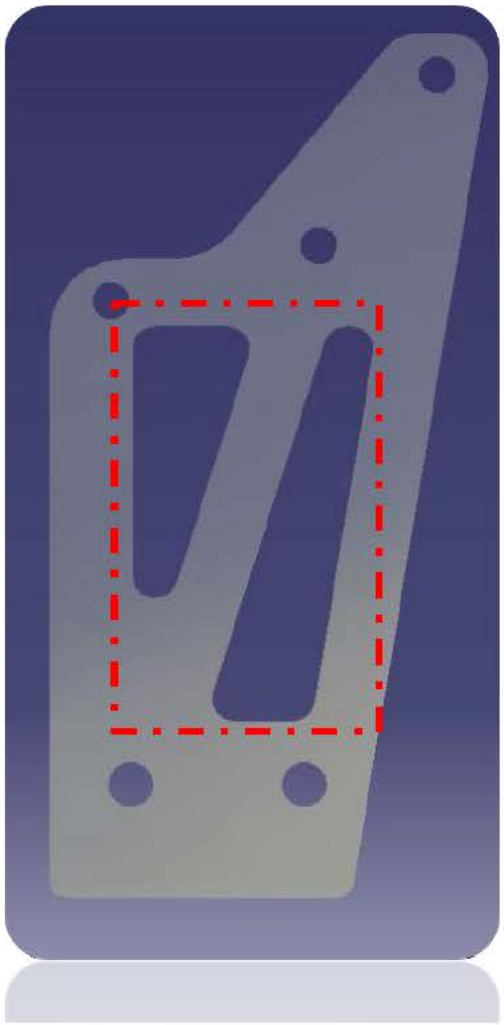


FIGURE 5 Path on Rendered Design

3-axis Programming Example

G code for internal pockets (cont.)

```
N55 X75.935 Y47.443
N56 X81.395
N57 X87.428 Y83.647
N58 X81.682 Y65.545
N59 G0 Z.5
N60 G1 Z-2.
N61 X75.935 Y47.443
N62 X81.395
N63 X87.428 Y83.647
N64 X81.682 Y65.545
N65 G0 Z-.5
N66 G1 Z-3.
N67 X75.935 Y47.443
N68 X81.395
N69 X87.428 Y83.647
N70 X81.682 Y65.545
N71 G0 Z-1.5
N72 G1 Z-4.
N73 X75.935 Y47.443
N74 X81.395
N75 X87.428 Y83.647
N76 X81.682 Y65.545
N77 G0 Z-2.5
N78 G1 Z-5.
N79 X75.935 Y47.443
N80 X81.395
N81 X87.428 Y83.647
N82 X81.682 Y65.545
N83 G0 Z-3.5
N84 G1 Z-6.
N85 X75.935 Y47.443
N86 X81.395
N87 X87.428 Y83.647
N88 X81.682 Y65.545
N89 G0 Z-4.5
N90 G1 Z-7.
N91 X75.935 Y47.443
N92 X81.395
N93 X87.428 Y83.647
N94 X81.682 Y65.545
N95 G0 Z-5.5
N96 G1 Z-8.
N97 X75.935 Y47.443
N98 X81.395
N99 X87.428 Y83.647
N100 X81.682 Y65.545
N101 Z2. F1000.
N102 M5
N103 M30
N104 M2
N105 M30
%
```

3-axis Programming Example

G code for drilling the holes

```
%
O1000
N1 G49 G64 G17 G21 G80 G0 G90 G40 G99
( IMSPPCC_MILL PPTABLE 06-13-2003 )
N2 X60.802 Y35. S70 M3
N3 Z15.
N4 G1 G94 Z5. F300.
N5 G83 X60.802 Y35. Z-8. R5. Q2. F10.
N6 X83.802
N7 G80
N8 G1 Z15. F1000.
N9 M5
N10 M30
N11 M2
N12 M30
%
```

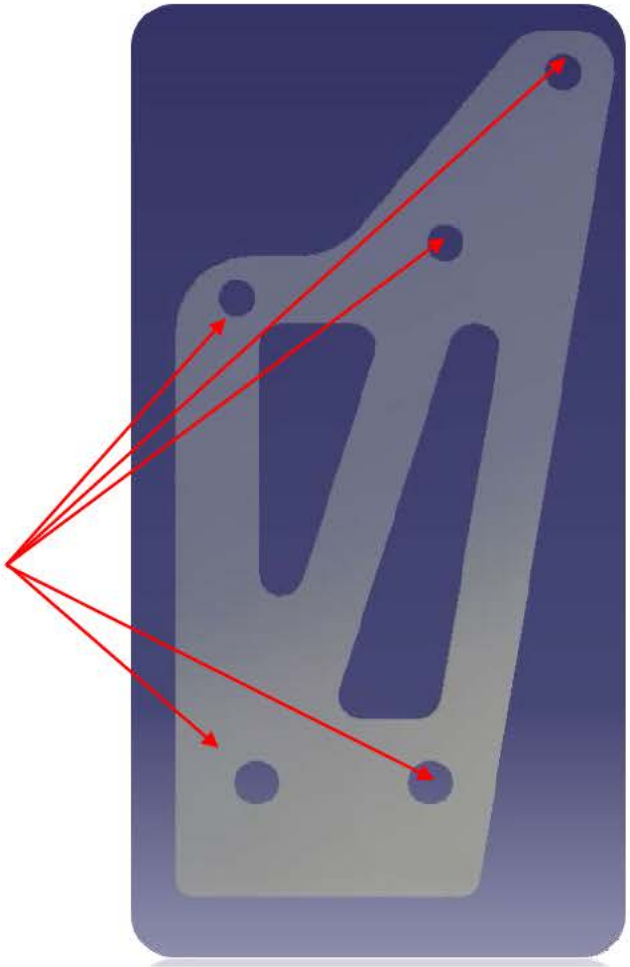


FIGURE 6 Designation of Holes on Rendered Design

3-axis Programming Example

G code for contour

```
%
O1000
N1 G49 G64 G17 G21 G80 G0 G90 G40 G99
( IMSPPCC_MILL PPTABLE 06-13-2003 )
( T4 END MILL D 8 )
N2 T0004 M6
N3 X46. Y22. S70 M3
N4 G43 Z9. H1
N5 G1 G94 Z-1. F30.
N6 Y94.989 F300.
N7 G2 X60. Y108.989 I14. J0
N8 G1 X66.657
N9 G3 X71.266 Y111.148 I0 J6.
N10 G1 X91.768 Y135.751
N11 G2 X98.682 Y138.989 I6.914 J-5.762
N12 G1 X103.263
N13 G2 X112.14 Y128.51 I0 J-9.
N14 G1 X94.224 Y21.014
N15 G2 X88.306 Y16. I-5.918 J.986
N16 G1 X52.
N17 G2 X46. Y22. I0 J6.
N18 G1 Y94.989
N19 G2 X60. Y108.989 I14. J0
N20 G1 X66.657
N21 G3 X71.266 Y111.148 I0 J6.
N22 G1 X91.768 Y135.751
N23 G2 X98.682 Y138.989 I6.914 J-5.762
N24 G1 X103.263
N25 G2 X112.14 Y128.51 I0 J-9.
N26 G1 X94.224 Y21.014
N27 G2 X88.306 Y16. I-5.918 J.986
N28 G1 X52.
N29 G2 X46. Y22. I0 J6.
N30 G0 Z.5
N31 G1 Z-2. F30.
N32 Y94.989 F300.
N33 G2 X60. Y108.989 I14. J0
N34 G1 X66.657
N35 G3 X71.266 Y111.148 I0 J6.
N36 G1 X91.768 Y135.751
N37 G2 X98.682 Y138.989 I6.914 J-5.762
N38 G1 X103.263
N39 G2 X112.14 Y128.51 I0 J-9.
N40 G1 X94.224 Y21.014
N41 G2 X88.306 Y16. I-5.918 J.986
N42 G1 X52.
N43 G2 X46. Y22. I0 J6.
N44 G1 Y94.989
N45 G2 X60. Y108.989 I14. J0
N46 G1 X66.657
N47 G3 X71.266 Y111.148 I0 J6.
N48 G1 X91.768 Y135.751
N49 G2 X98.682 Y138.989 I6.914 J-5.762
N50 G1 X103.263
N51 G2 X112.14 Y128.51 I0 J-9.
```

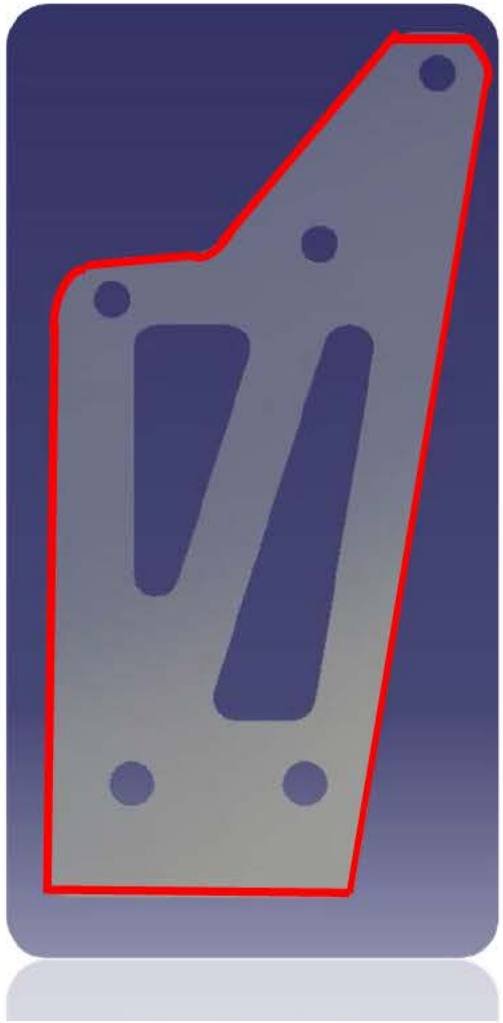


FIGURE 7 Designation of Contour on Rendered Design

3-axis Programming Example

G code for contour (cont.)

```
N52 G1 X94.224 Y21.014
N53 G2 X88.306 Y16. I-5.918 J.986
N54 G1 X52.
N55 G2 X46. Y22. I0 J6.
N56 G0 Z-.5
N57 G1 Z-3. F30.
N58 Y94.989 F300.
N59 G2 X60. Y108.989 I14. J0
N60 G1 X66.657
N61 G3 X71.266 Y111.148 I0 J6.
N62 G1 X91.768 Y135.751
N63 G2 X98.682 Y138.989 I6.914 J-5.762
N64 G1 X103.263
N65 G2 X112.14 Y128.51 I0 J-9.
N66 G1 X94.224 Y21.014
N67 G2 X88.306 Y16. I-5.918 J.986
N68 G1 X52.
N69 G2 X46. Y22. I0 J6.
N70 G1 Y94.989
N71 G2 X60. Y108.989 I14. J0
N72 G1 X66.657
N73 G3 X71.266 Y111.148 I0 J6.
N74 G1 X91.768 Y135.751
N75 G2 X98.682 Y138.989 I6.914 J-5.762
N76 G1 X103.263
N77 G2 X112.14 Y128.51 I0 J-9.
N78 G1 X94.224 Y21.014
N79 G2 X88.306 Y16. I-5.918 J.986
N80 G1 X52.
N81 G2 X46. Y22. I0 J6.
N82 G0 Z-1.5
N83 G1 Z-4. F30.
N84 Y94.989 F300.
N85 G2 X60. Y108.989 I14. J0
N86 G1 X66.657
N87 G3 X71.266 Y111.148 I0 J6.
N88 G1 X91.768 Y135.751
N89 G2 X98.682 Y138.989 I6.914 J-5.762
N90 G1 X103.263
N91 G2 X112.14 Y128.51 I0 J-9.
N92 G1 X94.224 Y21.014
N93 G2 X88.306 Y16. I-5.918 J.986
N94 G1 X52.
N95 G2 X46. Y22. I0 J6.
N96 G1 Y94.989
N97 G2 X60. Y108.989 I14. J0
N98 G1 X66.657
N99 G3 X71.266 Y111.148 I0 J6.
N100 G1 X91.768 Y135.751
N101 G2 X98.682 Y138.989 I6.914 J-5.762
N102 G1 X103.263
N103 G2 X112.14 Y128.51 I0 J-9.
N104 G1 X94.224 Y21.014
N105 G2 X88.306 Y16. I-5.918 J.986
```

3-axis Programming Example

```
N106 G1 X52.
N107 G2 X46. Y22. I0 J6.
N108 G0 Z-2.5
N109 G1 Z-5. F30.
N110 Y94.989 F300.
N111 G2 X60. Y108.989 I14. J0
N112 G1 X66.657
N113 G3 X71.266 Y111.148 I0 J6.
N114 G1 X91.768 Y135.751
N115 G2 X98.682 Y138.989 I6.914 J-5.762
N116 G1 X103.263
N117 G2 X112.14 Y128.51 I0 J-9.
N118 G1 X94.224 Y21.014
N119 G2 X88.306 Y16. I-5.918 J.986
N120 G1 X52.
N121 G2 X46. Y22. I0 J6.
N122 G1 Y94.989
N123 G2 X60. Y108.989 I14. J0
N124 G1 X66.657
N125 G3 X71.266 Y111.148 I0 J6.
N126 G1 X91.768 Y135.751
N127 G2 X98.682 Y138.989 I6.914 J-5.762
N128 G1 X103.263
N129 G2 X112.14 Y128.51 I0 J-9.
N130 G1 X94.224 Y21.014
N131 G2 X88.306 Y16. I-5.918 J.986
N132 G1 X52.
N133 G2 X46. Y22. I0 J6.
N134 G0 Z-3.5
N135 G1 Z-6. F30.
N136 Y94.989 F300.
N137 G2 X60. Y108.989 I14. J0
N138 G1 X66.657
N139 G3 X71.266 Y111.148 I0 J6.
N140 G1 X91.768 Y135.751
N141 G2 X98.682 Y138.989 I6.914 J-5.762
N142 G1 X103.263
N143 G2 X112.14 Y128.51 I0 J-9.
N144 G1 X94.224 Y21.014
N145 G2 X88.306 Y16. I-5.918 J.986
N146 G1 X52.
N147 G2 X46. Y22. I0 J6.
N148 G1 Y94.989
N149 G2 X60. Y108.989 I14. J0
N150 G1 X66.657
N151 G3 X71.266 Y111.148 I0 J6.
N152 G1 X91.768 Y135.751
N153 G2 X98.682 Y138.989 I6.914 J-5.762
N154 G1 X103.263
N155 G2 X112.14 Y128.51 I0 J-9.
N156 G1 X94.224 Y21.014
N157 G2 X88.306 Y16. I-5.918 J.986
N158 G1 X52.
N159 G2 X46. Y22. I0 J6.
N160 G0 Z-4.5
N161 G1 Z-7. F30.
N162 Y94.989 F300.
N163 G2 X60. Y108.989 I14. J0
N164 G1 X66.657
```

3-axis Programming Example

G code for contour (cont.)

```
N165 G3 X71.266 Y111.148 I0 J6.
N166 G1 X91.768 Y135.751
N167 G2 X98.682 Y138.989 I6.914 J-5.762
N168 G1 X103.263
N169 G2 X112.14 Y128.51 I0 J-9.
N170 G1 X94.224 Y21.014
N171 G2 X88.306 Y16. I-5.918 J.986
N172 G1 X52.
N173 G2 X46. Y22. I0 J6.
N174 G1 Y94.989
N175 G2 X60. Y108.989 I14. J0
N176 G1 X66.657
N177 G3 X71.266 Y111.148 I0 J6.
N178 G1 X91.768 Y135.751
N179 G2 X98.682 Y138.989 I6.914 J-5.762
N180 G1 X103.263
N181 G2 X112.14 Y128.51 I0 J-9.
N182 G1 X94.224 Y21.014
N183 G2 X88.306 Y16. I-5.918 J.986
N184 G1 X52.
N185 G2 X46. Y22. I0 J6.
N186 G0 Z-5.5
N187 G1 Z-8. F30.
N188 Y94.989 F300.
N189 G2 X60. Y108.989 I14. J0
N190 G1 X66.657
N191 G3 X71.266 Y111.148 I0 J6.
N192 G1 X91.768 Y135.751
N193 G2 X98.682 Y138.989 I6.914 J-5.762
N194 G1 X103.263
N195 G2 X112.14 Y128.51 I0 J-9.
N196 G1 X94.224 Y21.014
N197 G2 X88.306 Y16. I-5.918 J.986
N198 G1 X52.
N199 G2 X46. Y22. I0 J6.
N200 G1 Y94.989
N201 G2 X60. Y108.989 I14. J0
N202 G1 X66.657
N203 G3 X71.266 Y111.148 I0 J6.
N204 G1 X91.768 Y135.751
N205 G2 X98.682 Y138.989 I6.914 J-5.762
N206 G1 X103.263
N207 G2 X112.14 Y128.51 I0 J-9.
N208 G1 X94.224 Y21.014
N209 G2 X88.306 Y16. I-5.918 J.986
N210 G1 X52.
N211 G2 X46. Y22. I0 J6.
N212 G1 Z2. F1000.
N213 M5
N214 M30
N215 M2
N216 M30
%
```

3-axis Programming Example

Final Part

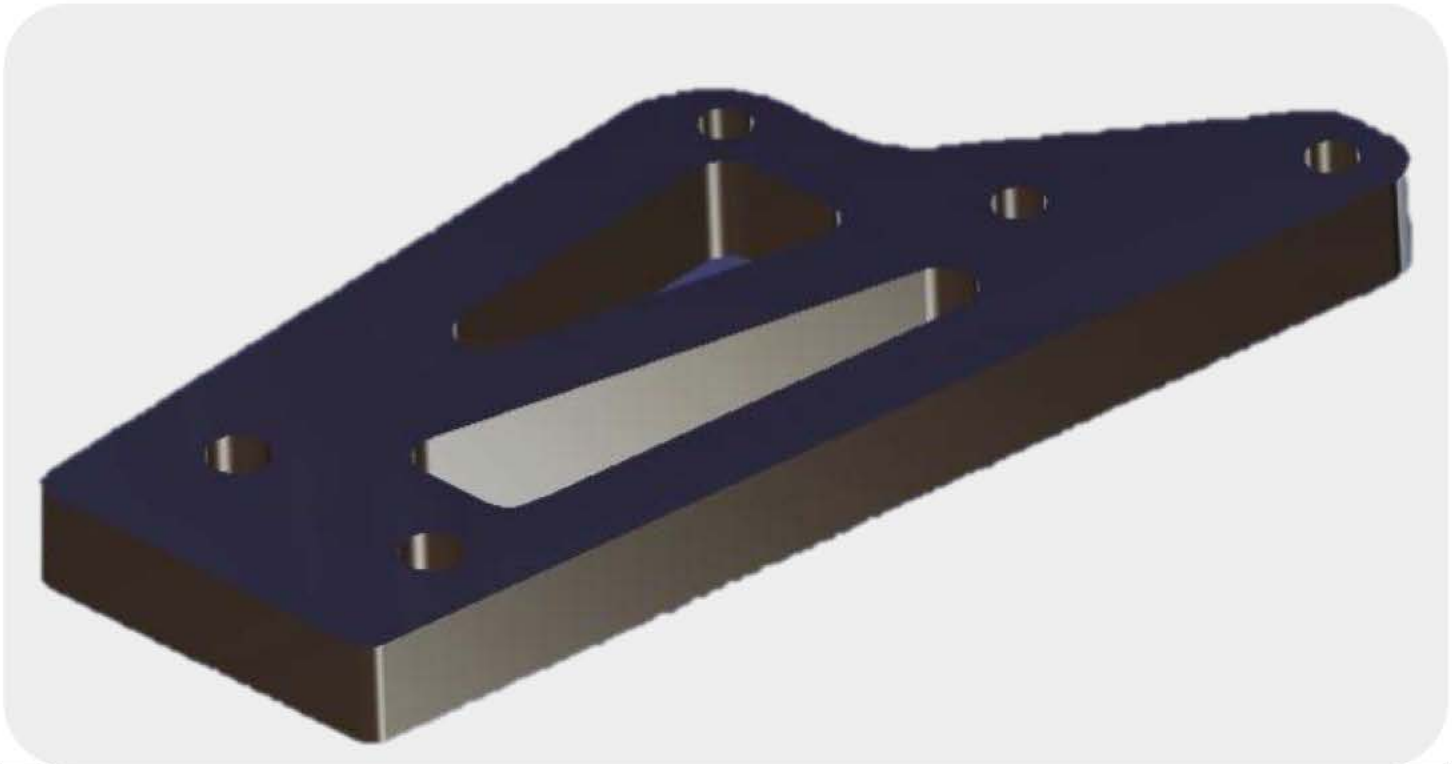


FIGURE 8 Final Part

Summary 1/2

- The important concepts presented in this section are:

- **Tool lengths** in three-axis machines must be **pre-set** by the operator. On some controls they can be pre-set in the program

- ***The initial level is the Z-axis spindle position*** when an 80 series canned cycle commences

- A **reference (or rapid) level is the Z-axis feedrate engagement point**, selected by the programmer

- **G98** selects a **return to initial level**, and **G99** selects a **return to reference level** when using 80 series G codes (canned cycle codes)

- **Canned cycles** are **routines built into the controller to simplify programming**.

Summary 2/2

- Values, called **parameters**, are passed to the control indicating **how the cycle is to perform, where the cycle is to begin, and how it should repeat**
- Positioning the spindle in two axis, then feeding with the third is called **$2^1/_2$-axis programming**
- Feeding with all three axes simultaneously is called **3-axis programming**
- **Indexers** often are used on CNC machinery. Positioning an index usually is just a matter of calling out the axis designator and a coordinate (i.e., AC, B270., A135.)

Vocabulary Introduced in this section

- $2^1/_2$-axis programming
- 3-axis programming
- 4-axis programming
- Canned cycle
- Indexer
- Initial level
- Rapid level
- Reference level
- Rotary table

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.

Third Parties' Works Note

This Work makes use of the following works:

Figure 2: slide 12, CC BY-SA 3.0, https://commons.wikimedia.org/wiki/File:Roland-CAMM-3-PNC-3000-desktop-3d-milling-machine.jpg