cnc webschool
knowledge to improve

Appendix 1: Programming on Siemens Control

Y-Axis Lathe Programming

Paper Size: 170x244mm (Book Size)

Contents

1.1 Introduction

This appendix will cover the programming of a lathe with 4 axes (X, Z, Y, C).

The machine is essentially the same as that analysed in the book with the added possibility of moving the turret along the transversal axis on plane G18.

As already seen in paragraph 4.5, the Y-axis gives the lathe greater flexibility when executing milling operations.

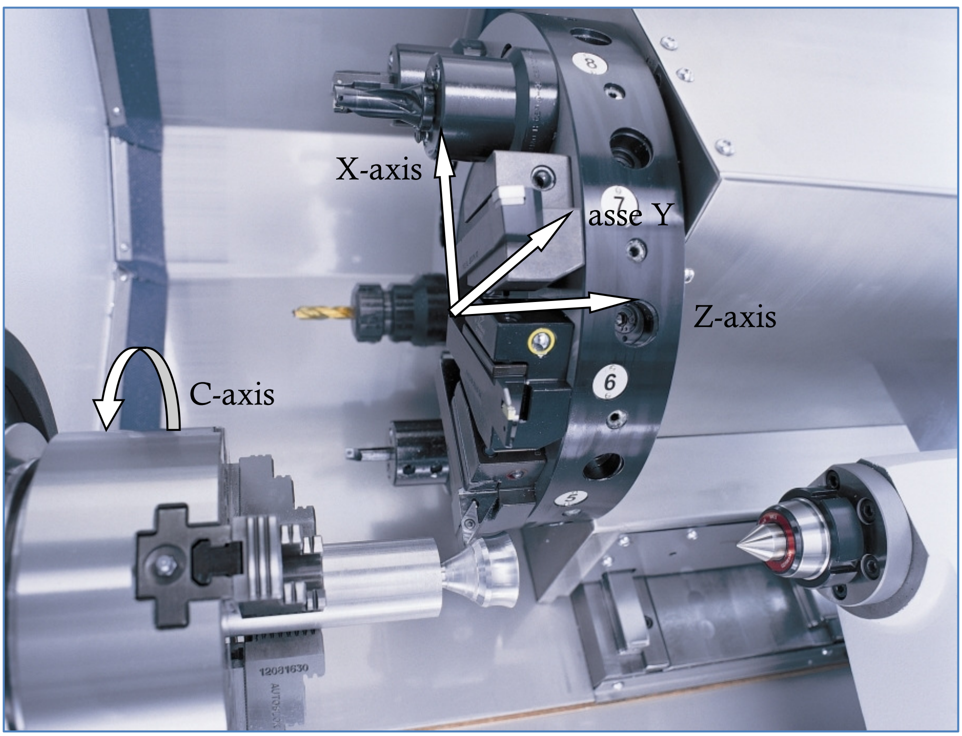

Fig. 1. Example of a lathe with 3 axes and motorised tools

The ISO functions presented during the course were first used on the X-Z (G18) turning plane; then in Chapter 32 they were used to define the profile to be milled on the X-Y (G17) plane; now they are used to program the milling operations carried out on the Y-Z (G19) plane.

The programming concepts for each single function are exactly those already presented in the chapters contained within the book.

A programming scheme remains to be established to determine:
- the **clockwise and anticlockwise directions** of the circle arcs,
- the **right and left position of the tool** compared to the profile of the part,
- the **value of the angles** to be used in direct programming.

The programming scheme to be used is that already analysed in paragraph 4.9.

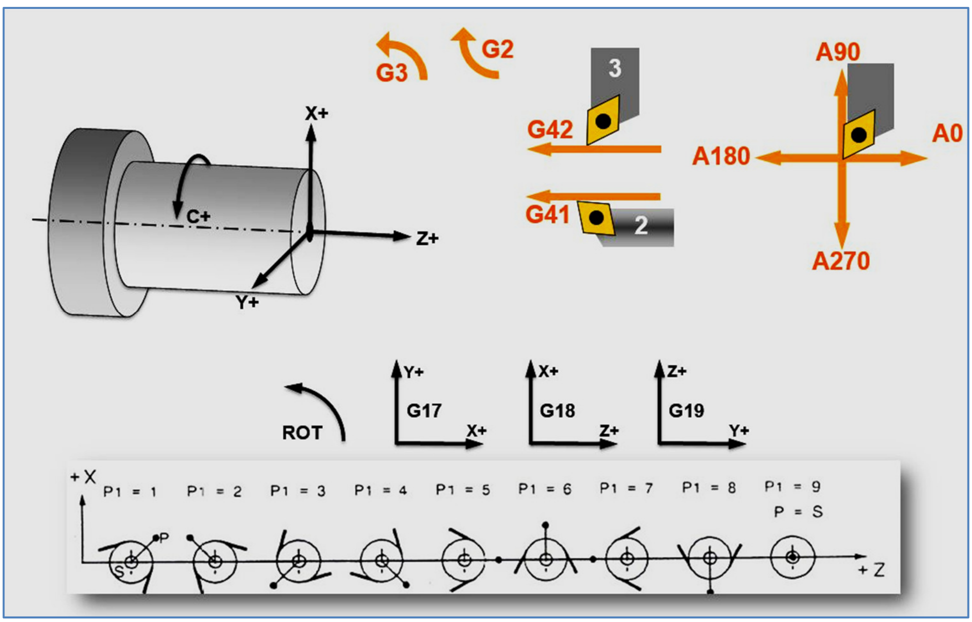

Fig. 2. Programming scheme for all work planes

1.2 Specific programming scheme for plane G19

So as to use correctly the programming scheme, it must be positioned according to the positive direction of the axes present in the machine, the tool's intended trajectory must be evaluated and the correct function to be used must be chosen.
Below is the specific scheme to be used on the Y-Z work plane.

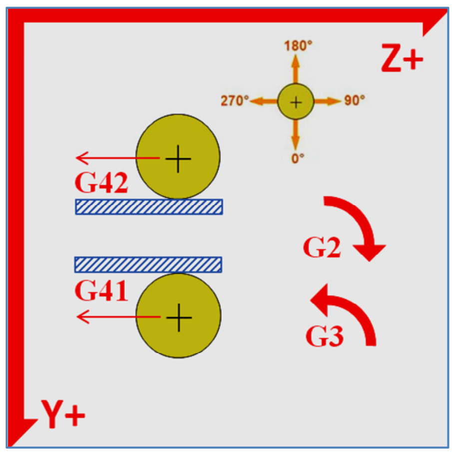

Fig. 3. Specific programming scheme for the Y-Z plane

1.3 Programming example

Programming for the following work piece starts with the application of the scheme in Figure 3.

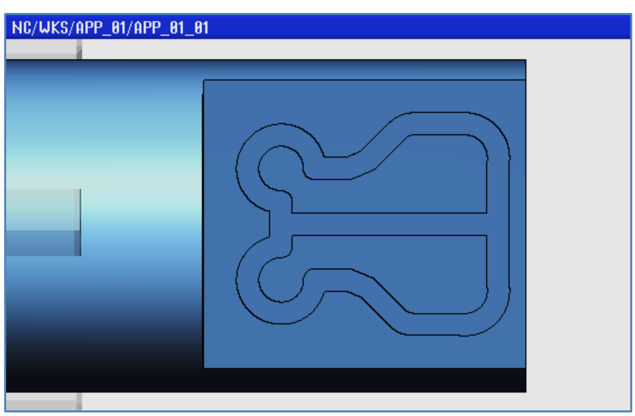

Fig. 4. Programming example with work on the Y-Z plane

1.3.1 Y-axis positive direction

In a lathe, the zero point of the Y-axis is located (as for the X-axis) on the rotation axis of the part.
The tool's direction of movement (indicated by the red arrow) shows that the mill is moving from Y negative to Y positive.

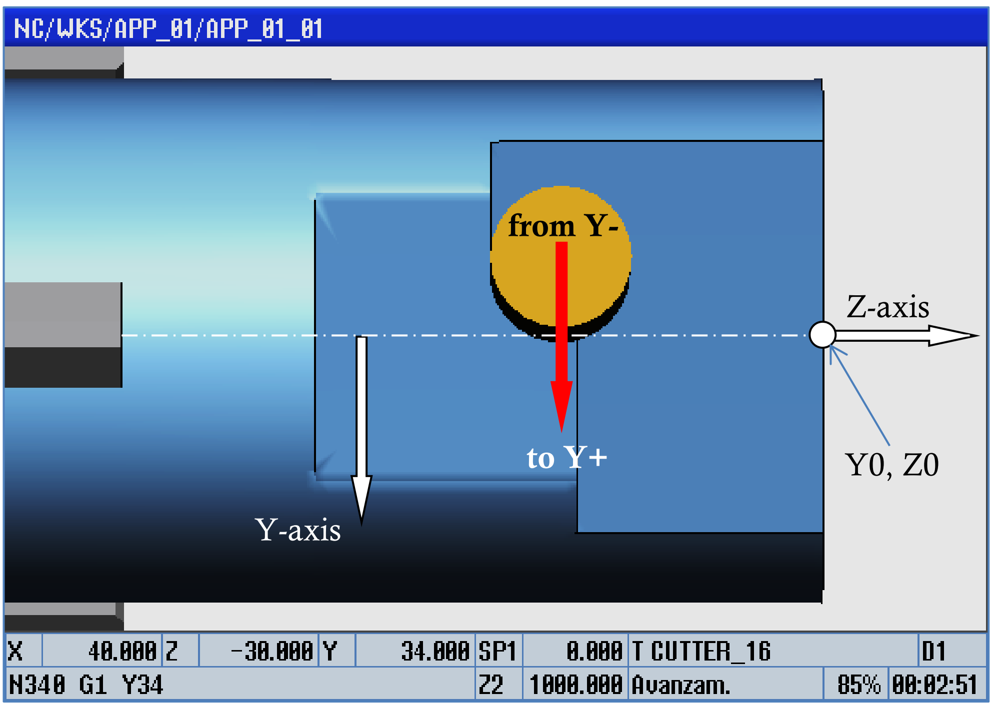

Fig. 5. Flat milling on the Y-Z plane

Analyse the example of a program that carries out a flattening of the part, considering that the position of the milling plane is determined by the value programmed into X.

```
; dimensions of the workpiece:
; XA = 80 diameter of the bar
; ZA = 0 machining allowance on front face
; ZI = -120 length of the finished part
; ZB = -80 protuberance from the jaws

N10 WORKPIECE(,,,"CYLINDER",192,0,-120,-80,60)

N20 G54
```

```
N30 G0 X400 Z500
N40 M8
N50 SETMS(1)
N60 SPOS=0
N70 SETMS(3)

N80 G19

N90 T19 D1 G0 Y0 ;RADIAL MILL D.16
N100 G95 S2000 M3

; FIRST FLATTENING A X50
N110 G0 Y28
N120 Z0
N130 X50
N140 G1 Y-26 G94 F200

N150 G0 Z-10
N160 G1 Y28

N170 G0 Z-20
N180 G1 Y-28

N190 G0 Z-30
N200 G1 Y28

N210 G0 Z-40
N220 G1 Y-28

N230 G0 Z-50
N240 G1 Y-28


; SECOND FLATTENING AT X40
N250 G0 Y34
N260 Z0
N270 X40
N280 G1 Y-34 G94 F200

N290 G0 Z-10
N300 G1 Y34

N310 G0 Z-20
N320 G1 Y-34

N330 G0 Z-30
N340 G1 Y34 (BLOCK SHOWN IN FIGURE 5)
```

```
N350 G0 Z-40
N360 G1 Y-34

N370 G0 Z-50
N380 G1 Y-34

N390 G0 Y50


; THIRD FLATTENING AT X32
N400 G0 Y39
N410 Z0
N420 X30
N430 G1 Y-39 G94 F150

N440 G0 Z-10
N450 G1 Y39

N460 G0 Z-20
N470 G1 Y-39

N480 G0 Z-30
N490 G1 Y39

N500 G0 Z-40
N510 G1 Y-39

N520 G0 Z-50
N530 G1 Y-39

N540 G0 Y50

N550 G0 X200
N560 G0 Z200
```

1.3.2 Circular interpolation

As can be seen in Figure 4, the path to be followed is symmetrical relative to the Z-axis.
After flattening, the program continues with the completion of the remaining half in the positive Y quadrant.
The red arrow indicates that the mill, to make the radius, turns clockwise; based on the programming scheme, the function to be used is therefore G2 (see Chapter 13).

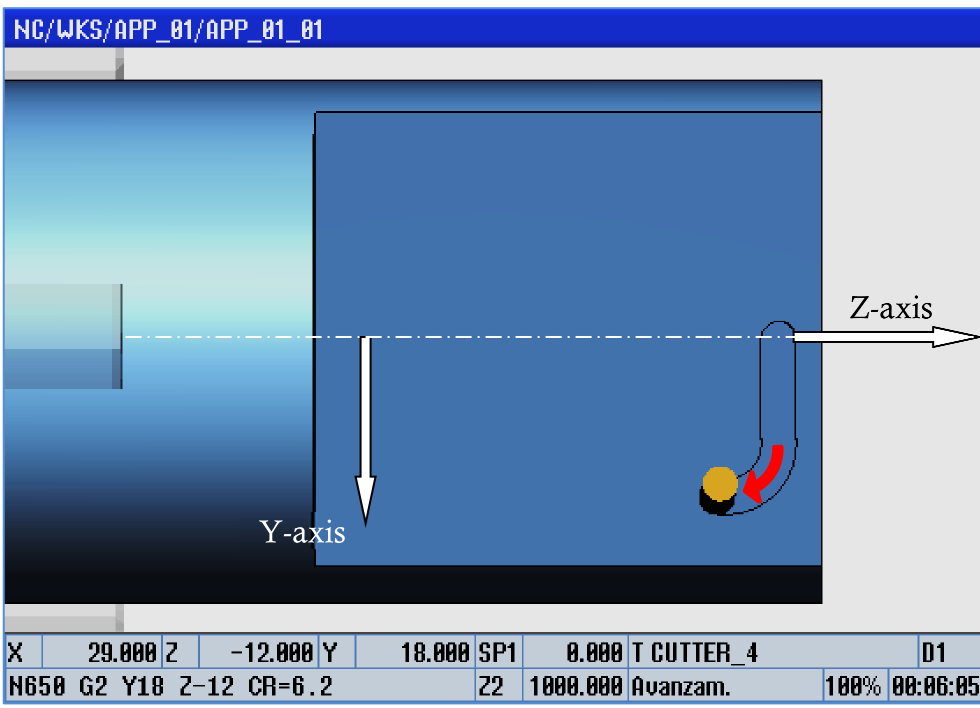

Fig. 6. Circular interpolation on the Y-Z plane

```
N570 T10 D1 G0 Y0 ;RADIAL MILL D.4
N580 G95 S4000 M3

; PROFILE START
N590 G0 Y0
N600 Z-5
N610 X34
N620 G1 X29 G94 F120

N630 START:
```

```
N640 Y11
N650 G2 Y18 Z-12 CR=6.2
```

1.3.3 Direct programming of radii, chamfers and angles

After the radius, the profile continues with a line parallel to the Z-axis which connects with a radius to the following straight line inclined at 45°.

The direct programming of radii, chamfers and angles occurs according to the same concepts and functions present in chapter 12.

Aligning the programming scheme in Figure 3 with the positive direction of the axes, it results that the angle value to be programmed is 225°.

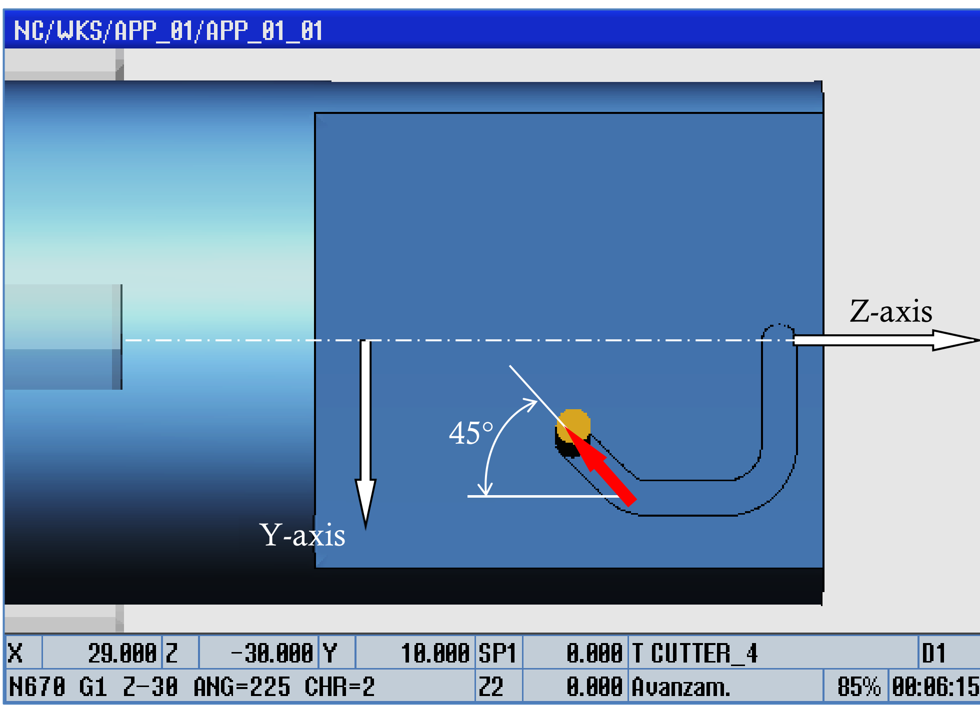

Fig. 7. Direct programming of radii, chamfers and angles on the Y-Z plane

```
N660 G1 Z-22 RND=4
N670 G1 Z-30 ANG=225 CHR=2
```

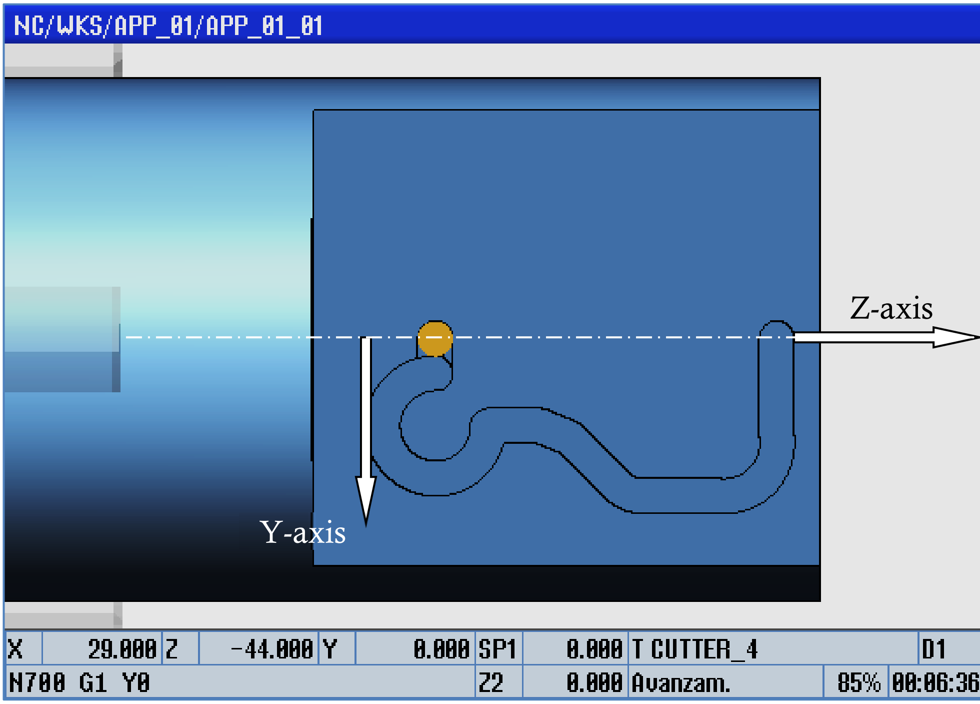

Fig. 8. Completion of half of the profile

The program completes the lower half of the profile and withdraws the mill to X32.

```
N680 G1 Z-38
N690 G2 Z=IC(-6) Y=IC(-6) K-6 J0
N700 G1 Y0
N710 END:

N720 G0 X32
```

1.3.4 MIRROR: Programmable mirror function

The upper part of the profile happens to be a perfect mirror image of the lower part.

The MIRROR function allows for the inversion of the positive direction of one or more axes.

In order to create the upper profile, the program repositions the mill at the start point of the lower profile (co-ordinates Y0, Z-5, X29), activates the mirroring of the Y-axis (MIRROR Y0) and repeats the blocks programmed between the buttons ‘START:’ and ‘END:’.
The MIRROR function (programmed without the name of any axis) deactivates the mirror function.

```
N730 G0 Y0 Z-5
N740 G1 X29
```

`N750 MIRROR Y0 ; Y-AXIS MIRROR ACTIVATION`

```
N760 REPEAT START END
```

Once the path is complete, Y-axis mirroring (MIRROR) deactivates and the block G1 Z-5 is carried out to complete the profile.

`N770 MIRROR ; MIRROR DEACTIVATION`

```
N780 G1 Z-5

N790 G0 X400
N800 G0 Z200
N810 G0 Y0

N820 M30
```