CNC PART PROGRAMMING (turning and milling)

PART 1 (Turning)

Generate a manual part program for the part shown in Figure 1 for the NUM Flexium 68. This task involves the use of G63, turning/facing cycle (additional info is given in pages 5-10). This cycle is used to generate a profile from a workpiece with an arbitrary shape. The maximum depth of cut you can use is 2 mm. The maximum spindle speed should not exceed 3500rpm (G92) and the constant surface speed and feed rate is 200 m/min, 0.25 mm/rev for roughing and 250 m/min, 0.1 mm/rev for finishing. The tool holder and offset information is saved in T01 D01 for roughing and T03 D03 for finishing. The tool change should occur in X150 Z150. The maximum groove penetration angle, finishing allowance, positioning clearance, and minimum chip value should be set at -145 degrees, 0.4 mm, 2 mm, and 1 mm, respectively.

The program should follow the following steps:

- Set absolute dimension, referenced to the measurement origin, rapid to X150 Z150
- Change tool to T01
- Set spindle speed to 900 rpm, counter clockwise
- Rapid to X100 Z50
- Set maximum spindle speed to 3500 rpm
- Set constant surface cutting speed 200 m/min
- Set feedrate 0.25 mm/min
- Start the roughing cycle
- Referenced to the measurement origin, move to Z150 Z150
- Change tool to T03
- Set the right radius offset for the tool, move to point a
- Set constant surface cutting speed to 250 m/min
- Start the finishing process by moving to point b
- Move to point c, d, e ...
- Turn off tool radius correction, go to X150 Z150 (referenced to the measurment origin), turn off the spindle speed.
- End the program.

PART 2 (Milling)

Generate a part program using the software “Walli” for the part shown in Figure 2 for the Cybermill CNC milling machine. Once the program is completed, simulate the program for verification, and then run the program on the Cybermill using the tool.

The diameter of the end milling you will be using is 6 mm (use T word for this). Include the following values in the program.

Spindle speed: 7 (3500 rpm)
Spindle rotation clockwise: M03
Feed rate: 4 (250 mm/min)
Workpiece clamp: M68

The Cybermill has predetermined spindle speed and feed rate as follows:

	0	1	2	3	4	5	6	7	8	9
Feed rate (mm/min)	50	100	150	200	250	300	350	400	450	500
Spindle speed (rpm)	Off	500	1000	1500	2000	2500	3000	3500	4000	4500

The home position of the Cybermill is the upper left corner (from your view), positive X and Y directions are to your right and toward you facing the machine. The distance from the tip of the tool to the top of the workpiece will be provided by the TA. The coordinate of the upper left corner of the workpiece is (3,3) after the part is clamped in position. Limit the maximum depth of cut per pass to 3 mm.

For the outside step machining, use two paths to remove all the material. First cut off the outside of the part, then cut the rectangular shape. The groove is 5mm deep and it needs to be made with two cuts: first 3mm, then 2mm.

Before drilling the hole, bring the tool to 10 mm above the drilling end point. For drilling, use G82 and D (2 seconds for dwell time) and set the feed to 1 (100 mm/min) (G82 Z10 F1 D2...where the Z coordinate should be the relative distance)

For slotting of the middle section, start the cutting from the left side and complete the cycle.

Take the diameter of the end mill into account when determining the tool movement.

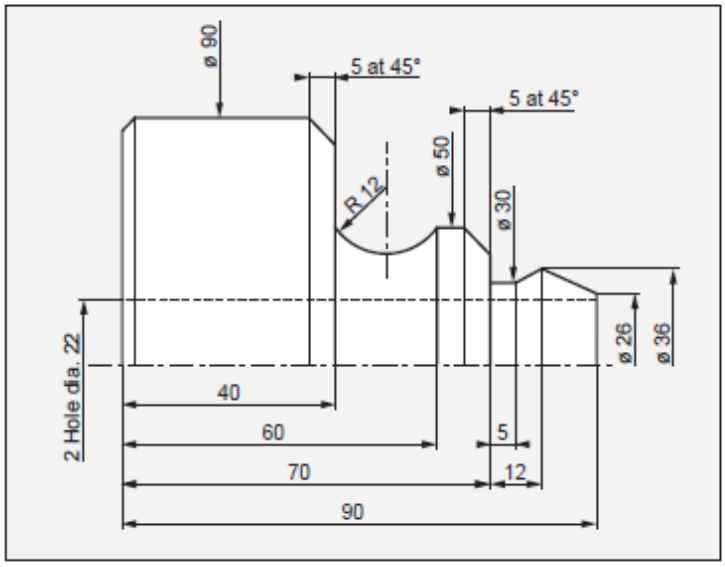
Note: lift up the cutting tool before you move to the next position

Once the part program is completed, show it TA for running the program.

The report must contain a cover page, a brief objective of lab., printouts of part programs and corresponding drawings and parts, your discussion and conclusions. The report must be typed.

Figure 1 (turning):

Rough and finish turning of an external profile



Machining paths

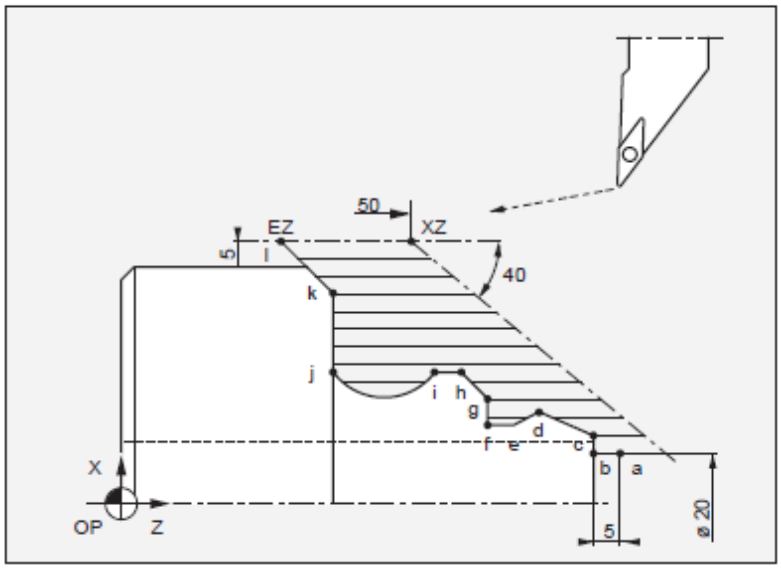
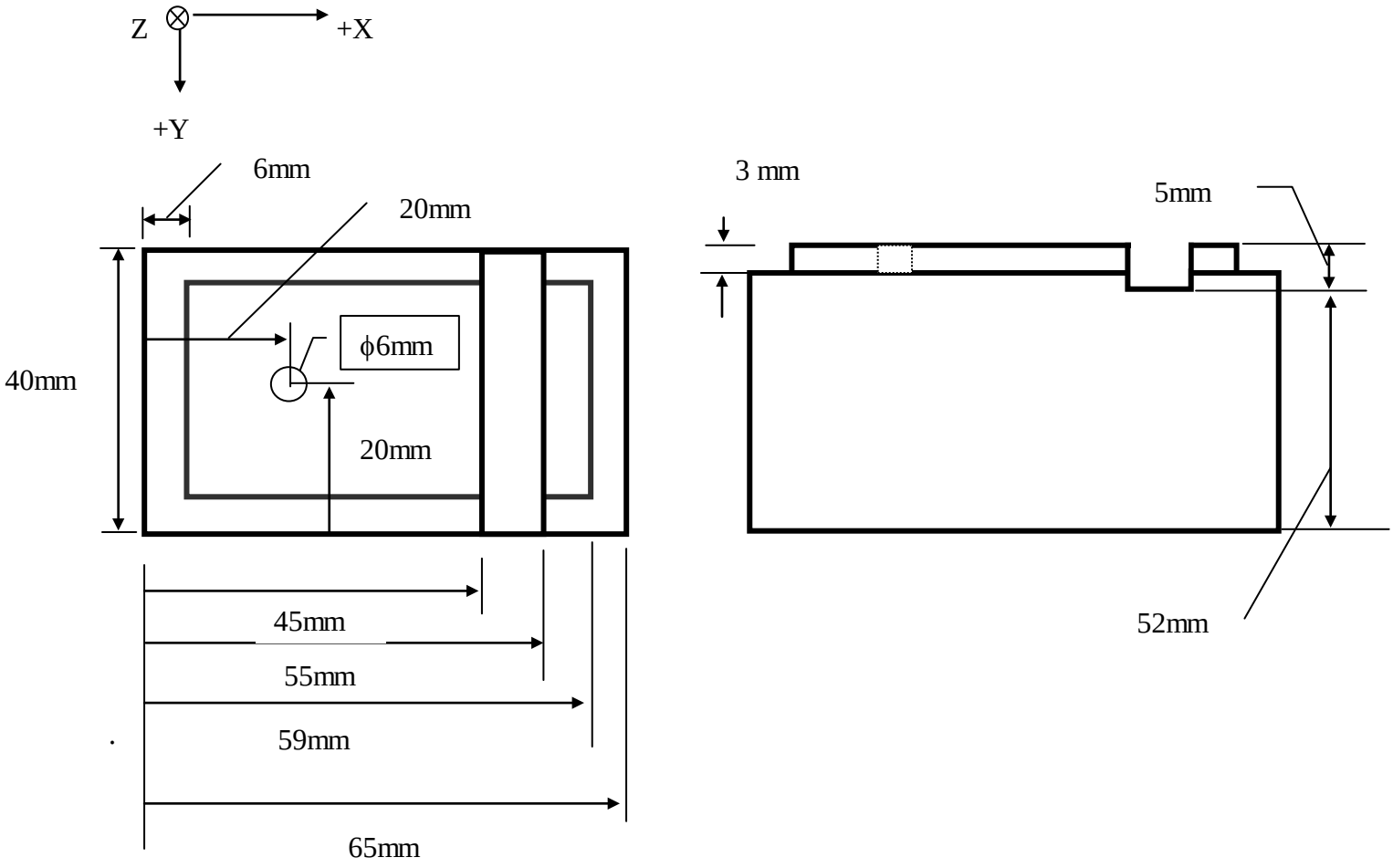


Figure 2 (milling)

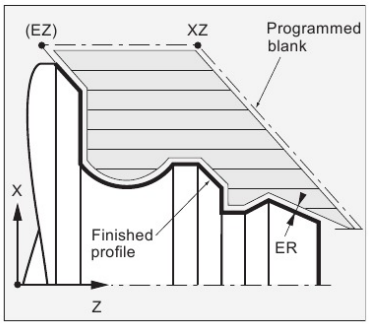


5.13.7 Roughing Cycle with Groove ► G63

G63	Roughing cycle with groove.

This function is used for roughing a volume of material located between the definition of a blank and a finished profile.

The cycle will machine a groove whose profile is compatible with the geometry and radius of the tool used. It can be carried out by facing or turning and for external or internal machining.



Syntax

N.. **G63** [N.. N..] / X.. Z.. EX.. / EZ.. P.. / R.. EA.. / EU.. / EW.. [EB..] [EC..] [ER..] [Q..] [EQ..] [EF..]

G63	Roughing cycle with groove.
N.. N..	First and last block numbers defining the finished profile (maximum 95 blocks).
X.. Z.. (or U.. W..)	Cycle start point.
EZ.. / EX..	Pass end point on the roughing axis: - EZ for rough turning along Z. - EX for rough facing along X. Direction of roughing pass execution: - Z+ if EZ > Z Z- if EZ < Z - X+ if EX > X X- if EX > Z
P.. / R..	Pass depth: - P: value along X (rough turning) - R: value along Z (rough facing).
EA../EU../EW..	Roughing pass start positions (see Figs. 1 and 2). These arguments are programmed to define a start angle for cut application and can be combined: EA EU, EA EW or EU EW.
EB..	Maximum groove penetration angle. Angle used to take into account the groove profile geometry and the tool rake angle (α) (see Fig. 3). - By default, EB is parallel to the roughing axis.
EC..	Maximum cut exit angle. Angle used to take into account the groove profile geometry and the tool angle of attack (K) (see Fig. 4). - By default, EC is perpendicular to the roughing axis.
ER..	Finishing allowance. Value applied normal to the profile. - The default value of ER = 0.
Q..	Positioning clearance. Approach distance at feed rate before contact with the material (for rapid to next cut depth). - The default value of Q = 0 (no clearance).
EQ..	Minimum chip value. - Below the programmed value, the pass is not executed.
EF..	Penetration feed rate in the material. - The default value of EF is the same as the modal feed rate F.

Property of the Function

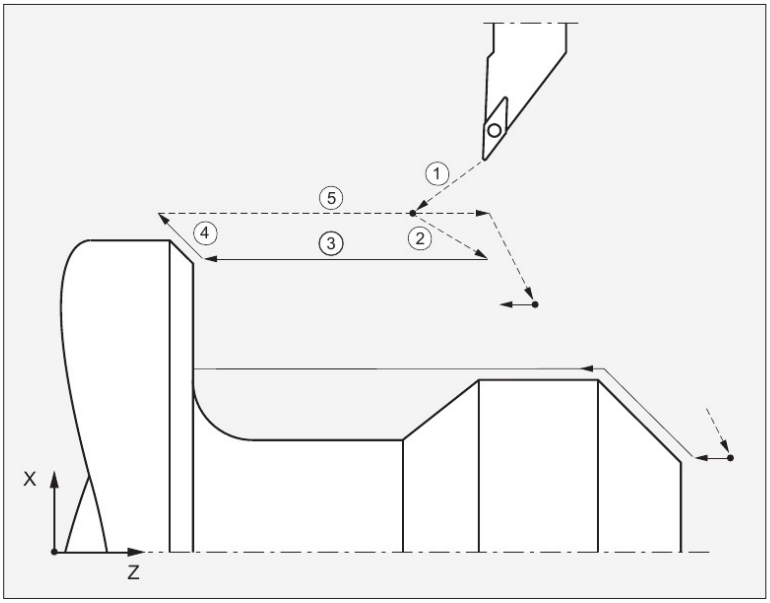
Function G63 is nonmodal.

Cancellation

Function G63 is cancelled at the end of the block.

Cycle Steps

Description of the first rough turning and grooving passes

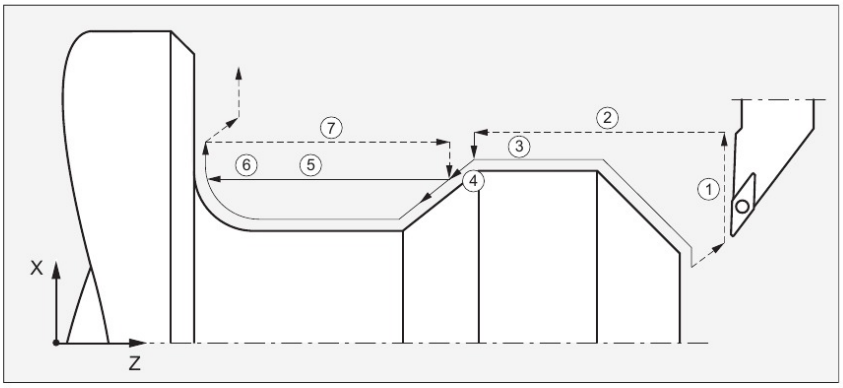


Rough turning pass

- Step 1: Tool positioning to X Z.
- Before the cycle begins, the system reads the declared tool radius.
- Step 2: Rapid to first cut depth P (for turning).
- Step 3: Execution of the first pass at the machining feed rate.
- Step 4: Lift along the profile at the machining feed rate.
- Step 5: Rapid return to the cycle start point.

The following passes are executed in the same way as steps 2 to 5.

After execution of the last roughing pass, the tool is retracted from the surface and positioned in X.



Grooving pass

- Step 1: Rapid positioning in X.
- Step 2: Rapid positioning in Z.
- Step 3: Positioning in X at the machining feed rate.
- Step 4: Cut application at the machining feed rate along the groove profile.
- Step 5: Execution of the pass at the machining feed rate.
- Step 6: Lift at the machining feed rate along the groove profile.
- Step 7: Rapid return to the cycle start point.

The following passes are executed in the same way as steps 3 to 6.

After execution of the last pass, the tool is retracted from the surface and positioned in X.

Notes

The number of blocks from N.. to N.. must not exceed 95. Otherwise, the system returns error message 92.

When the cycle is programmed, the system must be in state G40 (G41 or G42 tool radius offset cancelled).

At the end of the cycle, the system is left in G0 mode.

The finished profile definition can be programmed before the cycle call, but in this case it is necessary to use the branch function G79 (see chapter 5.14.3).

The cycle arguments defining positive angles are specified counterclockwise.

The feed rate function and argument can be programmed in the cycle block, e.g.:

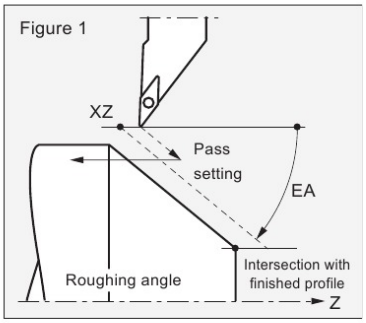
```
N.. G63 N.. N.. X.. Z.. EZ.. P.. EA.. EB.. ER.. Q.. EQ.. EF.. G95 F0.3
```

Notes on arguments EA, EU and EW

- EA: Angle limiting the start of cut application between point XZ (cycle start) and intersection with the finished profile.
  (If there is no intersection, the last pass is limited to the first point of the finished profile)

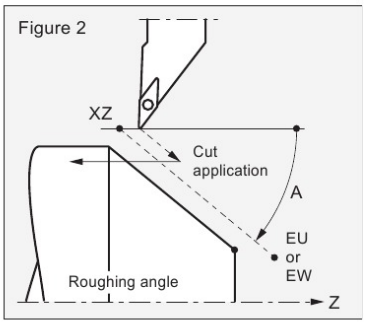
After positioning to point XZ, the system reads the declared tool radius before carrying out the first cut application.

Figure 1



- EU or EW: End point of the last roughing pass. If argument EA is not programmed, the values declared with EU and/or EW and XZ (cycle start) are used to define the cut application start angle (A).

Figure 2

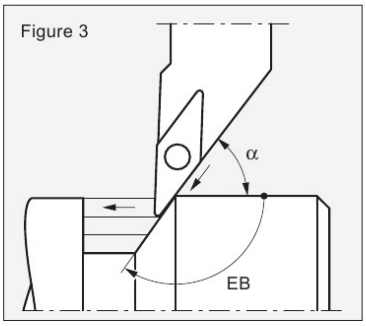


The angle is also used to differentiate between applying the cut at feedrate or rapid for a groove or not, respectively:

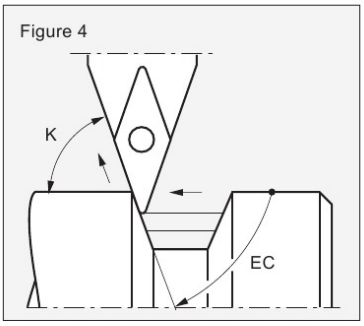
- Machining feed rate (in the material) if the angle between the cut application line and the roughing axis is $\geq$ 90 degrees.
- Traversing rate (off the material) if the angle between the cut application line and the roughing axis is < 90 degrees.

5.13.7.1 Definition of EB and EC

Angle defined by EB

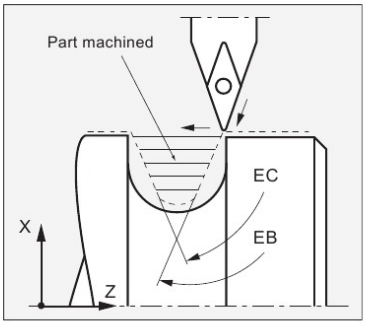


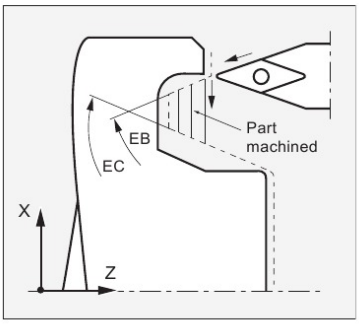
Angle defined by EC



Machining of a groove limited by declaration of angles EB and EC

When the groove profile and tool geometry are completely incompatible, the groove is not executed.

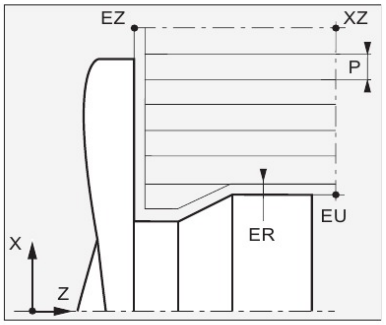




5.13.7.2 Cycle differences between turning and facing

Rough turning

```
N70 ...
N80 G63 N.. N.. X.. Z.. EZ.. P.. EU.. EB.. ER..
N90 G52 X.. Z..
N..
```



Rough facing

```
N70 ...
N80 G63 N.. N.. X.. Z.. EX.. R.. EW.. EC.. ER..
N90 G52 X.. Z..
N..
```