TYPES OF CNC MACHINE CONTROL UNITS

- FANUC (**F**actory **A**utomatic **Nu**merical **C**ontrol-Japan)
- SIEMENS
- GSK- China
- MACH3 (USA)

The Home of Mach3

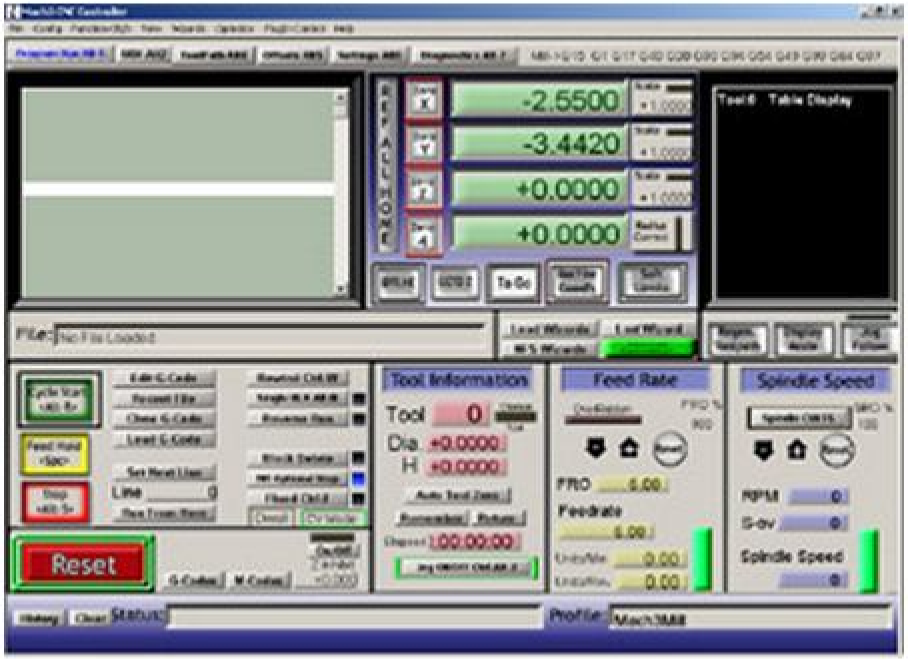
Artsoft has been in the CNC business since 2001.

In this time, the Mach series of CNC software has evolved into the best available PC-based CNC software on the market.

Not only is it extremely affordable to the hobbyist and industry alike, it is pioneering in its features and continuing development.

There are over 10000 users of Mach who swear by its ease of use, great features, and outstanding support.

Here are some of the basic features and functions provided by Mach3:



-Converts a standard PC to a fully featured, 6-axis CNC controller,

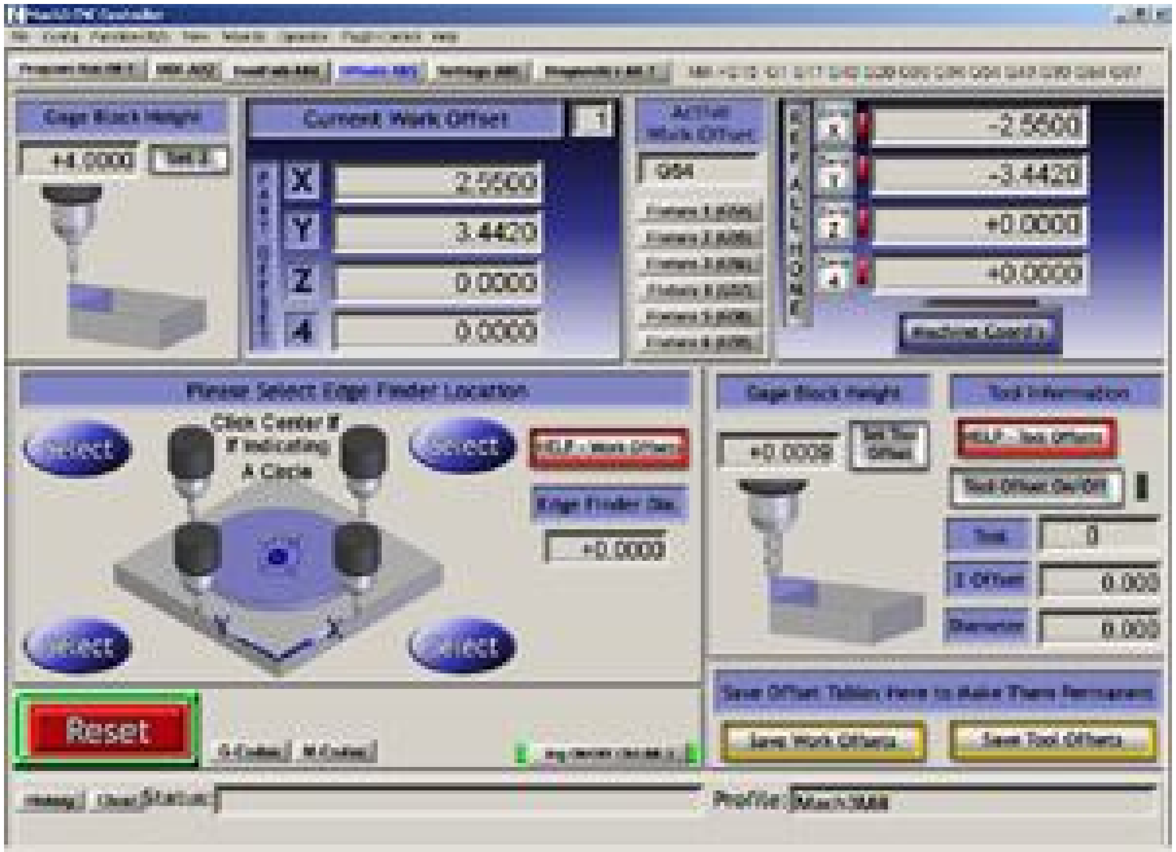
-Allows direct import of DXF, BMP, JPG and HPGL files through LazyCam,

-Visual G-code display,

-Generates G-code via LazyCam or Wizards,

-Fully customizable interface,

-Customizable M-Codes and Macros using VBscript



- Spindle Speed control

- Multiple relay control

-Manual pulse generation

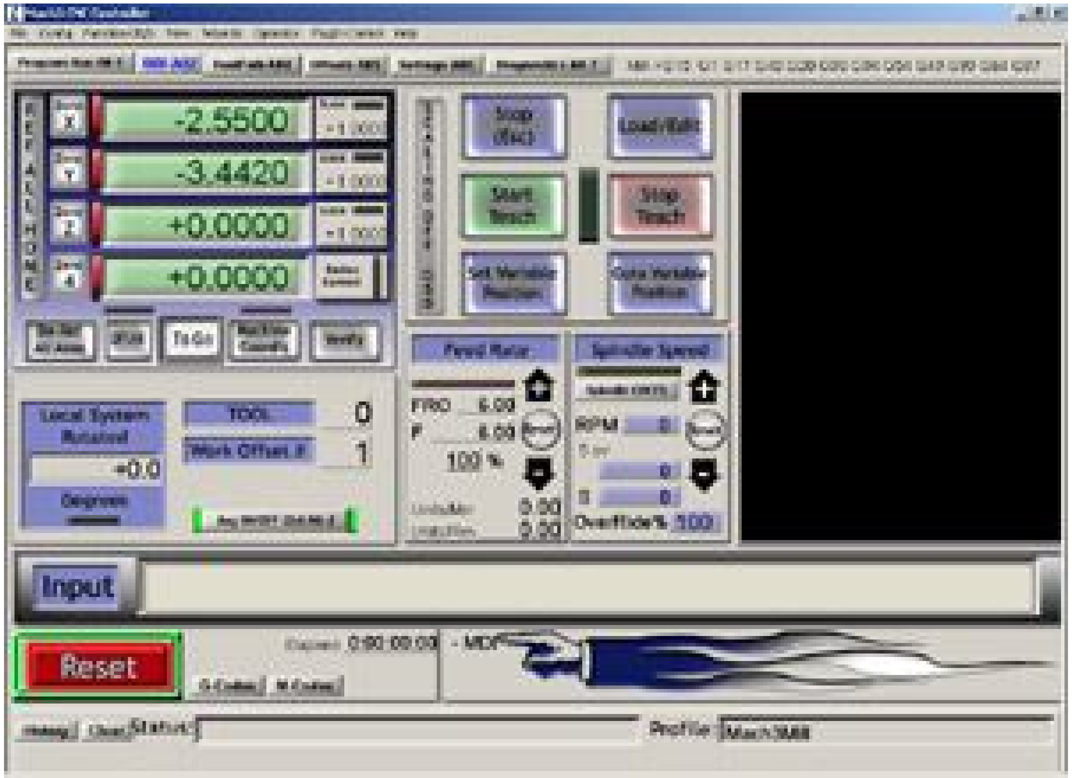
-Video display of machine

-Touch screen ability

-Full screen eligibility

Mach3 has successfully been used to control the following types of equipment:

- ***Lathes***
- ***Mills***
- ***Routers***
- ***Lasers***
- ***Plasma***
- ***Engravers***
- ***Gear cutting***



Wizards

Wizards are "mini-programs" that extend the capabilities of Mach3.

Wizards can be written by anybody and are designed to allow users to quickly perform routine or convenient operations without the need to have a pre-written G-Code file.

Included with Mach3 are free Wizards for such things as:

- *Gear cutting*
- *Digitizing*
- *Holes*
- *Slots and keyways*
- *Text engraving*
- *Many standard shapes*
- *Surfacing*
- *MANY MORE*

There Are Many Dialects of G-Code

Now that you have some basics under your belt, it's time to talk about some of the complications that arise from different dialects of G-Code.

Some wag once joked that the great thing about standards is there are so many to choose from. So it is from G-Code.

While much of it remains the same from controller to controller (setting aside alternatives to G-Code from things like Mazatrol, Heidenhain's CNC language, and others), there are important details and defaults you need to be aware of to understand the particular dialect of G-code your controller needs to be happy.

In terms of sheer numbers of users, the Fanuc dialects of G-Code are probably the most common among professionals and Mach3 among hobbyists.

This is not to say they are better than other G-Code dialects, just that they are more common and so if you're going to talk to other machinists or move around from job to job and machine to machine, it may be helpful if you're familiar with those dialects and how they differ if your machine doesn't use one of these two controllers.

G-Code has an extremely long history.

The first attempts at standardizing it came out of the Electronics Industry Association's RS-274 standard which has evolved to NIST's RS-274NGC standard.

The original EIA standards work was begun in the 1960's but the first standard wasn't released until 1980.

Even though there are now standards (ISO has one too), it isn't clear how many controllers out there are purely standards based.

How Are the Dialects Different?

G-Code dialects differ in a variety of ways.

Most manufacturers have added their own little bells and whistles to make their dialect better for competitive and marketing reasons.

For example, Haas has a series of special g-codes for pocket milling, as well as some special parameters and capabilities on some standard G-Codes.

It pays to understand the special capabilities of your machine because they were probably put there to save time based on feedback the manufacturer got from its customers.

In general, we see the following categories of differences between G-Code dialects:

-Which G-Codes are Supported. Not all controllers support all G-Codes. For example, many early lathe controls do not support the G71 and similar roughing cycles.

-G-Code mappings. Sometimes the same function will be supported by different g-code numbers on different controls.

-Parameters and Macro Programming. Parametric programming with macros is something that emerged after the basic standards were in place. Fanuc Macro B is probably the most common standard for it.

Many controls are very limited in their capabilities around Macro Programming and there are a lot of detail differences around exactly how Macros work.

-Parameters. Many G-Codes need additional information to do their job, so they use other words (letters) to collect that information. Exactly which words collect which information can vary from one control to the next.

-Formatting. Some controls allow G0 or G00. Some insist on G00.

Some allow numbers with no decimal, others insist on a decimal or even a trailing zero. "1", "1.", and "1.0" are all variations that may be accepted, rejected, or required when specifying the number 1.

We'll talk more shortly about what all of this means, but for now, be aware that these differences exist.

For simple programs and MDI use, obviously a lot of this won't matter. But, for writing complex hand-written G-Code or trying to understand why the G-Code your CAM program emits isn't quite right, you'll need to be aware of the dialect issues.

Standard Part programming language: RS 274-D (Gerber, GN-code)

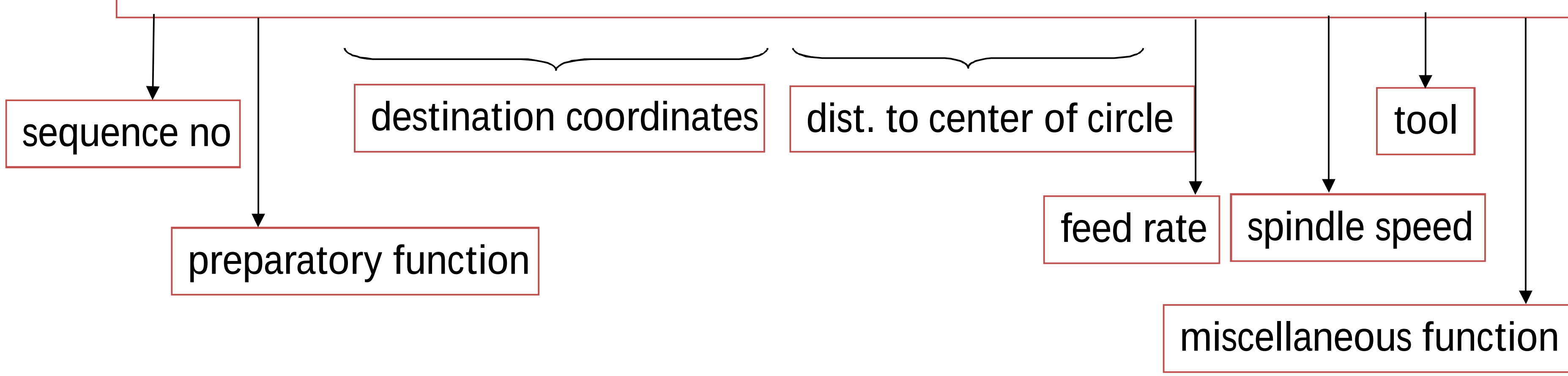
The RS274-D is a **word address format**

Each line of program == 1 **block**

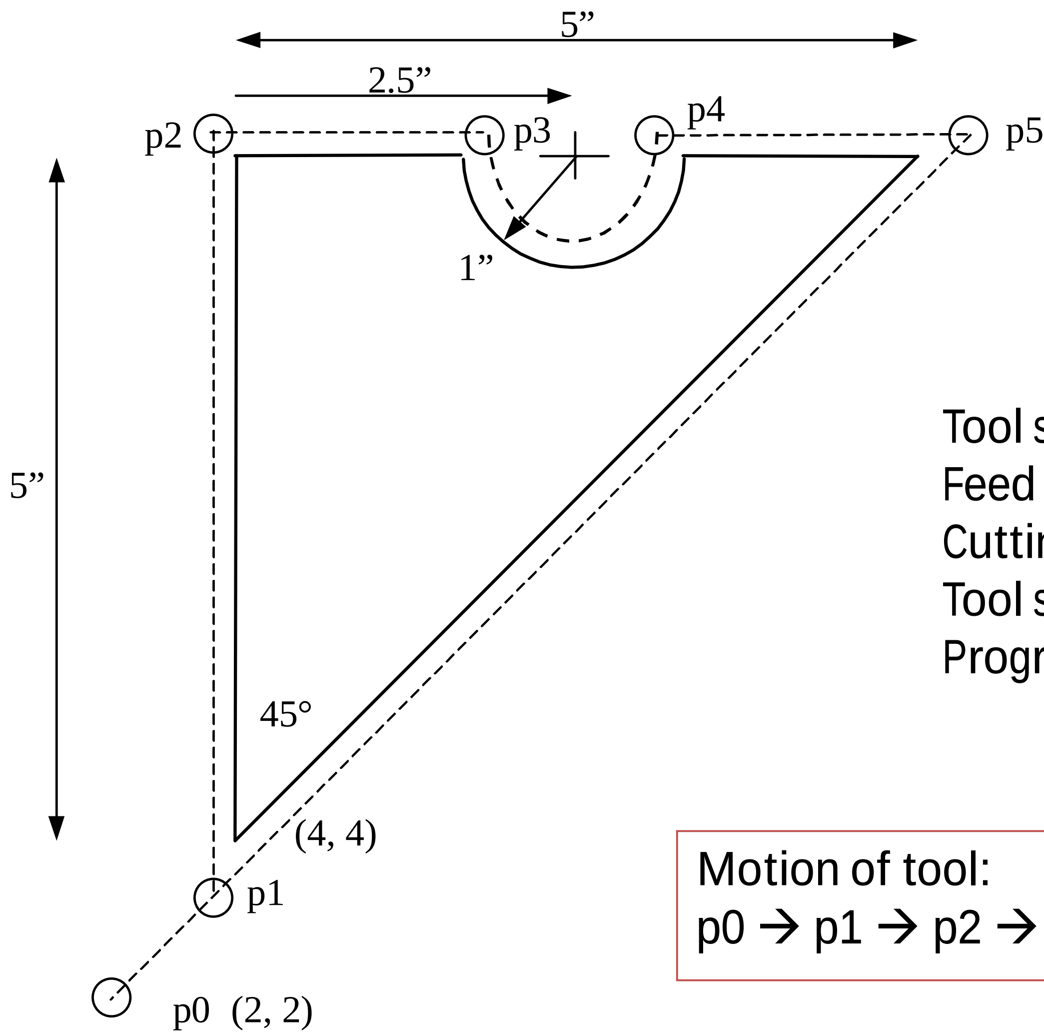
Each block is composed of several instructions, or (words)

Sequence and format of words:

N3 G2 X+1.4 Y+1.4 Z+1.4 I1.4 J1.4 K1.4 F3.2 S4 T4 M2



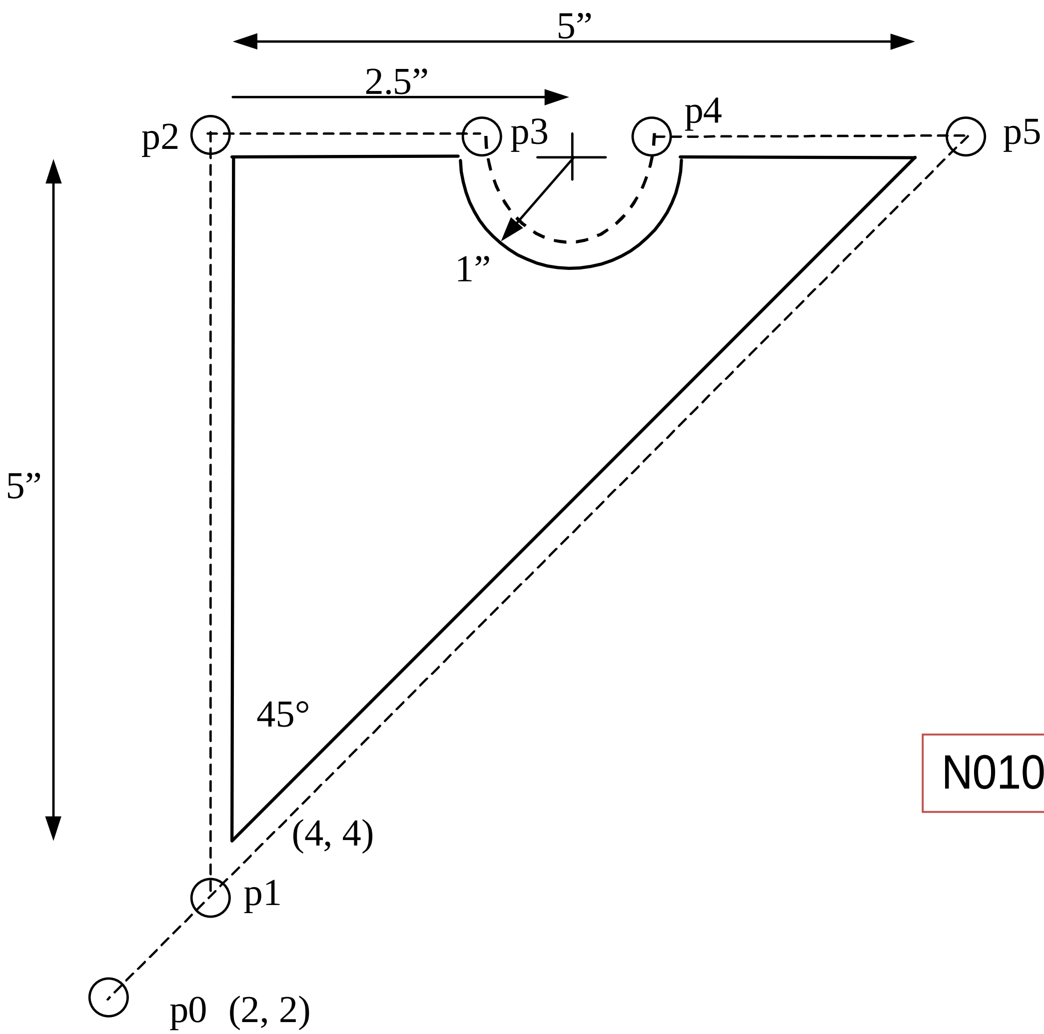
Manual Part Programming Example



Tool size = 0.25 inch,
Feed rate = 6 inch per minute,
Cutting speed = 300 rpm,
Tool start position: 2.0, 2.0
Programming in inches

Motion of tool:
p0 → p1 → p2 → p3 → p4 → p5 → p1 → p0

1. Set up the programming parameters



Programming in inches

Use absolute coordinates

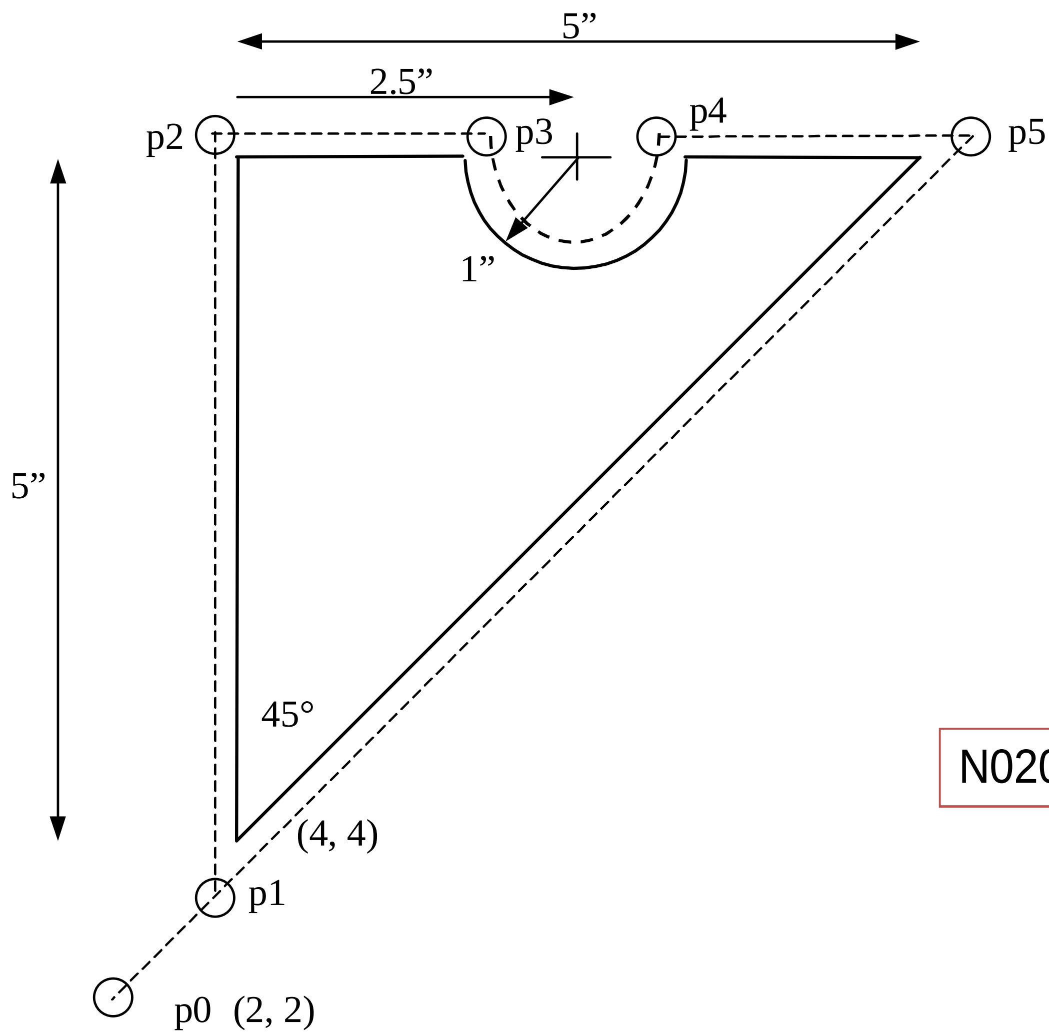
Feed in ipm

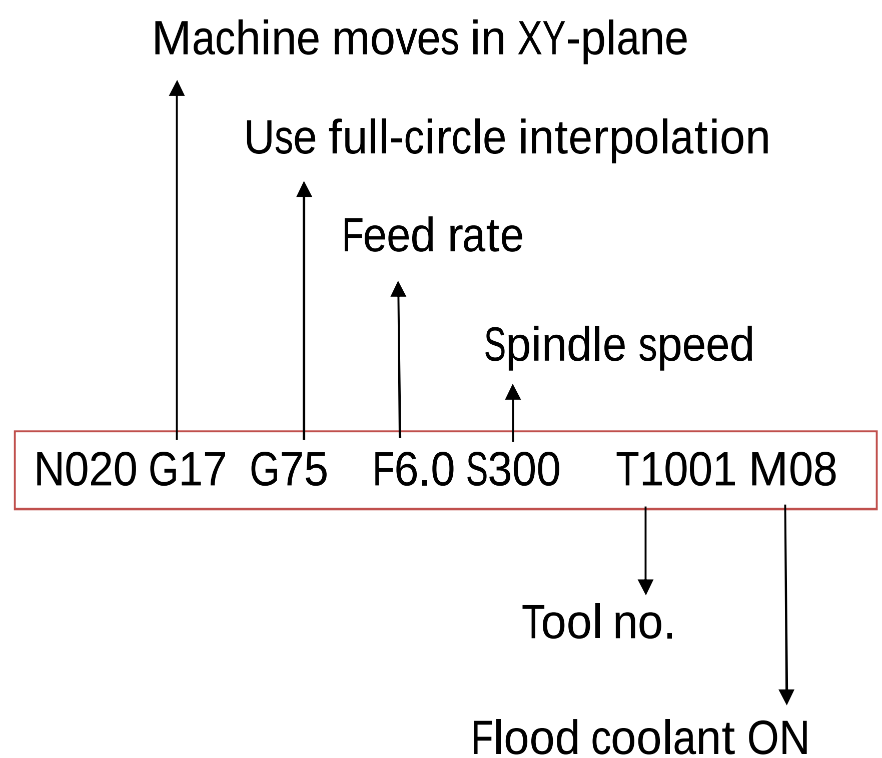
N010 G70 G90 G94 G97 M04

Spindle speed in rpm

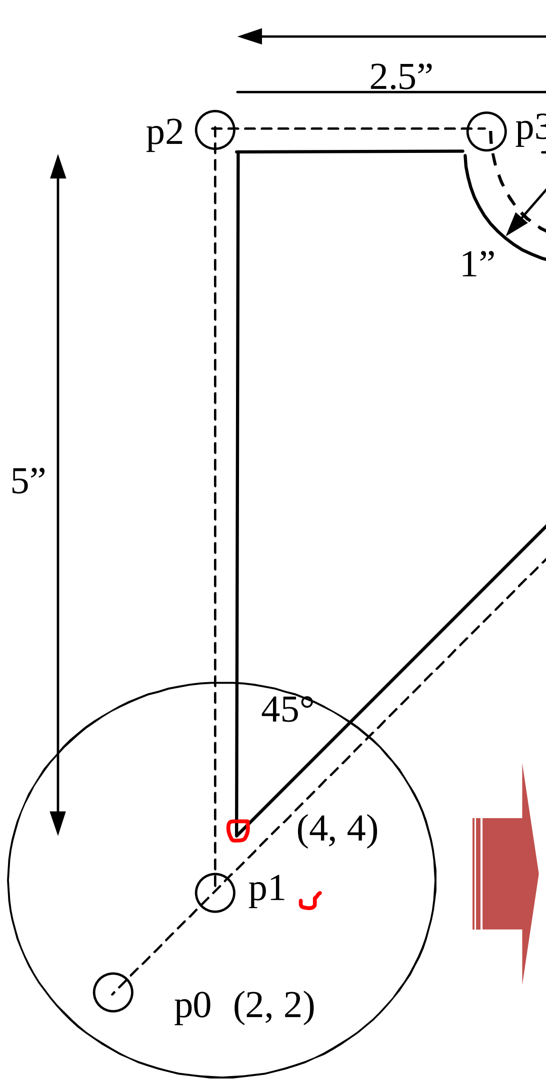
Spindle CCW

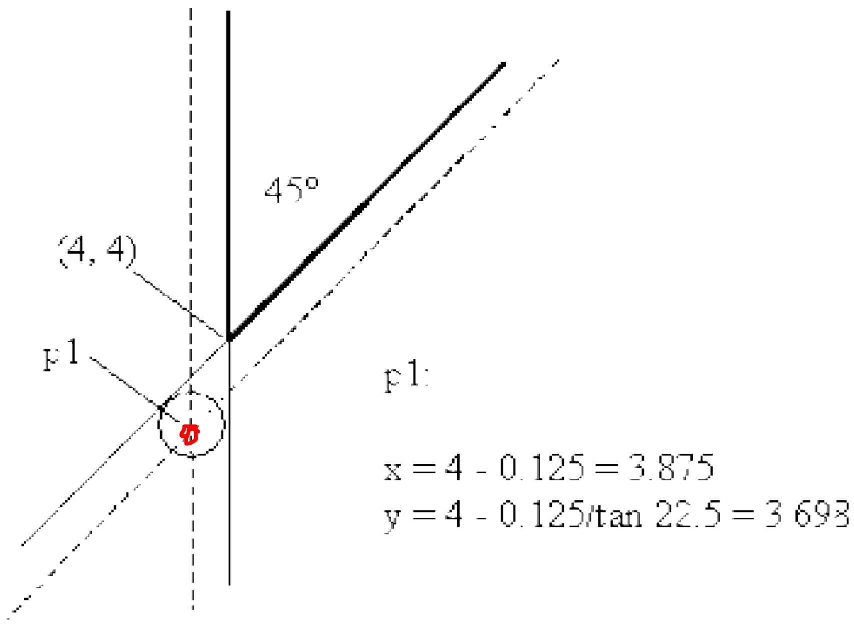
2. Set up the machining conditions





3. Move tool from p0 to p1 in straight line



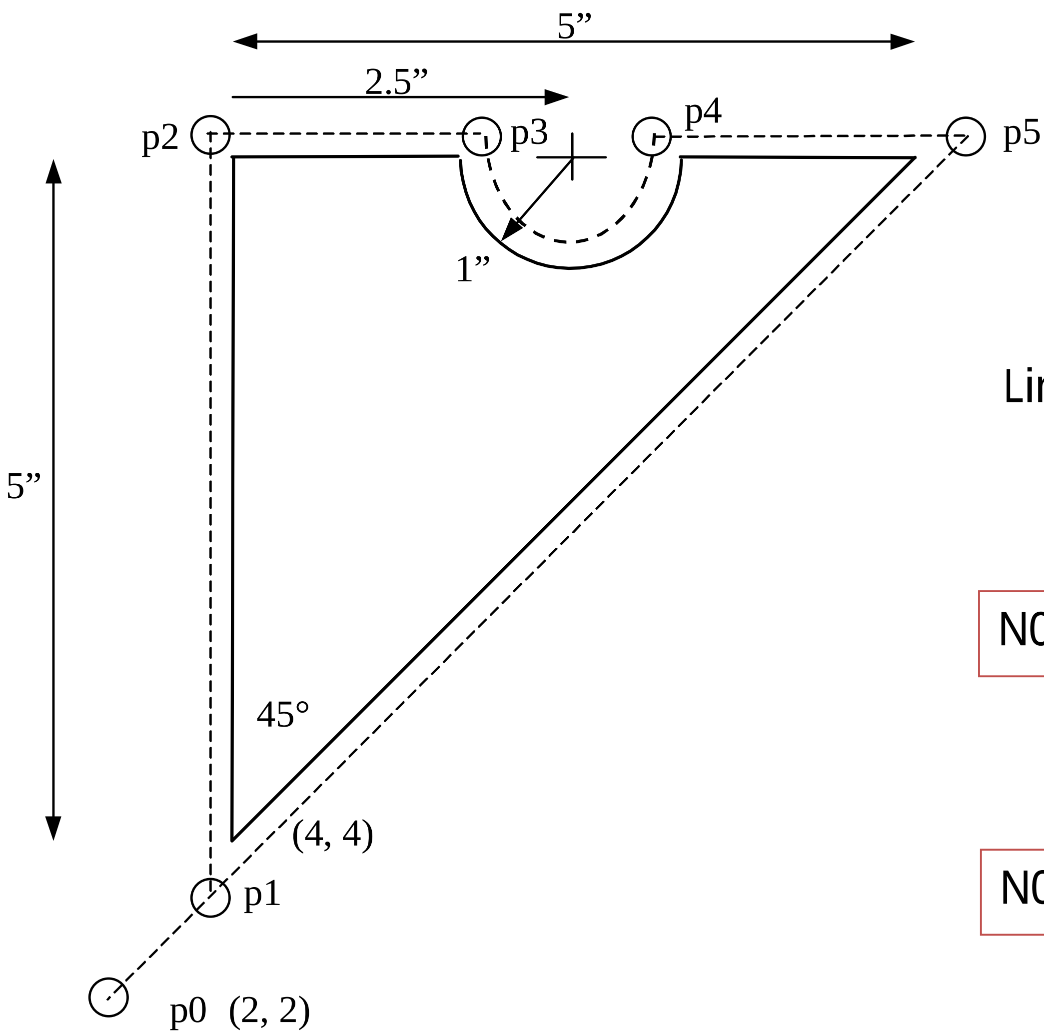


Linear interpolation

target coordinates

N030 G01 X3.875 Y3.698

4. Cut profile from p1 to p2



Linear interpolation

target coordinates

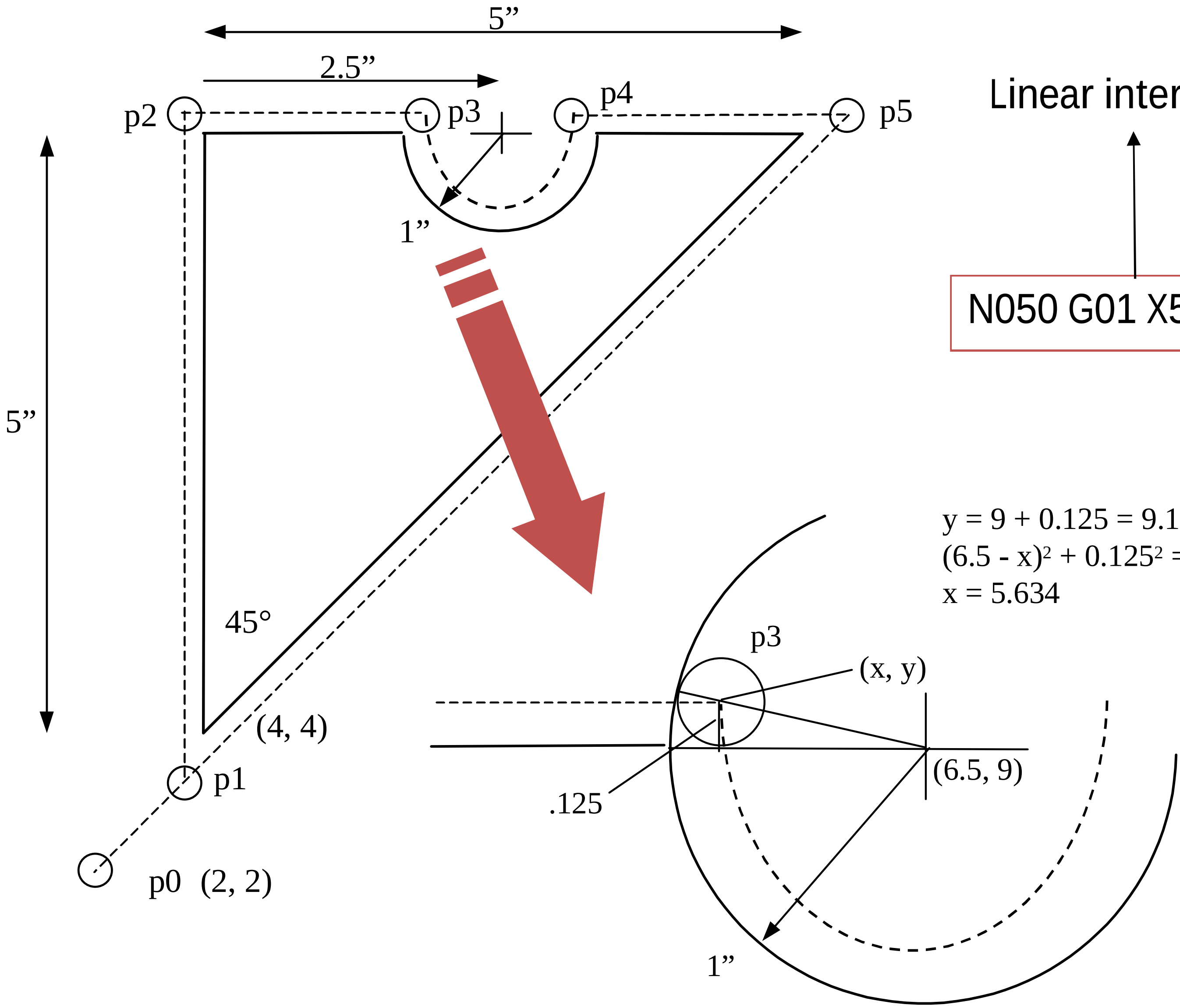
N040 G01 X3.875 Y9.125

or

N040 G01 Y9.125

X-coordinate does not change ➔ no need to program it

5. Cut profile from p2 to p3



$y = 9 + 0.125 = 9.125$

$(6.5 - x)^2 + 0.125^2 = (1 - 0.125)^2$

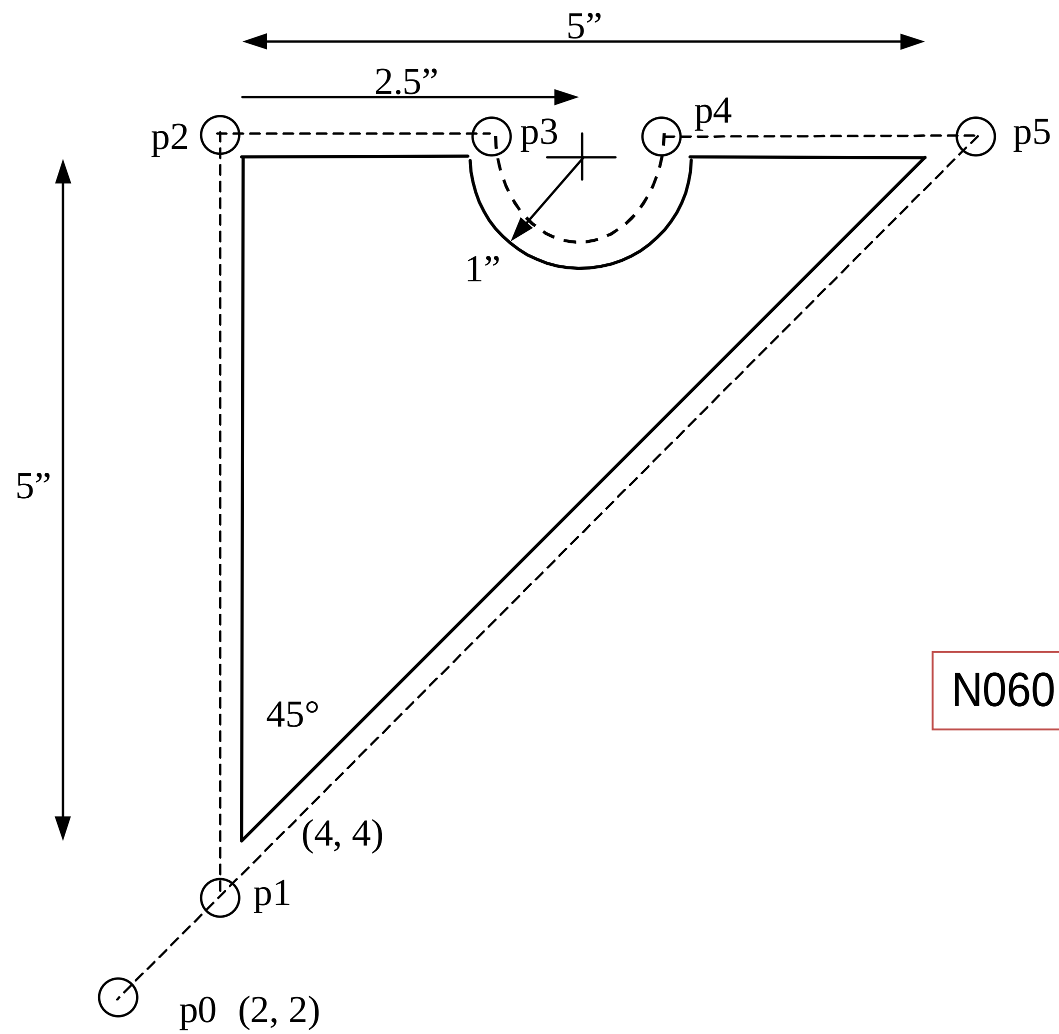
$x = 5.634$

Linear interpolation

target coordinates

N050 G01 X5.634 Y9.125

6. Cut along circle from p3 to p4



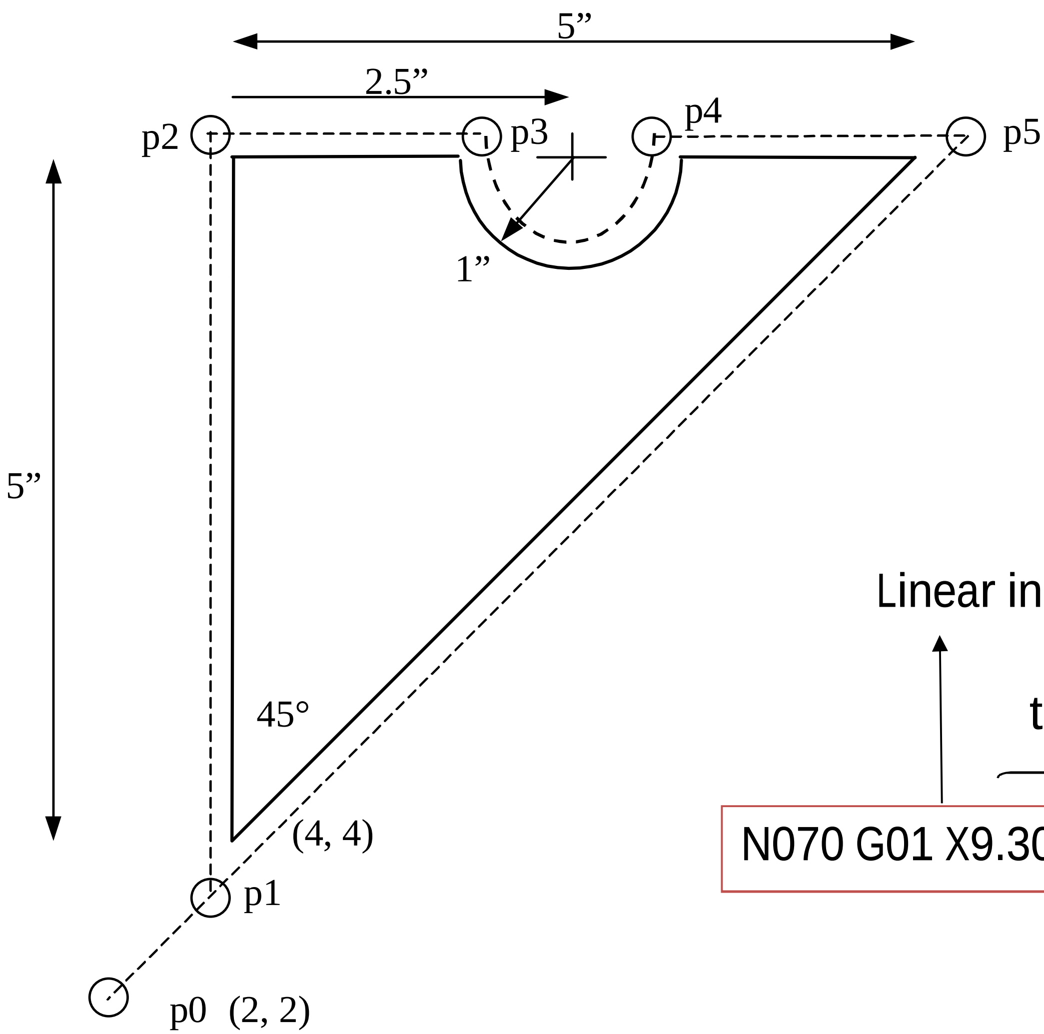
circular interpolation, CCW motion

target coordinates

N060 G03 X7.366 Y9.125 I6.5 J9.0

coordinates of center of circle

7. Cut from p4 to p5

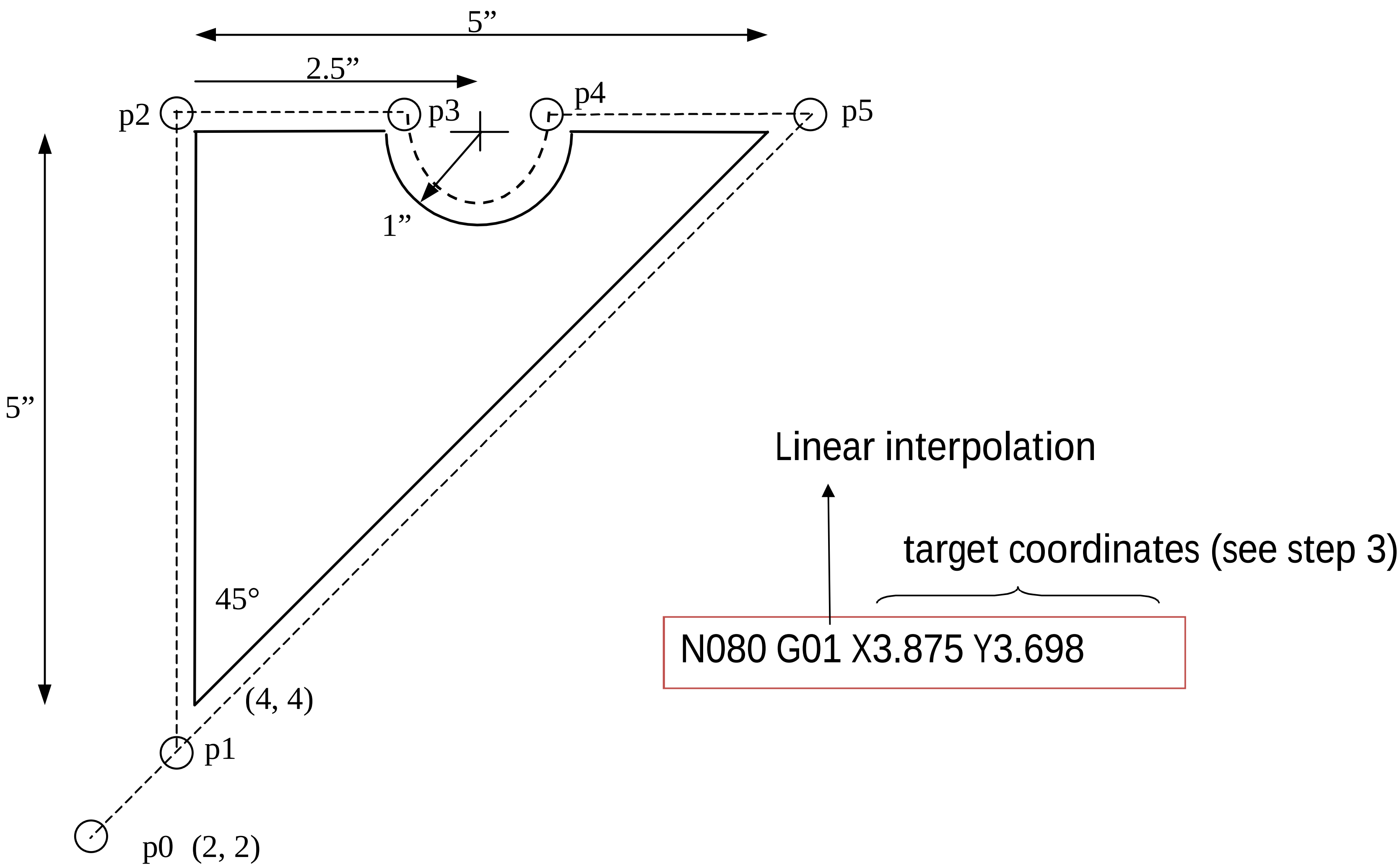


Linear interpolation

target coordinates (Y is unchanged)

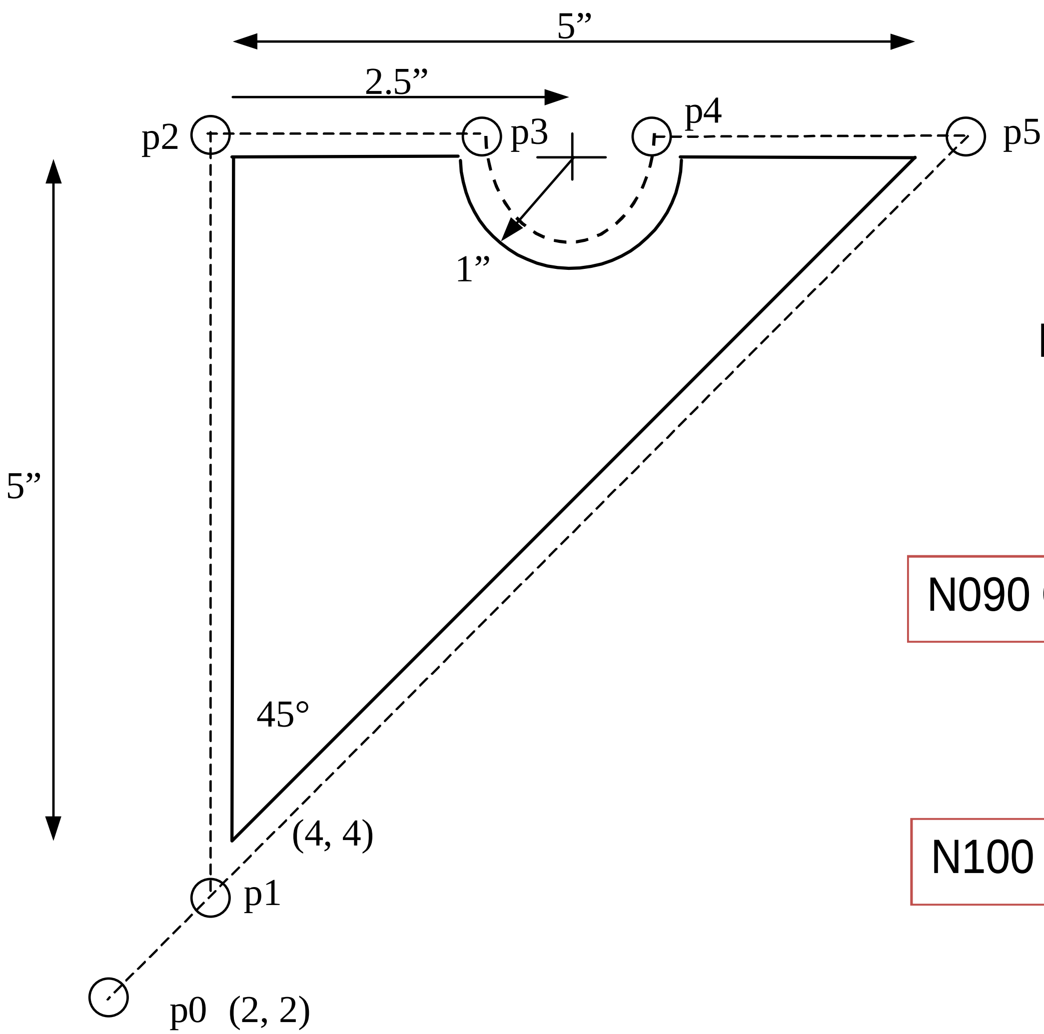
```
N070 G01 X9.302
```

8. Cut from p5 to p1



N080 G01 X3.875 Y3.698

9. Return to home position, stop program



Linear interpolation

target coordinates (see step 3)

N090 G01 X2.0 Y2.0 M30

end of data

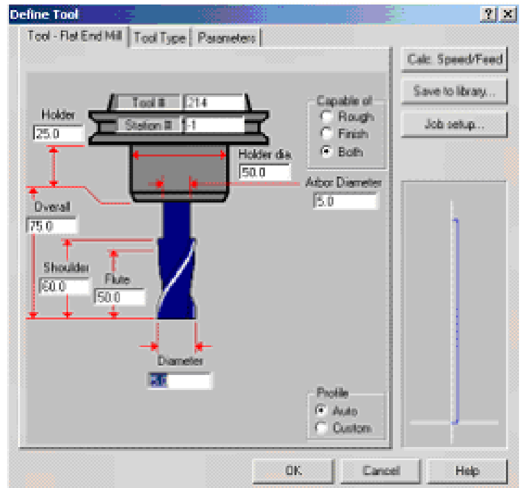
N100 M00

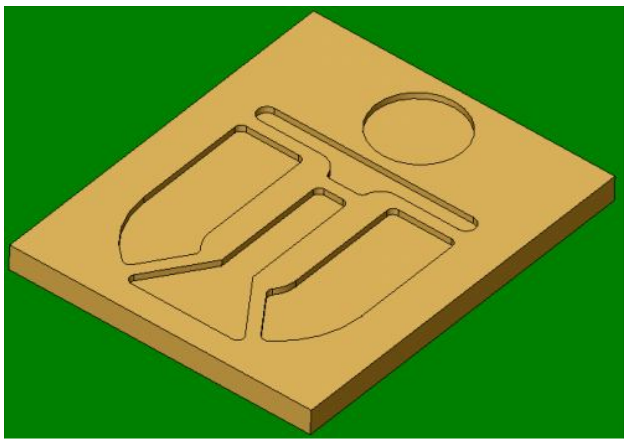
program stop

Automatic Part Programming

Software programs can automatic generation of CNC data

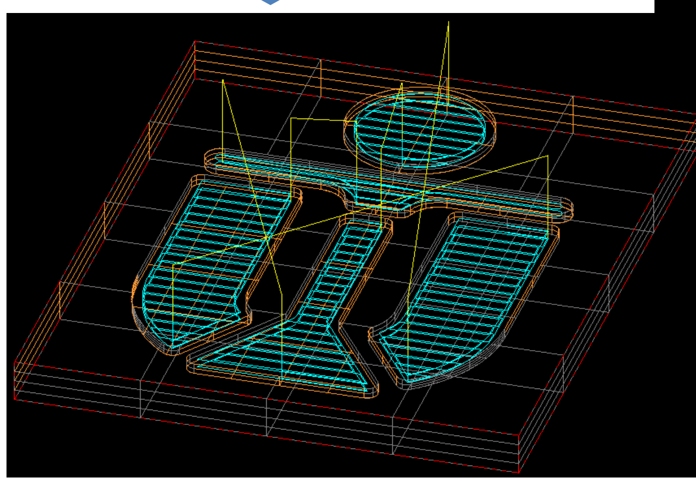
Define Tool

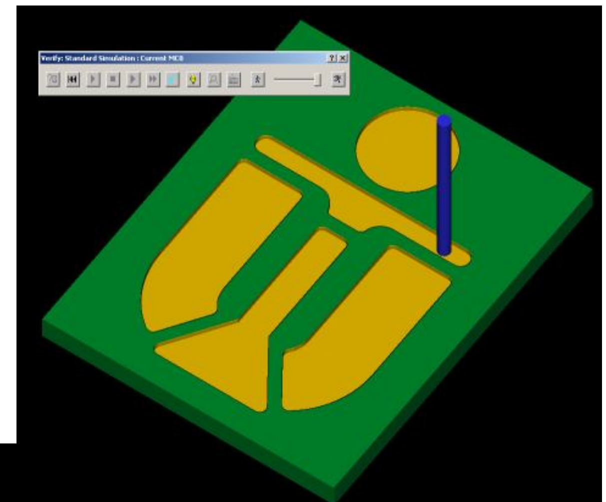




Make 3D model

CNC data



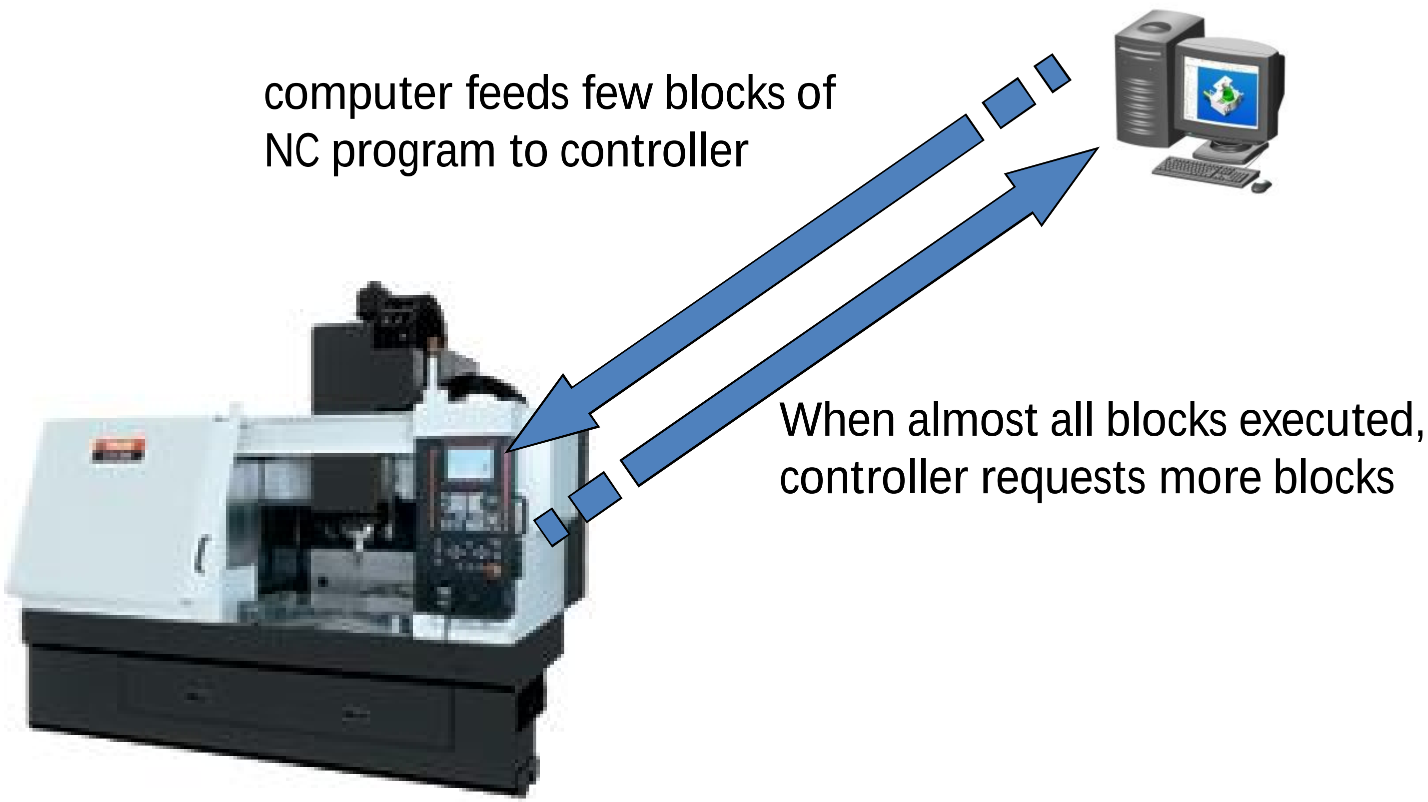


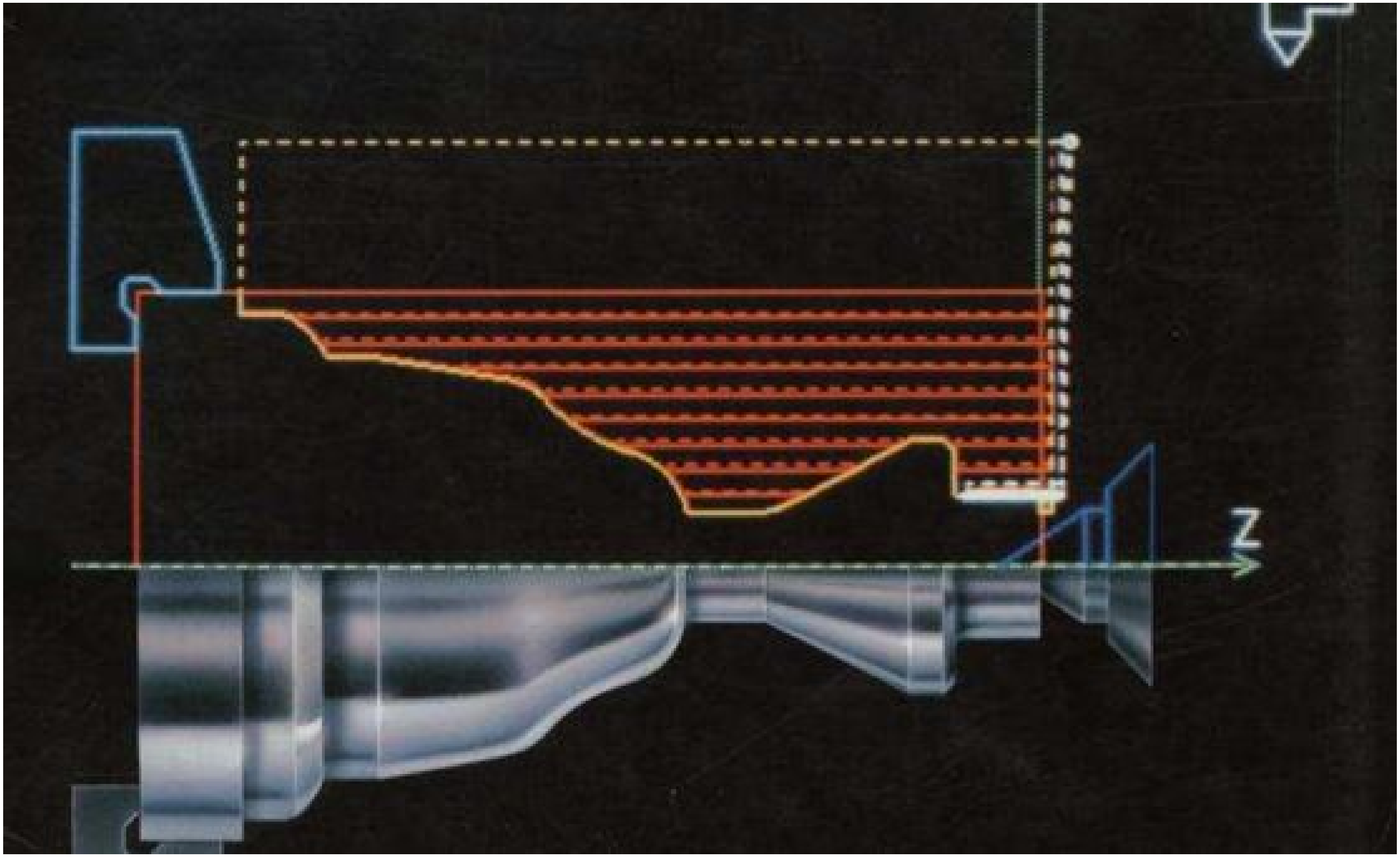
Simulate cutting

Automatic part programming and DNC

Very complex part shapes ➔ very large NC program

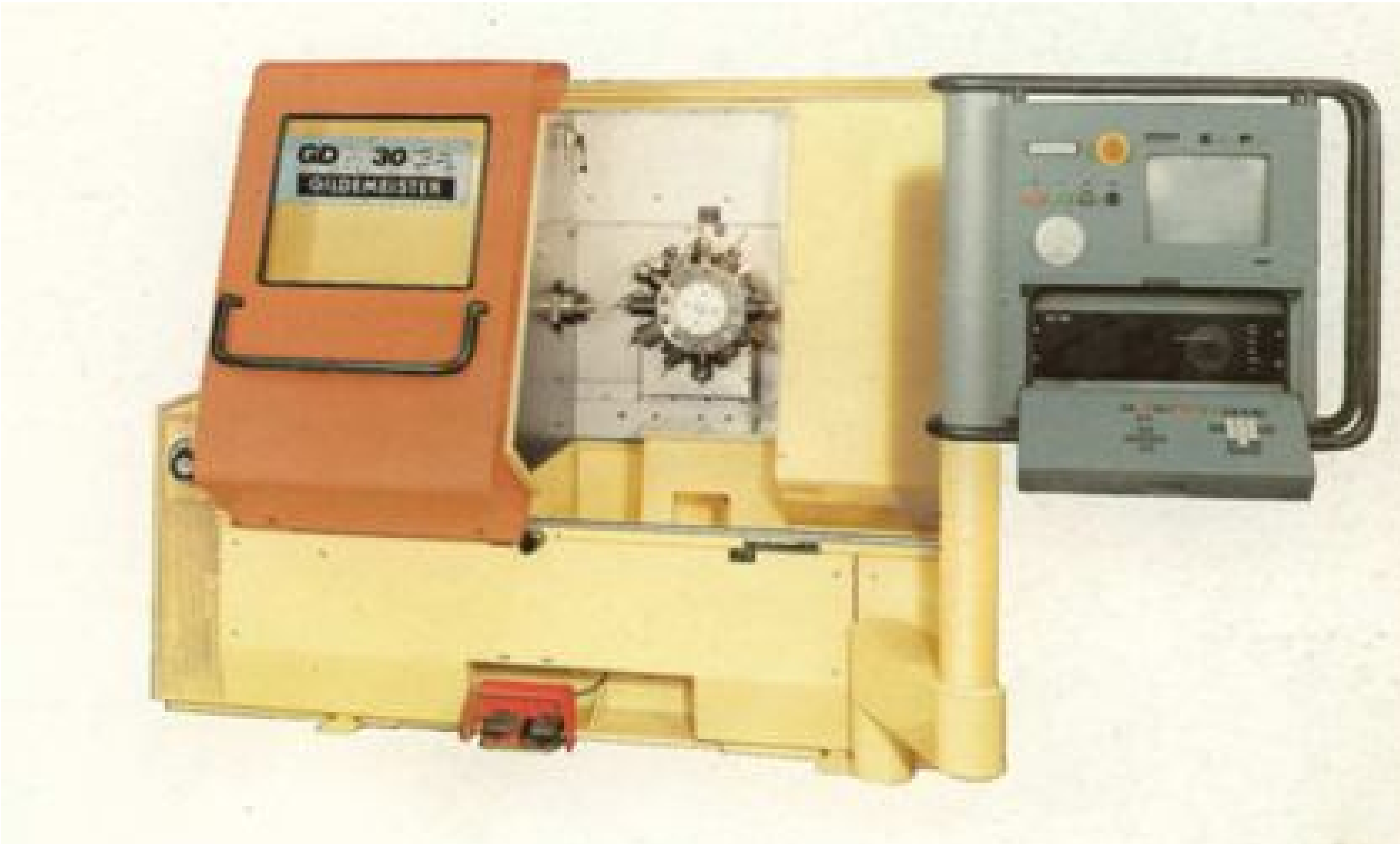
NC controller memory may not handle HUGE part program




Z

AUTOMATION

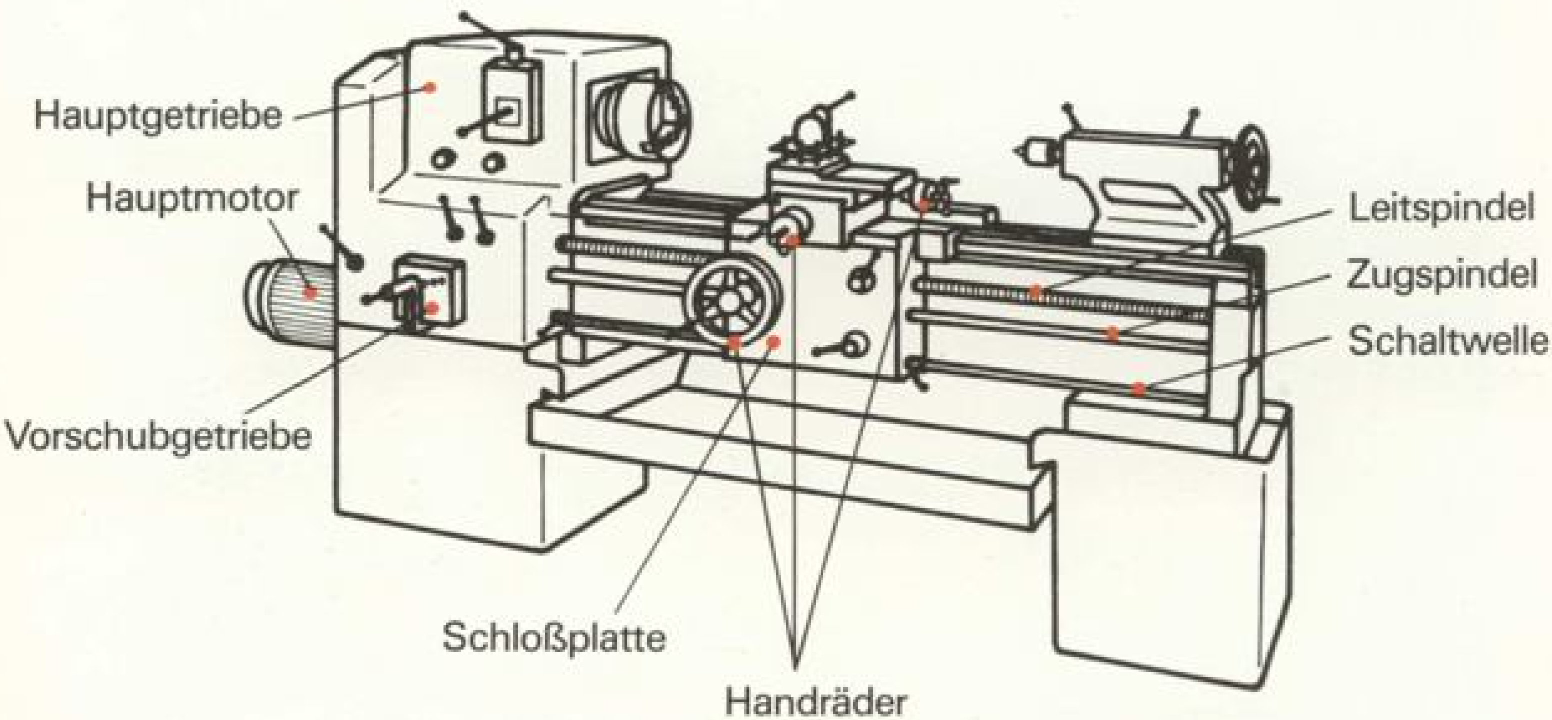
Gildemeister Drehmaschinen



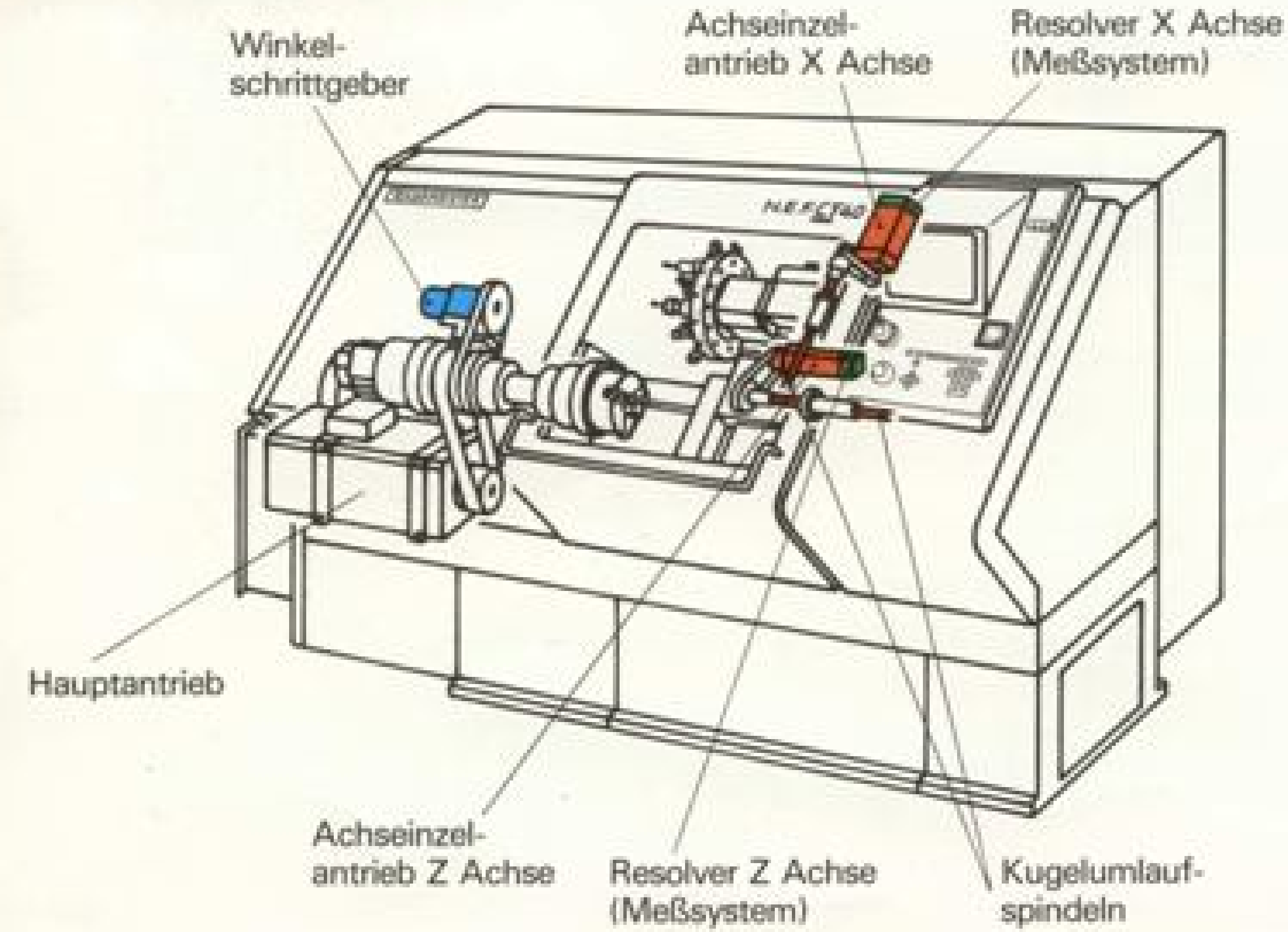

MD5iT

Gildemeister-Ausbildungssystem Datapilot

Herkömmliche Drehmaschine



CNC-Drehmaschine



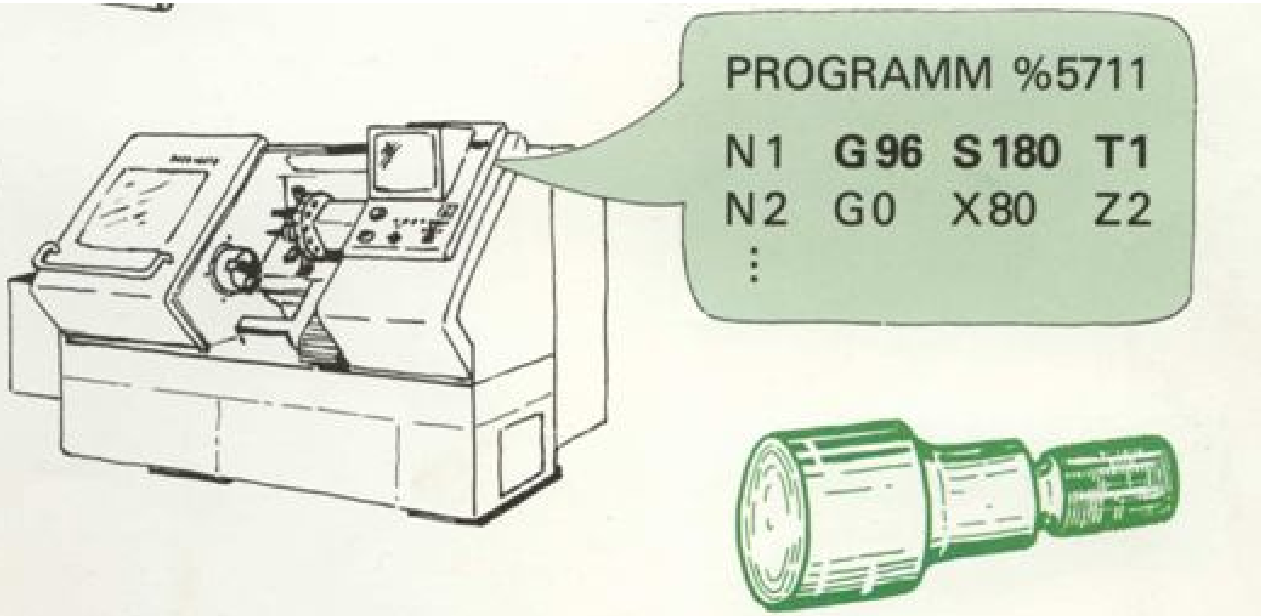
Von der Zeichnung zum Werkstück

1. Zeichnung lesen

2. Programmieren

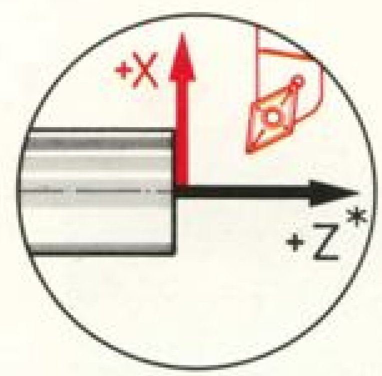
3. Programm eingeben

4. Fertigen



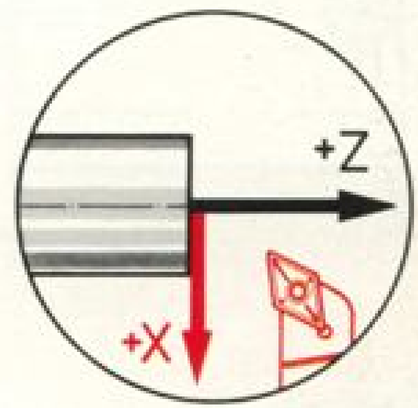
Achsen und Achsrichtungen

1. Werkzeug ist **hinter** der Drehmitte



DIN 66 217

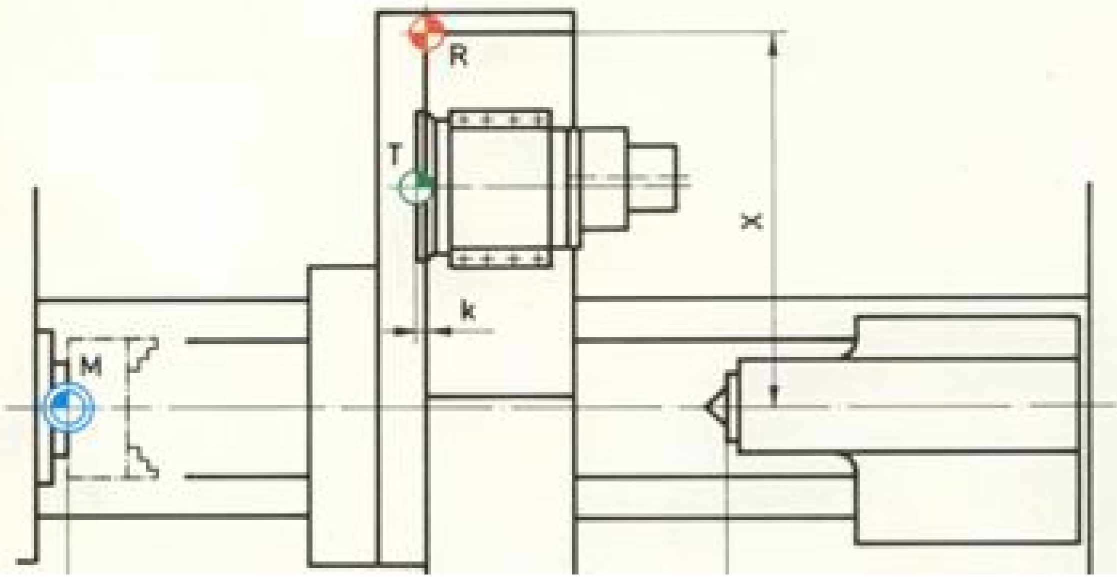
2. Werkzeug ist **vor** der Drehmitte

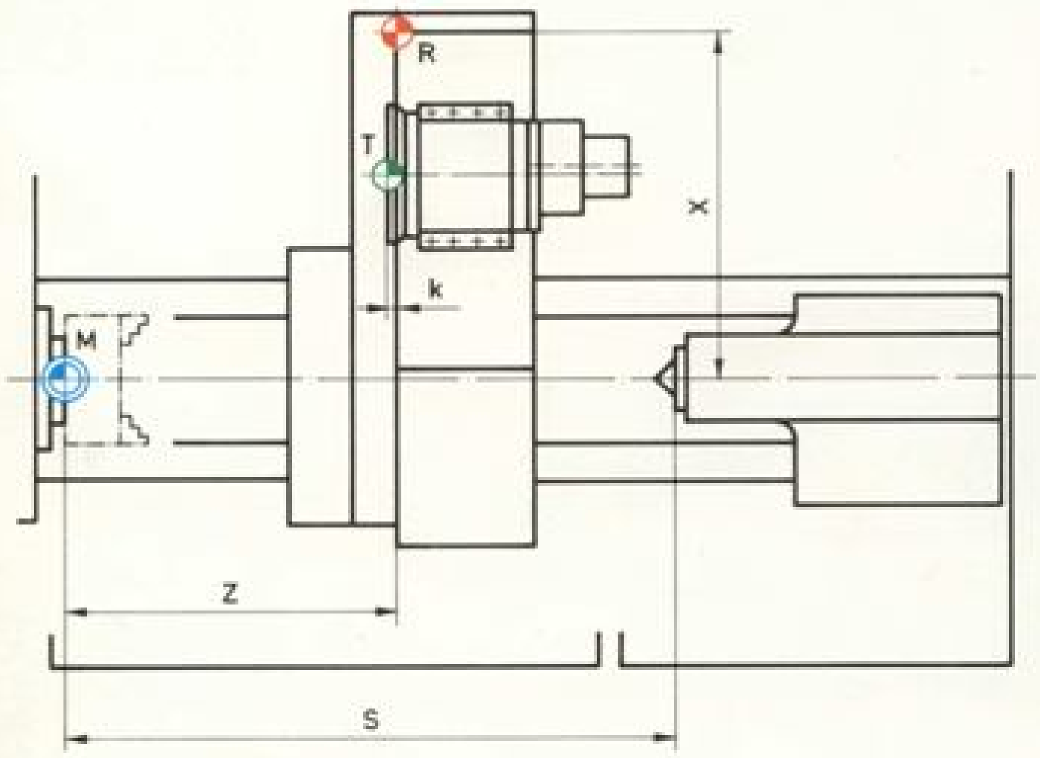


Bezugspunkte im Maschinenraum

Die wichtigsten Bezugspunkte im Maschinenraum sind:

- **M** Maschinennullpunkt
- **R** Referenzpunkt
- **T** Werkzeugträgerbezugspunkt



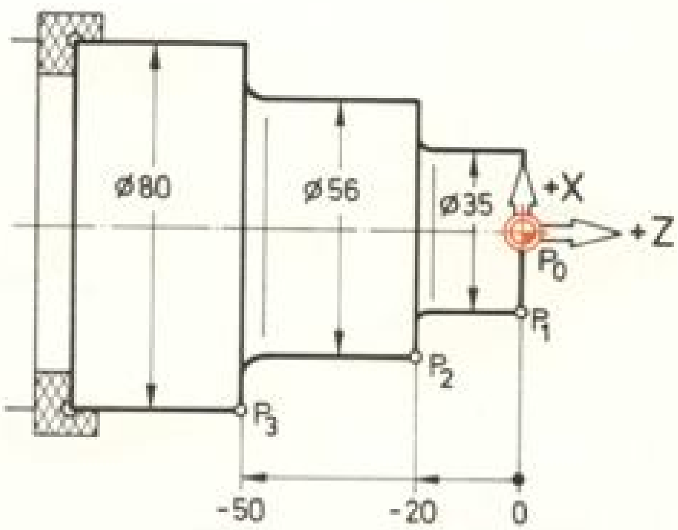


	X			Z			S	K	
	min	max	Hub	min	max	Hub	max		
CT40	275	535	260	183,5	823,5	640	900	10,5	[mm]

Werkstücknullpunkt

Der Werkstücknullpunkt (Symbol ⊕) ist der Bezugspunkt für die geometrischen Daten des Programmes.

1. Werkstücknullpunkt an der **Planfläche**

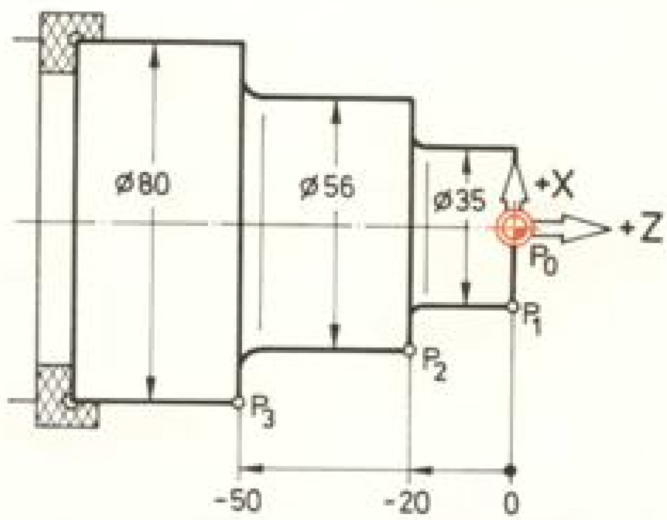


	X	Z
P_0	0	0
P_1	35	0
P_2	56	−20
P_3	80	−50

Werkstücknullpunkt

Der Werkstücknullpunkt (Symbol) ist der Bezugspunkt für die geometrischen Daten des Programmes.

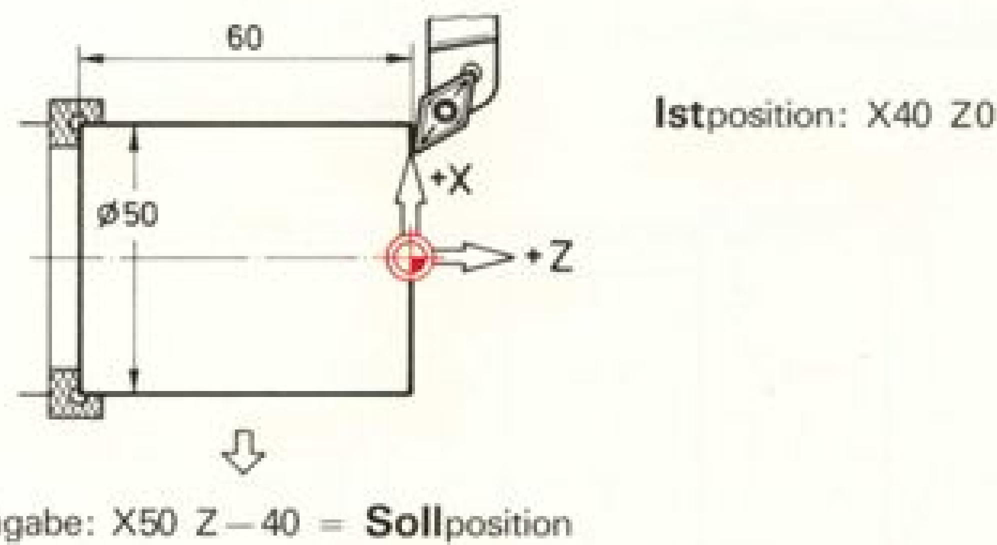
1. Werkstücknullpunkt an der **Planfläche**



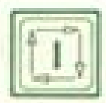
	X	Z
P_0	0	0
P_1	35	0
P_2	56	−20
P_3	80	−50

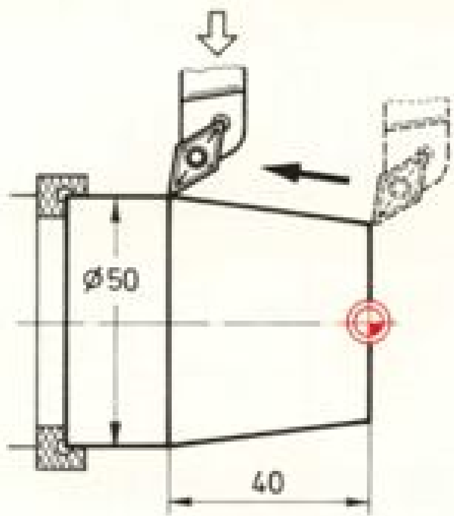
Bezugsmaßangabe (Absolut G90)

Absolute Maßangaben beziehen sich immer auf den **Werkstücknullpunkt** (deshalb **Bezugs**maße):



START – Taste drücken



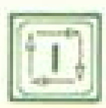


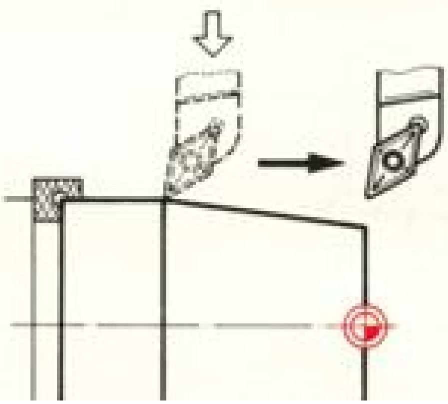
G90* bedeutet:
Das Werkzeug fährt
AUF die angegebene Position.

Istposition: X50 Z – 40

Eingabe: Z2 = **Soll**position

START – Taste drücken



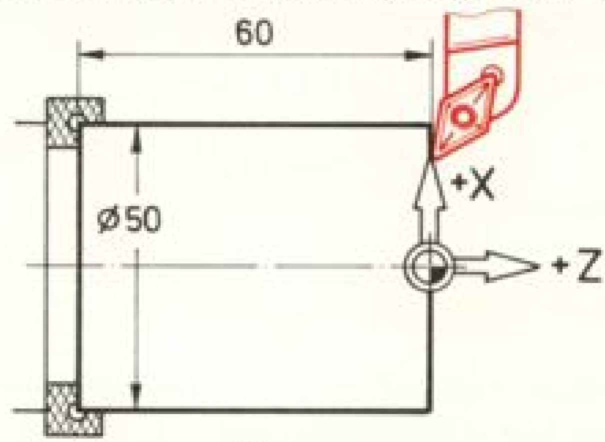


Istposition: X50 Z2

Kettenmaßangabe (Inkremental G91)

Bei inkrementalen Maßangaben geht man **NICHT** vom Werkstücknullpunkt aus.

Die programmierten X und Z Werte beziehen sich auf die **LETZTE Werkzeugposition**.



Istposition:* X40 Z0

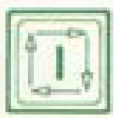
Achtung:
Radius-Wert!

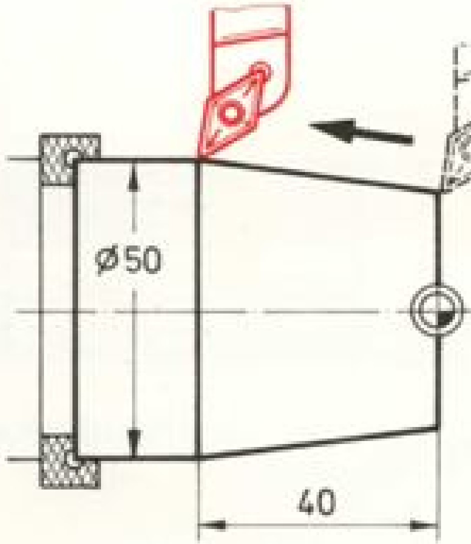
Eingabe: X5 Z−40

Achtung:
Radius-Wert!

Eingabe: X5 Z − 40

START – Taste drücken



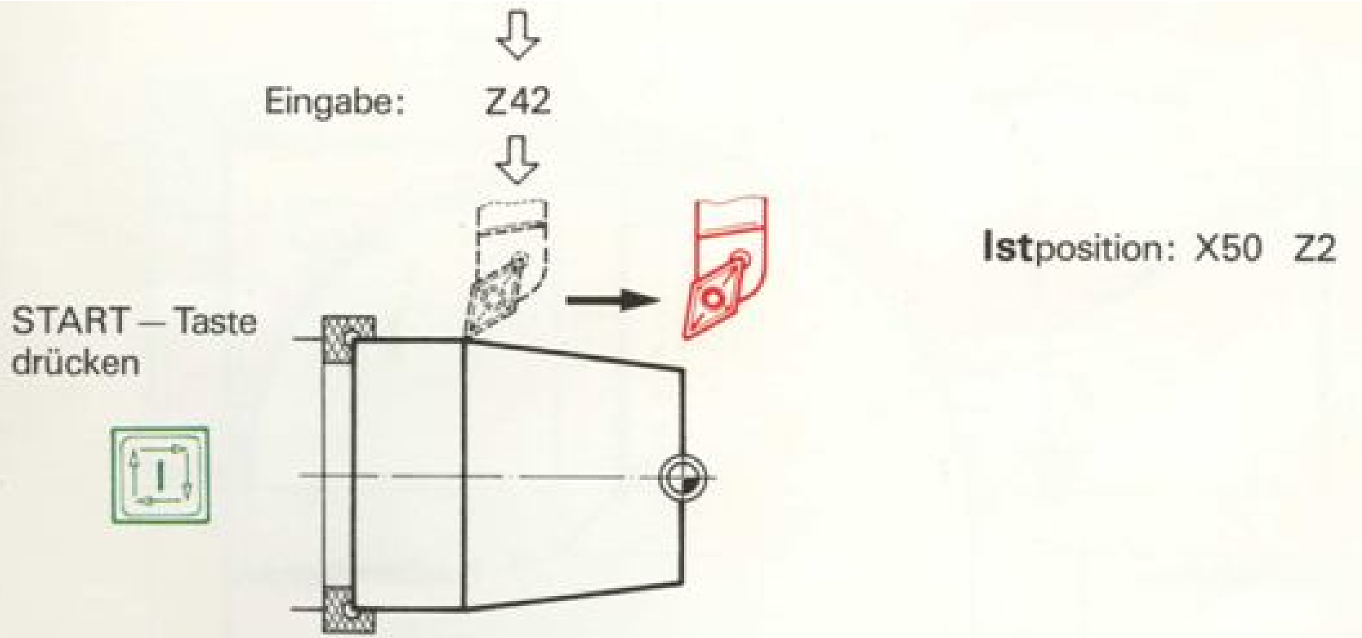


G91* bedeutet:
Das Werkzeug fährt
UM... mm in die angegebene Richtung

hier: X5 mm, + Richtung
Z40 mm, − Richtung

Istposition: X50 Z − 40

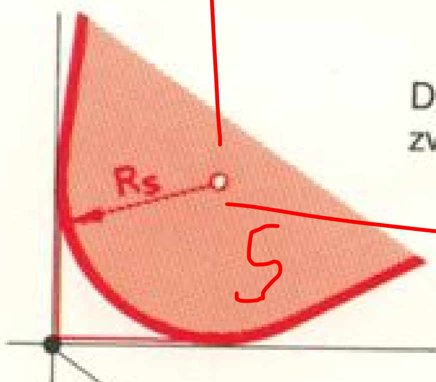
Eingabe: Z42



* Selbst wenn Kettenmaße programmiert wurden, erscheinen auf dem Bildschirm nach dem Verfahrweg die **Istpositionen** (Absolutmaße).

Auswirkungen des Schneidenradius

Damit die Standzeit des Drehmeißels nicht zu kurz ist, wird die Werkzeug-„Spitze" bei Schrupp- und bei Schlichtmeißeln abgerundet:

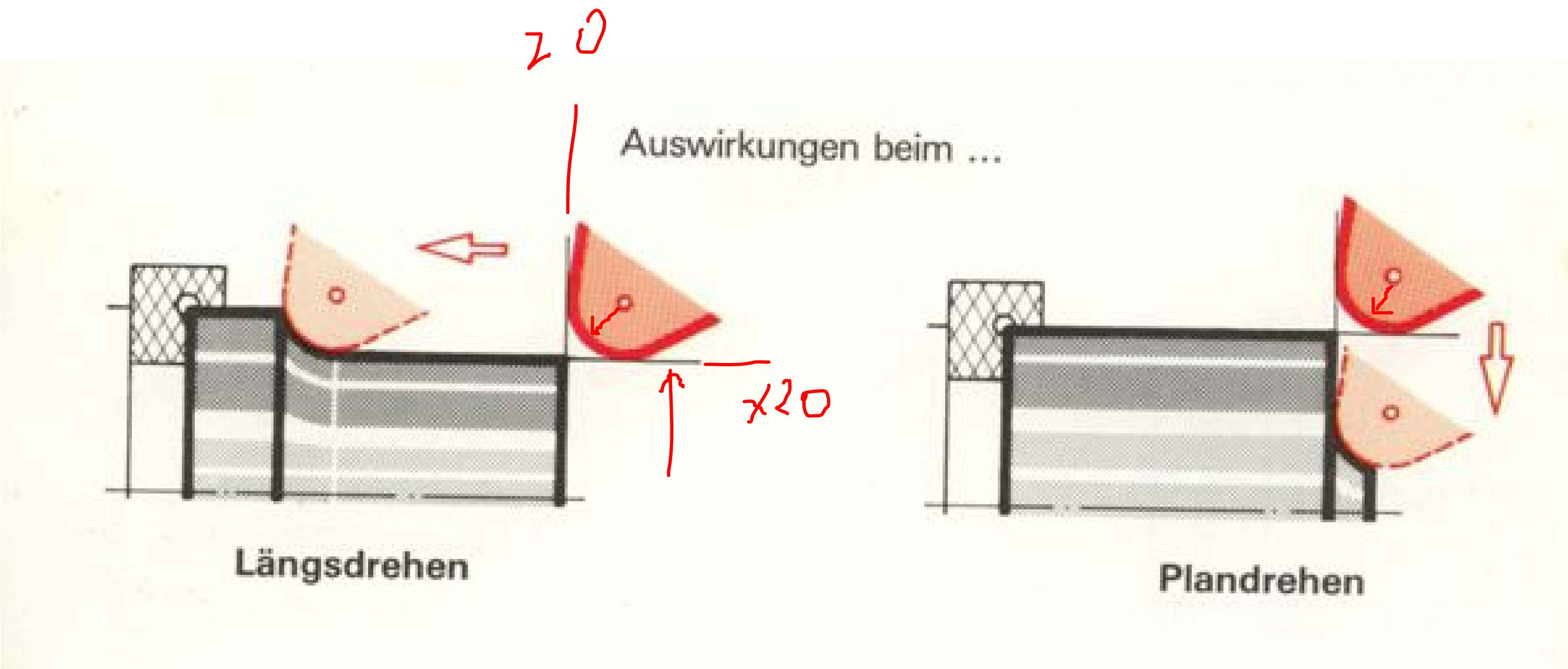


Der Schneidenradius R_S schwankt zwischen 0,4 und 1,6 mm.

theoretische Meißel-„Spitze"
= **Bezugspunkt für die Steuerung**

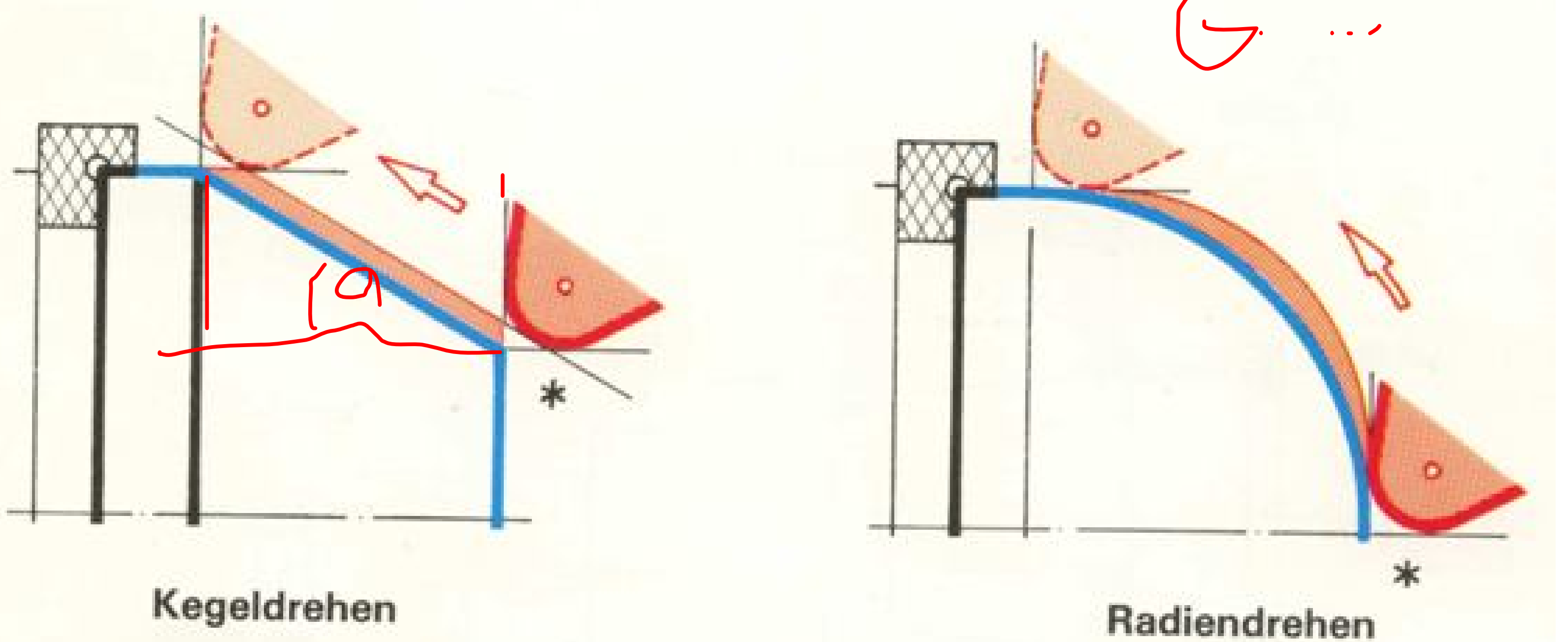
Auswirkungen beim ...

G1



G1

G41
G42
G...

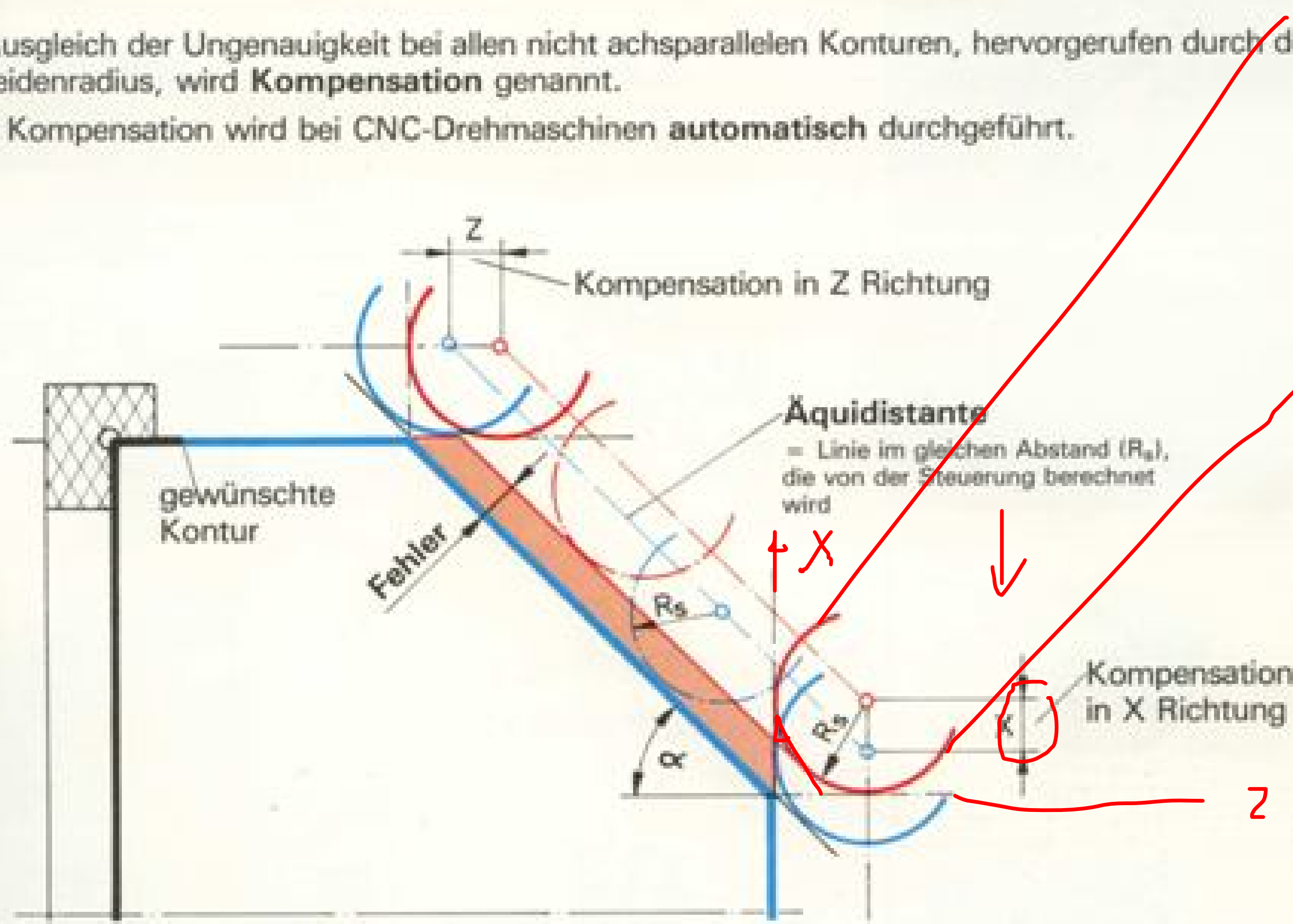


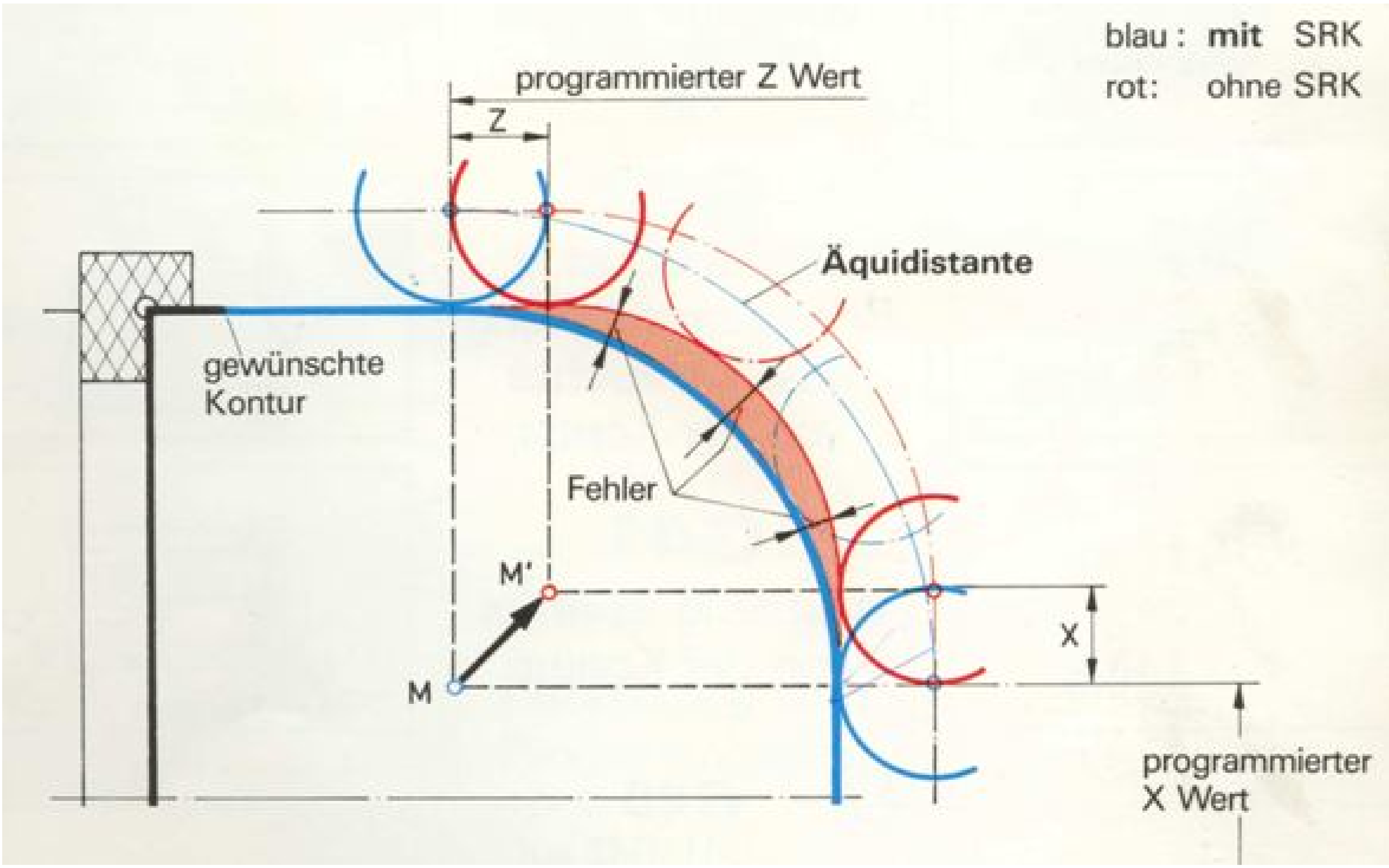
* Je größer der Schneidenradius, desto größer das „Aufmaß“ (die Ungenauigkeit).

Die Kompensation des Schneidenradius (SRK)

Der Ausgleich der Ungenauigkeit bei allen nicht achsparallelen Konturen, hervorgerufen durch den Schneidenradius, wird **Kompensation** genannt.

Diese Kompensation wird bei CNC-Drehmaschinen **automatisch** durchgeführt.




blau : mit SRK
rot: ohne SRK
programmierter Z Wert
Z
Äquidistante
gewünschte
Kontur
Fehler
M'
M
X
programmierter
X Wert

1.8 TOOL FIGURE AND TOOL MOTION BY PROGRAM

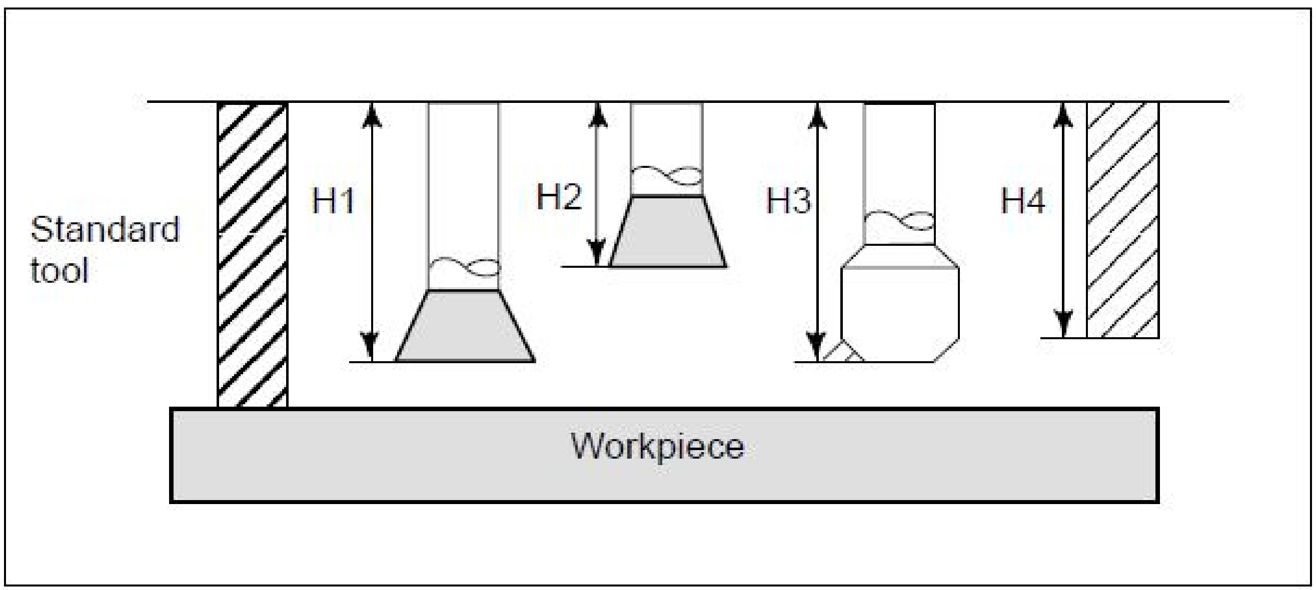
Machining using the end of cutter – Tool length compensation

Usually, several tools are used for machining one workpiece. The tools have different tool length. It is very troublesome to change the program in accordance with the tools.

Therefore, the length of each tool used should be measured in advance.

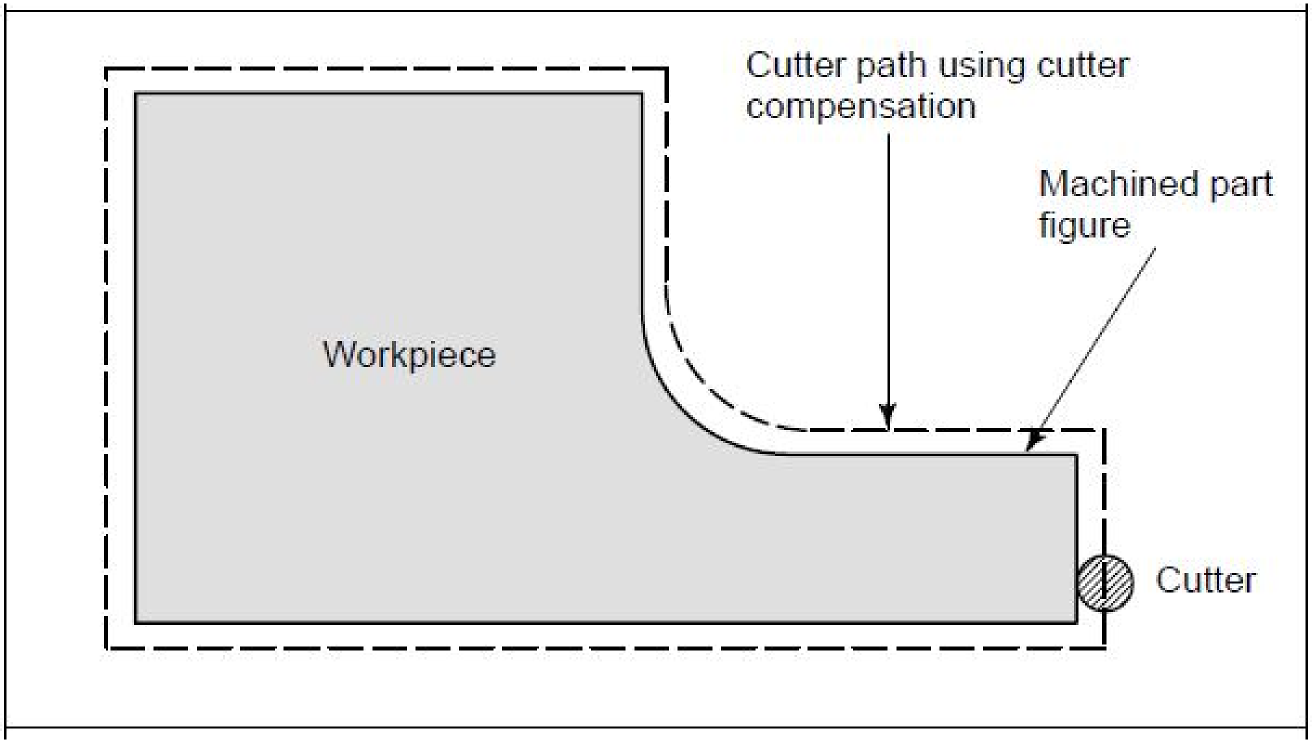
By setting the difference between the length of the standard tool and the length of each tool in the CNC (data display and setting : see III–11), machining can be performed without altering the program even when the tool is changed.

This function is called tool length compensation.



•Machining using the side of cutter – Cutter compensation function (See II–14.4,14.5,14.6)

Because a cutter has a radius, the center of the cutter path goes around the workpiece with the cutter radius deviated.



If radius of cutters are stored in the CNC (Data Display and Setting : see III–11), the tool can be moved by cutter radius apart from the machining part figure.
This function is called cutter compensation.

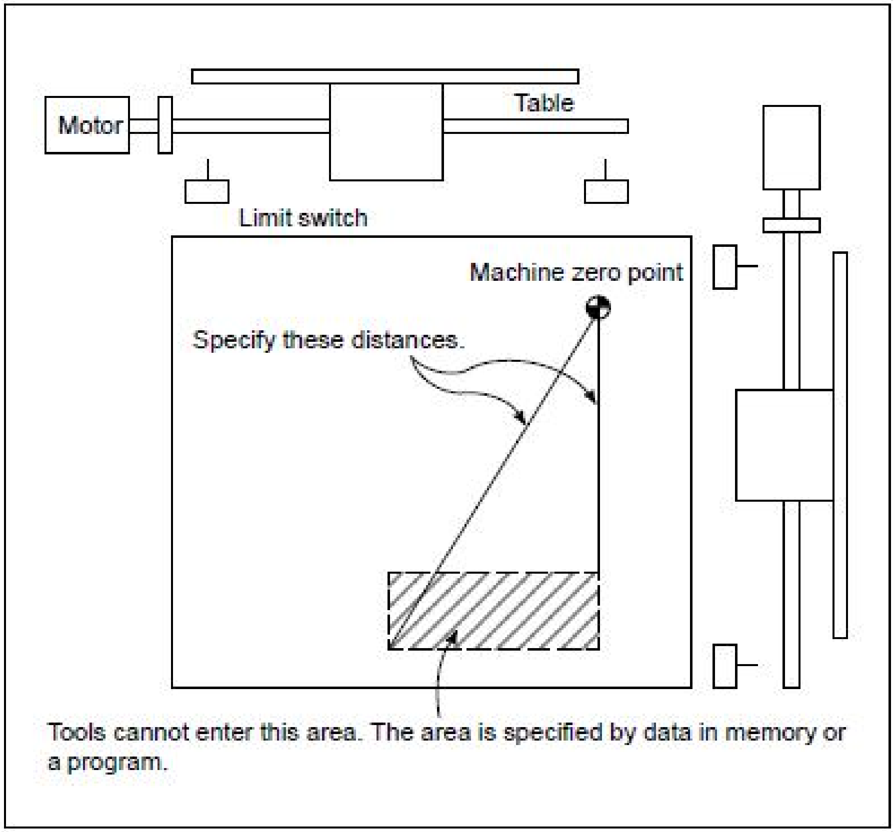
1.9 TOOL MOVEMENT RANGE – STROKE

Limit switches are installed at the ends of each axis on the machine to prevent tools from moving beyond the ends.

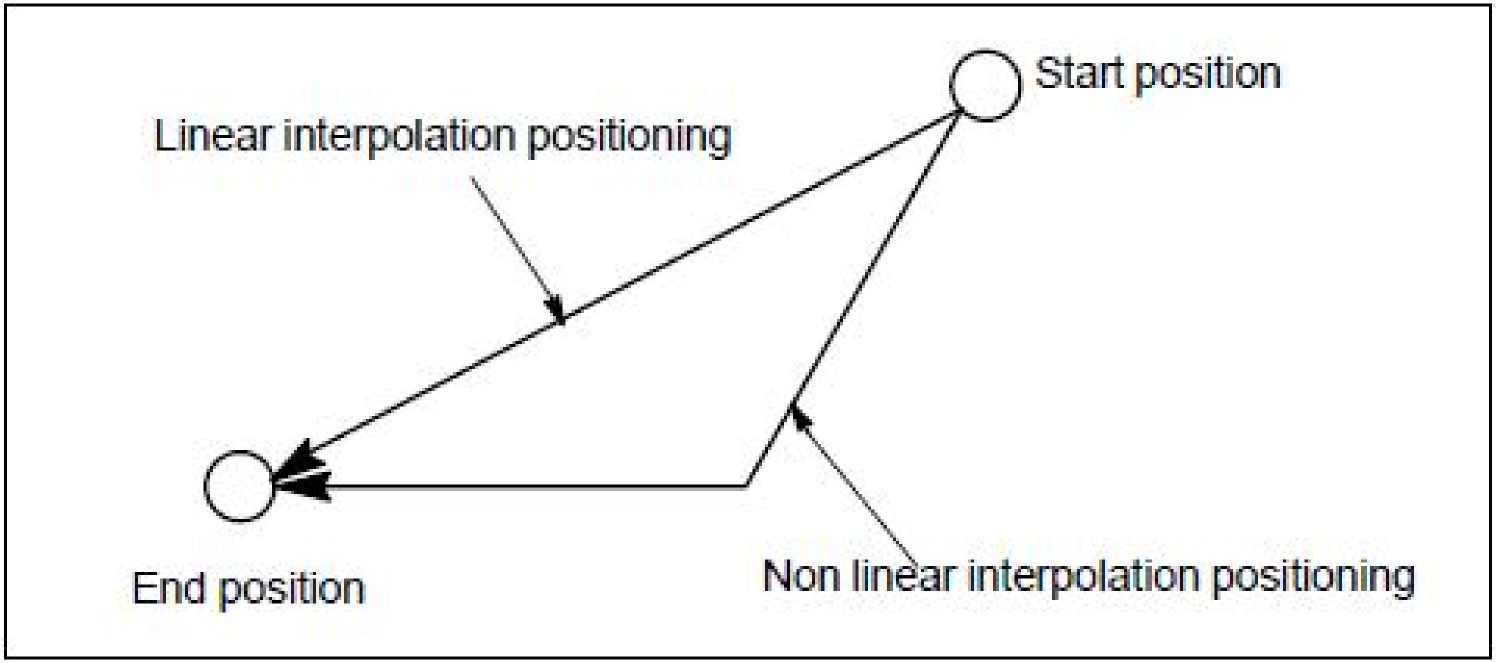
The range in which tools can move is called the stroke.

Besides strokes defined with limit switches, the operator can define an area which the tool cannot enter using a program or data in memory.

This function is called stroke check (see III–6.3).



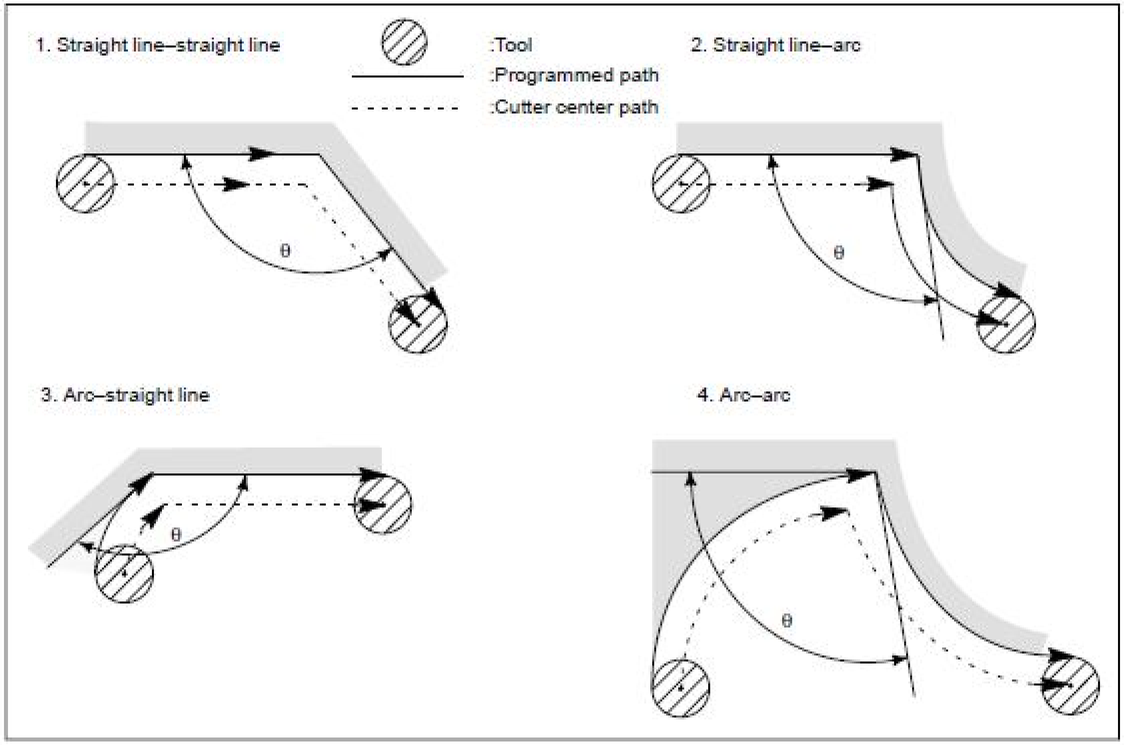
Linear interpolation positioning



5.4.2.1 Automatic Override for Inner Corners (G62)

When G62 is specified, and the tool path with cutter compensation applied forms an inner corner, the feedrate is automatically overridden at both ends of the corner.

There are four types of inner corners (Fig. 5.4.2.1 (a)).2,θθp178, in Fig. 5.4.2.1 (a) θp is a value set with parameter No. 1711. When θ is approximately equal to θp, the inner corner is determined with an error of 0.001,or less.

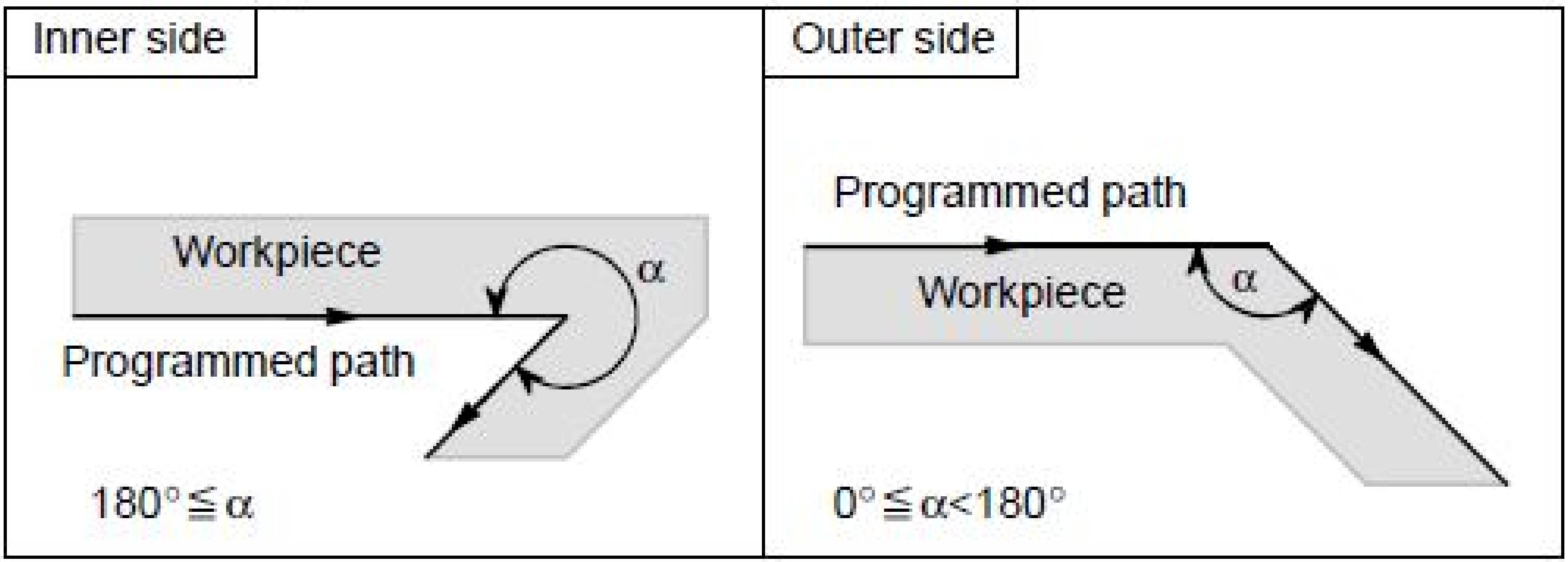


14.5 DETAILS OF CUTTER COMPENSATION C

14.5.1 General

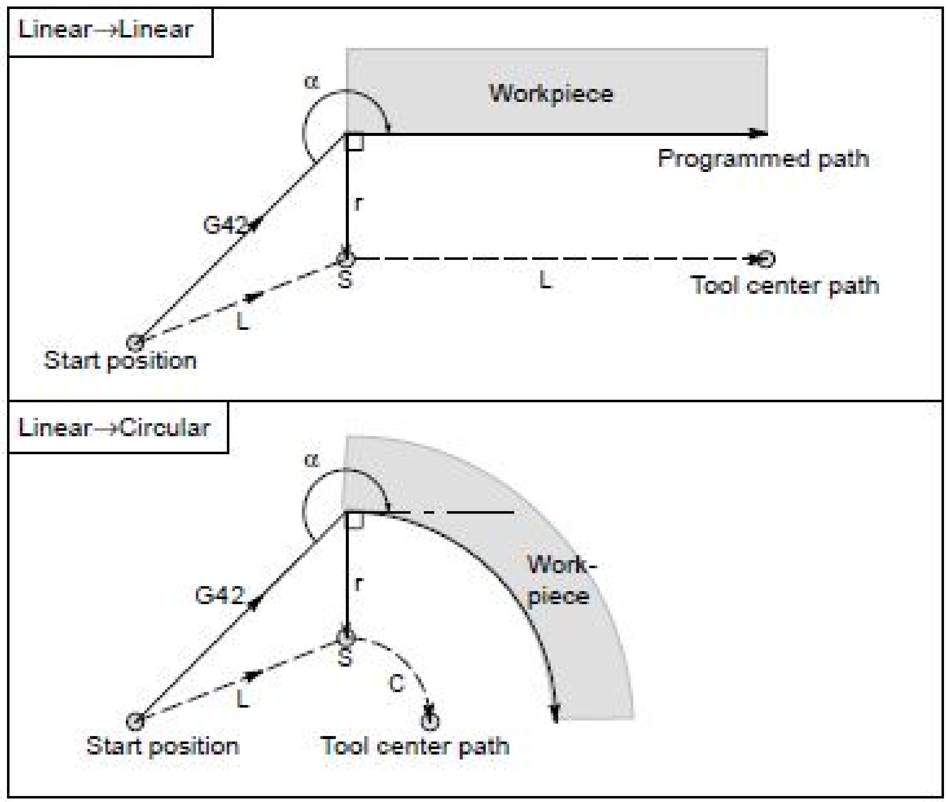
When an angle of intersection created by tool paths specified with move commands for two blocks is over 180°, it is referred to as "inner side."

When the angle is between 0° and 180°, it is referred to as "outer side."



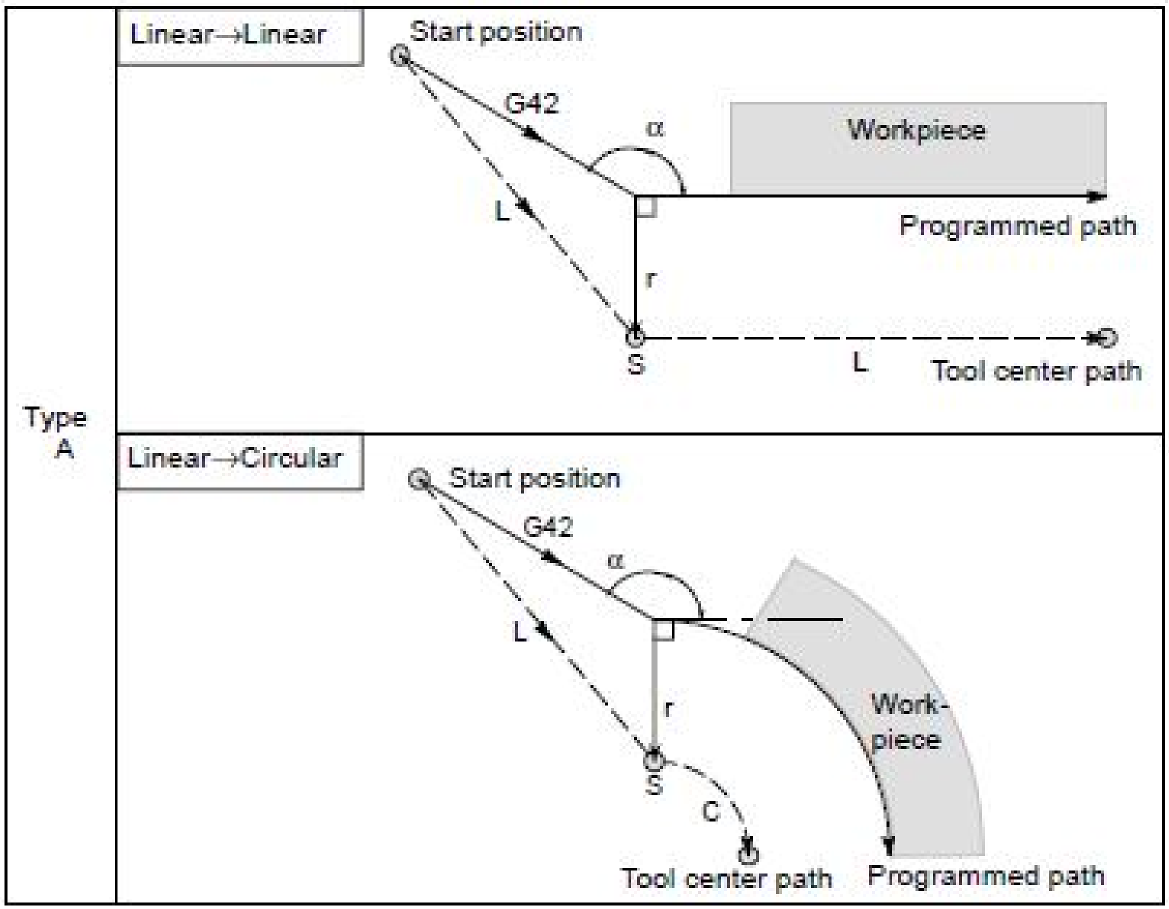
14.5.2 Tool Movement in Start–up

Tool movement around an inner side of a corner 180 ° ≤α



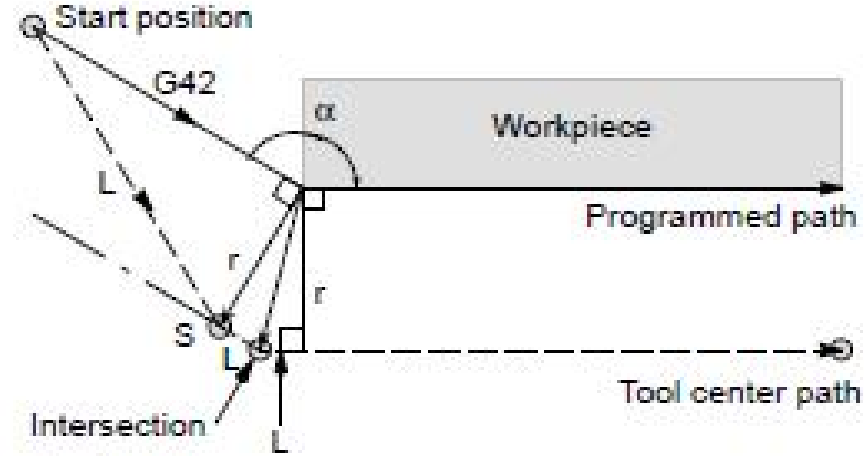
Tool movement around the outside of a corner at an obtuse angle 90 ≤°α<'180°

Tool path in start–up has two types A and B, and they are selected by parameter SUP (No. 5003#0).

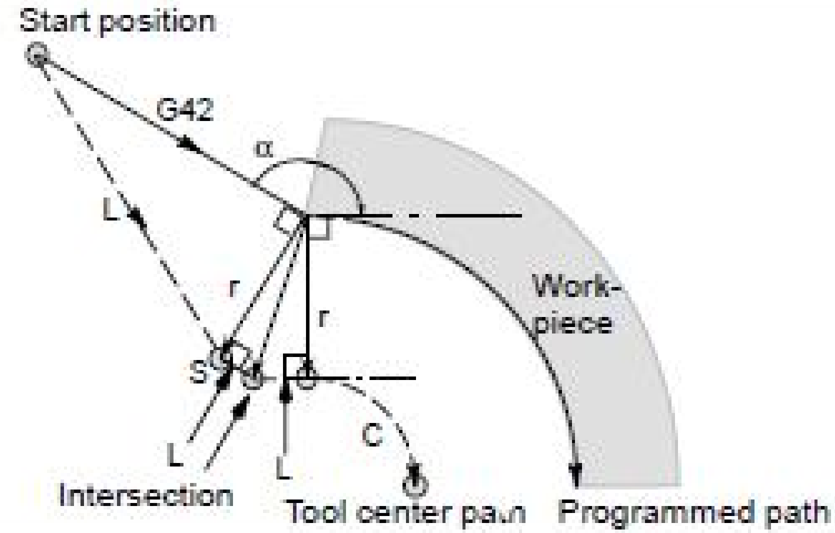


Type B

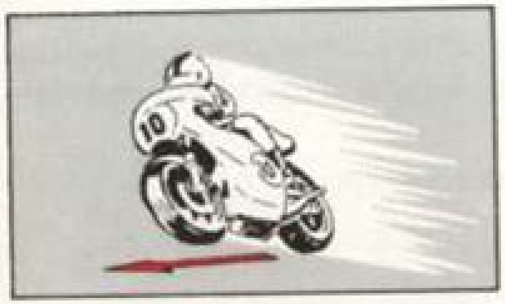
Linear→Linear



Linear→Circular

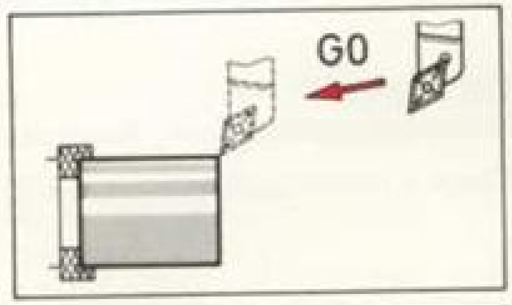


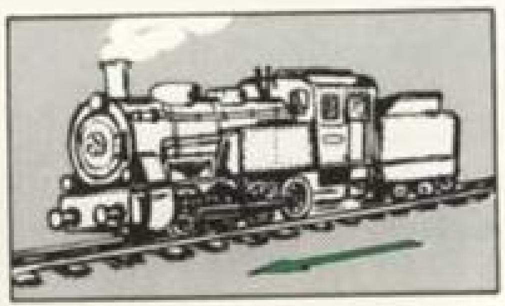
G0, G1, G2, G3; G41, G42, G40



G0

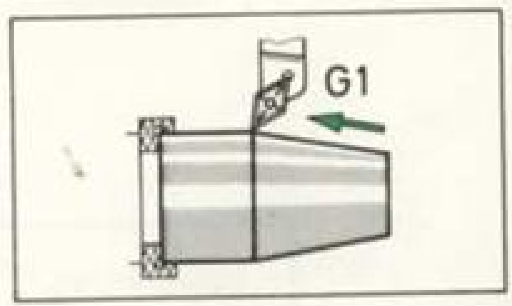
Eilgang

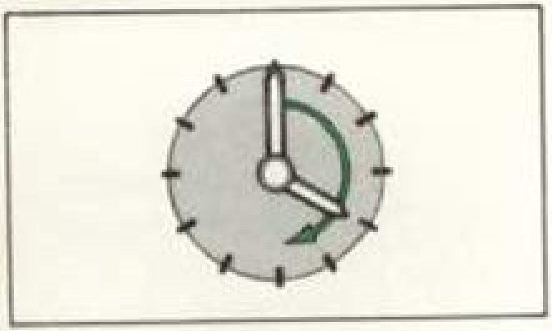




G1

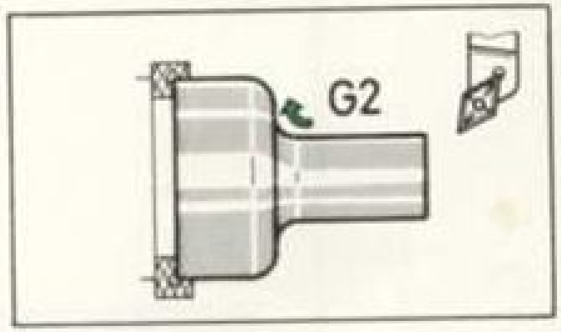
Vorschub auf
einer Geraden

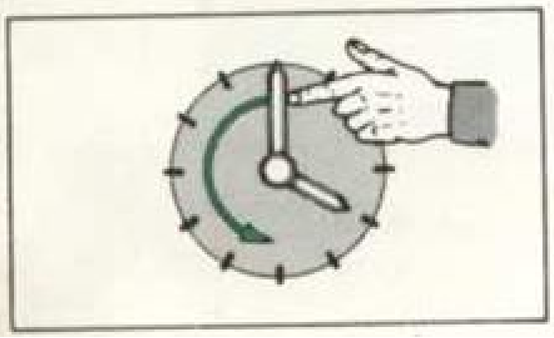




G2

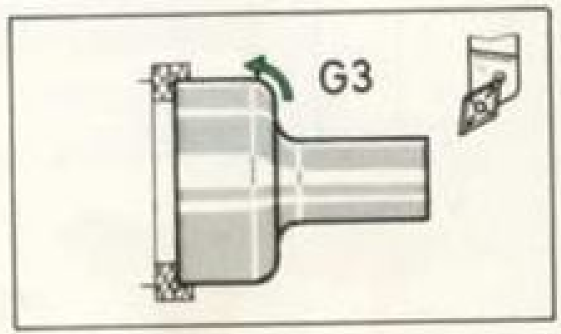
Vorschub **im** Uhrzeigersinn





G3

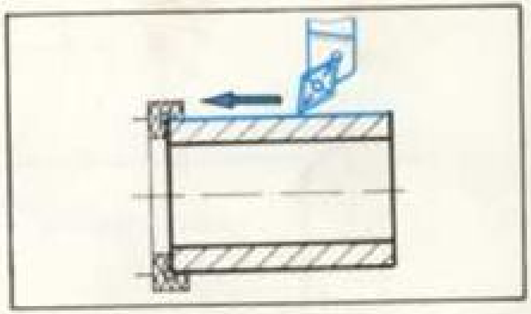
Vorschub **gegen** Uhrzeigersinn



G4**2**
=
Rechts

G42

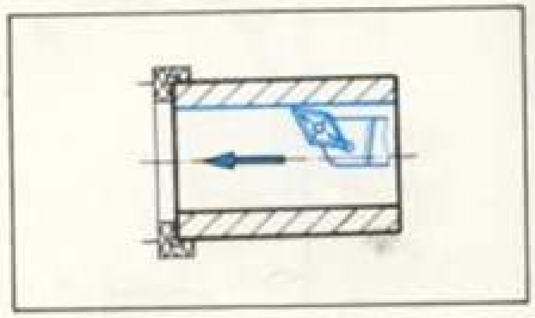
Werkzeug ist in Bewegungsrichtung gesehen **rechts** von der Kontur



G4**l**
=
links

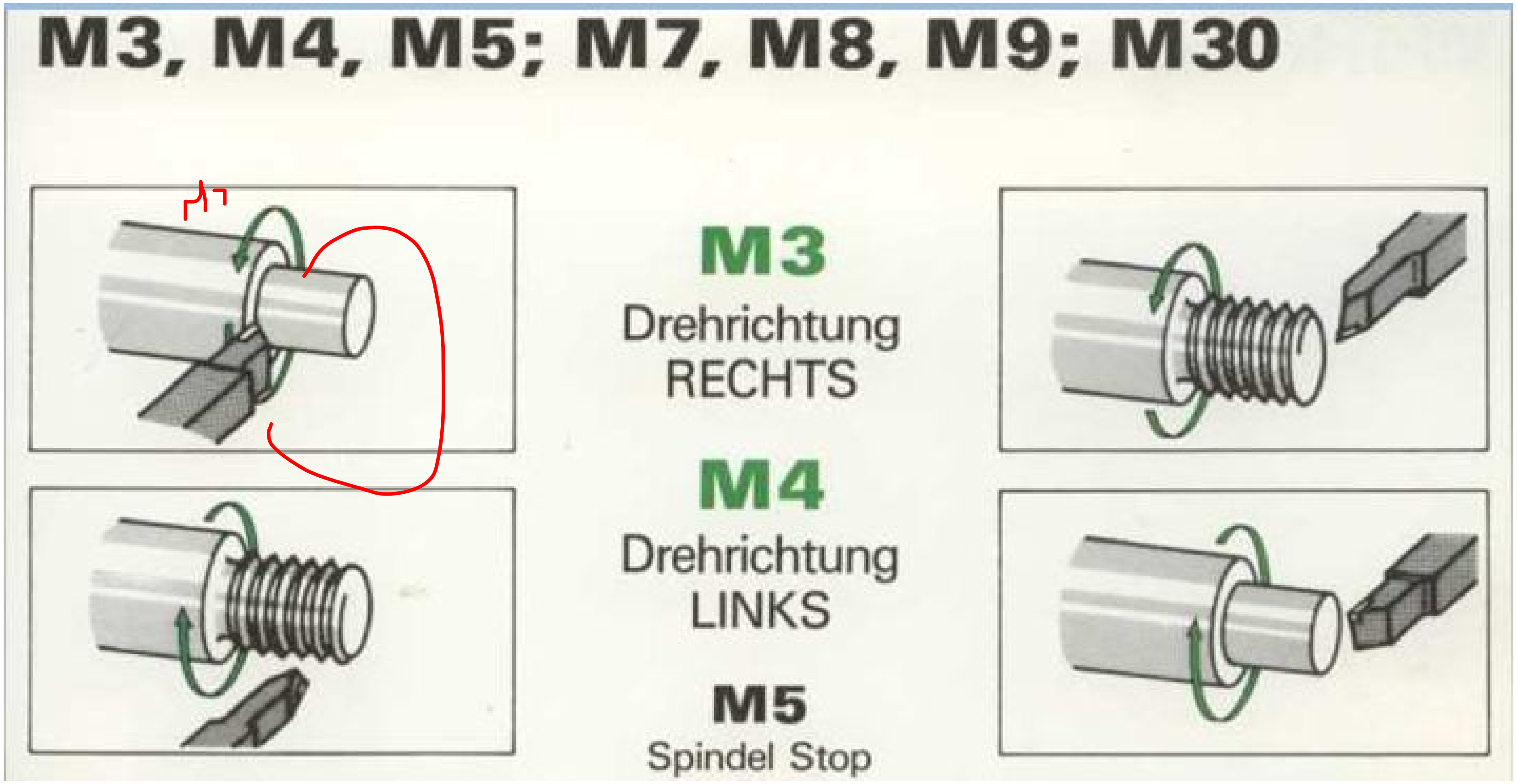
G41

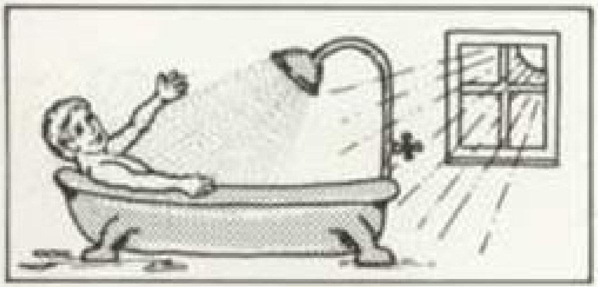
Werkzeug ist **links** von der Kontur



G40

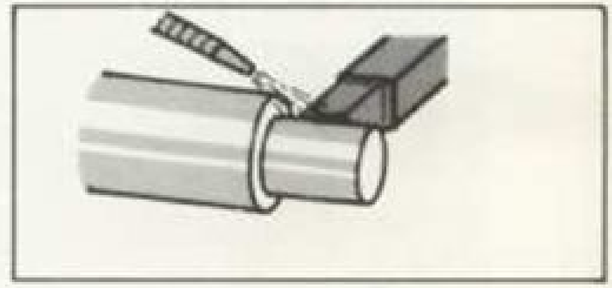
hebt G41/G42 auf

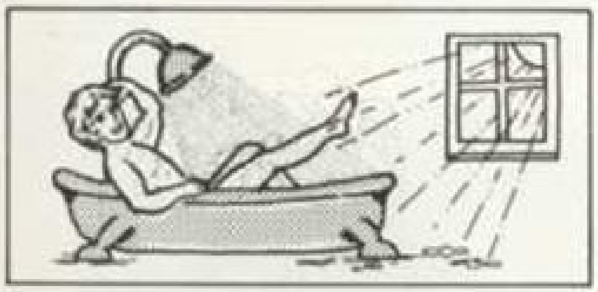

M3, M4, M5; M7, M8, M9; M30
M3
Drehrichtung
RECHTS
M4
Drehrichtung
LINKS
M5
Spindel Stop



M7

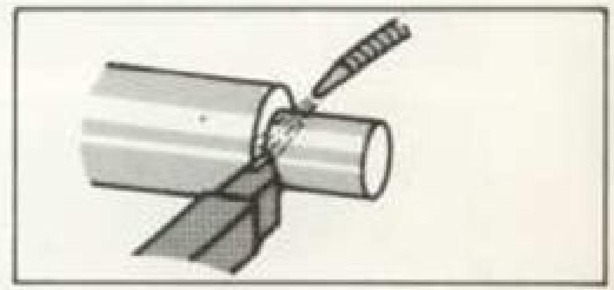
1. Kühlmittel EIN





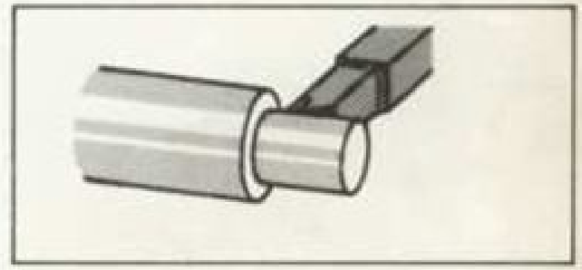
M8

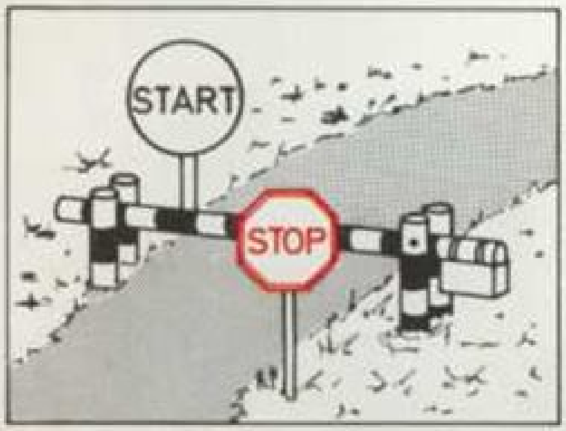
2. Kühlmittel EIN



M9

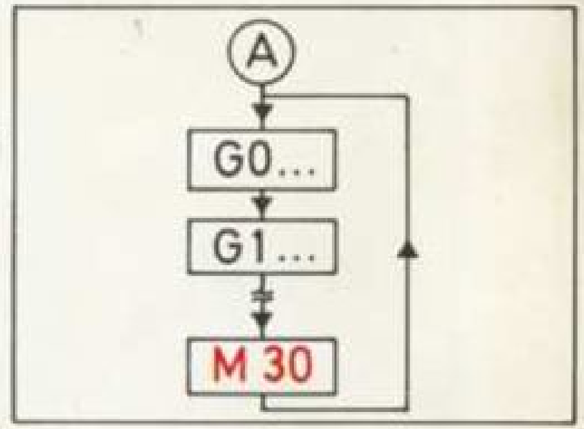
Kühlmittel
AUS



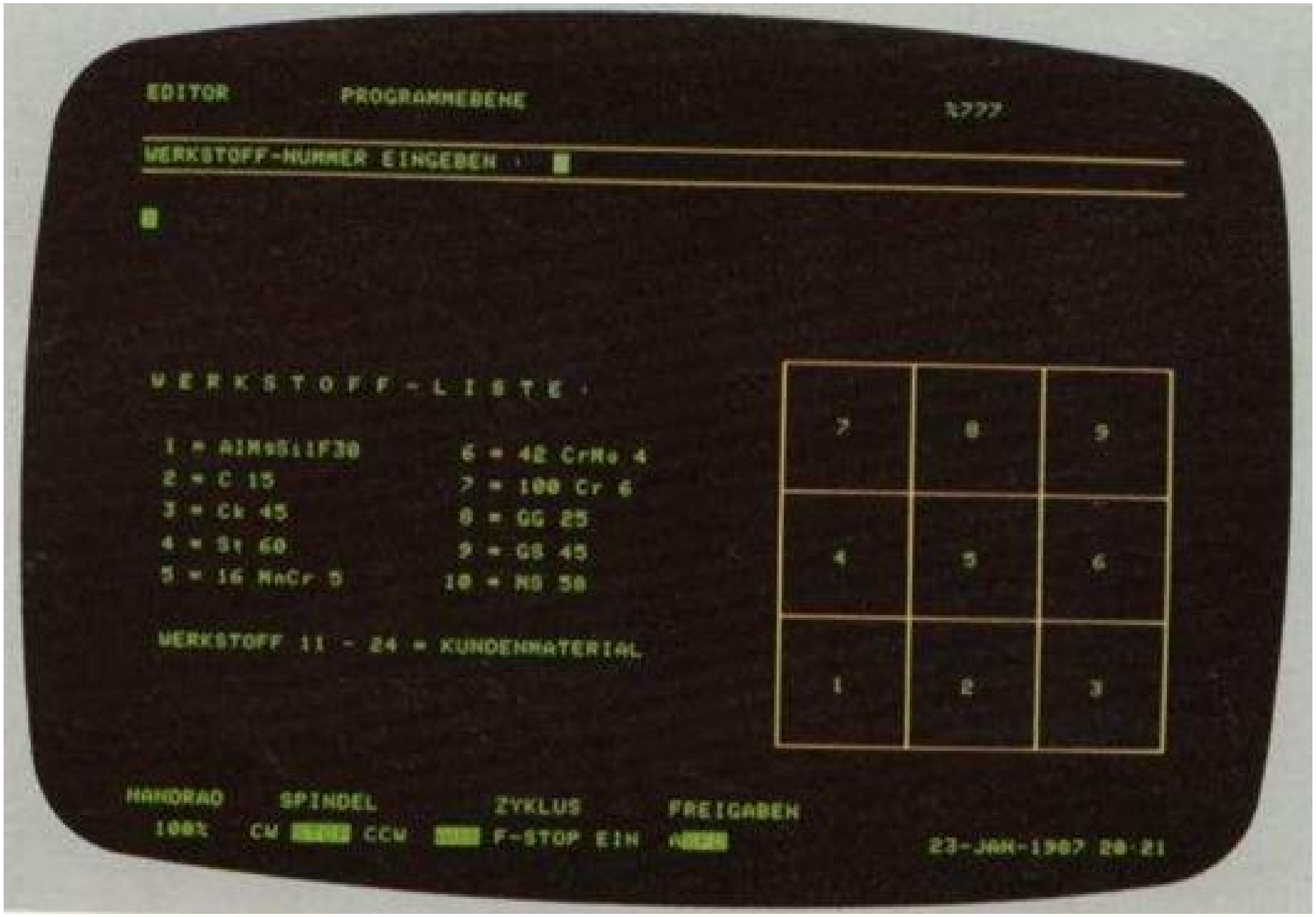


M30

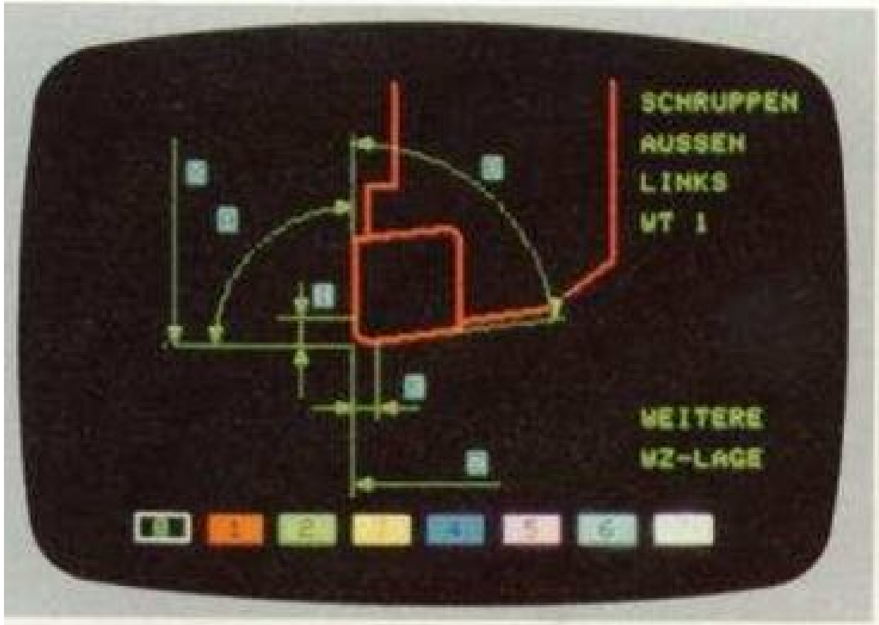
Programmende
+
Rücksprung zum
Programmbeginn

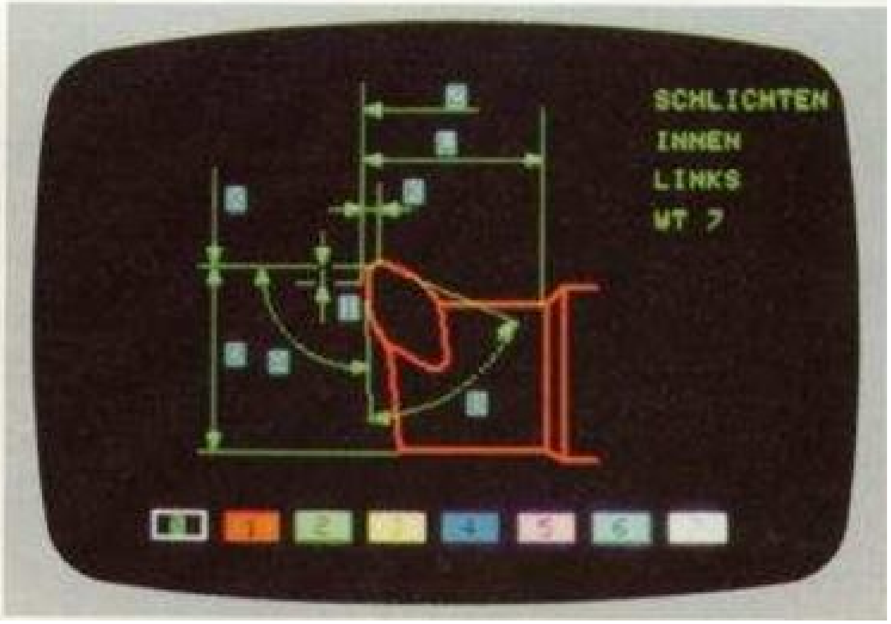



WERKSTOFFE
PARAMETER
N1303 W 3 ID 0 VR 150 VL 200 FR 0.500 FL 0.200 E 0.100
N1304 W 4 ID 0 VR 130 VL 250 FR 0.400 FL 0.100 E 0.000
N1305 W 5 ID 0 VR 100 VL 200 FR 0.400 FL 0.200 E 0.100
N1306 W 6 1000 VR 170 VL 260 FR 0.400 FL 0.200 E 0.100
ZR4.000 VB 45 FB 0.050 VG 10
WERKSTOFFDATEN:
VR = Schnittgeschwindigkeit Schruppen
VL = Schnittgeschwindigkeit Schlichten
FR = Vorschub Schruppen
FL = Vorschub Schlichten
E = Sondervorschub
ZR = Zustellungsrate
VB = Schnittgeschwindigkeit Bohren
FB = Vorschub Bohren
VG = Schnittgeschwindigkeit Gewindebohren
NR. ZURUECK
WERKZEUG EINGABE DIALOG
WERT ZURUECK
NR. SUCHEN
WERT VOR
WERT AENDERN
NR. VOR
HANDRAD
100%
SPINDEL
CW CCW
ZYKLUS
F-STOP EIN
FREIGABEN
23-JAN-1987 20:30


EDITOR
PROGRAMMEBENE
%777
WERKSTOFF-NUMMER EINGEBEN :
W E R K S T O F F - L I S T E :
1 = AlMgSi1F30
2 = C 15
3 = Ck 45
4 = St 60
5 = 16 MnCr 5
6 = 42 CrMo 4
7 = 100 Cr 6
8 = GG 25
9 = GS 45
10 = MS 58
WERKSTOFF 11 - 24 = KUNDENMATERIAL
7
8
9
4
5
6
1
2
3
HANDRAD
SPINDEL
ZYKLUS
FREIGABEN
100%
CW
CCW
F-STOP EIN
23-JAN-1987 20:21

WERKZEUGE


SCHRUPPEN
AUSSEN
LINKS
WT 1
WEITERE
WZ-LAGE


SCHLICHTEN
INNEN
LINKS
WT 7

WT = Werkzeugtyp

FC = Farbcode

X = Einstellmaß L (Radius-Wert)

Z = Einstellmaß Q

I = Lage des Schneidenmittelpunktes in X Richtung

K = Lage des Schneidenmittelpunktes in Z Richtung

A = Einstellwinkel $\varkappa$

B = Spitzenwinkel ε

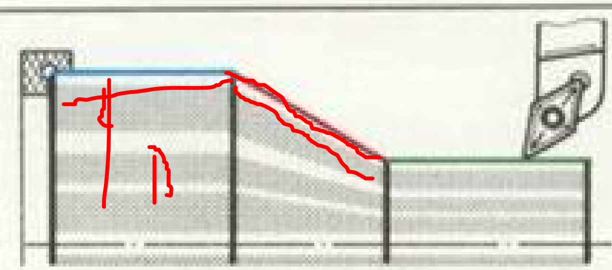
D = Durchmesser des Werkzeuges

L = Nutzbare Länge des Werkzeuges

Schnittgeschwindigkeit und Drehzahl

1. **G96** = konstante **Schnittgeschwindigkeit** in $\frac{\textbf{m}}{\textbf{min}}$

G96 S180



- große Drehzahl
- stufenlose Veränderung der Drehzahl
- kleine Drehzahl

$v = \frac{\pi D n}{1000}$

$v = c$

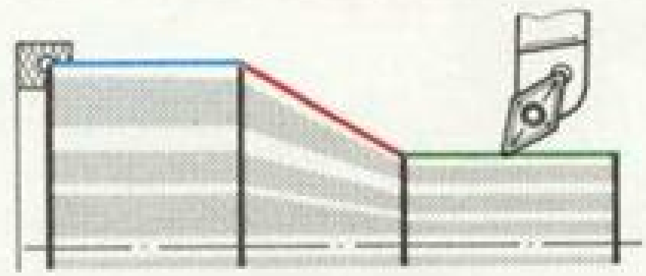
$n = \frac{1000 v}{\pi D}$

$n_D = \frac{c}{D}$

$n_d > n_D$

2. **G97** = konstante **Drehzahl** in $\frac{1}{min}$

G97 S2400



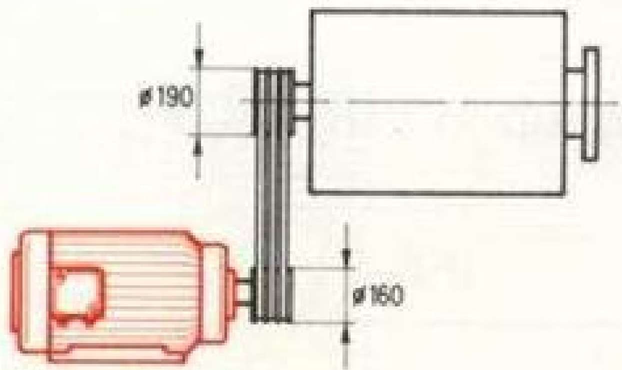
jeweils **gleiche** Drehzahl

Leistungsdiagramm einer CNC-Drehmaschine

CT 40:

$n_{min} = 20 \frac{1}{min}$

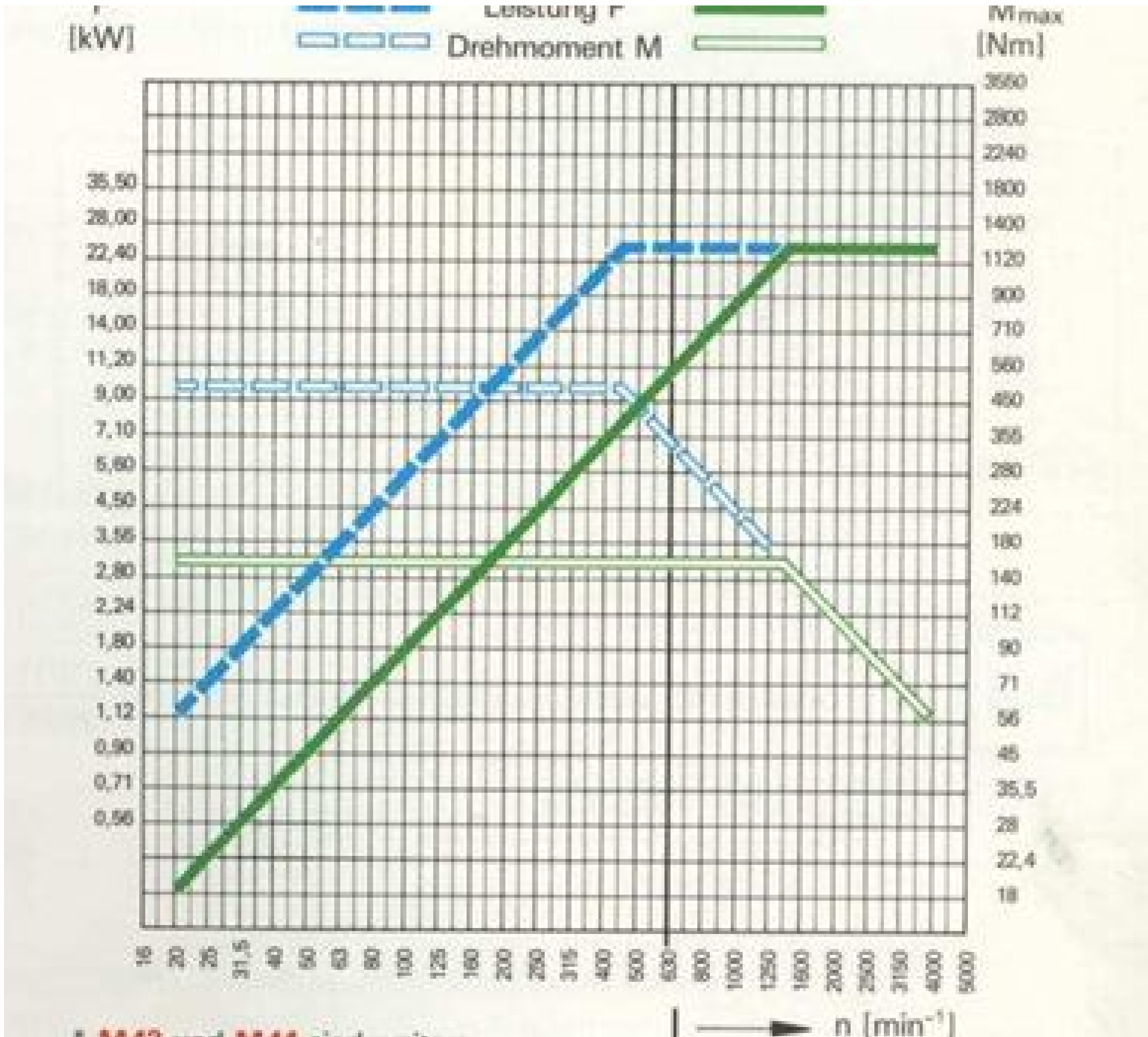
$n_{max} = 4000 \frac{1}{min}$



Stufe I: i_1 = 3,18 : 1 **(M41)***

Stufe II: i_2 = 1 : 1 **(M42)**

P [kW]	M41		M42	M_{max} [Nm]
	▬ ▬ ▬	Leistung P	▬▬▬	
	▭ ▭ ▭	Drehmoment M	▭▭▭	



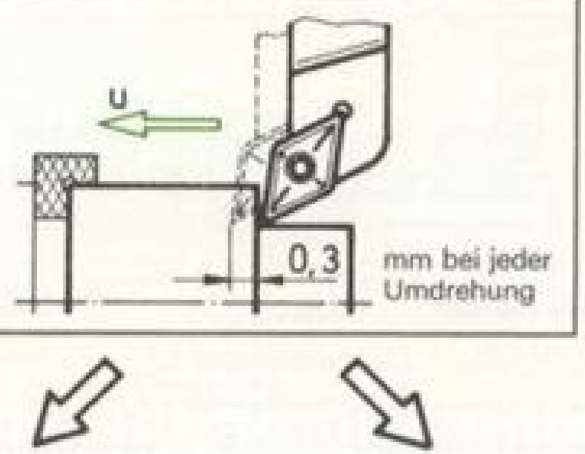
* **M43** und **M44** sind weitere Getriebestufen für größere Maschinen

G26 S630 = Drehzahl-Begrenzung

Vorschub und Vorschubgeschwindigkeit

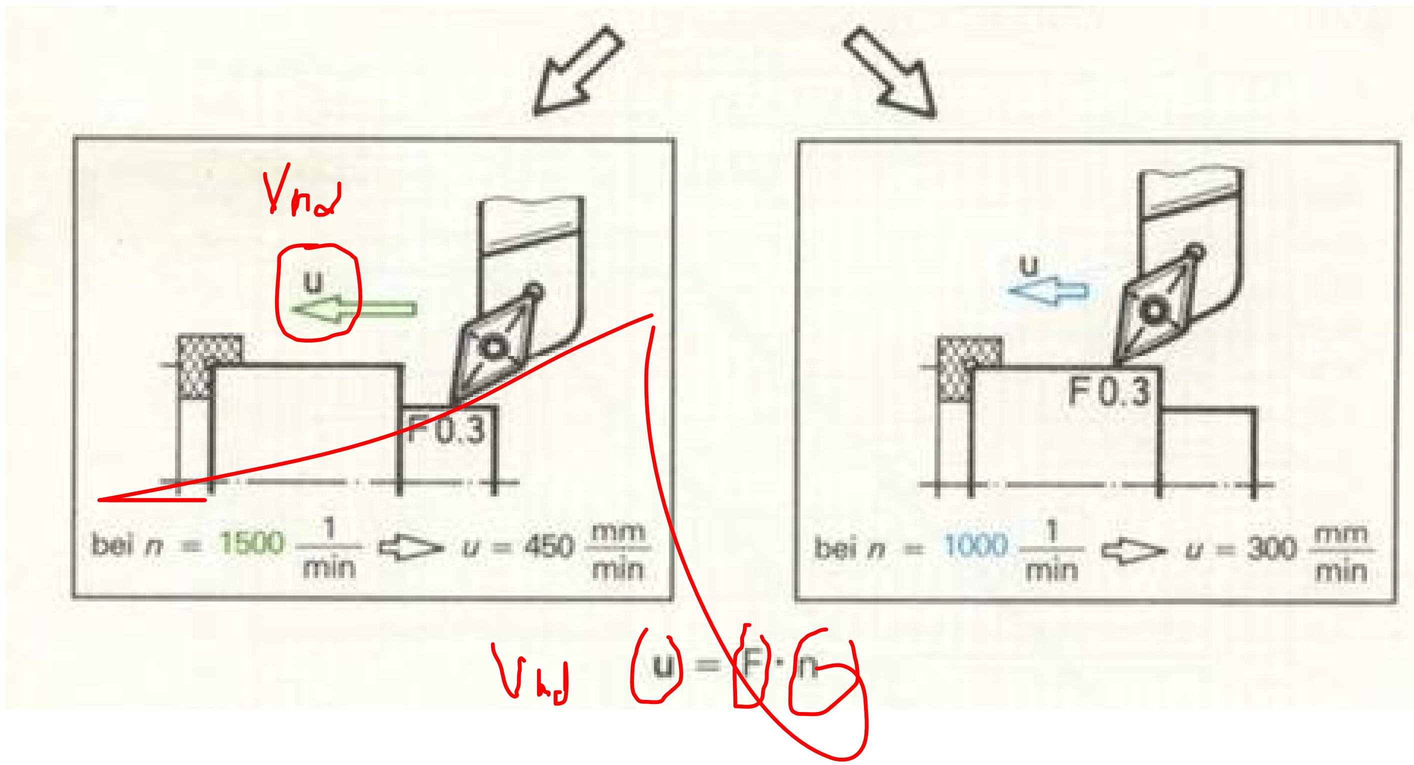
1. **G95** = **Vorschub** in **mm** $\left(\frac{mm}{U}\right)$

F0.3



s = 0,3 mm/U t_h = L / …

n · s = v n d

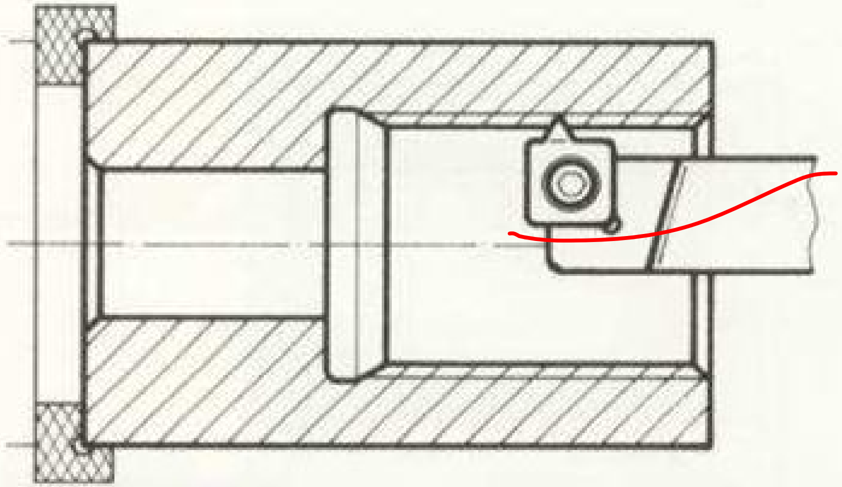


2. **G94** = konstante Vorschubgeschwindigkeit in $\frac{mm}{min}$

G 94 F 3000
Anwendung z. B. ● bei Teilegreifern ● bei angetriebenen Werkzeugen ● Nutenziehen

Programmaufbau und Satzformat

In die Buchse soll ein Gewinde geschnitten werden:



Gewindeschneiden
+
Bohren
+
Ausdrehen
+
Fasen

Arbeitsablaufplan:

1.	Bohren
2.	Ausdrehen
3.	Fasen
4.	Gewindeschneiden

Gliederung eines Programmes:

% 4710 (= Programm-Nummer)

N 1	G 97			F0.1	S 1200	T5 (Voll-Bohrer)	M3	M7
N 2	G 0	X 0	Z 2					
N 3	G 1		Z –72					

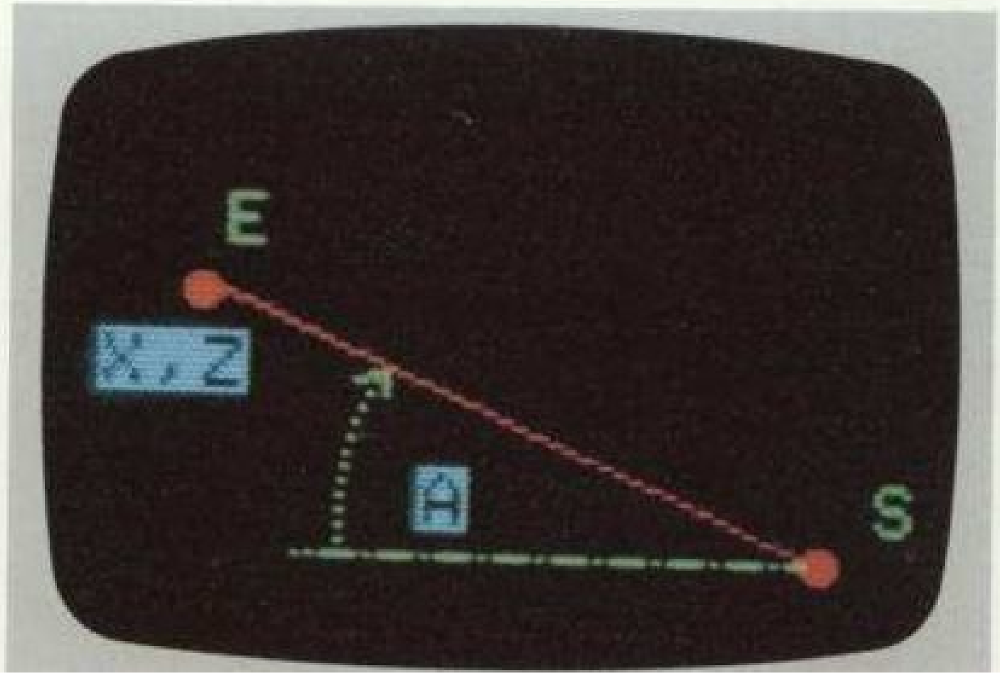
Programm-technische Befehle	**Geometrische** Befehle	**Technologische** Befehle

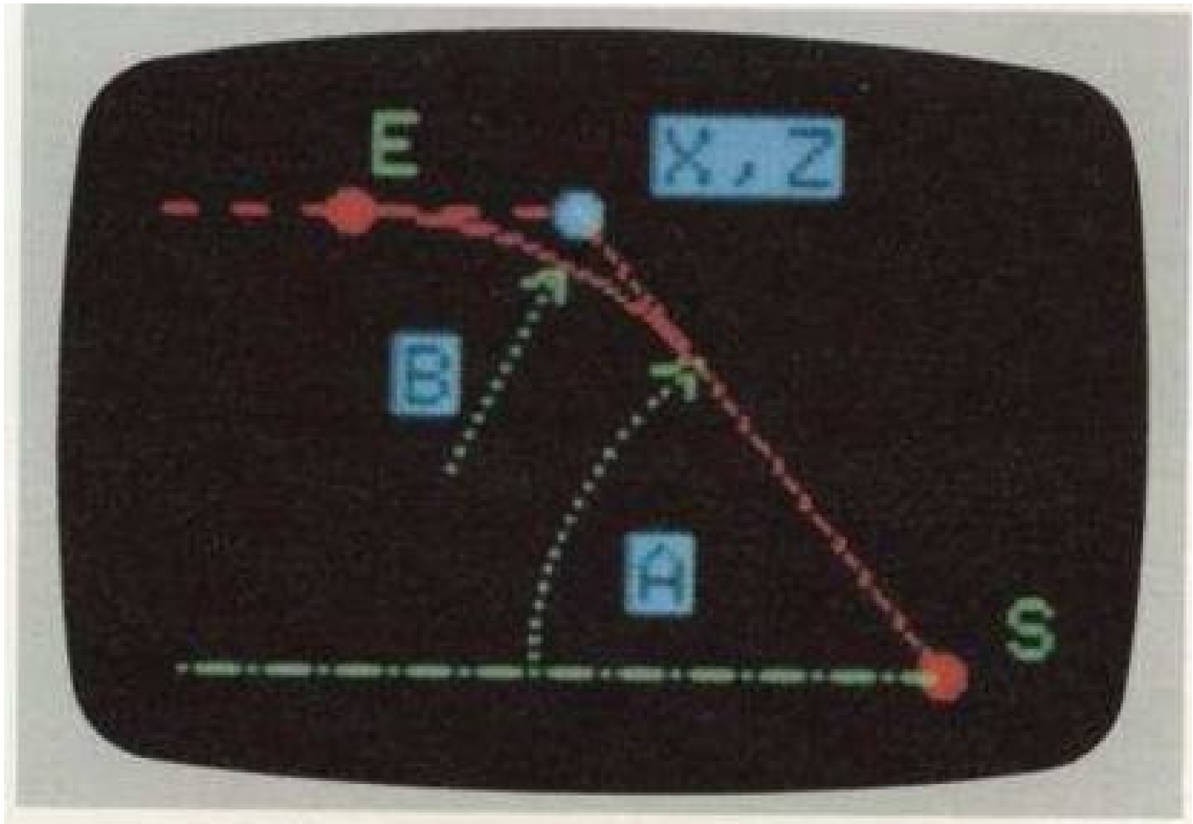
Das **PROGRAMM** besteht aus * **Sätzen.**

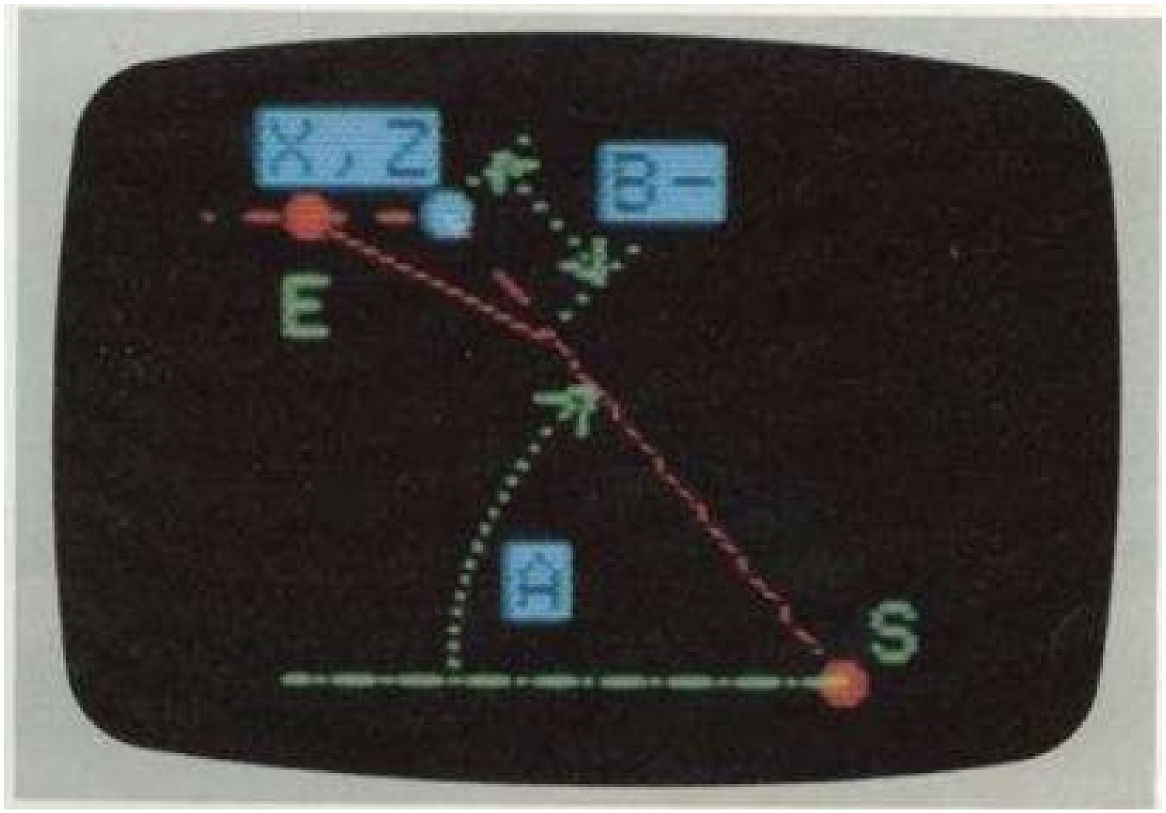
Der **SATZ** besteht aus * **Wörtern.**

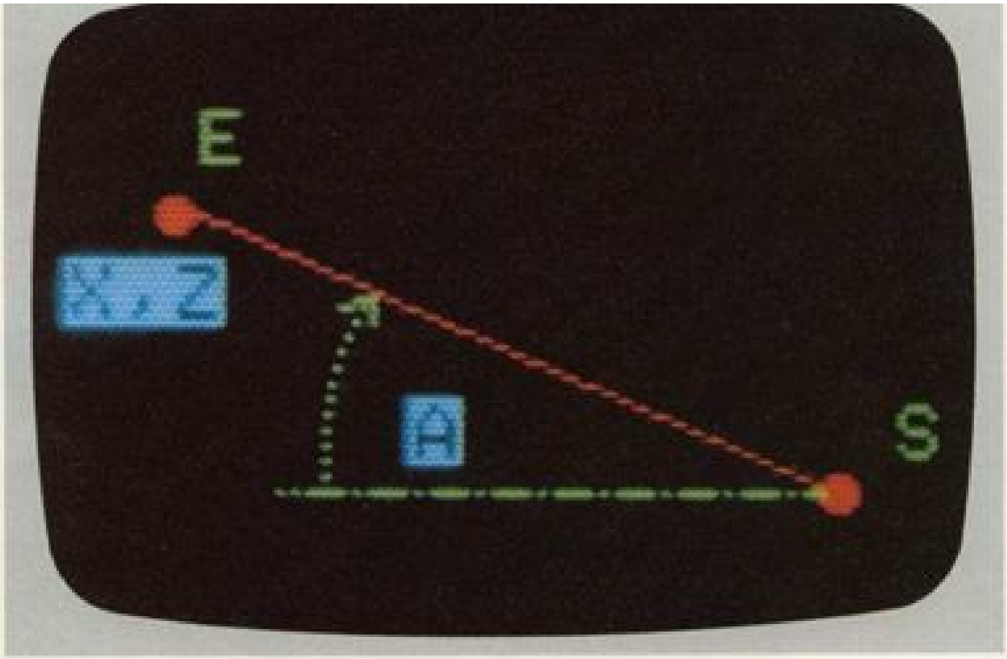
Das **WORT** besteht aus einer * **Adresse** und einer * **Zahl.**

Geometriemöglichkeiten mit G1




E
X, Z
B
A
S


X, Z
B-
E
S

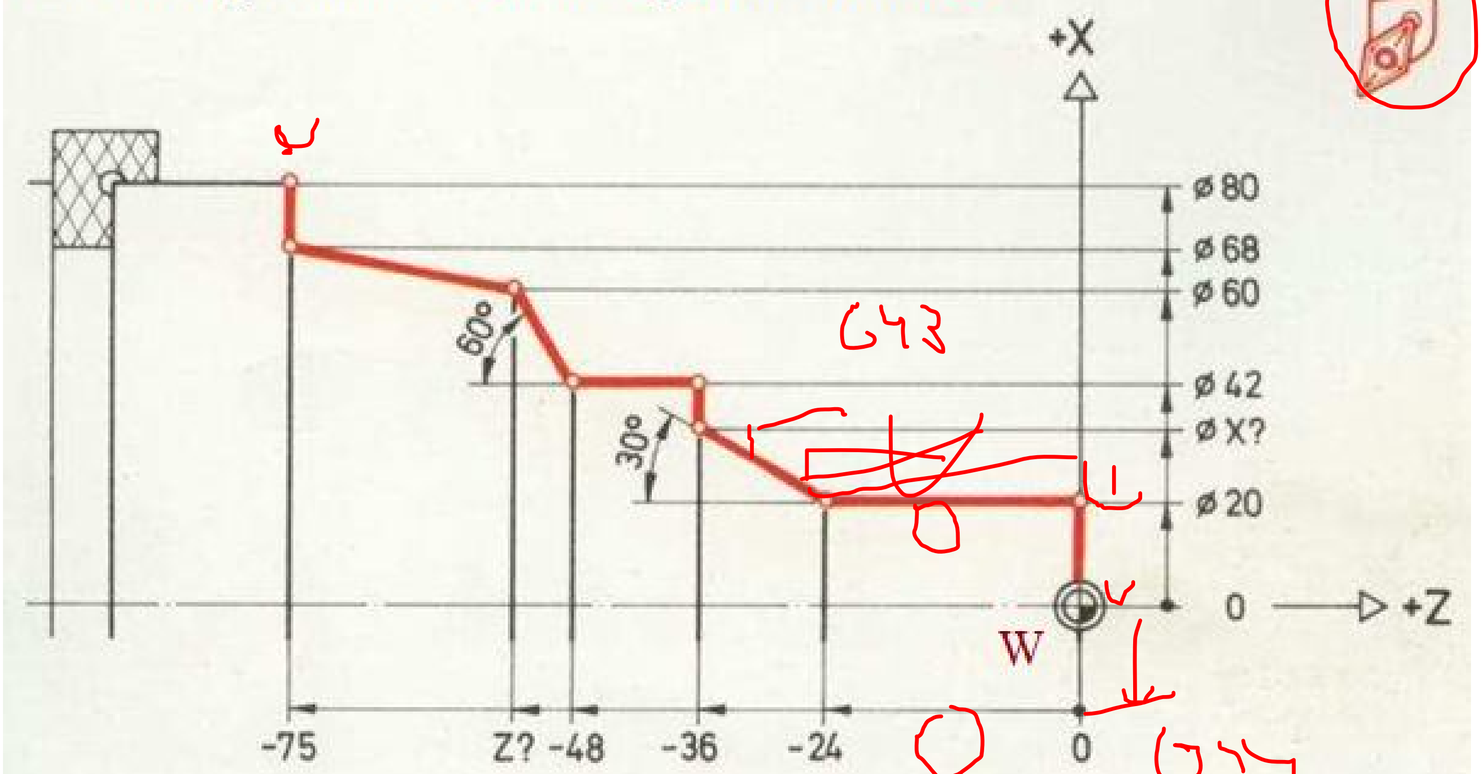


X = Enddurchmesser

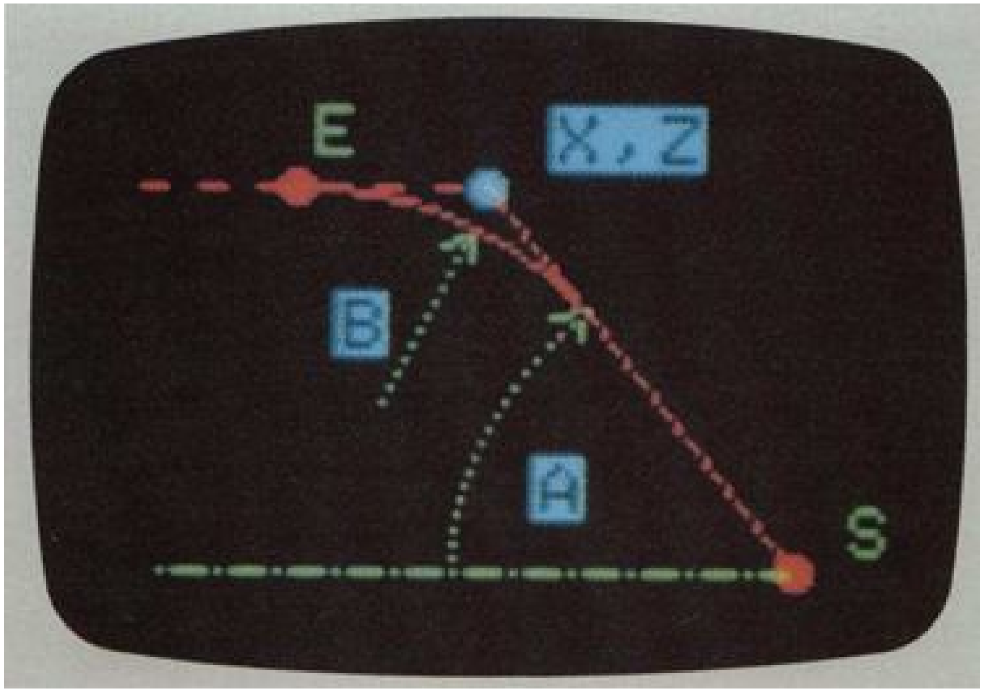
Z = Endlänge

A = Winkel

Zhvendosja linerare G1 me X,Z dhe kendin A



% 1001								
N	G	X	Z	Hilfsadressen	F	S	T	M
N1	G0	X0	Z2					
N2	G42 G1		Z0					
N3	G1	X20						
N4	G1		Z–24					
N5	G1	X?	Z–36	A30				
N6	G1	X42						
N7	G1		Z–48					
N8	G1	X60	Z?	A60				
N9	G1	X68	Z–75					
N10	G1	X80						
N11	G40 G1	X82						
N12								M30



X = Enddurchmesser

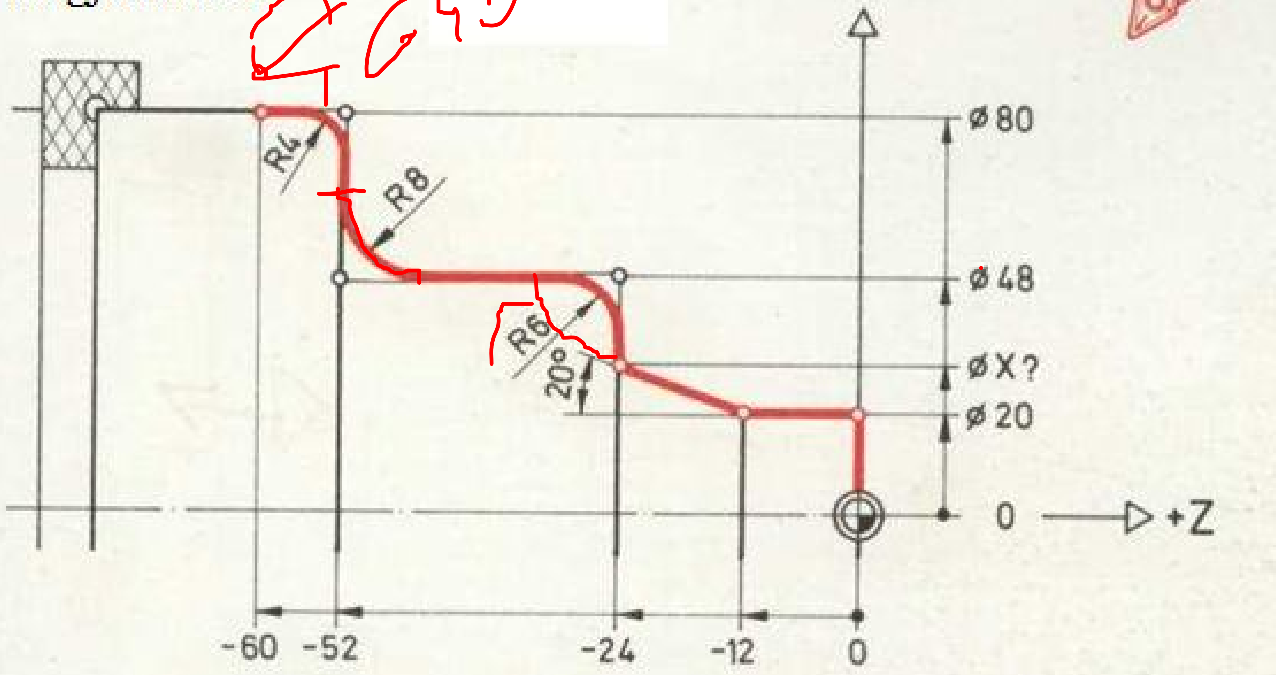
Z = Endlänge

E = tatsächlicher Endpunkt der Verrundung

A = Winkel

B = tangentiale Verrundung

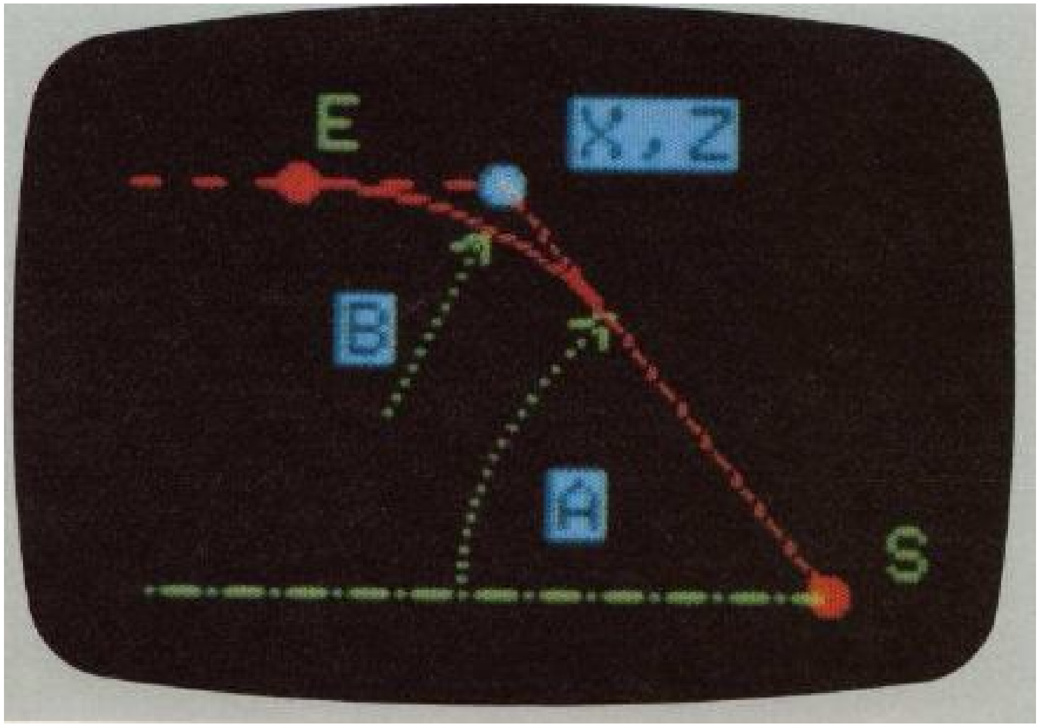
Zhvendosja G1 me Z,X,A dhe rrezen kendedrejt tangjenciale B



% 1002

N	G	X	Z	Hilfsadressen	F	S	T	M
N1	G0	X0	Z2					
N2	G42 G1		Z0					
N3	G1	X20						
N4	G1		Z−12					
N5	G1	X?	Z−24	A20				
N6	G1	X48		B6				
N7	G1		Z−52	B8				
N8	G1	X80		B4 E0.08*				
N9	G1		Z−60					
N10	G40 G1	X82						
N11								M30

* Sondervorschub für kleine Übergangs-Radien und Fasen.

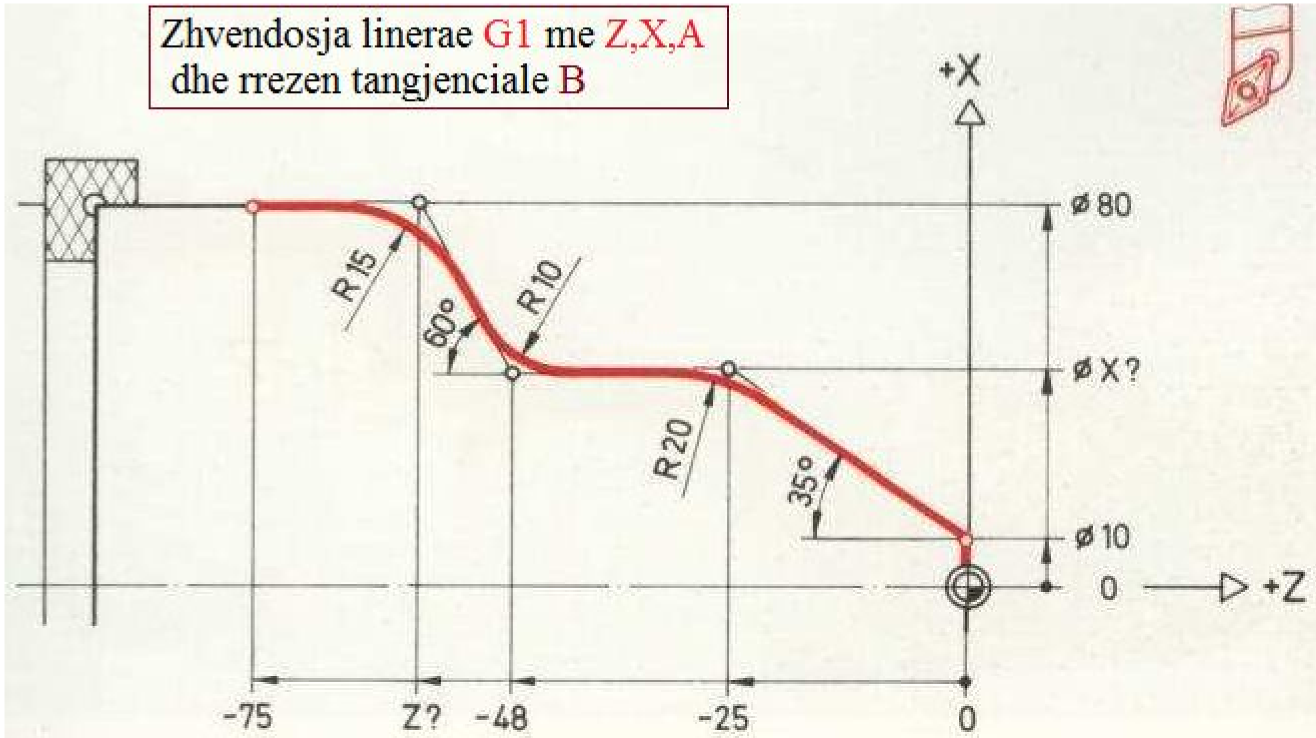


X = Enddurchmesser

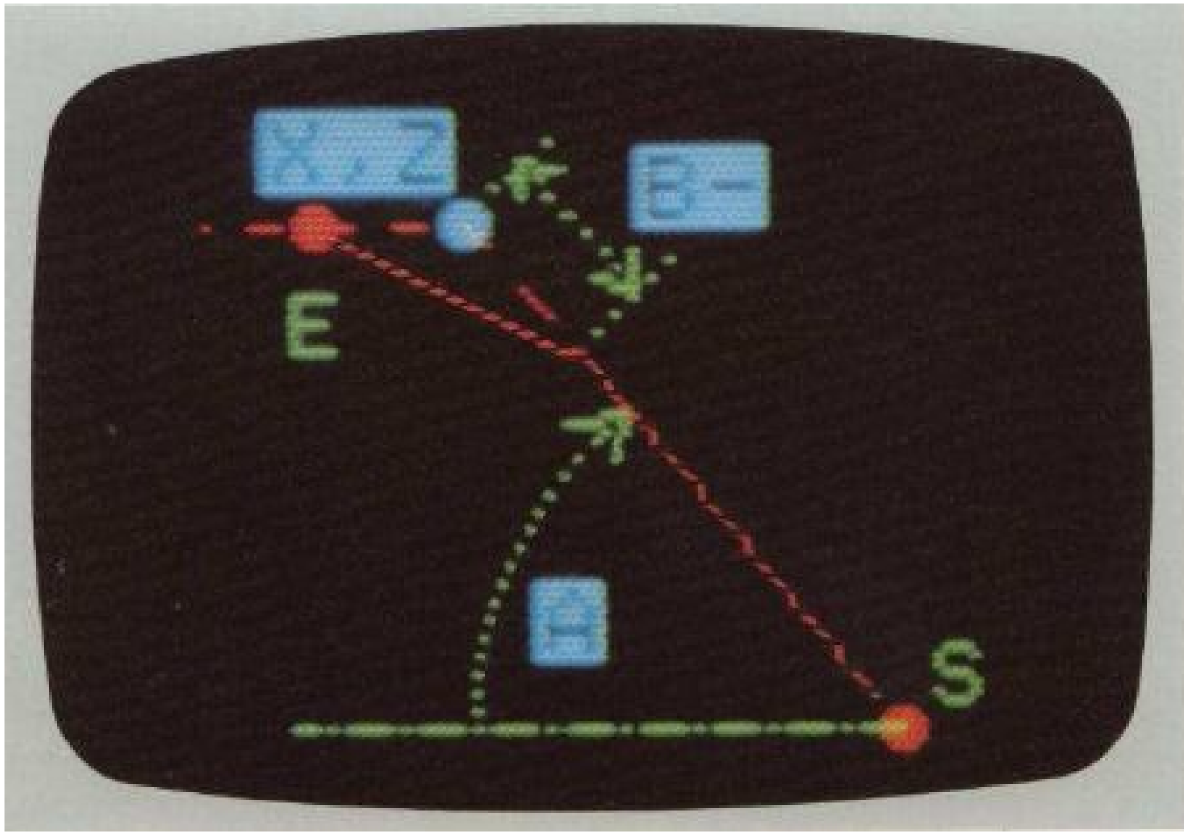
Z = Endlänge

E = tatsächlicher Endpunkt der Verrundung

A = Winkel



% 1003								
N	G	X	Z	Hilfsadressen	F	S	T	M
N1	G0	X0	Z2					
N2	G42 G1		Z0					
N3	G1	X10						
N4	G1	X?	Z−25	A35 B20 E0.08				
N5	G1		Z−48	B10				
N6	G1	X80	Z?	A60 B15				
N7	G1		Z−75					
N8	G40 G1	X82						
N9								M30



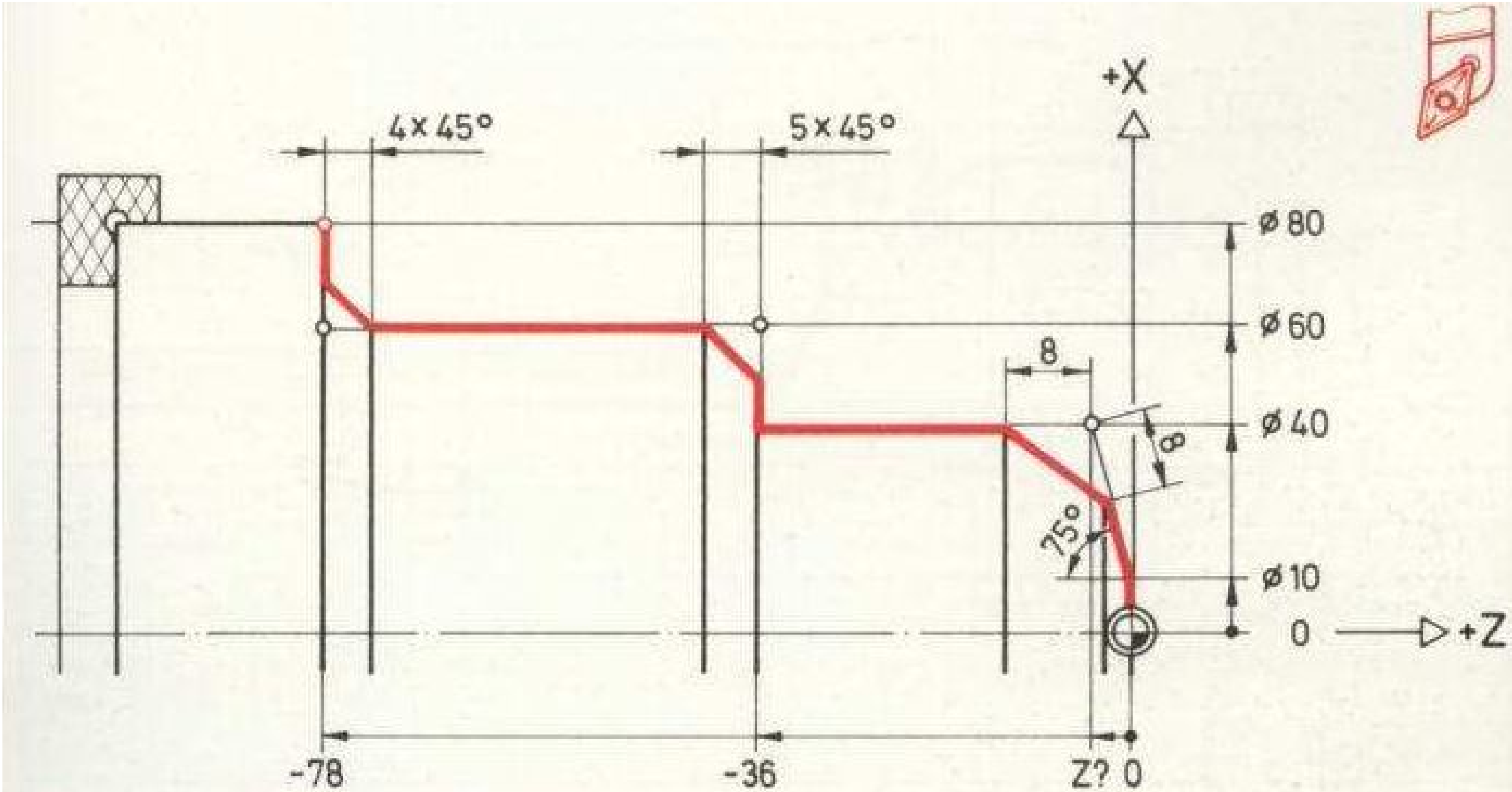
X = Enddurchmesser

Z = Endlänge

E = tatsächlicher Endpunkt der Fase

A = Winkel

B– = Fase

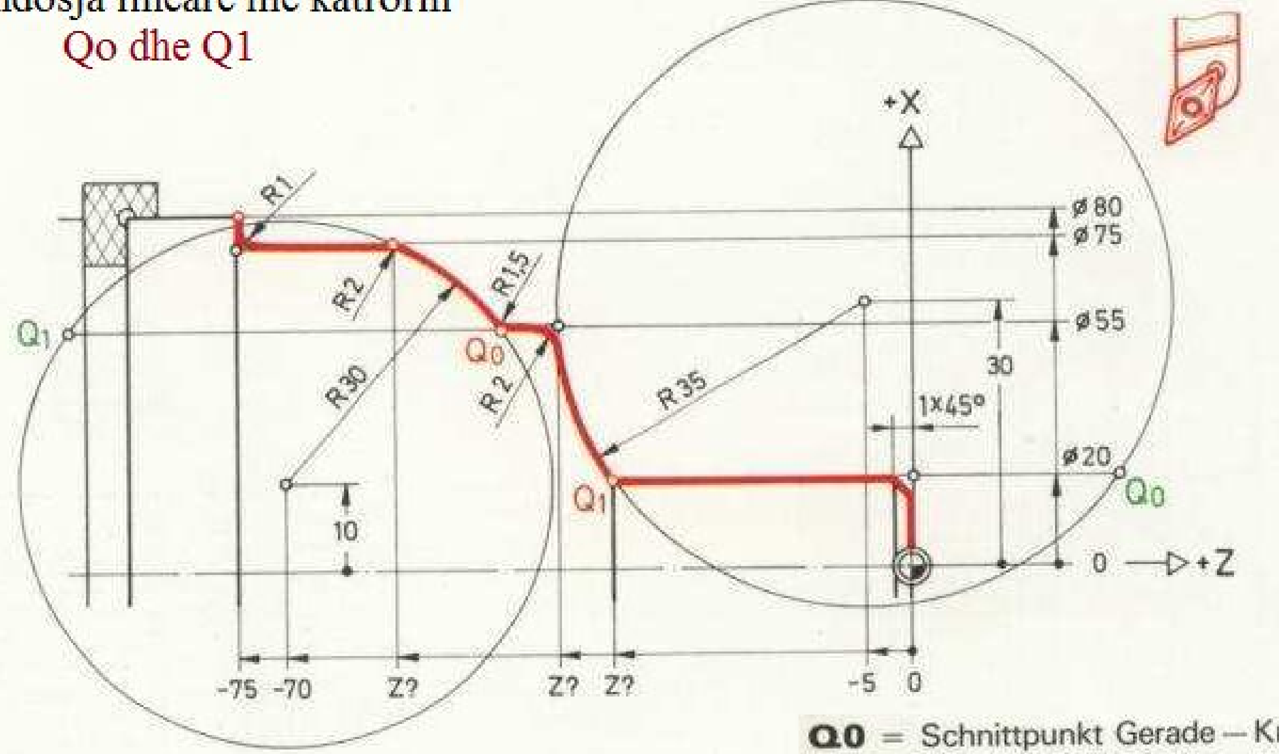


% 1004

N	G	X	Z	Hilfsadressen	F	S	T	M
N1	G0	X0	Z2					
N2	G42 G1		Z0					
N3	G1	X10						
N4	G1	X40	Z?	A75 B−8				
N5	G1		Z−36					
N6	G1	X60		B−5				
N7	G1		Z−78	B−4				
N8	G1	X80						
N9	G40 G1	X82						
N10								M30

Zhvendosja lineare me katrorin

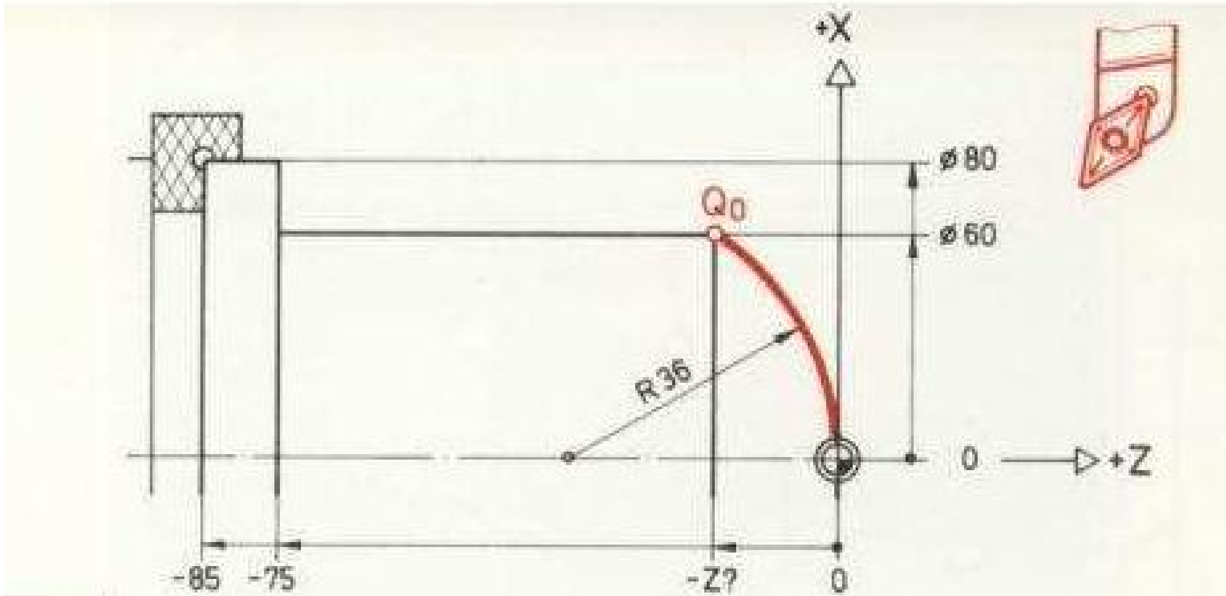
Qo dhe Q1



Q0 = Schnittpunkt Gerade — Kreis rechts
Q1 = Schnittpunkt Gerade — Kreis links

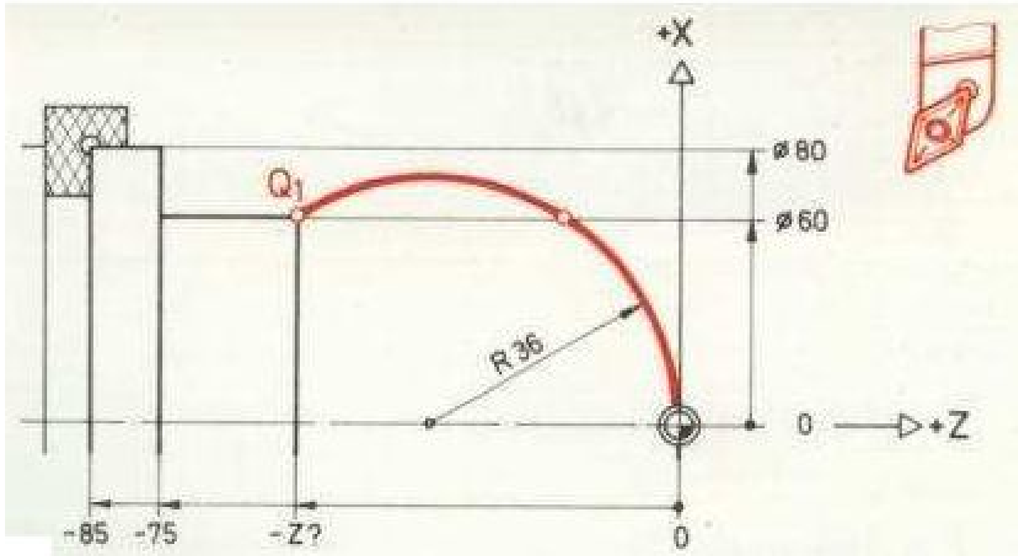
% 1005													
N	G		X	Z	Hilfsadressen					F	S	T	M
N1		G0	X0	Z2									
N2	G42	G1		Z0									
N3		G1	X20						B−1				
N4		G1		Z?	A0			Q1	B0				
N5		G12	X55	Z?	I30 K−5	R35			B2				
N6		G1		Z?	A0			Q0	B1.5				
N7		G13	X75	Z?	I10 K−70	R30			B2				
N8		G1		Z−75					B1				
N9		G1	X80										
N10	G40	G1	X82										
N11													M30

Levizja rrethore me katror Qo



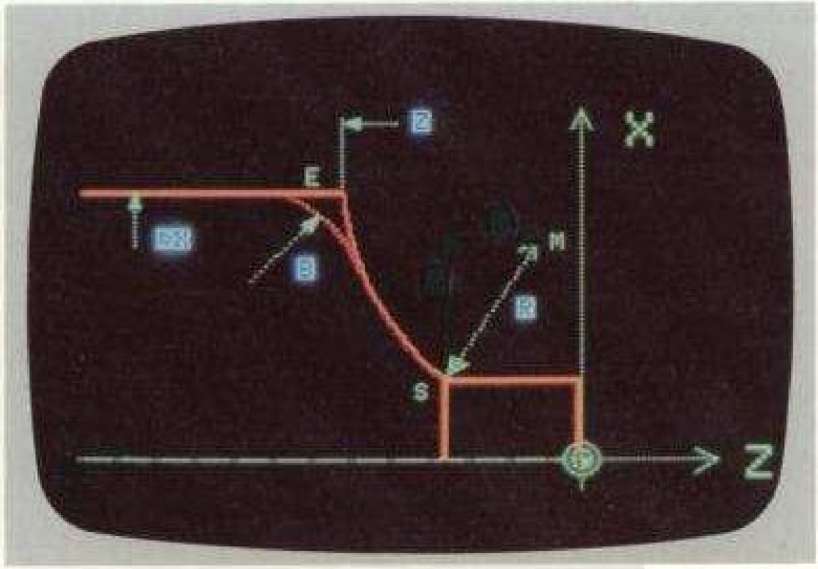
% 1006									
N	G	X	Z	Hilfsadressen	F	S	T	M
N1	G0	X0	Z2					
N2	G42 G1		Z0					
N3	G3	X60	Z?	I0 K−36 B0 Q0				
N4	G1		Z−75					
N5	G1	X80						
N6	G40 G1	X82						
N7								M30

Levizja rrethore me katror Q1

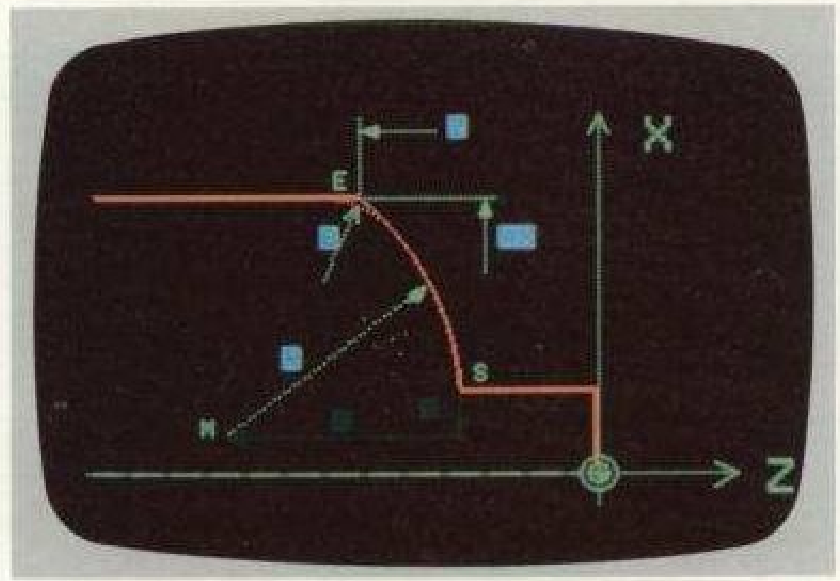


% 1007						
N	G	X	Z	Hilfsadressen	F S T	M
N1	G0	X0	Z2			
N2	G42 G1		Z0			
N3	G3	X60	Z?	I0 K−36 B0 Q1		
N4	G1		Z−75			
N5	G1	X80				
N6	G40 G1	X82				
N7						M30

G2



G3



X = **Enddurchmesser**

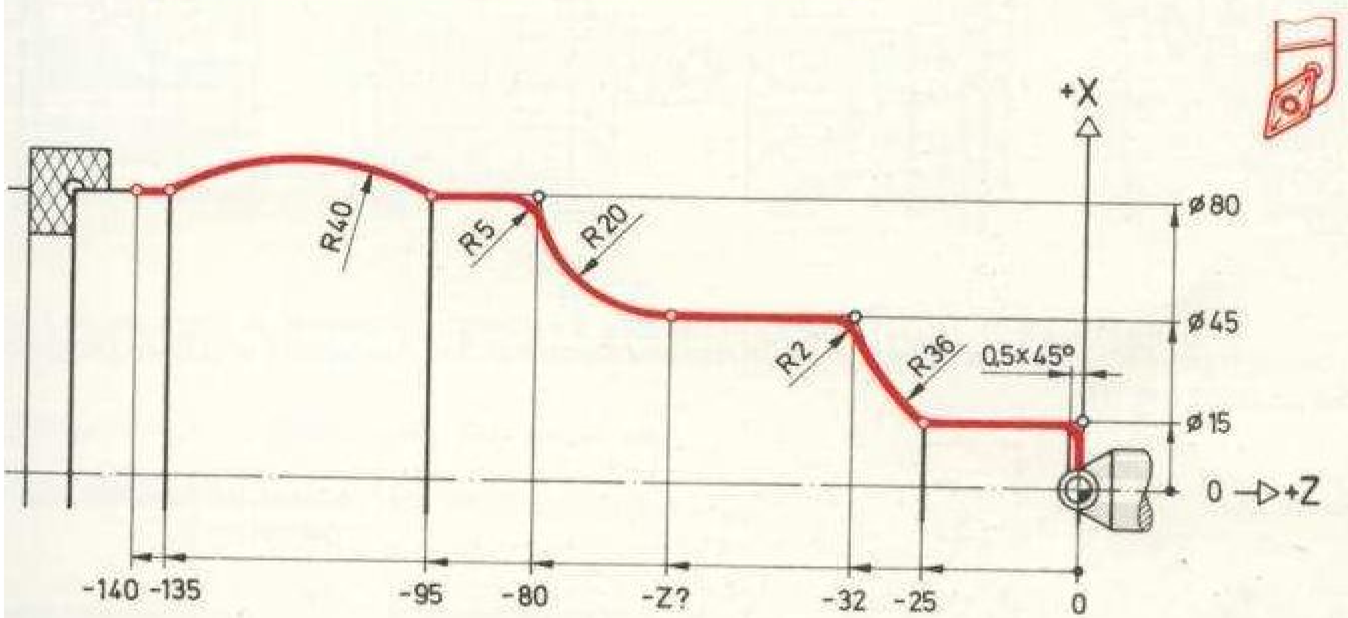
Z = **Endlänge**

R = **Radius-Wert**

B = **tangentiale Verrundung**

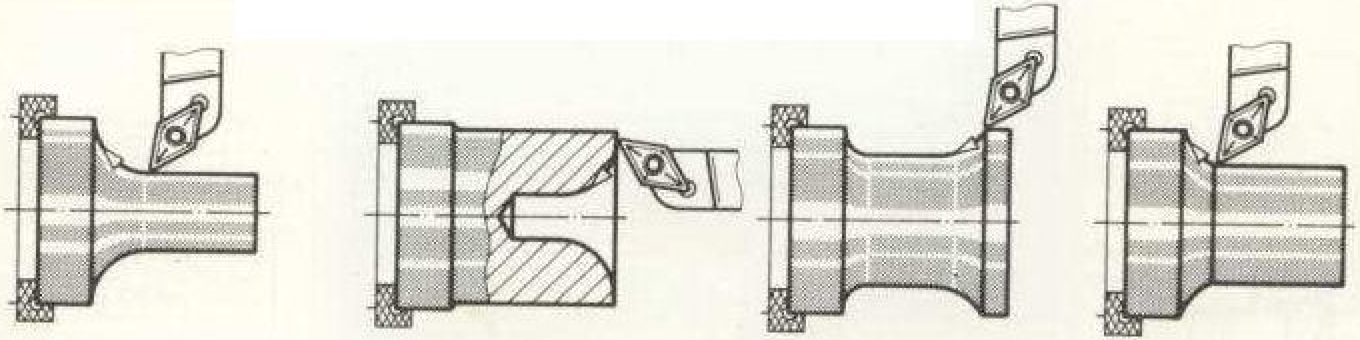
B = 0: Kennzeichen für **nicht**-tangentialen Übergang

Levizja rethore G2/G3 me X,Y,Z dhe B



% 1008								
N	G	X	Z	Hilfsadressen	F	S	T	M
N1	G0	X13	Z2					
N2	G42 G1		Z0					
N3	G1	X15		B−0.5 E0.05				
N4	G1		Z−25	B0				
N5	G2	X45	Z−32	R36 B2				
N6	G1		Z?	A0				
N7	G2	X80	Z−80	R20 B5				
N8	G1		Z−95	B0				
N9	G3	X80	Z−135	R40 B0				
N10	G1		Z−140					
N11	G40 G1	X82						
N12								M30

Percaktimi i gjatesive I dhe K tek G2/G3

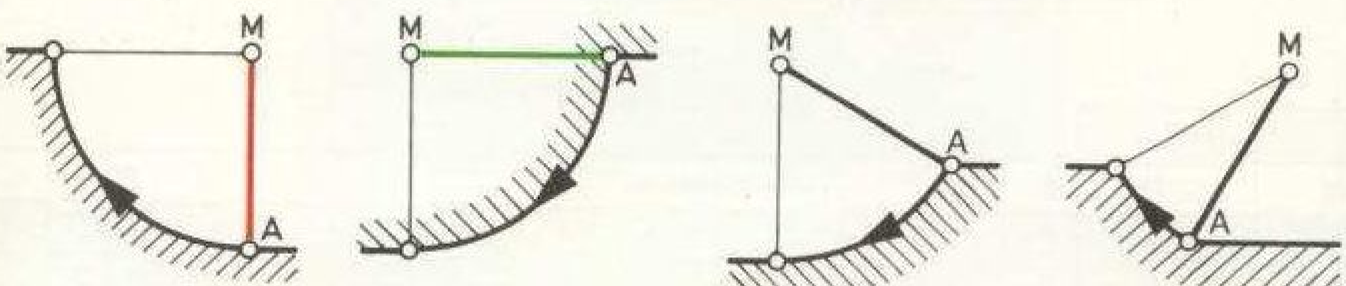


In bestimmten Fällen ist es erforderlich, die Kreisbewegung mit den Adressen **I** und **K** zu programmieren (statt mit R).

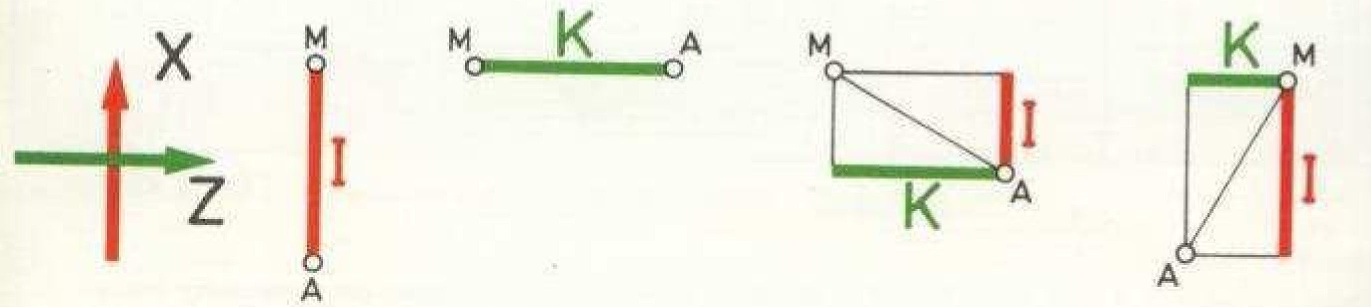
Satzformat: N... G... X... Z... **I***... **K***

* I gehört zu X („I wie Icks")
K gehört zu Z.

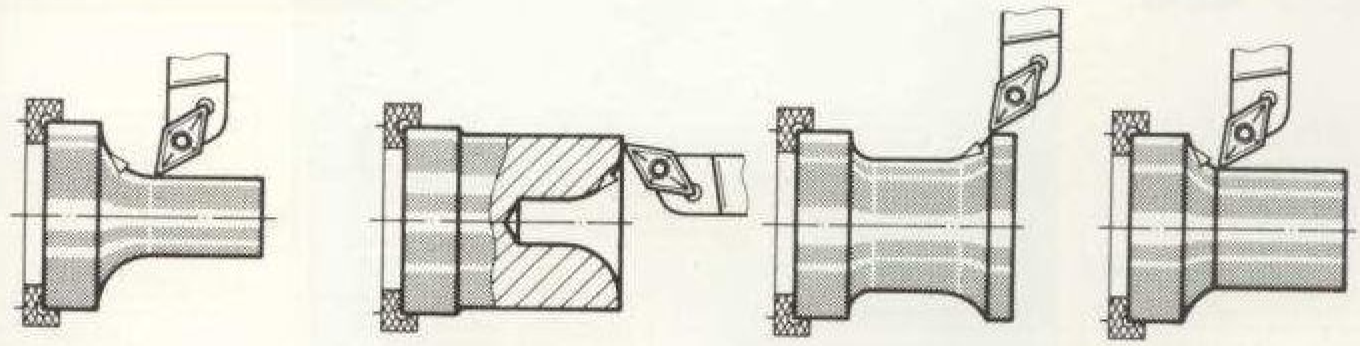
Für **I** und für **K** sind nur der **A**nfangspunkt **A** und der **M**ittelpunkt **M** von Interesse.



Die Längen **I** und **K** sind die **achsparallelen** Abstände in X und in Z Richtung zwischen **A** und **M**.

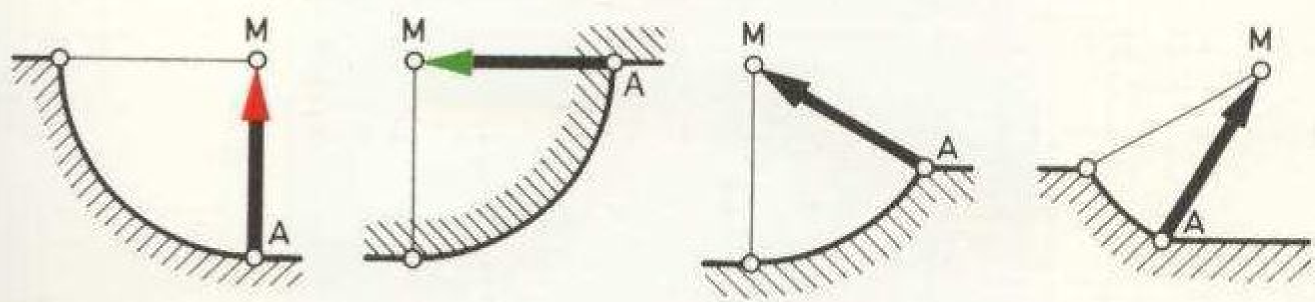


Percaktimi i parashenjes I dhe K tek G2/G3

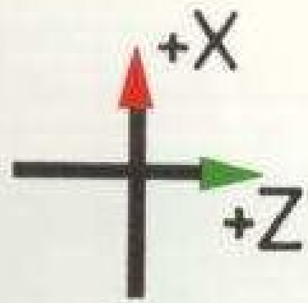


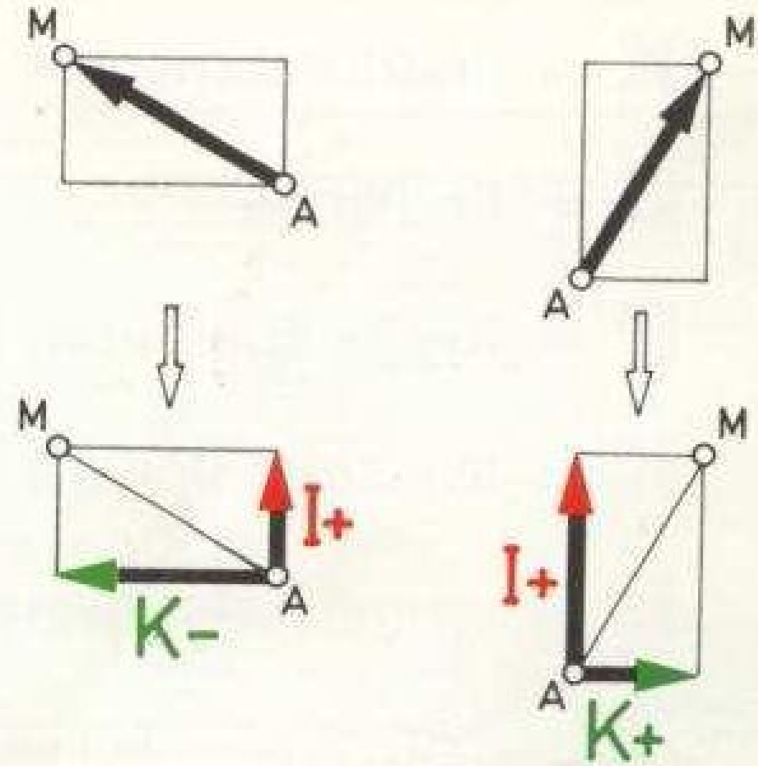
Die Längen **I** und **K** können ein **positives** oder ein **negatives** Vorzeichen haben.

1. Von **A** nach **M** schauen und Richtungspfeil eintragen.

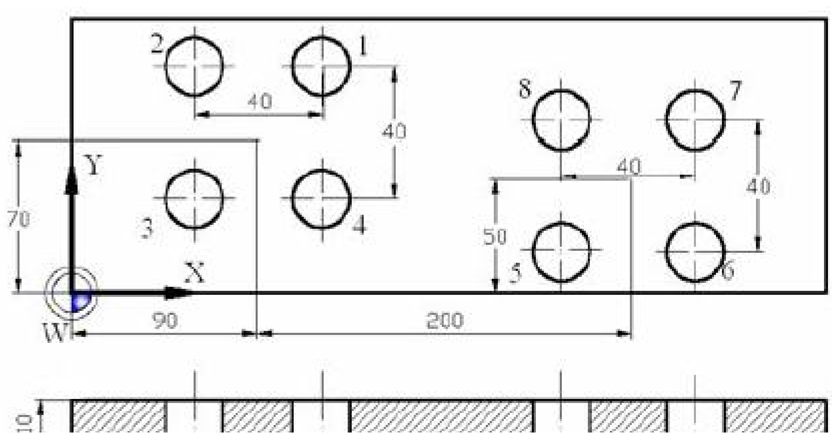


2. Wenn die Strecke $\overline{AM}$ **nicht** senkrecht oder waagerecht verläuft, muß diese Strecke „zerlegt" werden:



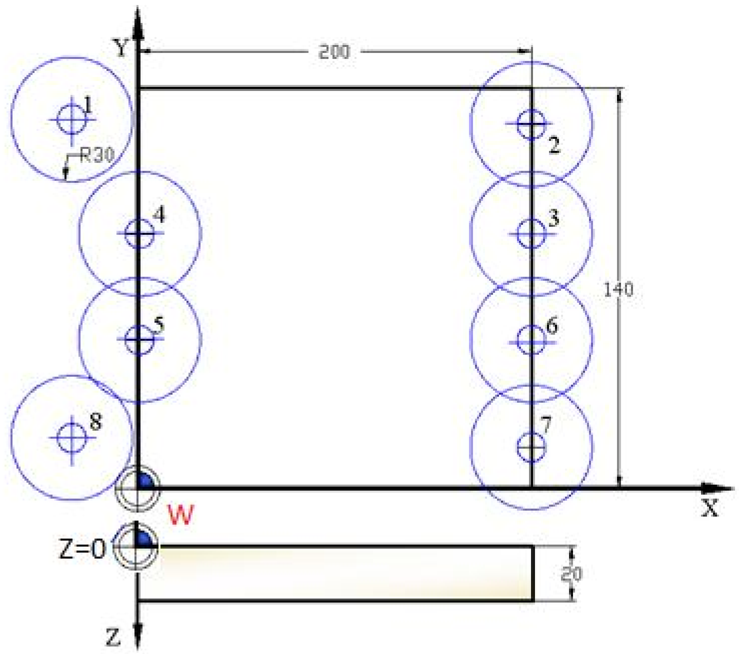


3. Die **Vorzeichen** für **I** und **K** ergeben sich durch den Vergleich mit den Achsenkreuzrichtungen **X** und **Z**.



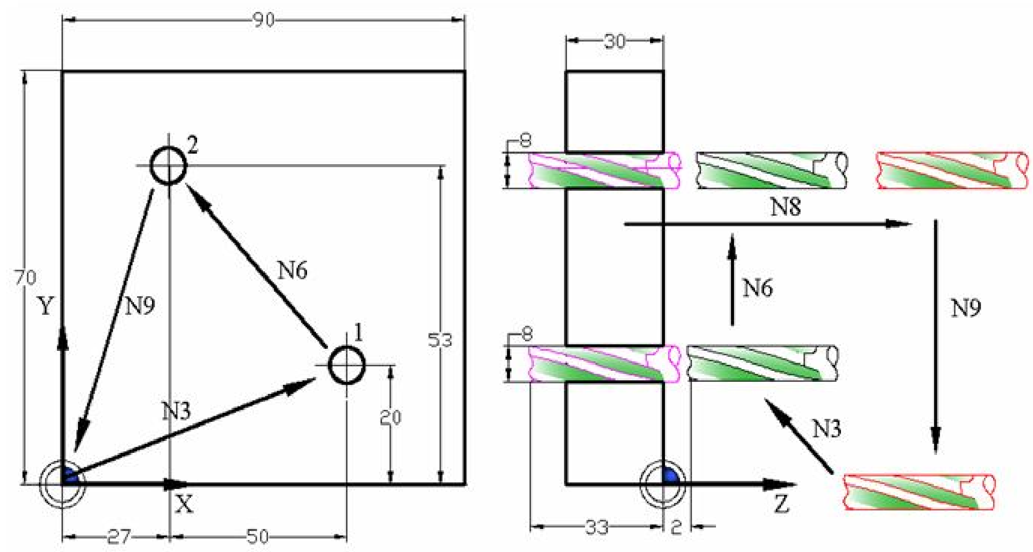

```
%PM

N 9001

N1  G17
    T1                    M6

N2  G81                   Y2          Z-10        F200                    S500

N3  G92    X90        Y70

N4  G79    X20        Y20        Z0

N5  G79    X-20

N6  G79                   Y-20

N7  G79    X20
```

FREZIMI PLANAR



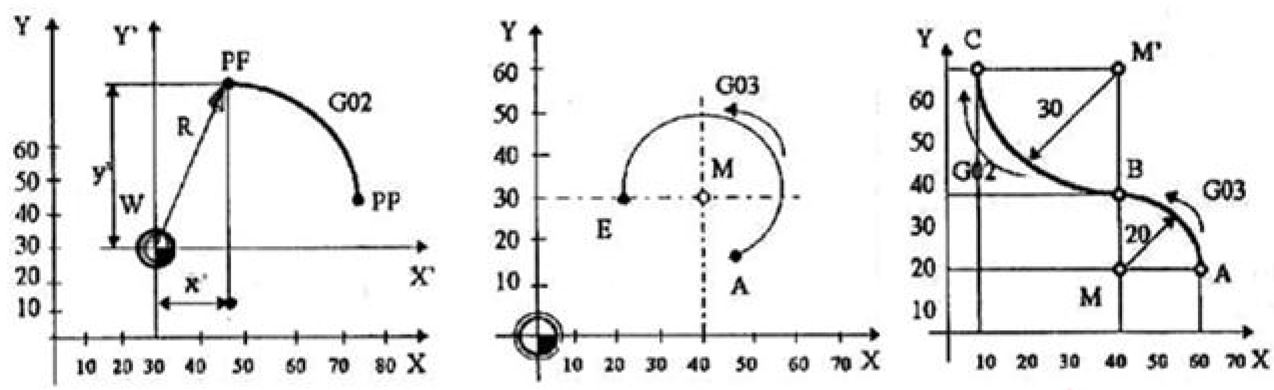
%PM								
N9001								
NI1	GI7							
					T1	M6		
N2	GO	X-35	Y130	Z-10		S500		M3
N3	G1	X200					F300	
N4			Y90					
N5		XO						
N6			Y40					
N7		X200						
N8			Y10					
N9		X-35						
N10	G0			Z10				M30

Shpimi i vrimës 1 dhe 2 në makinën shpuese horizontale



N9001							
N1	G17				TI		
							M66
N2	G54						
N3	(GO)	X65	Y20	Z2		S1200	
							M3
N4	G1			Z-33		F200	
							M8
N5	GO			Z2			
N6	(GO)	X27	Y53				
N7	G1			Z-33			
N8	GO			ZlOO			
N9	(GO)	X0	Y0				
							M30

G2 (G3) - INTERPOLIMI RRETHOR



Shembull: Programimi i harkut rrethor më të madh se 180° (fig.7.11.b).

Gjatë aplikimit të përmasave absolute, harku rrethor do të programohet kështu:

....
N20 G01 X45 Y155 F200
N21 G03 X20 Y30 I40 J30
....

Shpjegim:

N20: pika A është pika fillestare e harkut rrethor .

N21: harku rrethor është më i madh se 180^0 në kahjen e kundërt të akrepave të orës, me pikën e fundme E dhe pikën e qendrës së rrethit M.

Programi i përpunimit gjatë aplikimit të përmasave relative (inkremente) do të jetë:

....
N6 GO1 X45 Y15 F200 (A)
N7 G91
N8 G03 X-25 Y15 I-5 J15 (E)
......

Shpjegim:

N6: pika A, pika fillestare është e programuar me përmasa absolute, që të sqarohet përdorimi i përmasave relative në fjalinë N8.

N7: fillimi i programimit me përmasa relative

N8: një hark rrethor më i madh se 180^0 në kahjen CCW, me pikë të fundme E[(x,y), të cilat janë inkremente në krahasim me pikën A dhe pikë të qendrës së rrethit M(I dhe J janë inkremente në krahasim me pikën A) d.m.th.

I-5 = (40-45)

J15 = (30-15)

276.3.1. RREZJA RRETHORE ME KËND 90°

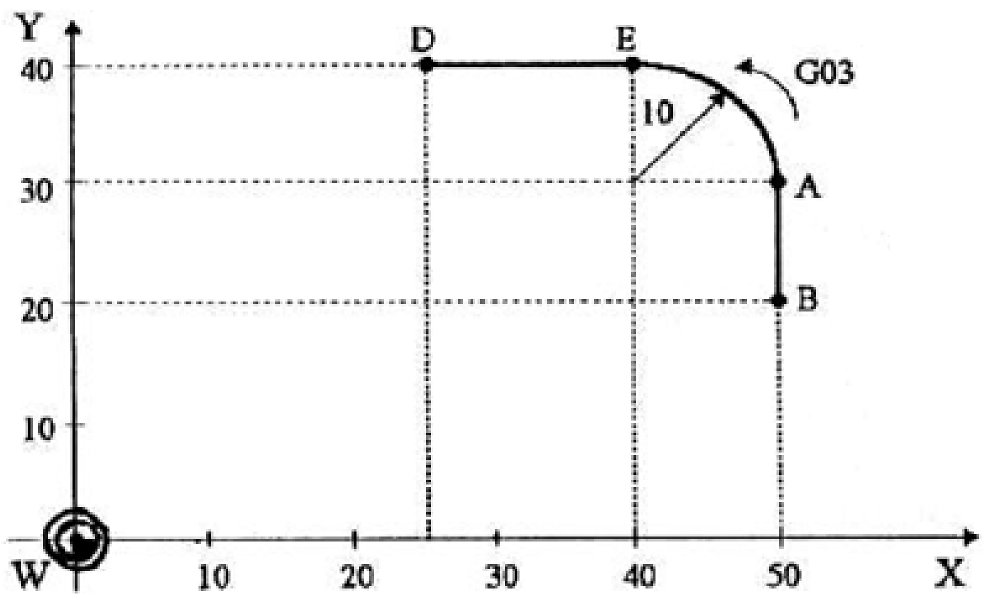
Rrezja e një harku rrethor më të vogël se 180^0 mundë të programohet pa parametra të interpolimit, me fjalë të adresës R (fig.7.12).

Programi:

N1 G0 X50 Y20
N2 G1 Y30 F200
N3 G3 X40 Y40 R10
N4 G1 X25

Sqarim:

N1: pika B është pika fillestare.
N2: pika A është pika e fundit e drejtëzës dhe pika fillestare e lakores rrethore.
N3: lakorja rrethore në kahjen e kundërt të akrepit të orës (G03)me pikën e fundit E dhe rreze R10.
N4: pika e fundit e rrafshit D.



7.6.3.2. RRETHI I PLOTË

Rrethi plotë realizohet duke programuar koordinatat e pikës së qendrës së rrethit. Pika fillestare (PF) d.m.th. pika në të cilën ndodhet instrumenti, është në të njëjtën kohë edhe pika përfundimtare (PP).

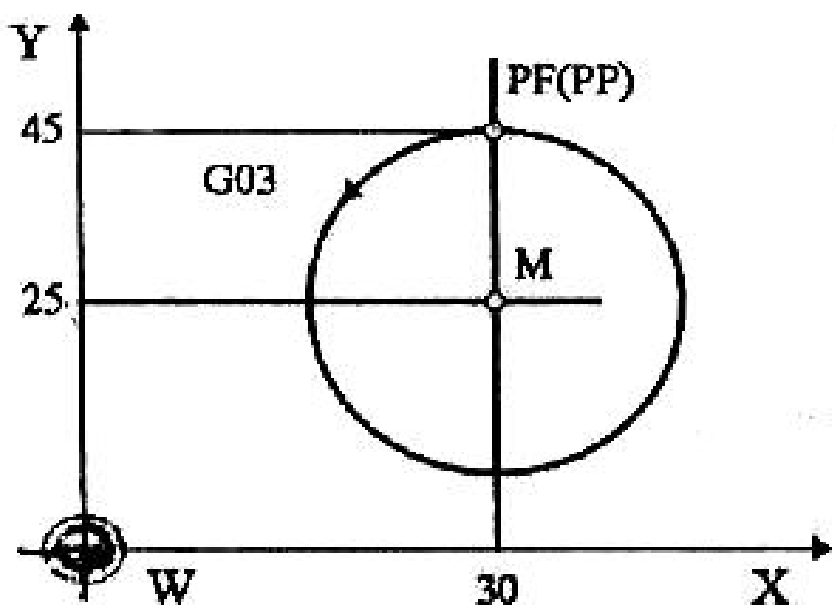
Rrezja llogaritet automatikisht nga njësia dirigjuese prej pikës fillestare dhe pikës se qendrës së rrethit.

Fig.7.13. Programimi i rrethit.

Shembull: Programimi i rrethit sipas (fig. 7. 13).

Programi

N1	G0	G90	X30.	Y45.					S2000
				M3					
N2	G3			X30	Y45	I30	J25	FI00	
				M8					
N3	G0			X0	Y0				



7.6.4. G04-KOHA E VONESËS

Koha e vonesës programohet direkt në sekonda me adresën F ose X. Fjalia e ardhshme kryhet pas kalimit të vonesës së programuar kohore.

Funksioni G04 është aktiv vetëm në një fjali dhe sipas nevojës mund të programohet përsëri.

Në fjalinë me vonesë kohore përveç funksionit G04 shkruhet edhe koha e programuar e vonesës d.m.th. stagnimi kohor i kufizuar i lëvizjes ndihmëse.

Për shembull 10 sekonda të vonesës kohore programohet në këtë mënyrë:

N100 G04 X1000, ose
N100 G04 F1000

7.7. FUNKSIONET E KORREKTURËS SË IMP
7.7.1. ZGJEDHJA E RRAFSHEVE

Tek makinat MAHO është e mundur që më përdorimin e një koke vertikale të frezës të gjendet instrumentit në një aks tjetër të lëvizjes. Për programimin e pjesëve mbetet konfiguracioni i akseve i pa ndryshuar, sepse shprehet për mes funksioneve G17, G18, G19 në të cilin aks gjendet instrumenti.

Dirigjimi përdor këtë funksion për llogaritjen e korrekturës të gjatësisë dhe ciklit të punës. Rrafshi i korrekturës të rrezes qëndron vertikalisht ndaj aksit të instrumentit.

Funksionet G17, G18, G19 përbëjnë një grup gjatë të cilës në fjalinë programuese mund të jetë i veprueshëm vetëm një funksion. Te aktivizimi i dirigjimit automatikisht aktivizohet G17. Pas urdhrit të një G-funksioni tjetër automatikisht anulohet korrektura e gjatësive në aksin paraprak, e realizohet në aksin e definuar më vonë.

Në asnjërin nga të dy akset nuk pason asnjë lëvizje vepruese.

G-funksioni	*Boshti i instrumentit*	*Rrafshi i korrekturës të rrezes*
G17	Z-boshti	XY-rrafshi
G18	Y-boshti	XZ-rrafshi
G19	X-boshti	YZ-rrafshi

Që të bihet instrumenti në pozitën e nevojshme duhet të merren parasysh dimensionet vertikale të kokës së frezës.

Kjo mund të rrjedh në këtë mënyrë:

- me ndihmën e zhvendosjes së pikës zero (G92 ose G93)
- me ndihmën e zhvendosjes së memoruar të pikës zero G54 deri në G57.

Shembull: Vrimat P_1 në rrafshin XY dhe P_2 në rrafshin XZ duhet të përpunohen (pa përdorimin e cikleve punuese). Përdoret koka vertikale e frezës, instrumenti i së cilës mund të lëviz ose në aksin X ose në aksin Y (fig.7.14).

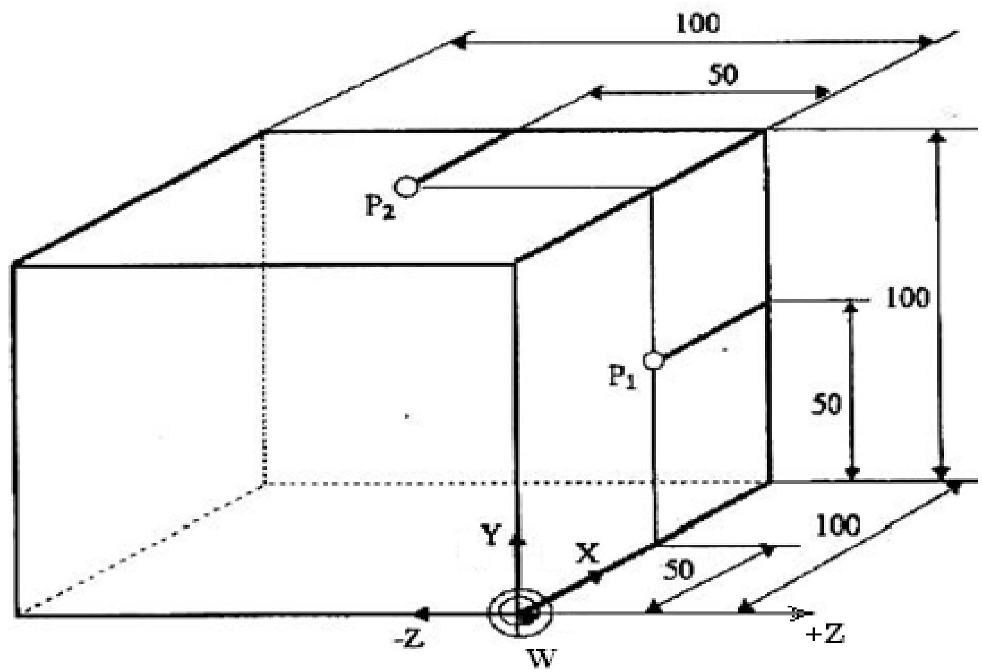
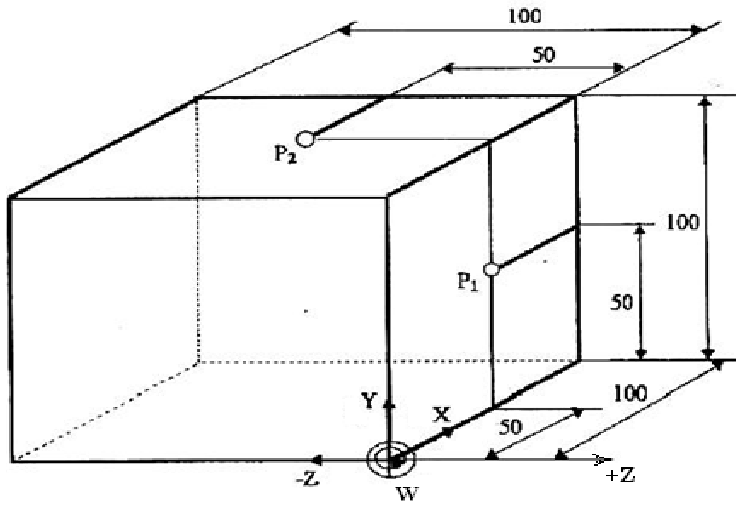


Fig.7.14.

Programimi i përpunimit mund të duket kështu:

```
%PM
N9001
N1    G17                               T1                   M6
N2    G0     X50    Y50    Z2
                                                  S1000      M3
N3    G1                   Z-10               F200

N4    G0                   Z100
N5                  Y200
                                                             M6
N6    G18                              T2                    M6
N7    G55
                                                             M3
N8    G0     X50    Yl02   Z-50
N9    G1            Y90
N10   G0            Y200                                     M30
```



Shpjegim:

N1: shtrëngohet instrumenti T1 i cili gjendet në aksin Z

N2: afrimi kah pika P_1

N3: zhvendosja (lëvizja) me hap me thellësinë aksin Z

N4: kthimi i instrumentit nga vrima

N5: kthimi i instrumentit në aksin Y ashtu që të mund të vijë në veprim koka vertikale, ndërsa programi ndalet.

N6: shtrëngohet instrumenti T2 i cili gjendet në aksin Y

N7: zhvendosja e memoruar e pikës zero që të mund të merret parasysh përmasat e ndryshuara a_1 dhe a_2 të kokës frezuese vertikale. Vlerat gjegjëse memorohen në dirigjim para fillimit të programit dhe mbesin të vendosura gjithnjë deri sa mos të anulohen.

N8: pika P_2 në rrafshin XZ aktivizohet.

N9: instrumenti zhvendoset me hap në thellësinë aksin Y
N10: instrumenti kthehet mbrapa prej vrimës.

7.7.2. KORREKTURA E RREZES SË FREZËS (G40, G41, G42)

Gjatë konsumit, mprehjes ndryshon diametri i frezës.

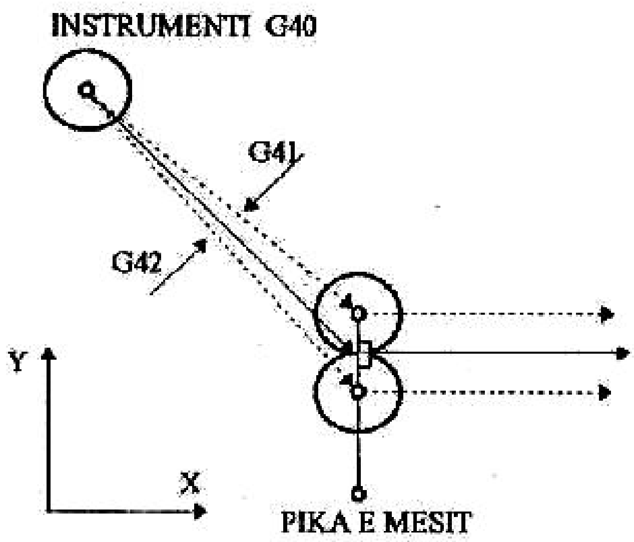
Madhësinë e këtij ndryshimi programuesi nuk mundë t'i parasheh më herët, ashtu që me ndihmën e funksionit për korrekturë G41 dhe G42 eliminohet dallimi në mes të diametrit të programuar dhe atij real.

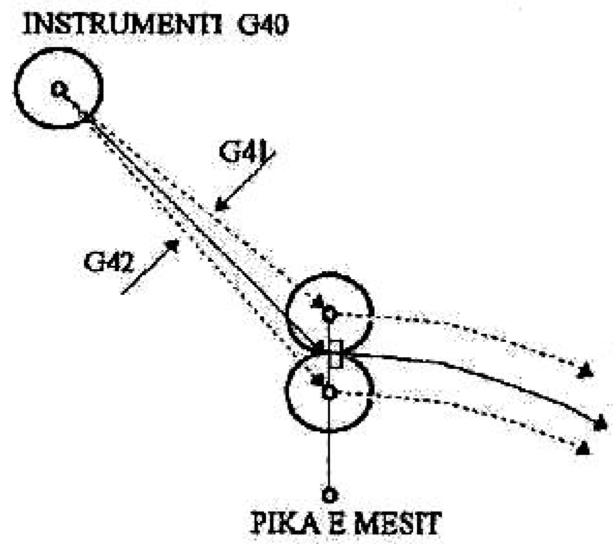
Trajektorja e korrekturës së frezës është paralele me atë të programuarën.

Nëse trajektorja e qendrës së frezës, shikuar në kahjen e lëvizjes gjendet majtas nga sipërfaqja e përpunuar, atëherë përdoret funksioni G41 gjegjësisht G42 nëse freza është djathtas nga sipërfaqja e përpunuar.

Gjatë thirrjes së parë të korrekturës së rrezes së frezës duhet të definohet numri gjegjës i korrekturës me adresën (D…).

Vlera e ndryshimit ose madhësia absolute e diametrit, ose të gjatësisë së instrumentit vihet në grupin e ndërprerësve dekad ose bartet përmes tastierës së sistemit dirigjues në memorien e korrekturës se instrumentit.





c) G17 G01 F ... ;

d) G17 G01 F ... ;

a) G17 G01 F ... ;

b) G17 G01 F ...

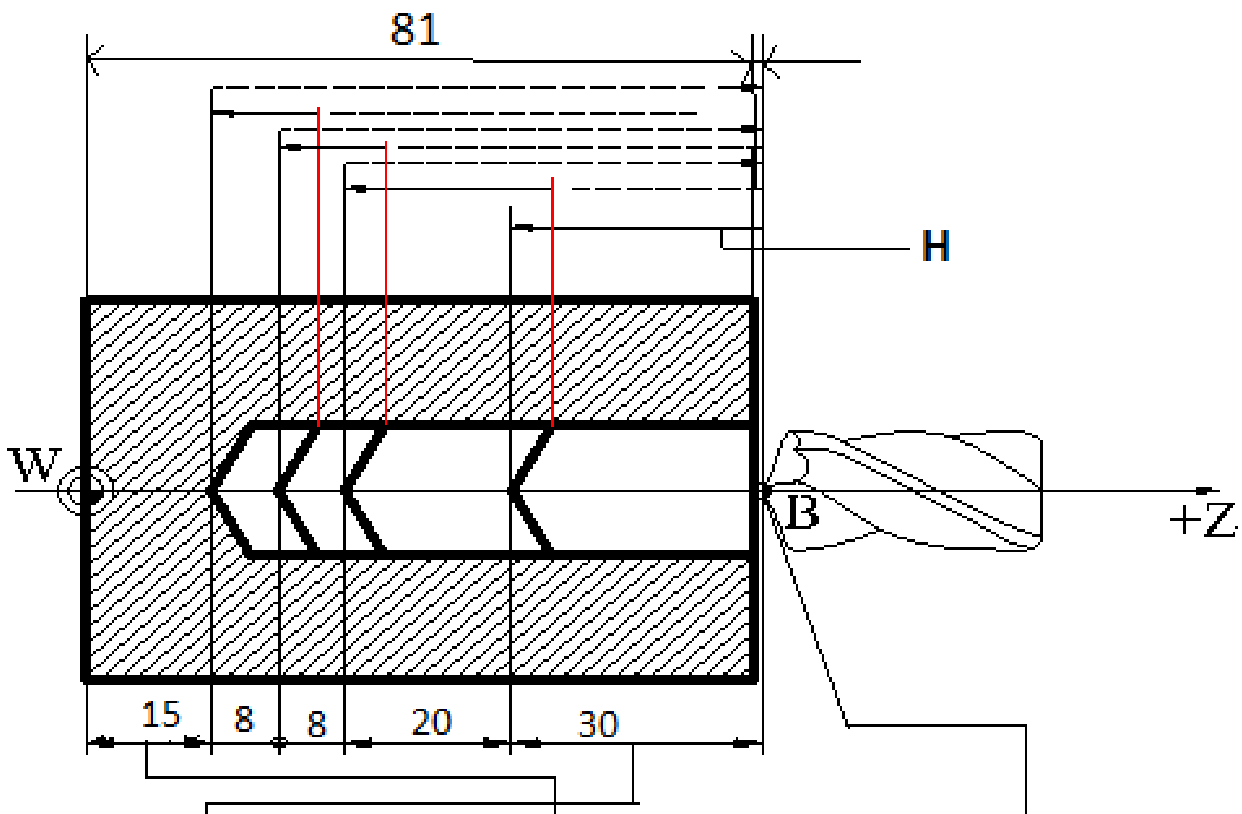
G41 (G42) D... X ... Y ...

G41 (G42) D... X ... Y... ;

X ... ;

G02 X... Y... J...;

CIKLI I SHPIMIT TË THELLË ME TORNO – L98



N10	R25 30	R22 81	R26 15	R24 10	R27 0,5	R28 1

R25 – thellesia fillestare e shpimit
R22 – pika fillestare
R26 – thellesia perfundimtare
R24 – ndryshimi
R27 – vonesa ne piken fillestare
R28 – vonesa ne piken e shpimit

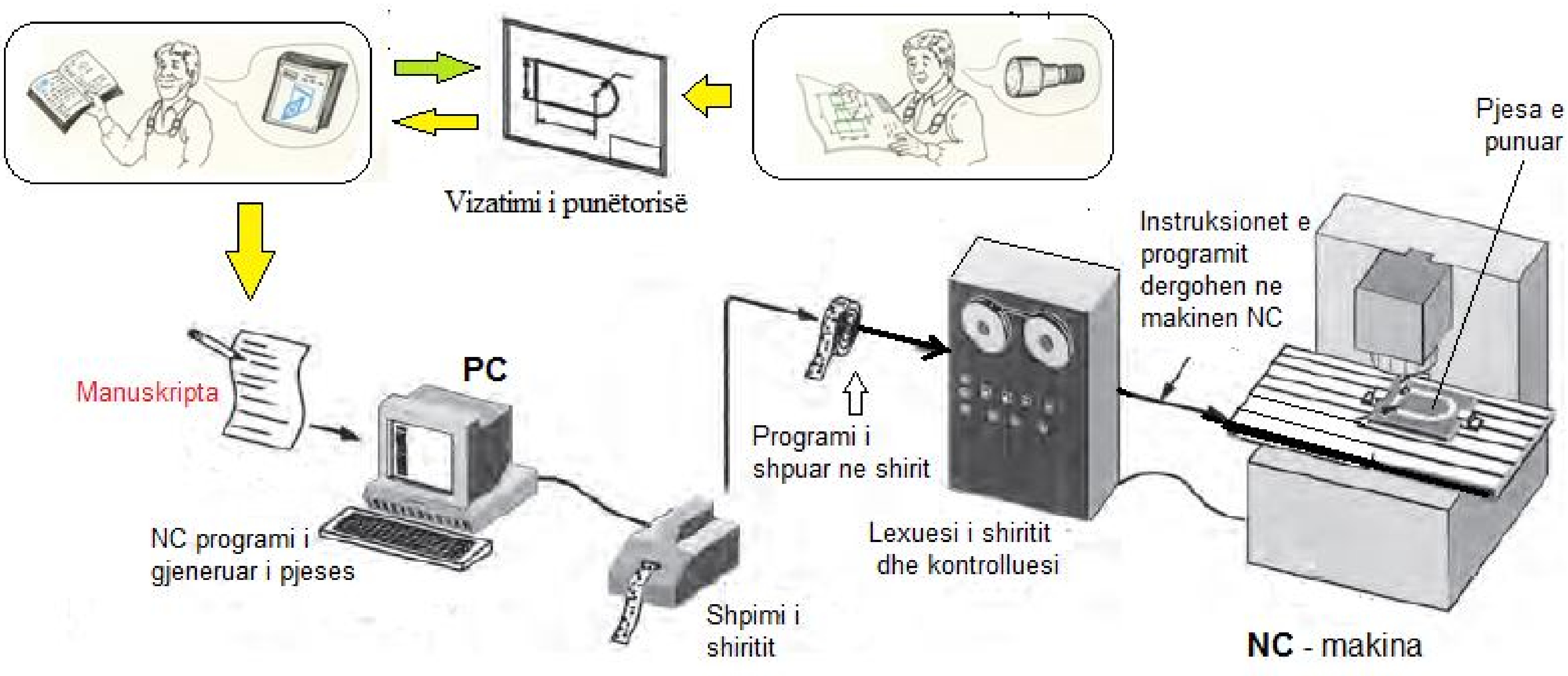
1.3 NUMERICAL CONTROL DEFINITION

NC has been used in industry for more than 40 years.

Simply put, NC is a method of automatically operating a manufacturing machine based on a code of letters, numbers, and special characters.

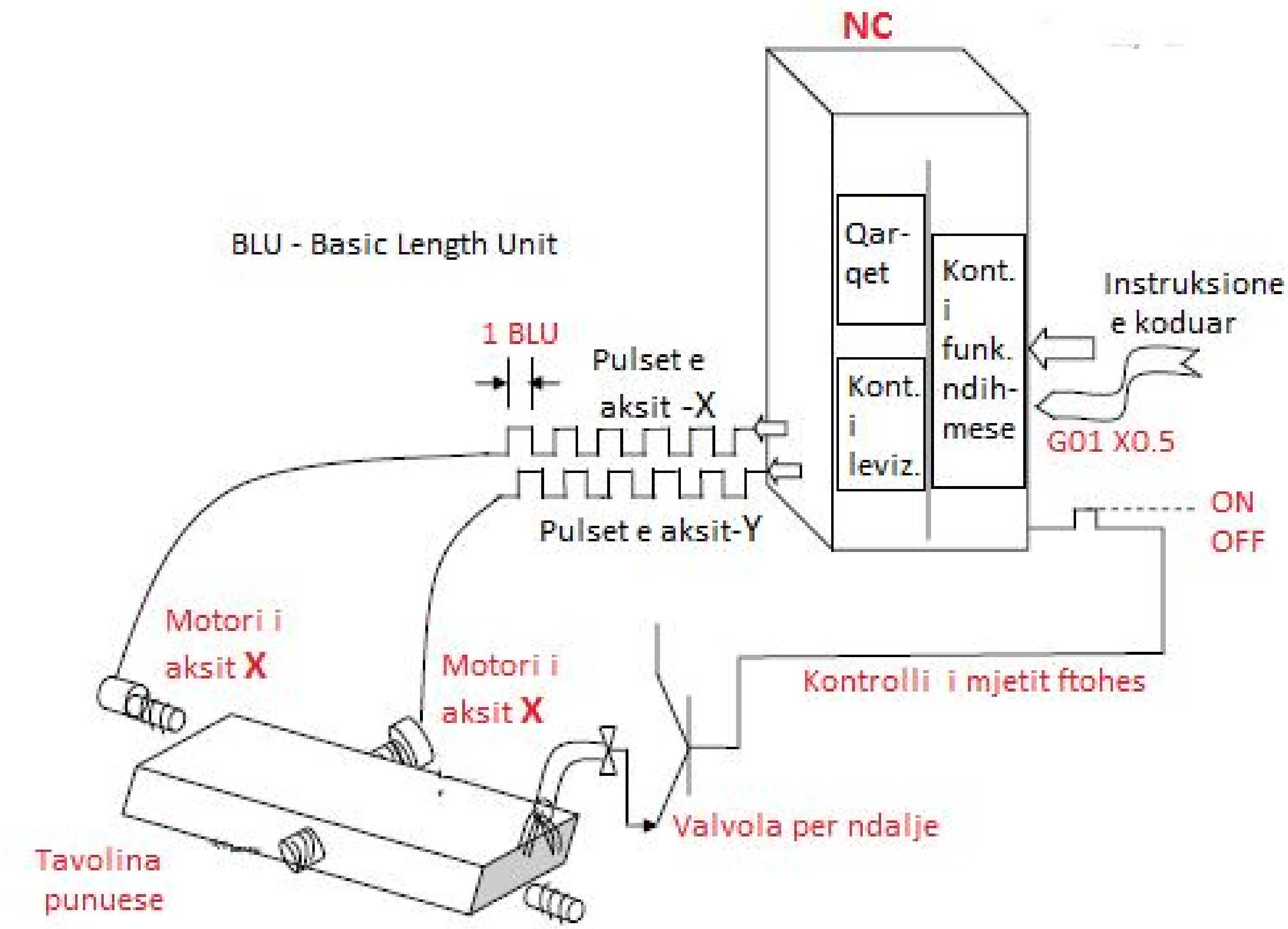
A complete set of coded instructions for executing an operation is called a program.

1.3 NUMERICAL CONTROL DEFINITION



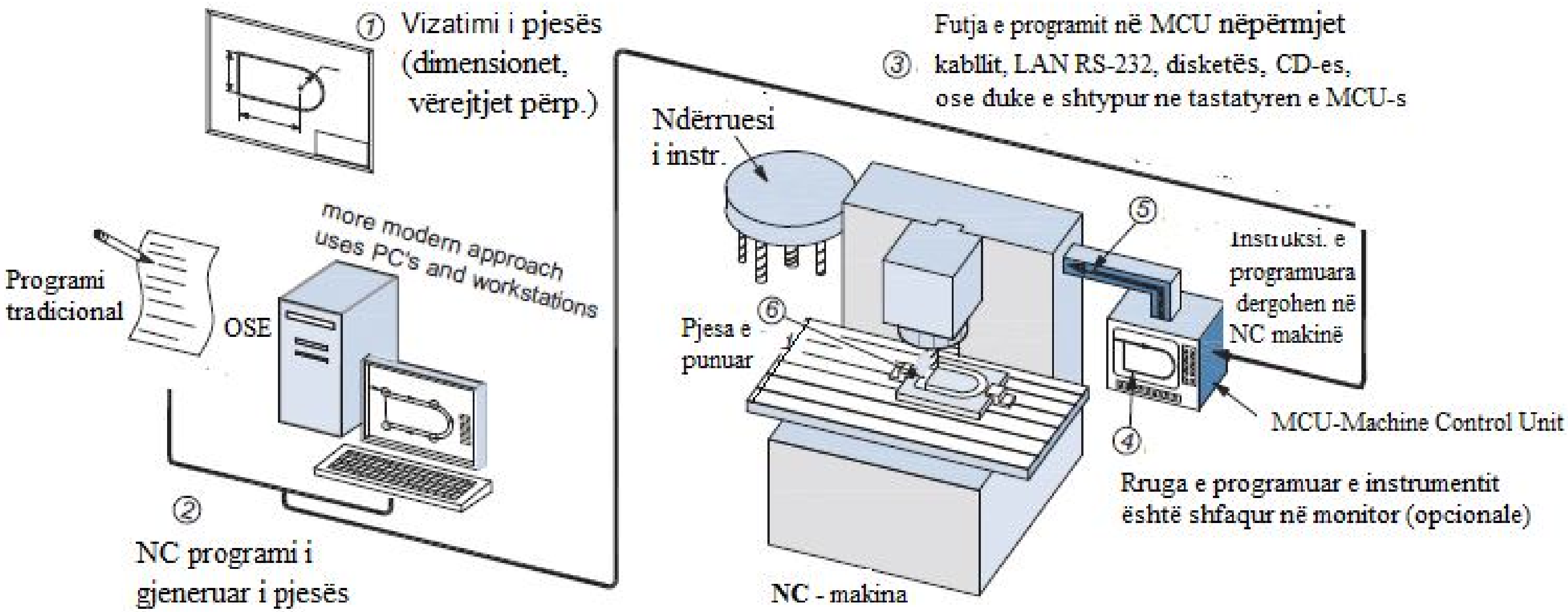
Komponentet e NC- sistemeve tradicionale

1.3 NUMERICAL CONTROL DEFINITION



The program is translated into corresponding electrical signals for input to motors that run the machine.

1.3 NUMERICAL CONTROL DEFINITION

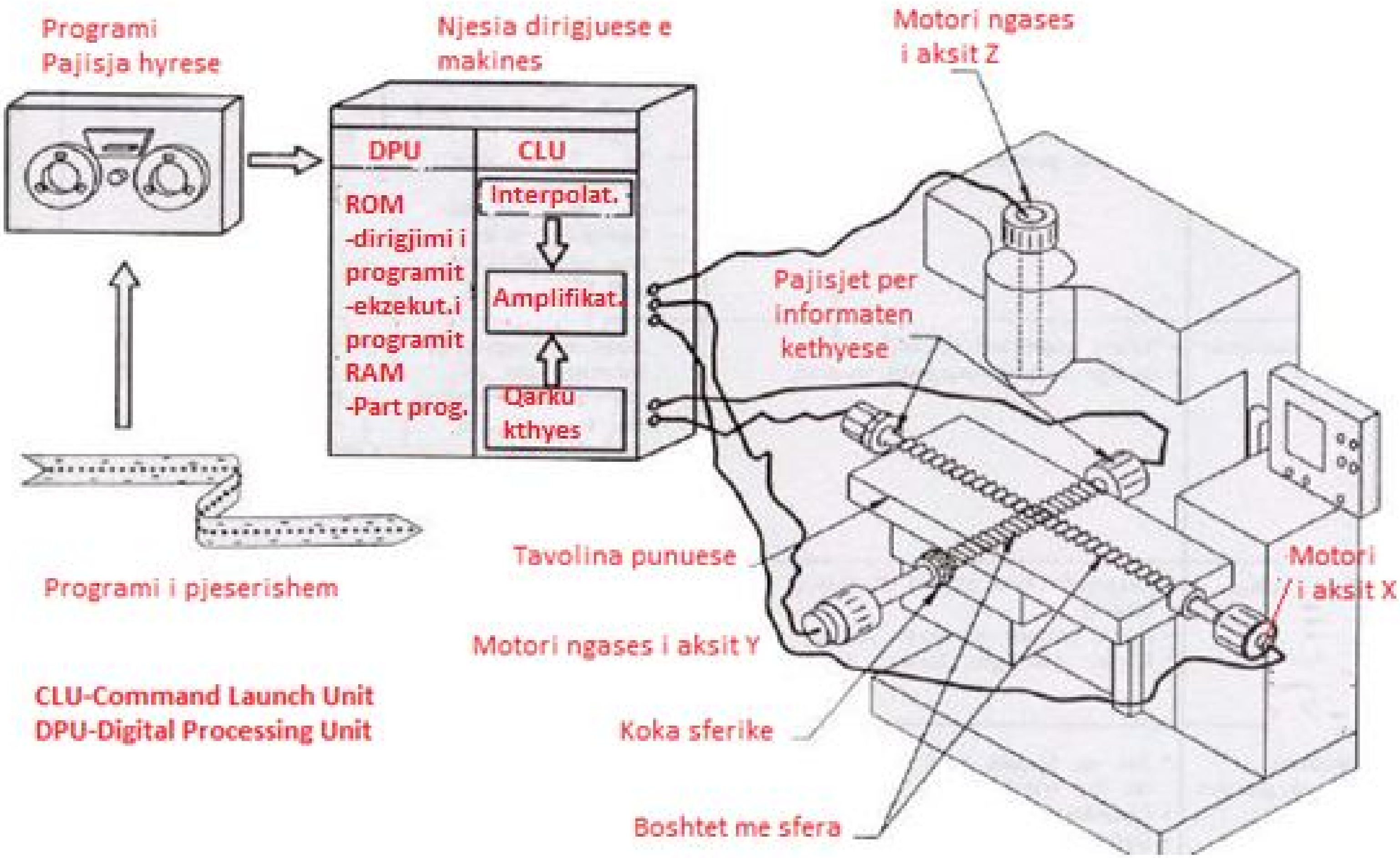


Komponentet e CNC sistemeve moderne

NC machines can be programmed manually.

If a computer isused to create a program, the process is known as computer-aided programming (CAP ose CNC).

1.3 NUMERICAL CONTROL DEFINITION



NC SYSTEMS

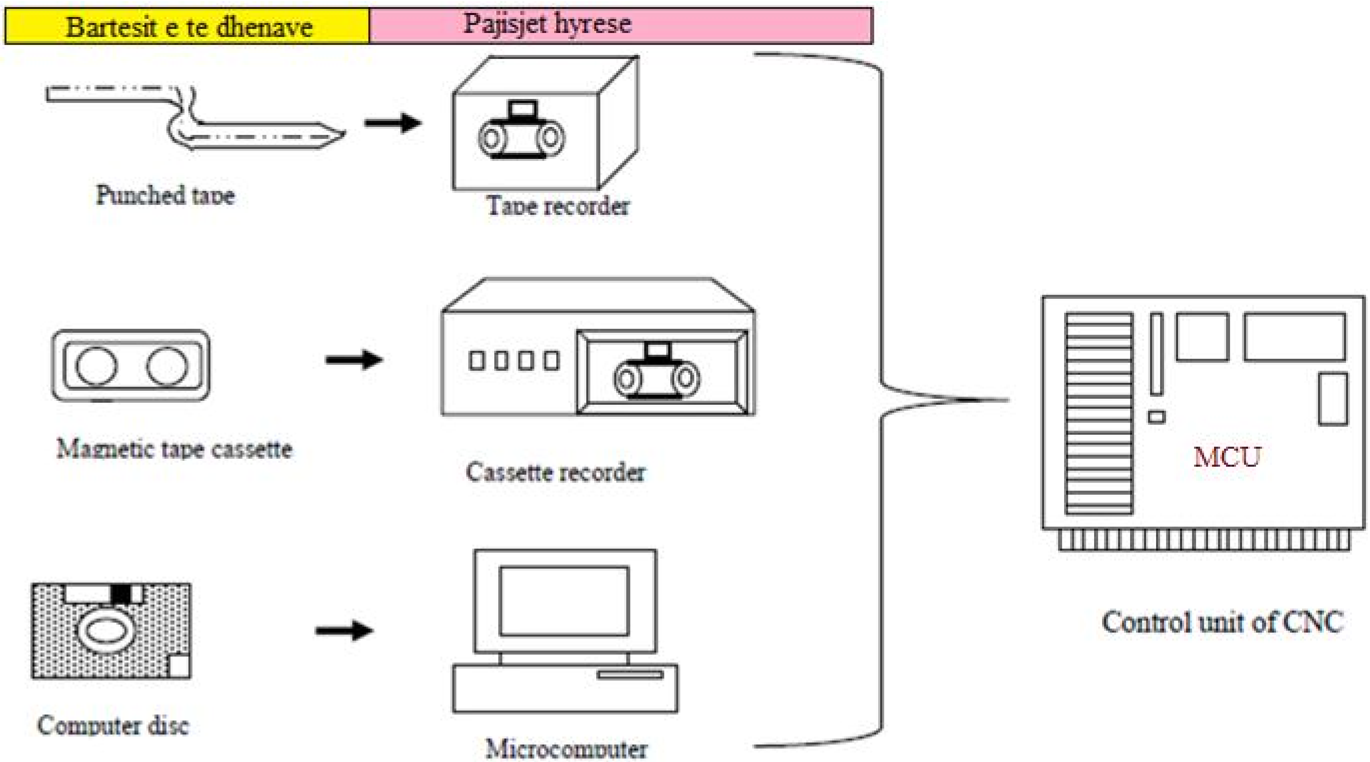
Traditionally, NC systems have been composed of the following components:

***Tape punch*:** converts written instructions into a corresponding hole pattern.

The hole pattern is punched into tape which is passed through the tape punch.

Much older units used a typewriter device called a Flexowriter, and later devices included a microcomputer coupled with a tape punch unit.

***Tape reader*:** reads the hole pattern on the tape and converts the pattern to a corresponding electrical signal code.



Bartesit e te dhenave
Pajisjet hyrese
Punched tape
Tape recorder
Magnetic tape cassette
Cassette recorder
Computer disc
Microcomputer
MCU
Control unit of CNC

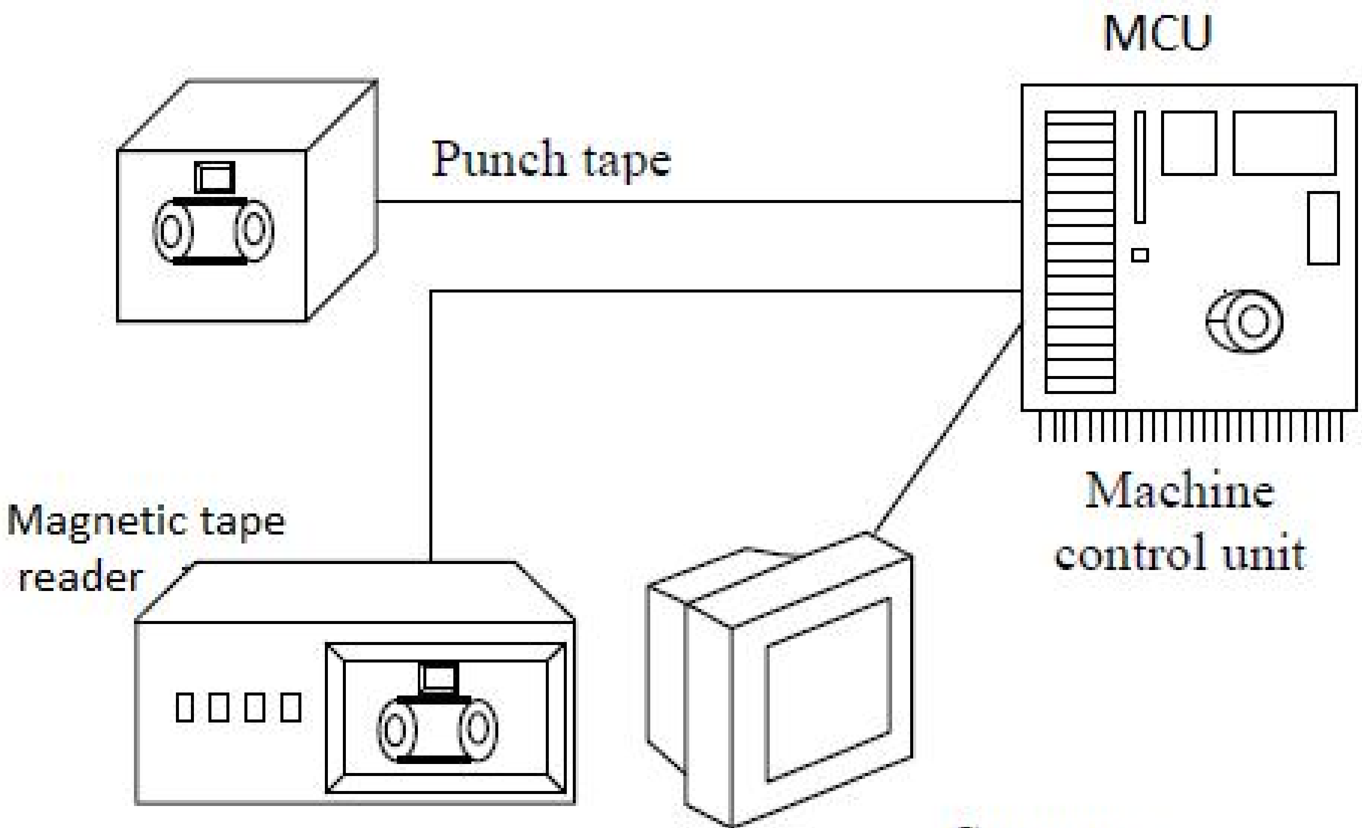


Figure 5.5: Program input device

NC SYSTEMS

Controller: receives the electrical signal code from the tape reader and subsequently causes the NC machine to respond.

NC machine: responds to programmed signals from the controller.

Accordingly, the machine executes the required motions to manufacture a part (spindle rotation on/off, table and or spindle movement along programmed axis directions, etc.). See Figure 1.1.

ADVANTAGES OF NC SYSTEMS COMPARED WITH MANUAL PRODUCTION METHODS

NC systems offer some advantages over manual production methods:

1. Better control of tool motions under optimum cutting conditions.

2. Improved part quality and repeatability.
3. Reduced tooling costs, tool wear, and job setup time.
4. Reduced time to manufacture parts.
5. Reduced scrap.
6. Better production planning and placement of machining operations in the hands of engineering.

1.4 DEFINITION OF COMPUTER NUMERICAL CONTROL (CNC) AND ITS COMPONENTS

A CNC machine is an NC machine with the added feature of an onboard computer.

The onboard computer is often referred to as the machine control unit or MCU.

Control units for NC machines are usually hardwired, which means that all machine functions are controlled by the physical electronic elements that are built into the controller.

The onboard computer, on the other hand, is "soft" wired, which means the machine functions are encoded into the computer at the time of manufacture, and they will not be erased when the CNC machine is turned off .

Computer memory that holds such information is known as ROM or read-only memory.

1.4 DEFINITION OF COMPUTER NUMERICAL CONTROL (CNC) AND ITS COMPONENTS

The MCU usually has an alphanumeric keyboard for direct or manual data input (MDI) of part programs.

Such programs are stored in RAM or the random-access memory portion of the computer.

They can be played back, edited, and processed by the control.

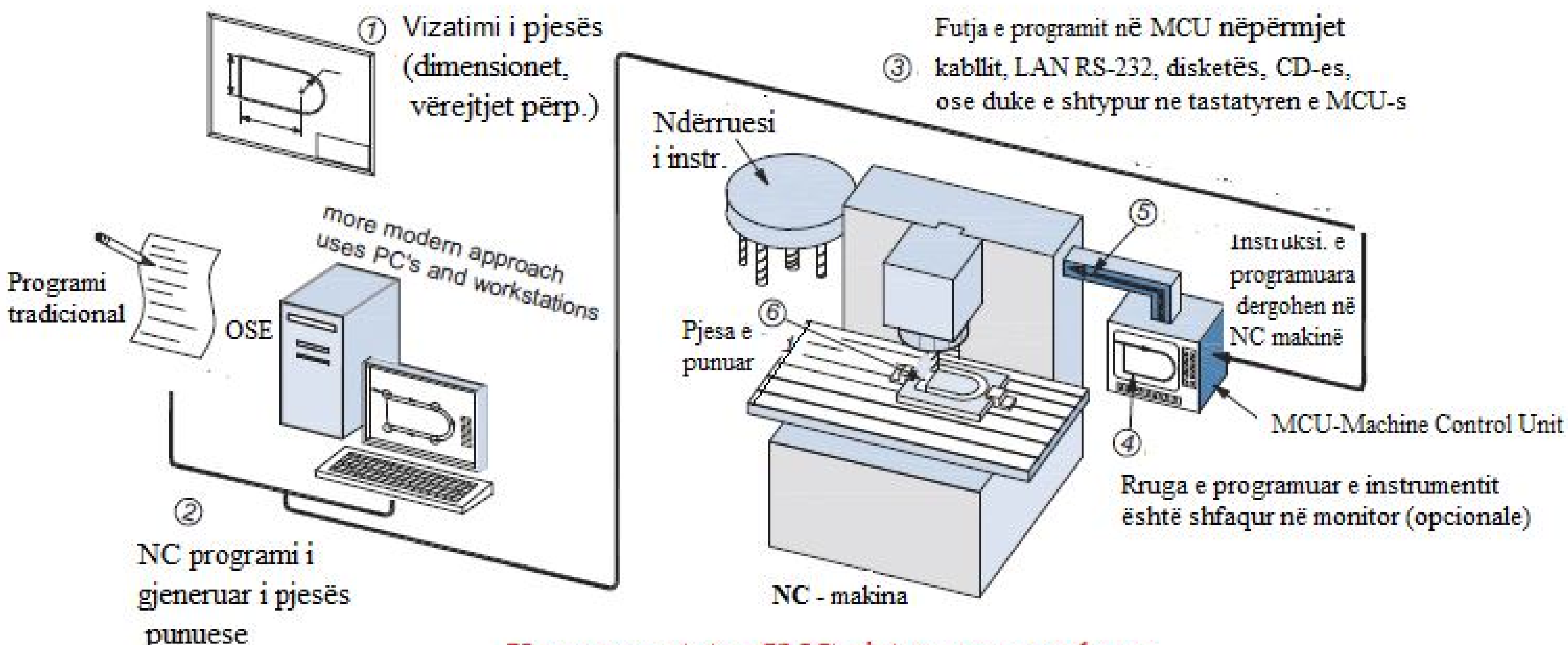
All programs residing in RAM, however, are lost when the CNC machine is off.

These programs can be saved on auxiliary storage devices such as , magnetic tape, or magnetic disk.

Newer MCU units have graphics screens that can display not only the CNC program but the cutter paths generated and any errors in the program.

The components found in many CNC systems are shown in Figure 1.2.

1.4 DEFINITION OF COMPUTER NUMERICAL CONTROL (CNC) AND ITS COMPONENTS



Komponentet e CNC sistemeve moderne

5.3.3 Machine Control Unit –(MCU)

The machine control unit (MCU) is the backbone of CNC systems .

Following six functions are being done by MCU:

1. Read coded instructions
2. Decode coded instructions
3. Implement interpolations to generate axis motion commands
4. Feed axis motion commands to amplifier circuits to drive axis mechanisms
5. Receive the feed back signals of position and speed for each drive axis
6. Implement auxiliary control functions such as coolant ON/OFF, spindle ON/OFF, and tool change.

(See Figure 1.3.)

5.3.3 Machine Control Unit –(MCU)

FIGURE 1.3 A modern Machine Control Unit.-MCU

1.5 ADVANTAGES OF CNC COMPARED WITH NC

CNC opens up new possibilities and advantages not offered by older NC machines.

1. Reduction in the hardware necessary to add a machine function.New functions can be programmed into the MCU as software.
2. The CNC program can be written, stored, and executed directly at the CNC machine.
3. Any portion of an entered CNC program can be played back and edited at will. Tool motions can be electronically displayed upon playback.
4. Many different CNC programs can be stored in the MCU.

1.5 ADVANTAGES OF CNC COMPARED WITH NC

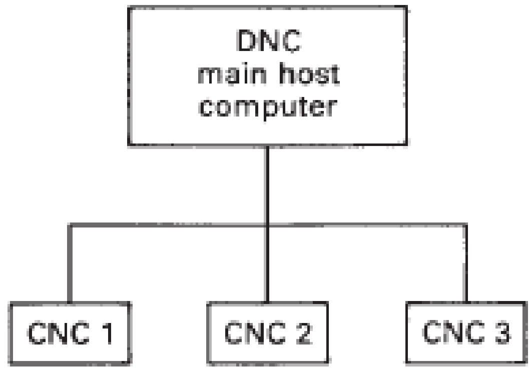
5. Several CNC machines can be linked together to a main computer.
Programs written via the main computer can be downloaded to any CNC machine in the network.

This is known as direct numerical control or . See Figure 1.4.

6. Several DNC systems can also be networked to form a large distributive numerical control system. Refer to Figure 1.5.

7. The CNC program can be input from flash or floppy disks or downloaded from local area networks.

DIRECT NUMERICAL CONTROL - DNC



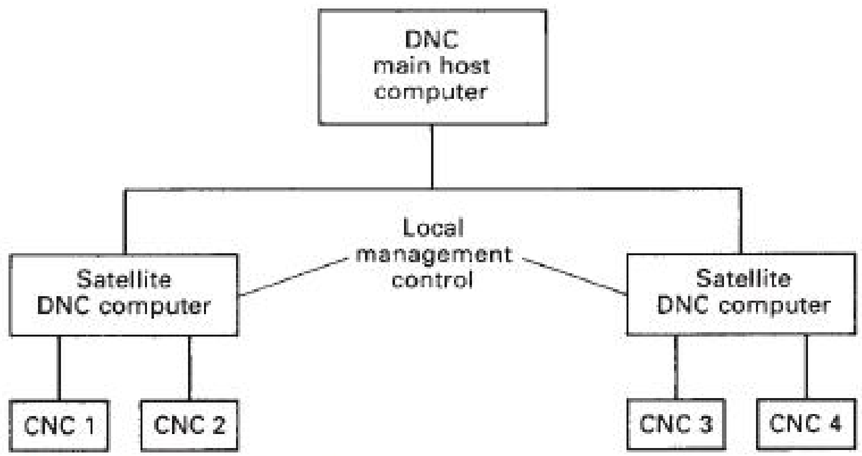


FIGURE 1.5 Distributive numerical control.

DIRECT NUMERICAL CONTROL - DNC

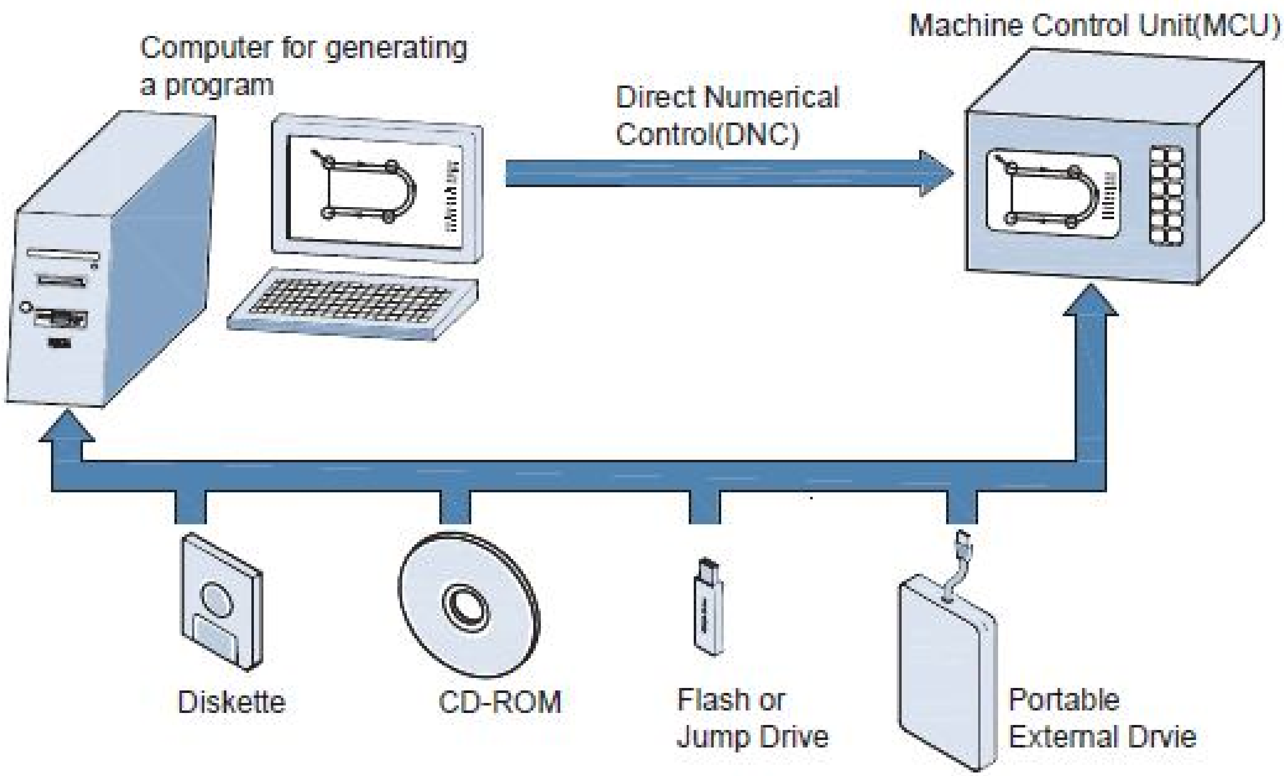


FIGURE 1.11 Modern CNC input and storage methods.

5.5 CONTROL SYSTEMS OF A CNC

5.5.1 Point-to-Point System

CNC controls can be either a point-to-point or continuous path system.

The point –to-point (PTP) control moves the tool to the programmed point, normally in rapid traverse, without engaging the workpiece.

The PTP system is also called the positioning system because its exact tool path normally cannot be controlled.

PTP tool path follow one of the three modes:

-axial path,

-45^0 line and l

-linear path. Figure 5.7 represents these three modes.

Figure 5.7 (a): Axial path

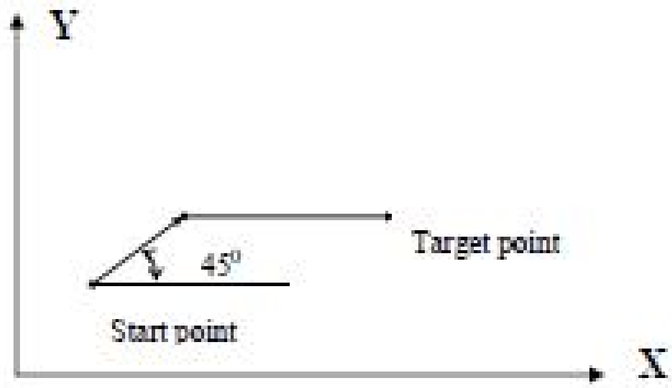


Figure 5.7 (b): 45⁰ Line path

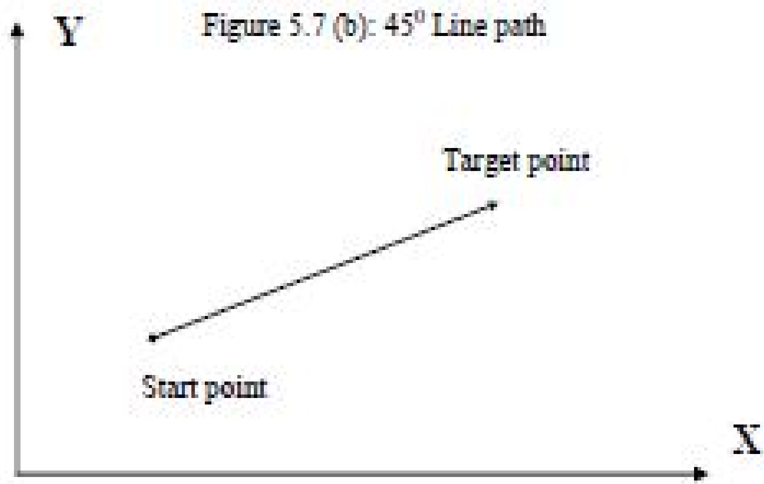


Figure 5.7 (c): Linear path

Fig. 5.7

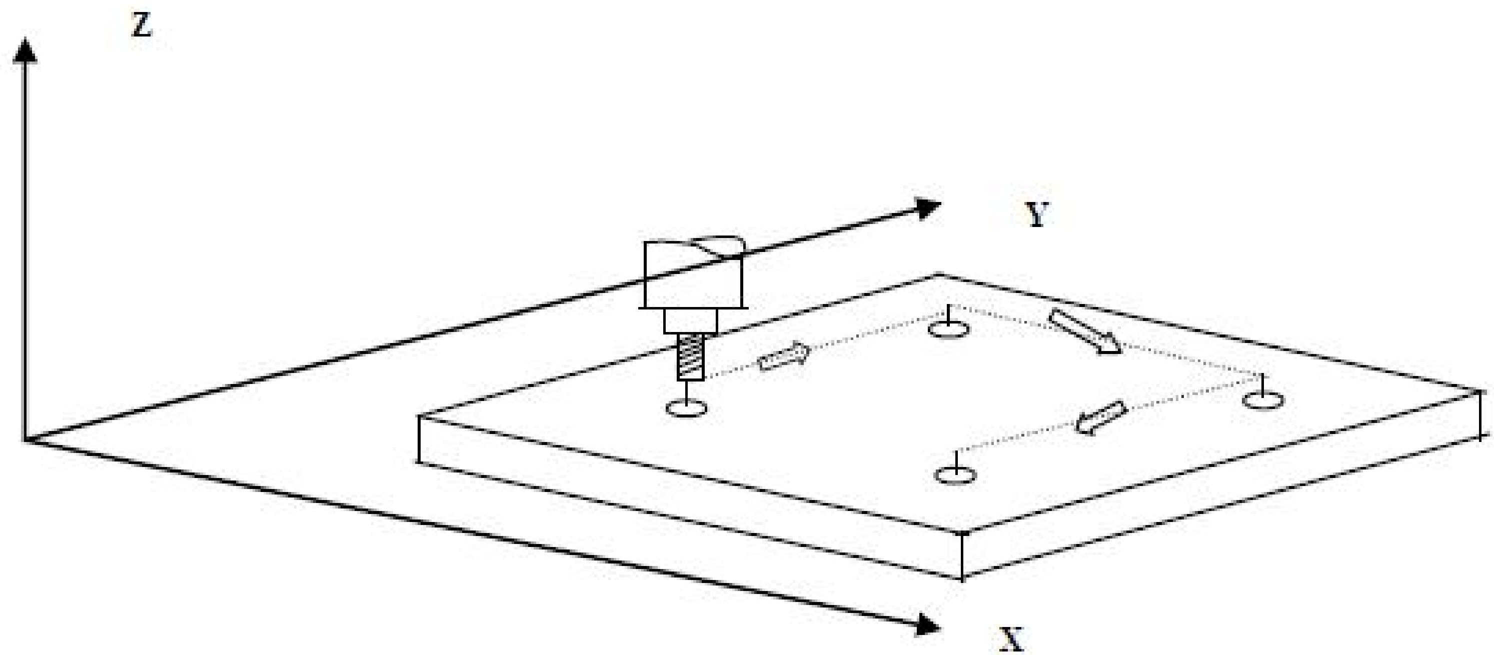


Figure 5.8: an application of a PTP control system

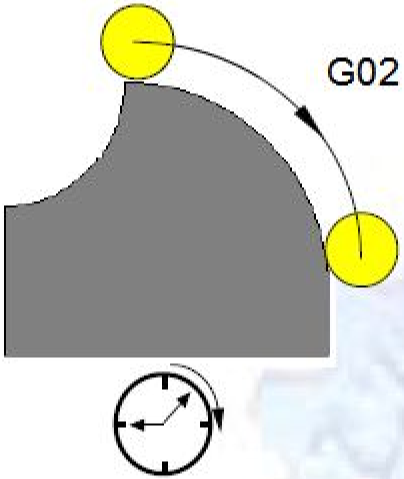
5.5.2 Continuous Path Control Systems

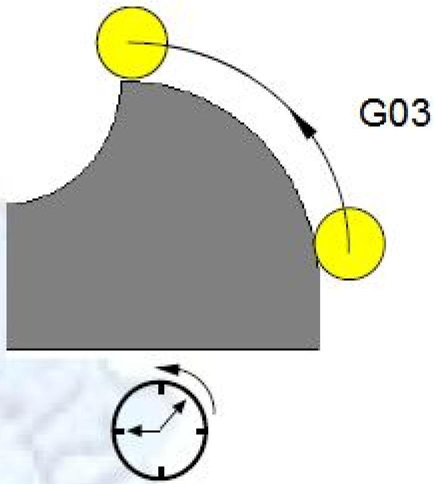
This is also known as contouring system. It is capable ofsynchronizing two or more axial drives to produce a desired path.

Contour systems are of five types, depending upon type of control:

5.5.2 Continuous Path Control Systems

These are: 2-D contouring	**It synchronizes feed only in two axes simultaneously**
2 ½-D contouring	Any two of three can be controlled simultaneously
3-D contouring	Capable of synchronizing three axes simultaneously
4-axis machining	Apart from 3 regular axis, 4th axis is rotary axis
5-axis machining	It incorporates 3 regular and 2 rotary axis





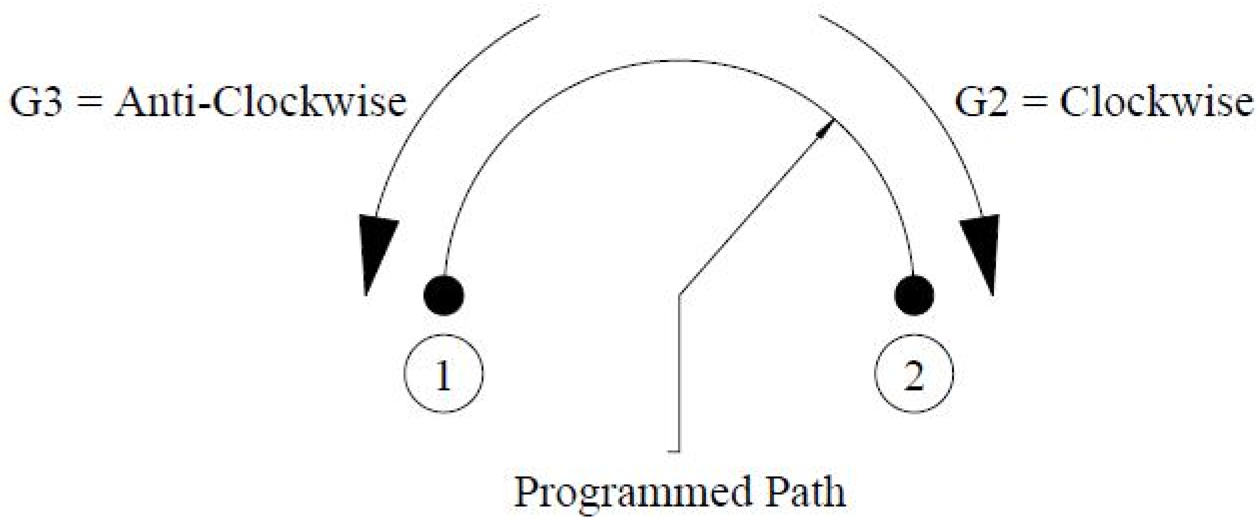
Format

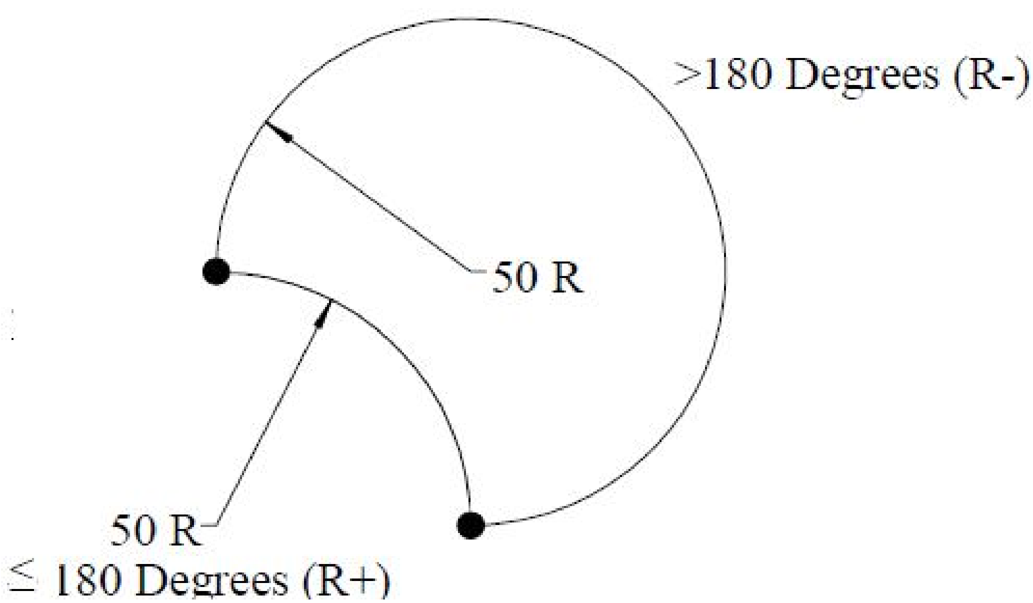
N__ G02/03 X__ Y__Z__ I__ J__K__ F__ using the arc center

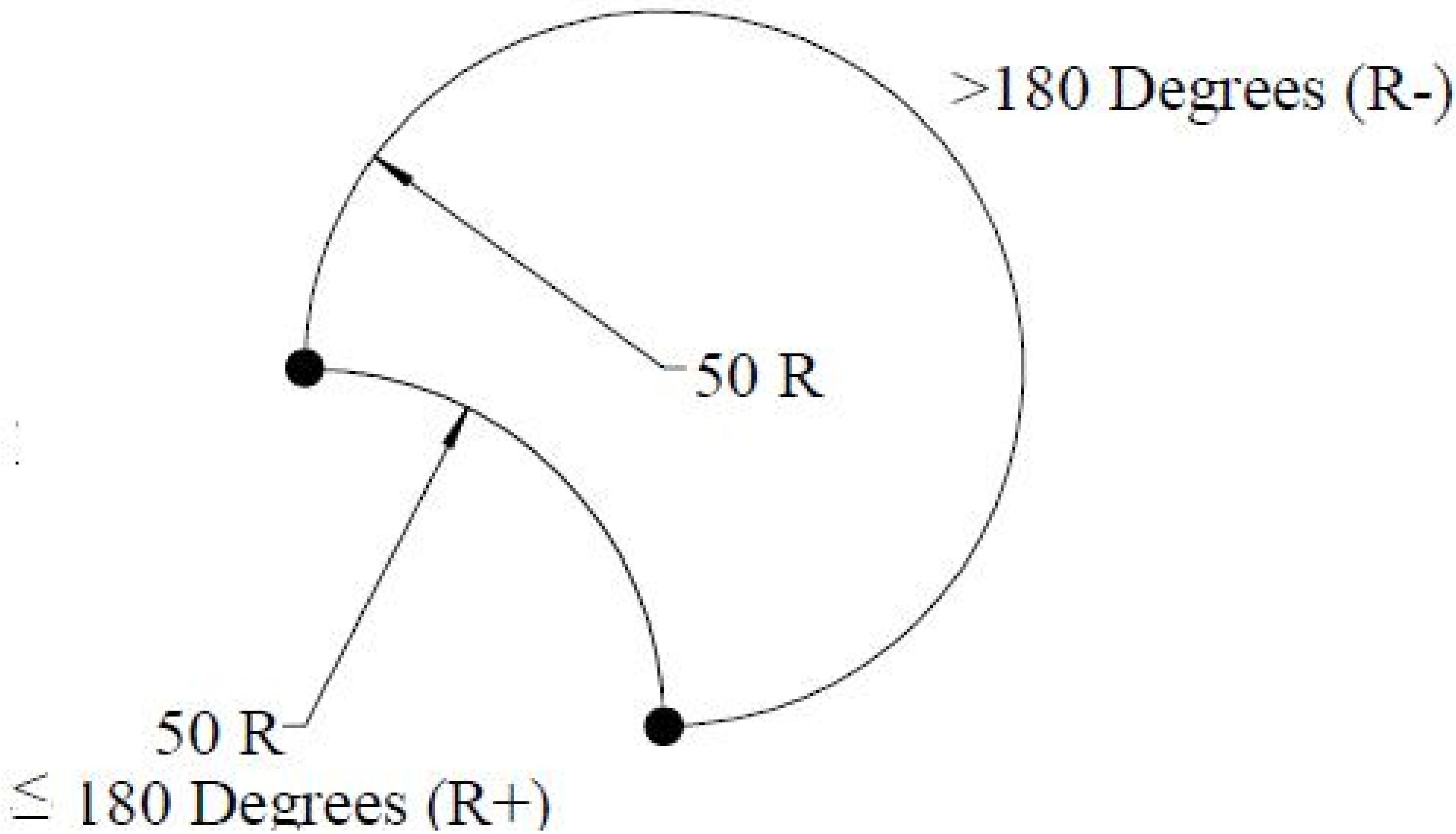
OR

N__ G02/03 X__ Y__Z__ R__ F__ using the arc radius

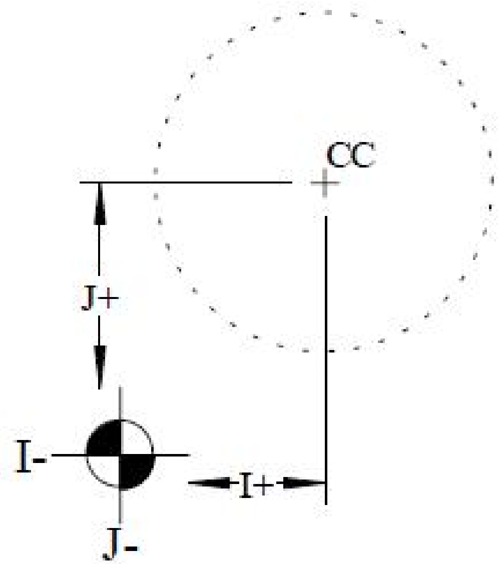
1) ARCS WITH A KNOWN RADIUS





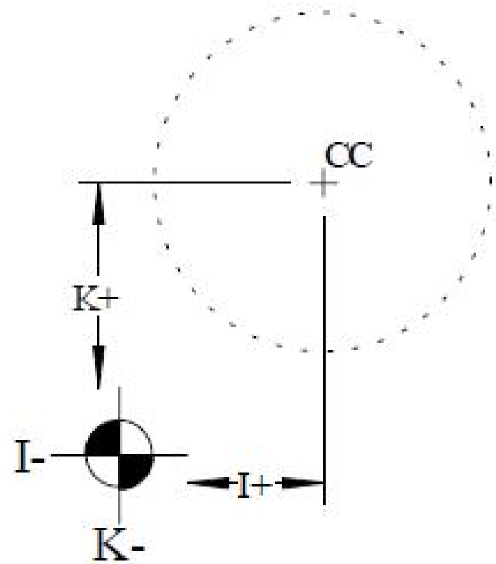


Fanuc



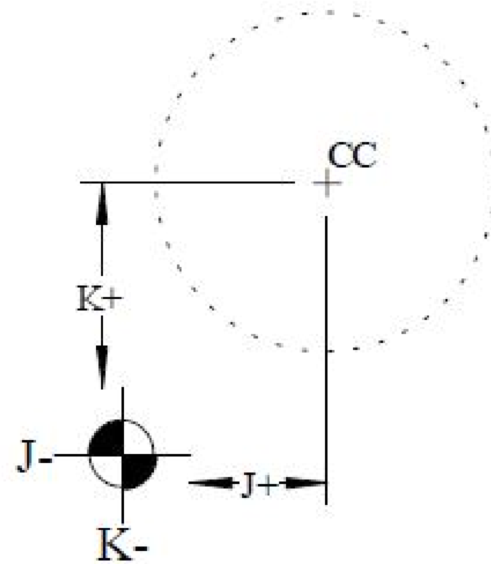
G17 Plan View (Z-)

I = X axis circle centre
J = Y axis circle centre



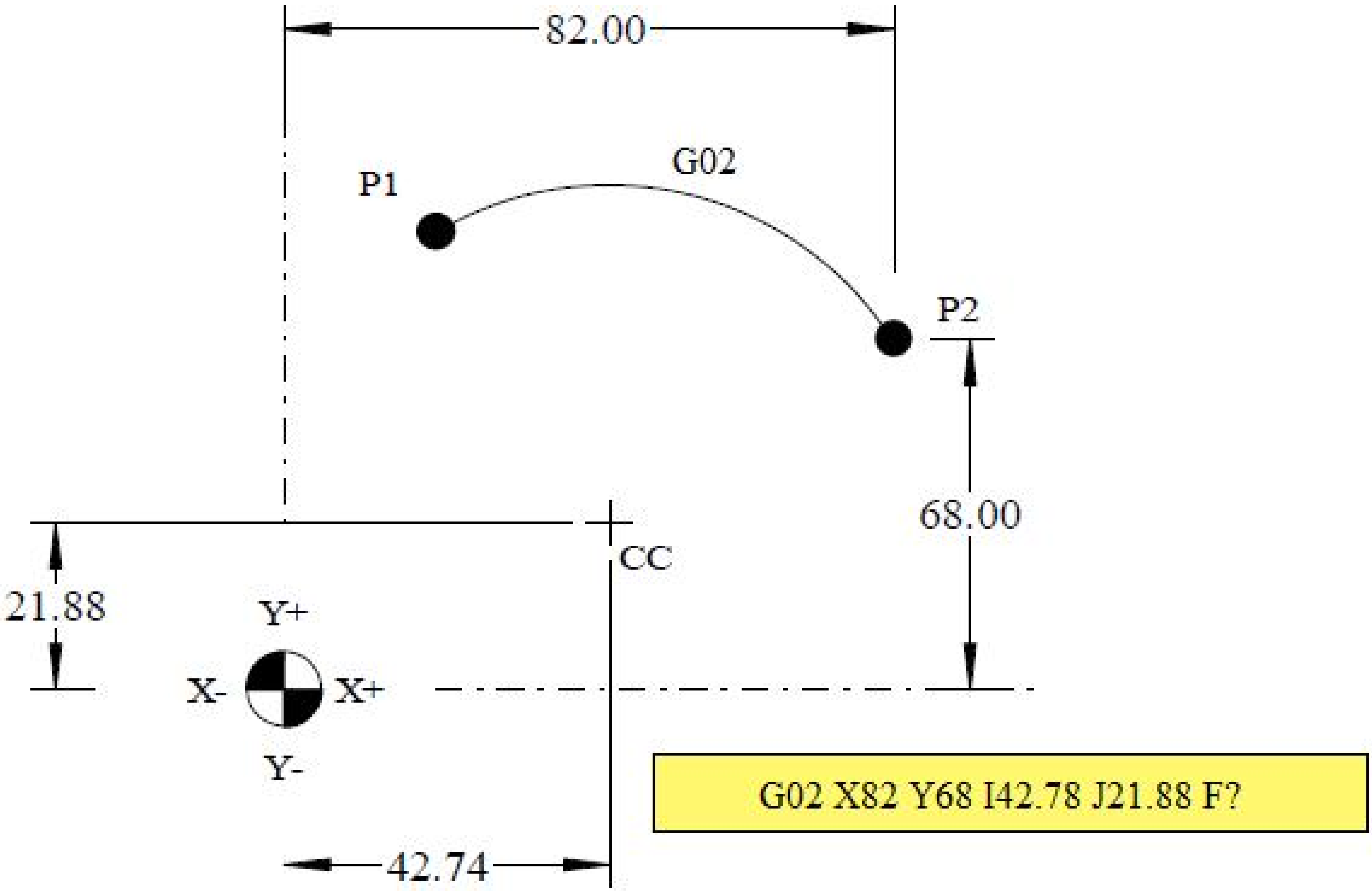
G18 Front View (Y-)

I = X axis circle centre
K = Z axis circle centre



G19 Side View (X-)

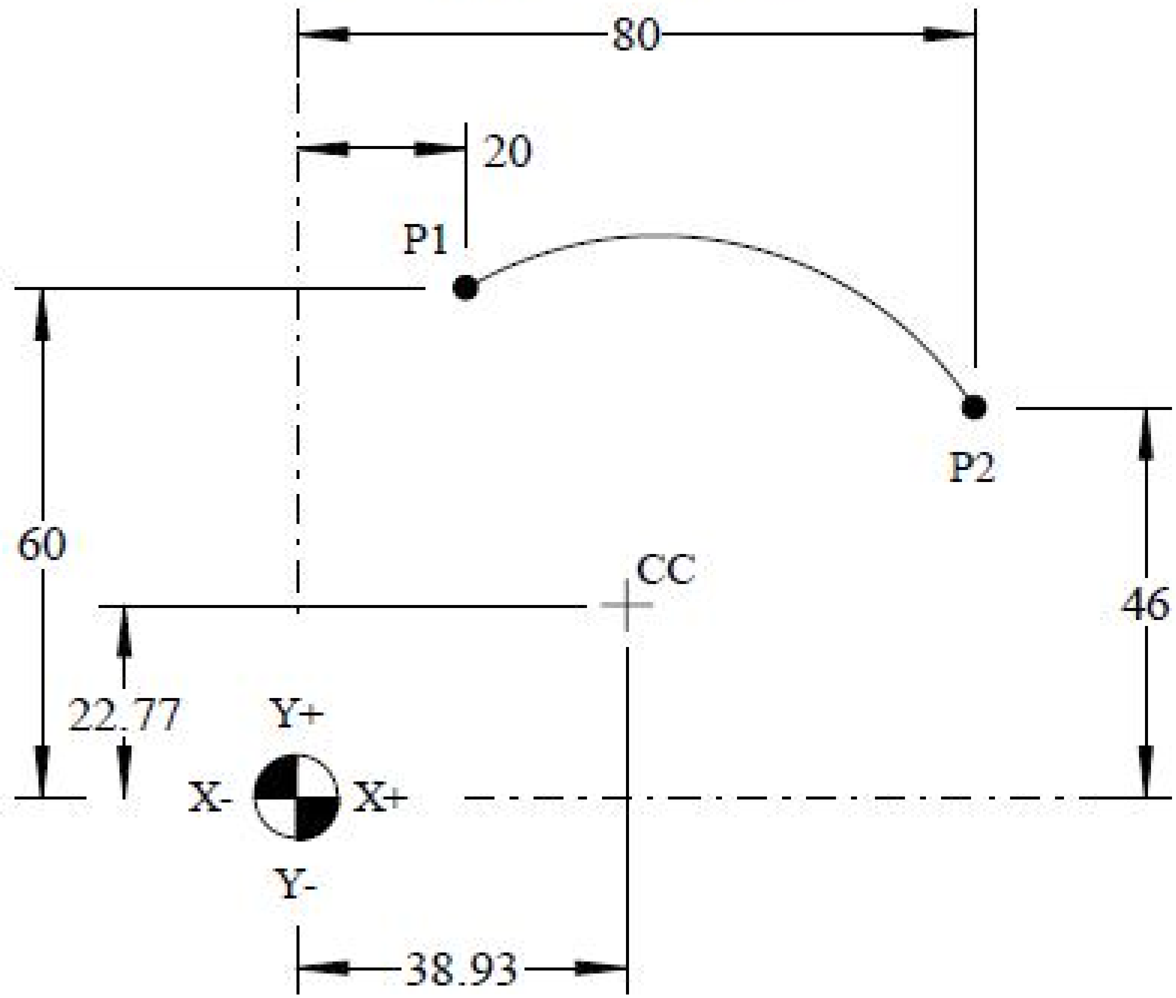
J = Y axis circle centre
K = Z axis circle centre



82.00
G02
P1
P2
68.00
CC
21.88
Y+
X-
X+
Y-
G02 X82 Y68 I42.78 J21.88 F?
42.74

Circular Programmed Movements – e.g. 5

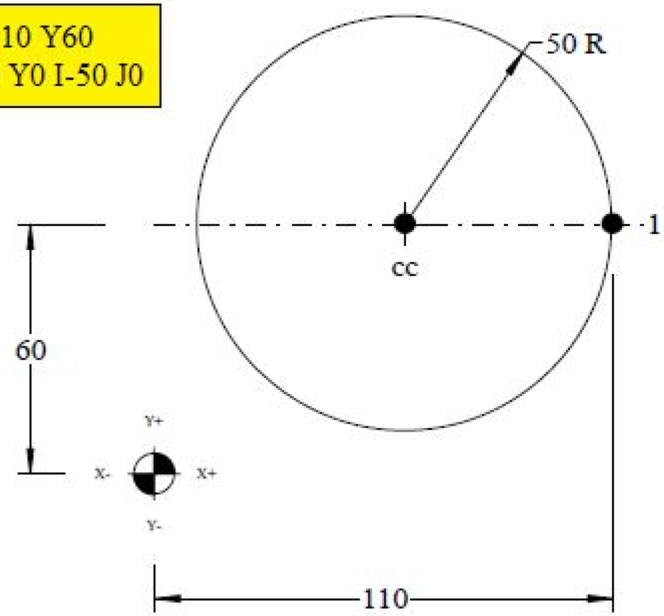
X, Y, I & J (Absolute)



Full Circular Movements – e.g.8

Inkremental:

```
G17 G00  X110 Y60
G91 G02  X0 Y0 I-50 J0
```



(Absolute Rapid XY to point 1)
(Clockwise to point 1)

Incremental

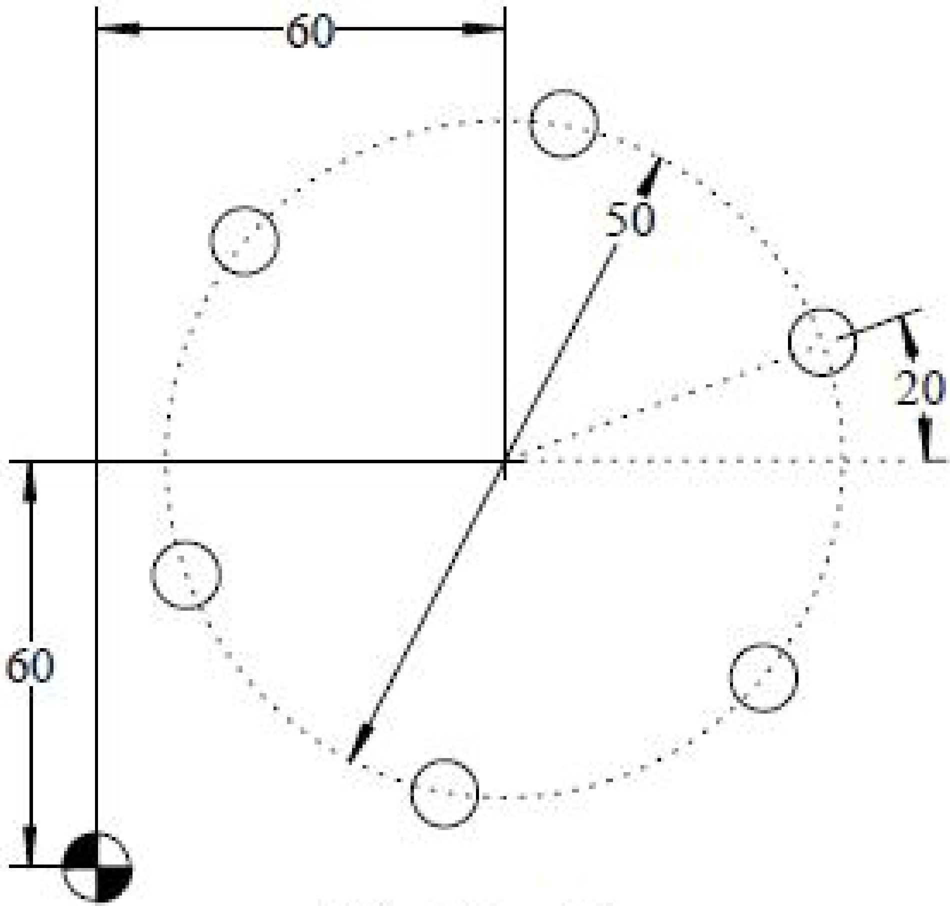
Note:

The end points and circle centre positions are taken from the "Start point"

(Absolute Rapid XY to point 1)
(Incremental Clockwise to point 1)

P.C.D. Pattern Cycle

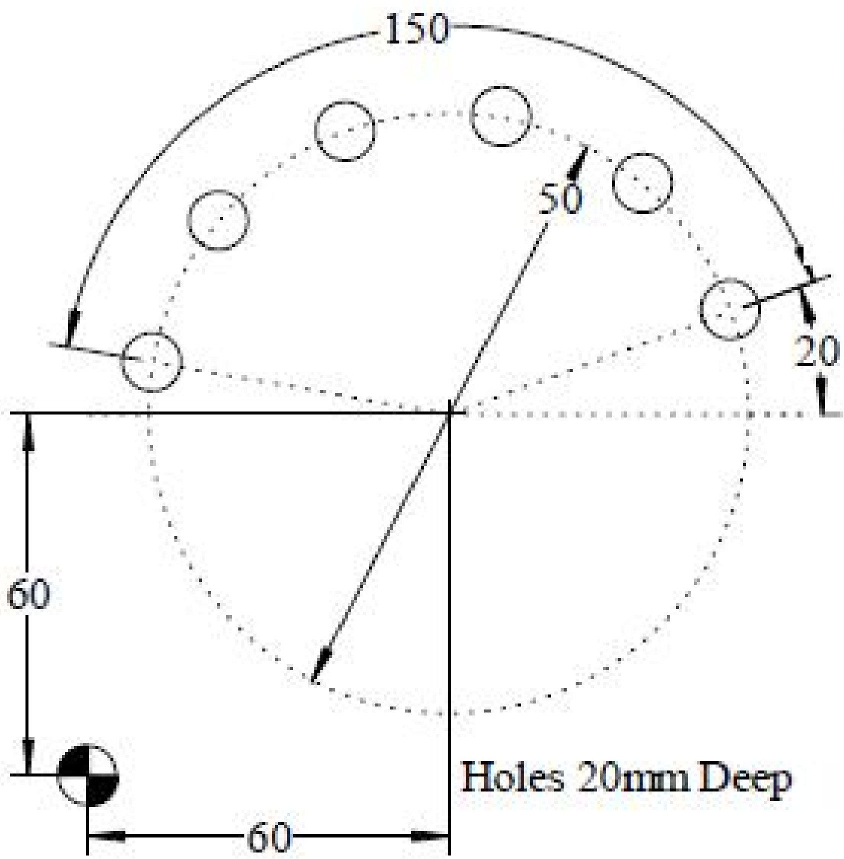
G39



Holes 20mm Deep

P.C.D. Arc Pattern Cycle

G39



: T1 M6
G0 G90 G40 G71 G17 G94
X60 Y60 Z100 H? S? M3
Z5
G39 U50 P20 O150 K6
G81 R? Z-20 F? M8
G37
G0 G90 Z100 M30

CIKLI PËR TORNIM GJATËSOR (TËRTHOR) G84

Cikli tipik përbëhet nga zhvendosja e pikës fillestare për shkak të arritjes së thellësisë së prerjes, tornimit gjatësor (tërthor), kthimit prapa për thellësi të prerjes dhe kthimi në pikën fillestare të ciklit.

Me aplikimin e funksionit G84 dukshëm shkurtohet programi dhe lehtësohet programimi, sidomos në rastet e tornimit me më shumë kalime. Funksioni G84 mundëson edhe tornim ciklik të konicitetit

Figurat më poshtë i tregojnë parametrat e nevojshëm për përshkrimin e këtij funksioni, ku kemi:

- **Cikli i tornimit me dalje vertikale**
- **Cikli i tornimi me lëvizje të pjerrët**

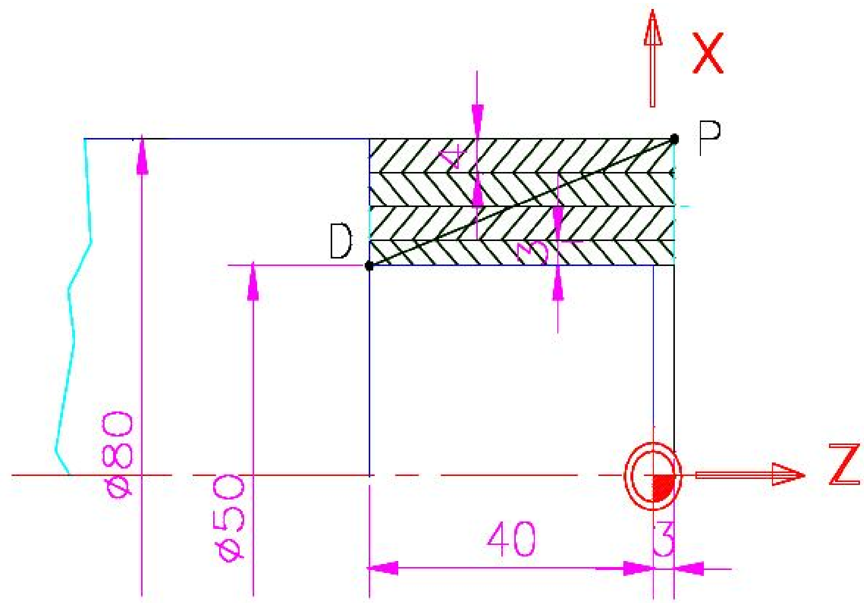
1.Cikli i tornimit me dalje vertikale

G00 X80.000 Z3.000 – koordinatat e pikës fillestare (P)

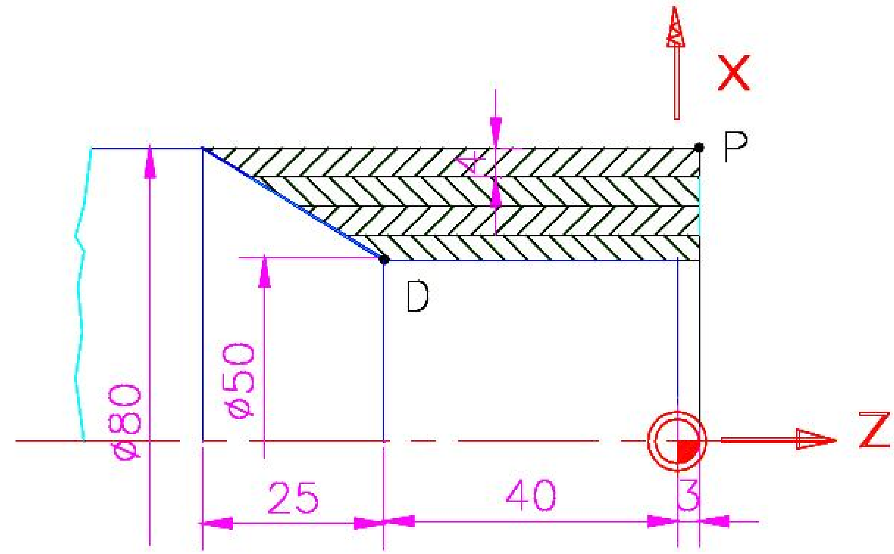
G84 X50.000 Z–40.000 D3=4000 F250

X, Z – koordinatat e pikës vertikale të ciklit (D)

D3 – trashësia e prerjes në mm, F – hapi në mm/rr



2.Cikli i tornimi me lëvizje të pjerrët



G84 X50.000 Z–40.000 P2=-25.000 D3=4000 F250

G00 X80.000 Z3.000

X,Z – koordinatat e pikës diagonale të ciklit

P2 – gjatësia e konicitetit në drejtimin Z

D3 – trashësia e prerjes në µm, F – hapi në mm/rr

Makina me të cilën po flasim **“Emco Turn 55” -** nuk e përmban këtë funksion por e shfrytëzon funksionin **Stock removal.**

Ky funksion shfrytëzohet p.sh. te makina tornuese **EMCOTRONIC T1 CNC**.

LËVIZJA RRETHORE E INSTRUMENTIT PRERËS

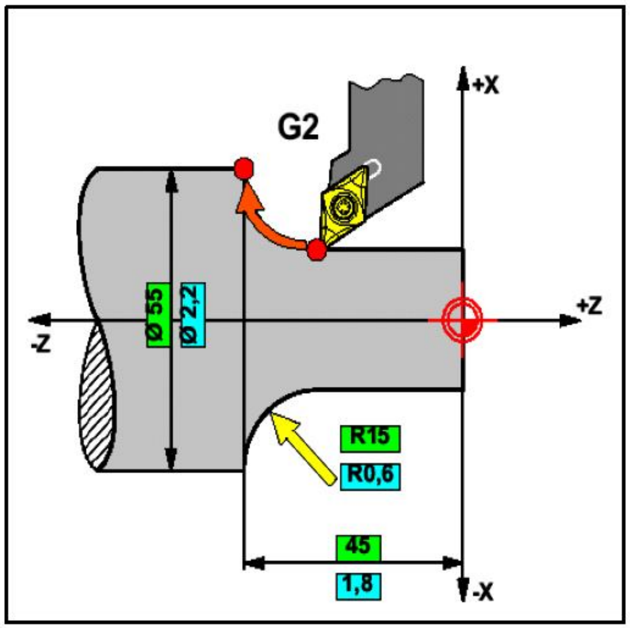
Makina tornuese CNC mundëson tornim të formësuar (rrezes) dhe rrezeve kalimtare duke shfrytëzuar dy funksione:

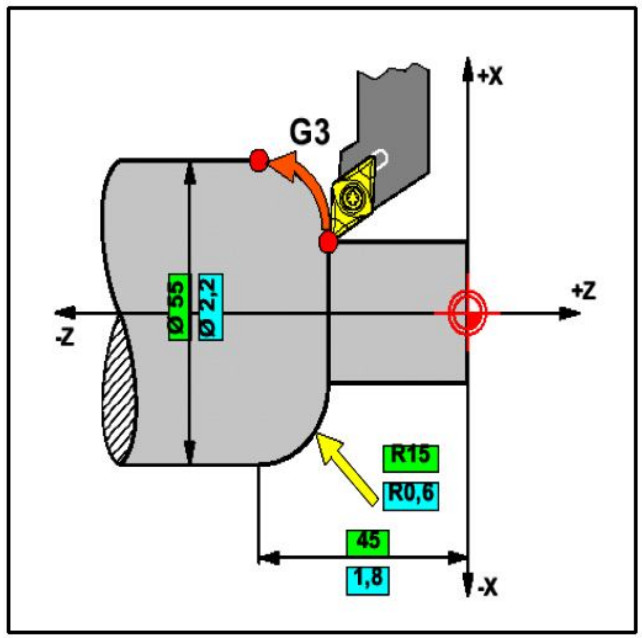
G02 (G2) – lëvizje rrethore në kahje të akrepave të orës (djathtas)

G03 (G3) – lëvizje rrethore në kahje të kundërt të akrepave të orës (majtas).

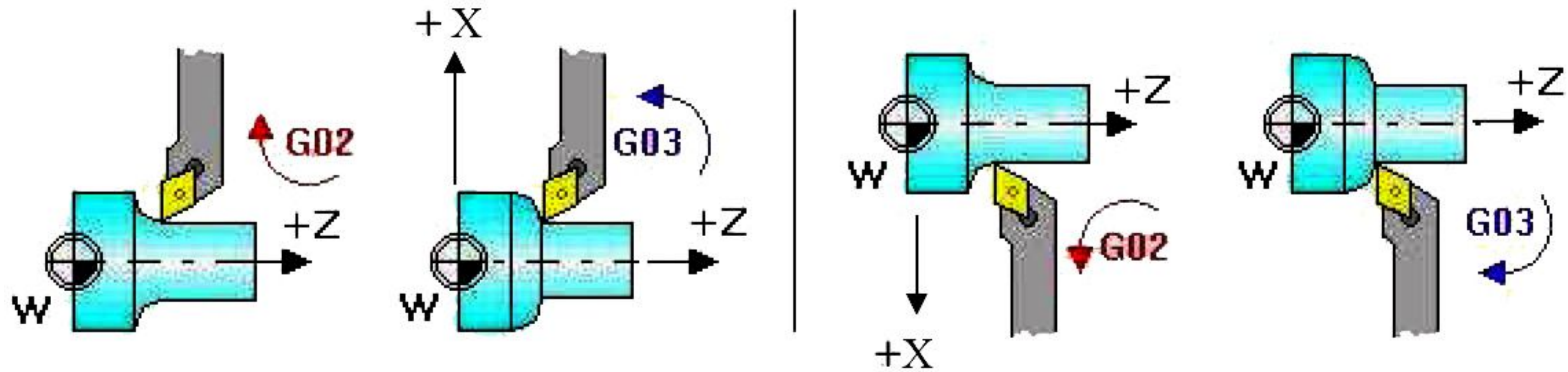
Udhëzime të shkurtra të programimit të G02, G3

1. **Sjellet thika në pozitën fillestare,**
2. **Caktohet kahja e rrotullimit për vendosjen e prapme – të përparme të thikës G02 ose G03**
3. **Definimi i pikës përfundimtare të rrezes (në diametër dhe gjatësi) – koordinatat e akseve X, Z,**
4. **caktimi i koordinatave të qendrës së rrezes I, K (distanca e qendrës në akset X dhe Z) – duhet pasur kujdes në parashenjat I dhe K.**





Funksionet G2/G3 me vendosje të thikës lart/poshtë



Makina tornuese Emco mundëson disa mënyra të ndryshme të programimit të lëvizjeve rrethore:

- Programimi me pikën fillestare (S – start), pikën përfundimtare (E– end) dhe rrezen e rrethit (CR)

<u>**Komandat**</u>:

G2 ose **G02 / G3** ose **G03** lëvizja rrethore në hapin punues

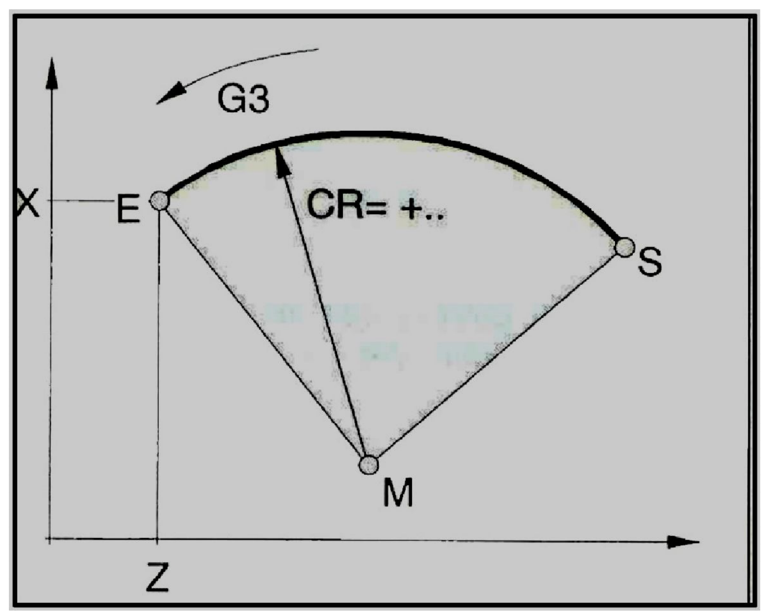
Mund të jepet sistemin kënddrejtë koordinativ si:

G2 (ose **G3**) **X... Z... CR=±...**

X, Z – pika përfundimtare (**E**)

CR –- rrezja e rrethit

CR=+ për këndet deri **180º, CR= –** për këndet mbi **180º.**



2. programimi me pikën fillestare (A=P1), me pikën e fundit (N=P2) dhe me qendrën e rrethit (S=Po)

<u>Komandat</u>:

G2 ose **G02 / G3** ose **G03** lëvizja rrethore në hapin punues

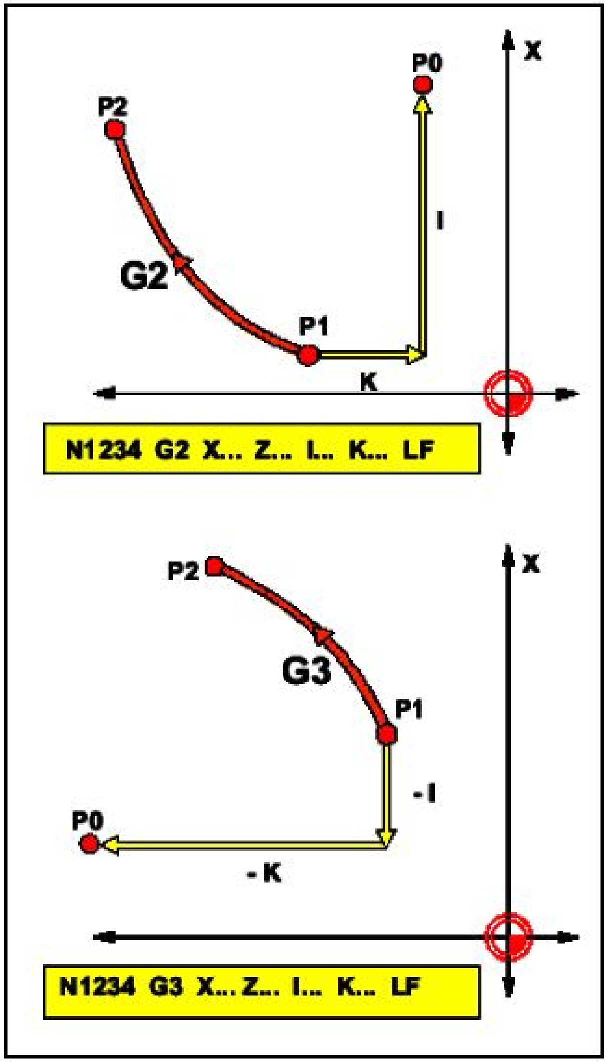
Mund të jepet sistemin kënddrejtë koordinativ si:

G2 X… Z… I… K…

X, Z - pika e fundit (P2)

I, K – koordinatat e qendrës së rrethit (S) në sistemin inkremental ose

I = AC(…), K = AC(…) në sistemin absolut nga pika punuese zero (0).



3. Programimi me pikën fillestare (A=P1), pikën përfundimtare (B=P2),pikën e qendrës së rrethit (S=P0) dhe këndin e harkut rrethor (AR)

Komandat:

G2 ose **G02 / G3** ose **G03** lëvizja rrethore në hapin punuese

Mund të jepen në sistemin kundrejt koordinativ si:

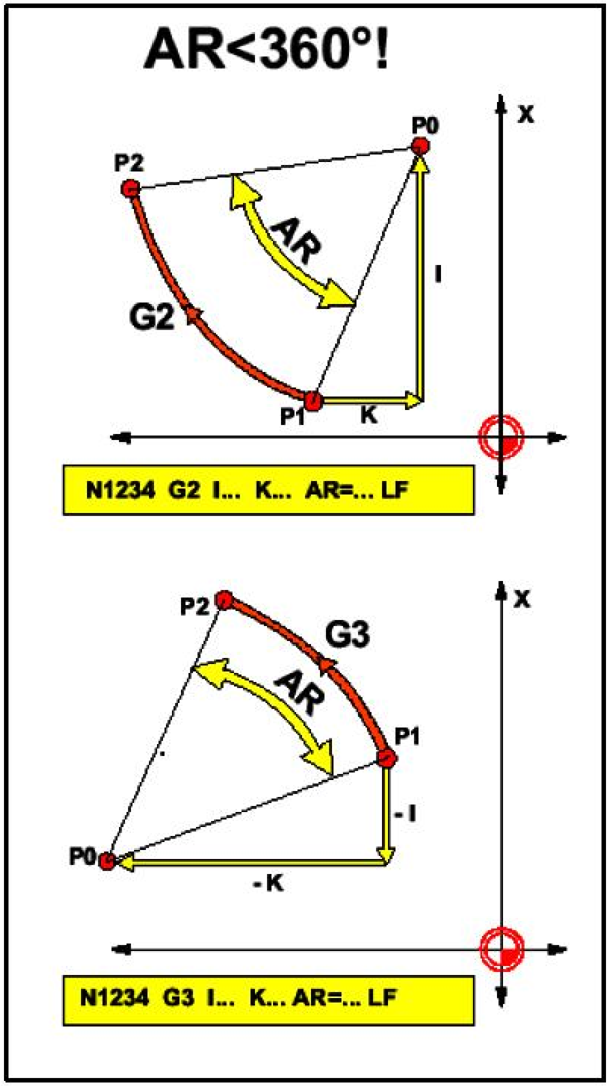
G3 X... Z... AR...

G3 I... K... AR..

X, Z – pika e fundit (**B**)

I, K – koordinatat e qendrës së rrethit në sistemin inkremental

AR – këndi i harkut rrethor



4. Programimi me pikën fillestare (A=P1), ndërmjet-pikës (M), pikës përfundimtare (B=P2)

<u>Komandat:</u>

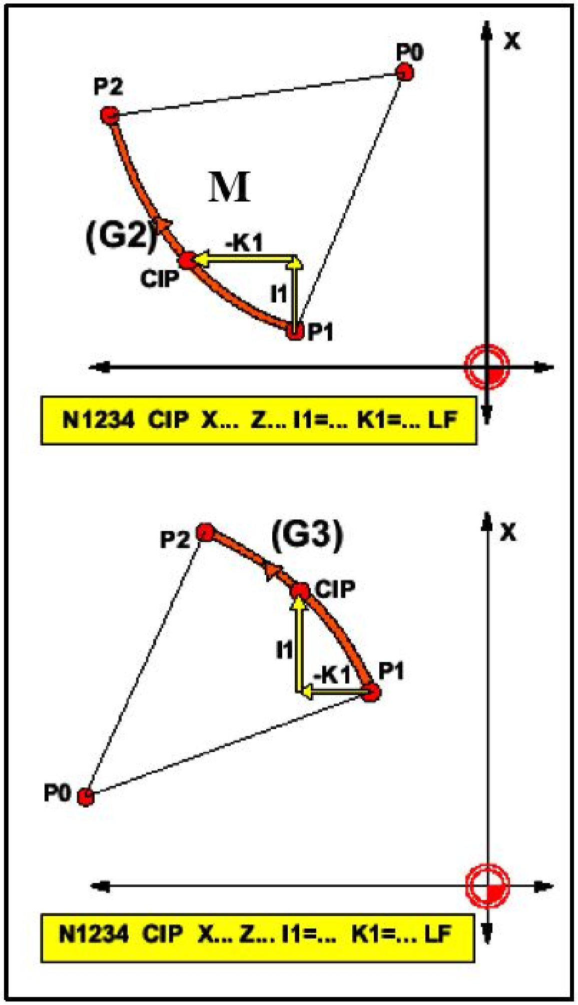
CIP lëvizja rrethore në hapin rrethor (**C**ircle through **P**oints) – rrethi përmes pikës

Janë dhënë:

CIP X.. Z.. I1=.. K1=..

X, Z – pika përfundimtare (P2)

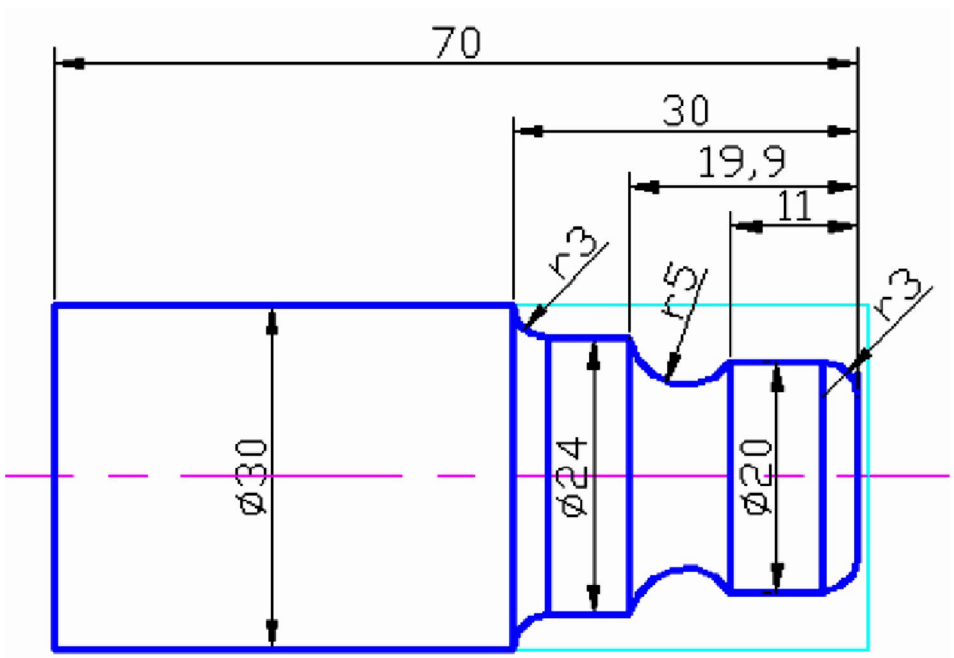
I1, K1 – koordinatat e ndërmjet-pikës së rrethit



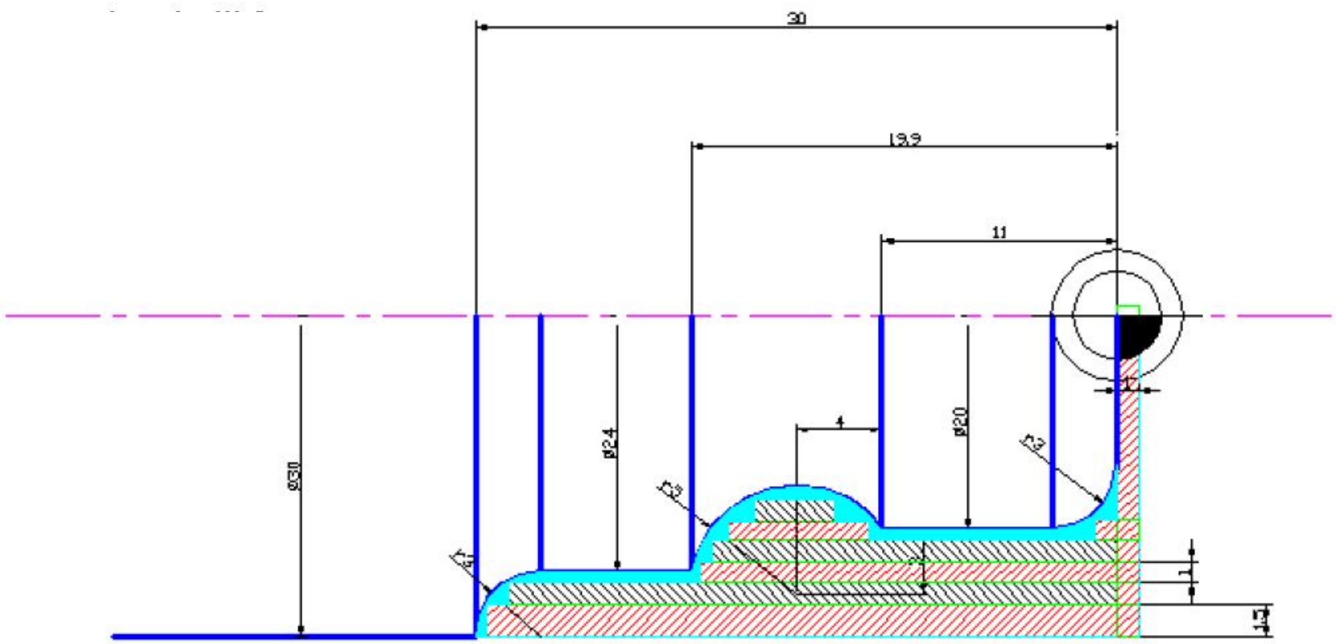
Punimi i programit për BOSHTIN 1 (funksionet G2, G3)

Te programimi i sipërfaqeve të rrumbullakuara me rreze, paraprakisht duhet patjetër ti afrohemi konturës të copës punuese duke shfrytëzuar funksionet G0 dhe G01. Te prerja e fundit ose e ashtuquajturi tornim kontural kur të arrimë te rrezja e aplikojmë funksionin G02 ose G03. Mund të shfrytëzohen edhe funksionet e ciklit për afrim te kontura e copës punuese (Stock removal) për të cilin do të flitet më vonë.

Për tornim përfundimtar (kontural) të shfrytëzohet thika neutrale!



Nr. Rend.	Përshkrimi i vendosjes - operacionit	Instrumenti prerës	Hapi mm/min	Numri i rrotullimeve rr/min
1	Shtrëngimi	Koka shtr.	0.05	1500
2	Tornim I ashpër gjatësor	ISO 6	0.05	1500
3	Përpunim I radiusit	ISO 6	0.05	1500



Pjesa programore

Nr. Rend.	**Funksionet e programit**												**Vërejtje**
%MP / **N9000**	**G**	**X**	**Z**	**F**	**S**	**M**	**T**	**D**	**CR**	**AR**	**I**	**J**	
N1	18												
N2	54												
N3	23												
N4	90	14	2	0.05	1500	3	1	1					
N5	1		0										
N6	3	20	-3						3				
N7	1		-11										
N8	2	24	-19.9						5				
N9	1		-27										
N10	2	30	30						3				
N11	1		-70										
N12	0	32											
N13			100										
N14						2							

KOMPENSIMI I RREZES SE INSTRUMENTIT PRERËS

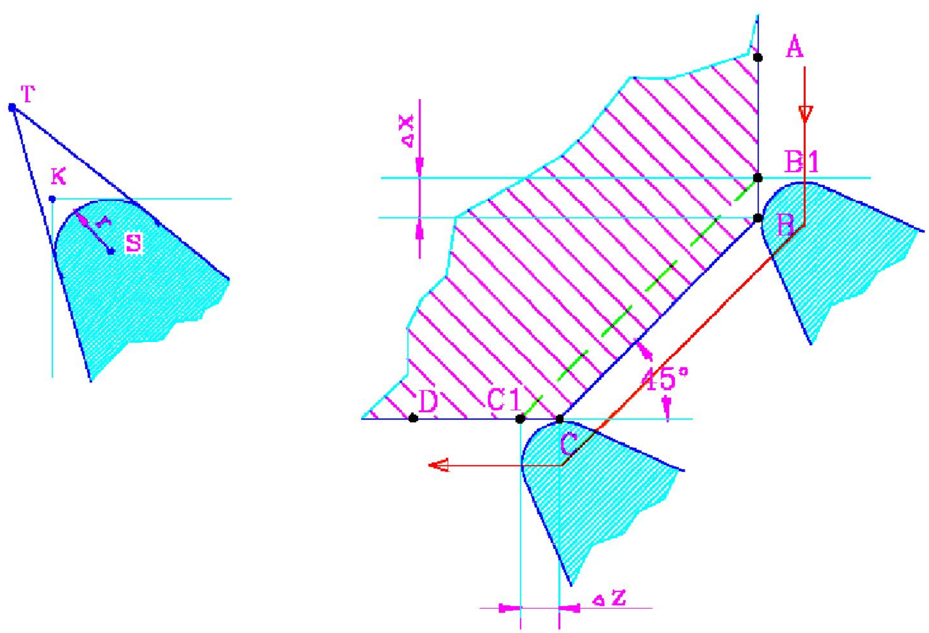
Maja e pllakës prerëse punohet me rreze të rrumbullakimit të cilat janë të standardizuara, p.sh. R 0,2 ose R 0,4.

Në procesin e tornimit të sipërfaqeve paralele dhe vertikalisht në aksin e rrotullimit, udhëzimi i majës imagjinare të instrumentit (K) sjell deri te tornimi i saktë i konturës së dhënë pa marrë parasysh rrumbullakimin e vërtetë të majës së tehut prerës së instrumentit

Kjo do me thënë se mund të përvetësohet rregulli vijues:

Gjatë tornimit të sipërfaqeve paralele dhe vertikale në aksin e rrotullimit nuk është e nevojshme të kompensohet rrezja e majës së tehut prerës së instrumentit pasi që udhëzimi i majës imagjinare i tehut prerës së instrumentit jep dimensione përfundimtare të sakta të copës punuese.

Gjatë tornimit të sipërfaqeve konike paraqitet shmangia e konturës së vërtetë nga kontura e kërkuar.



T - maja teorike e tehut prerës së instrumentit prerës

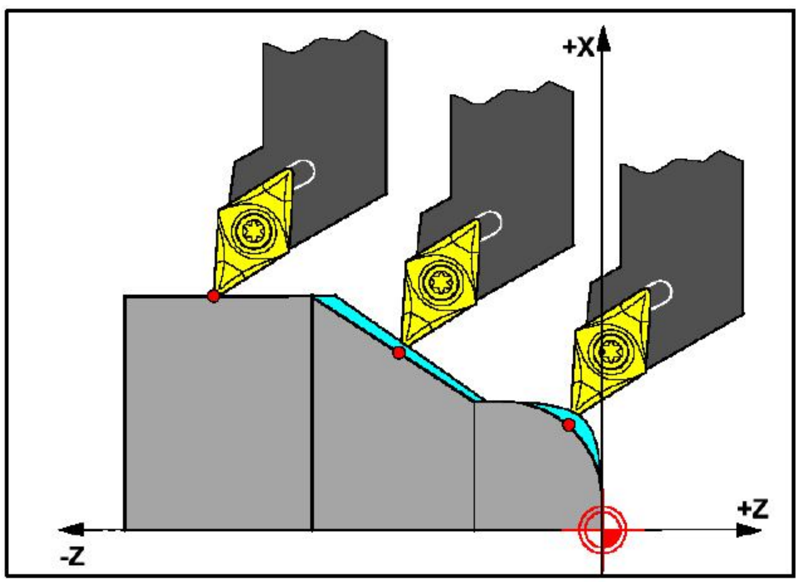
K - maja imagjinare e tehut prerës së instrumentit

S - qendra e rrumbullakimit të majës së tehut prerës së instrumentit

A-B-C-D rruga e lëvizjes së instrumentit prerës

Δx, Δz - kompensimi i rrezes së majës së instrumentit, e llogarit vetë njësia drejtuese e makinës

Gjatë tornimit të sipërfaqeve konike dhe kanaleve, për arsye të rrumbullakimit të tehut prerës dhe udhëzimit të majës imagjinare vjen deri te shmangia ndërmjet profilit të dhënë dhe të vërtetë, prandaj për arsye të tornimit korrekt të këtyre sipërfaqeve është e nevojshme të kompensohet rrezja e tehut prerës të instrumentit dhe të ndryshohen koordinatat e pikave të lëvizjes fillestare dhe përfundimtare.

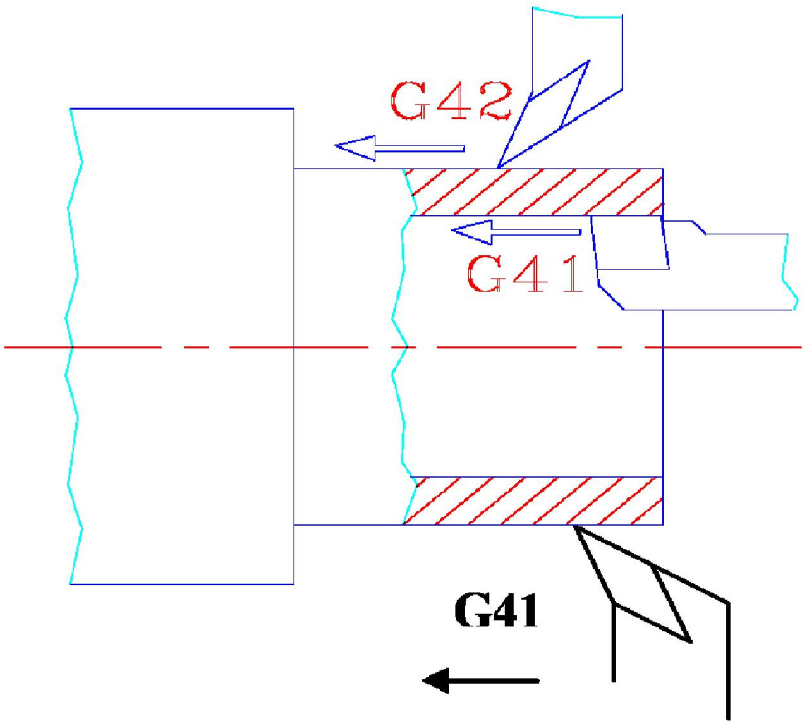


Ekzistojnë dy funksione për korrigjimin e rrezes së instrumentit **G41** dhe **G42** me të cilat njësisë drejtuese të makinës metalprerëse i jepen informata për lëvizjen e instrumentit dhe llogaritjen e korrigjimeve të nevojshme.

G42 – korrigjimi i rrezes së instrumentit **djathtas** nga kontura e dhënë

G41 – korrigjimi i rrezes së instrumentit **majtas** nga kontura e dhënë

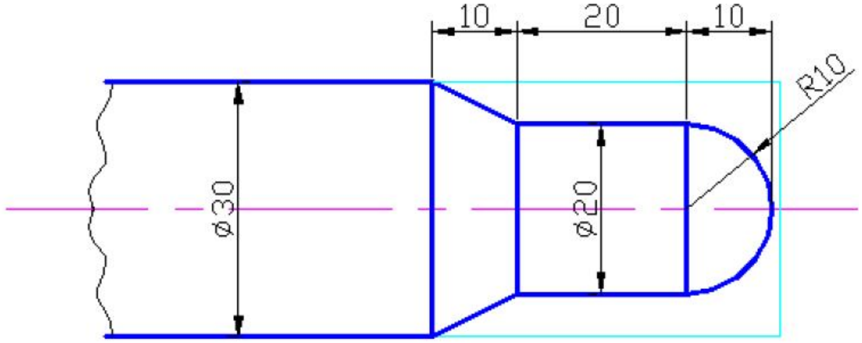
G40 – shkyçja e korreksionit të rrezes së instrumentit.

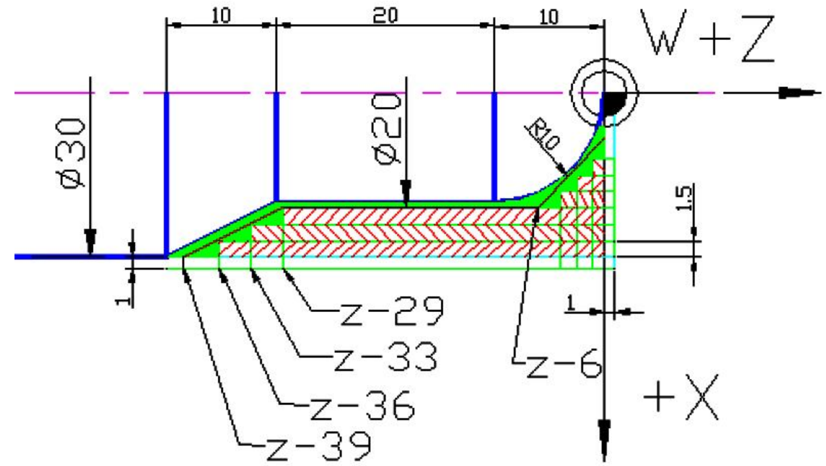


G42 – instrumenti ndodhet, shikuar në drejtim të hapit, djathtas nga kontura,

G41 – instrumenti ndodhet, shikuar në drejtim të hapit, majtas nga kontura.

PUNIMI I PROGRAMIT PËR BOSHTIN 2 (KOMPENSIMI ME RREZEN E INSTRUMENTIT PRERËS)



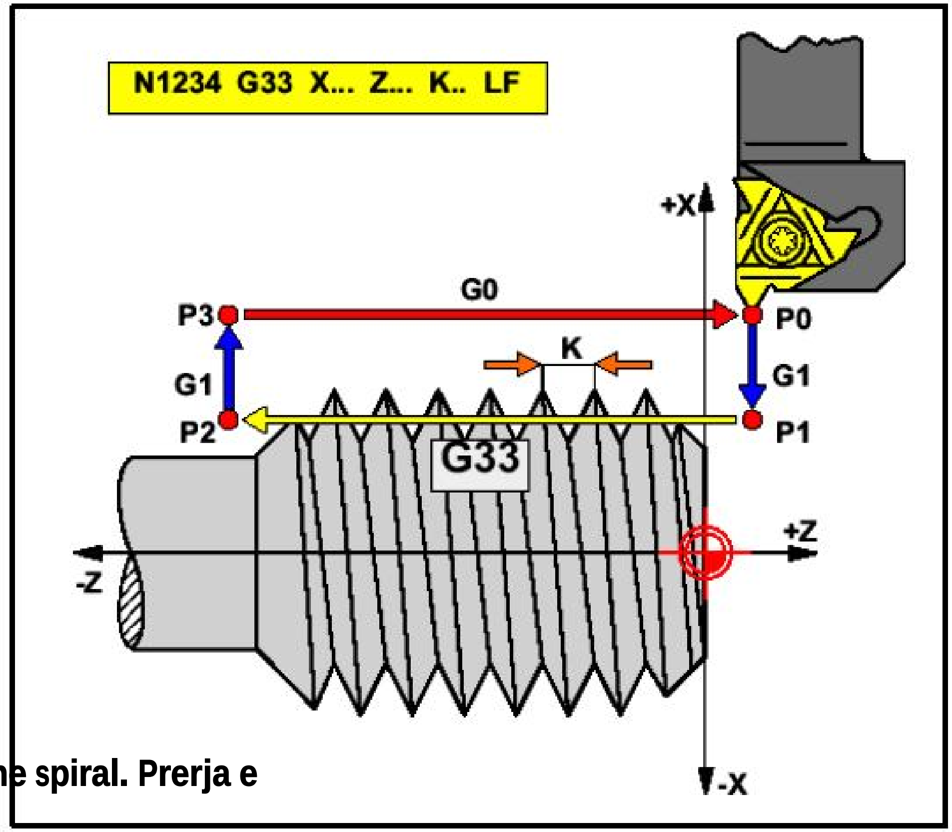


Nr. Rend.	Përshkrimi i vendosjes - operacionit	Instrumenti prerës	Hapi mm/min	Numri i rroullimeve rr/min
1	Shtrëngimi	Koka shtr.	0.05	1500
2	Tornim I ashpër gjatësor	ISO 6	0.05	1500
3	Përpunim I radiusit	ISO 6	0.05	1500

Pjesa programore

Nr. Rend.	Funksionet e programit											Vĕrejtje
%MP N9000	G	X	Z	F	S	M	T	CR	AR	I	J	
N1	18											
N2	54											
N3	23											
N4	90	0	2	0.05	1500	3	1					
N5	1		0									
N6	3	20	-10					10				
N7	1		-30									
N8		30	-40									
N9			-100									
N10	0	32										
N11			100									
N12						2						

TORNIMI I FILETËS (HAPJA E FILETËS SË JASHT



Në makinën tornuese CNC mund të tornohen filetat (
të majta.

Gjatë kësaj duhet të kemi parasysh pozitën e pik
përfundimtare të filetës, kahjen e rrotullimit të bosht
edhe standardet – të dhënat tabelare të vetë filetës (

Fileta mund të prehet duke shfrytëzuar funksionin
(përshtatur) me lëvizjen gjatësore të suportit, me r
filetës,por mund të shfrytëzohet funksioni i ciklit për

PRERJA E FILETËS

Është i mundur prerja e filetave të rrafshët, konik dhe spiral. Prerja e filetave realizohet me instrument prerës përkatës,

Hapi K ose I duhet të jetë i zgjedhur patjetër në varësi të drejtimit të filetës (gjatësor ose tërthor).

Shpejtësia e rrotullimit S dhe hapi F nuk janë aktiv te G33 (100%).

G33 Z... K... (K – hapi i filetës, Z – thellësia e filetës).

G97 shkyçja e shpejtësisë së prerjes konstante

Example 1: Longitudinal Thread (Cylindrical)

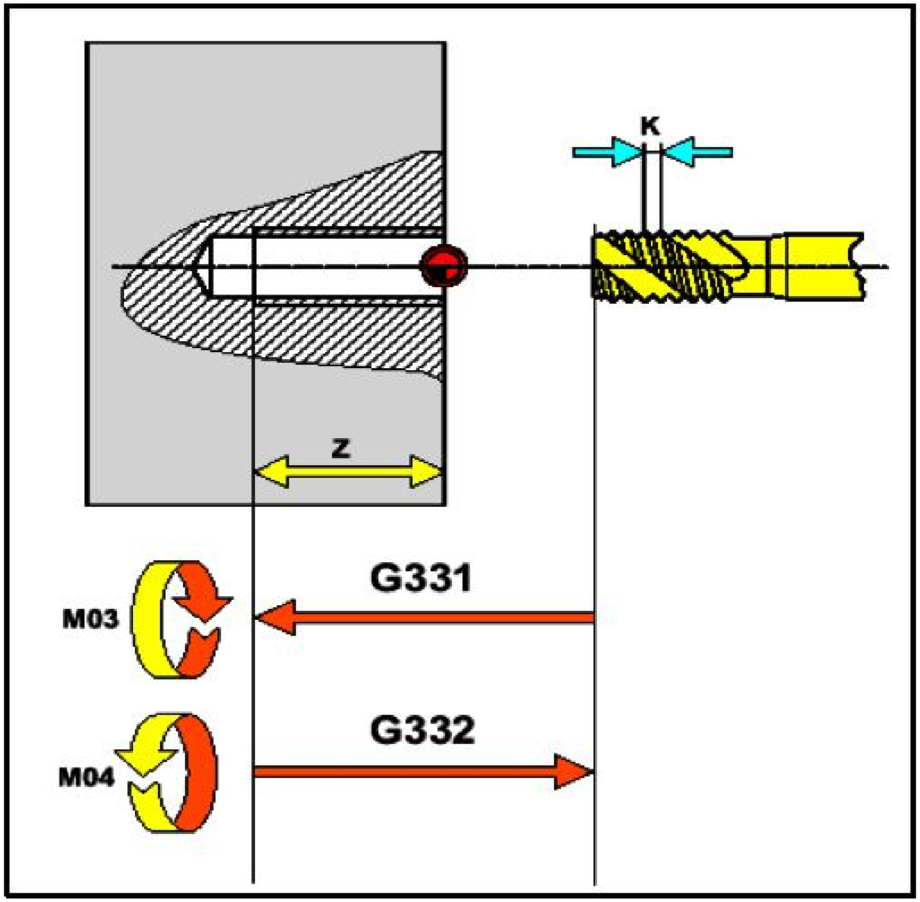
```
N0130 ...
N0140 G97 M03 S500
N0150 G00 X22 Z4
N0160 X18
N0170 G33 Z-28 K1
N0180 G00 X22
N0190 ...
```

G331 / G332 – PRERJA E FILETËS SE BRENDSHME PA KOMPENSIMIN E KOKËS SHTRËNGUESE

G331 – prerja e filetës së brendshme definohet me thellësinë e shpimit **Z** dhe hapin **K**

G332 – lëvizja kthyese te prerja e filetës definohet gjithashtu me thellësinë e shpimit **Z** dhe hapin **K**, ndërsa kahja e rrotullimit të boshtit ndryshon automatikisht.

Përpara fillimit të komandës **G331** patjetër duhet të hapet vrima paraprakisht me përmasa të pastra, gjegjësisht të caktohet pozicioni i saktë prej ku fillon komanda.



KORNIZAT (FORMAT) – FRAMES

Kornizat e ndryshojnë sistemin aktual koordinativ:

TRANS – ATRANS – translacioni (zhvendosja) i sistemit koordinativ

ROT – AROT – rrotullimi i sistemit koordinativ

SCALE – ASCALE – matësi i programuar

MIRROR – **AMIRROR** – pasqyrimi i sistemit koordinativ

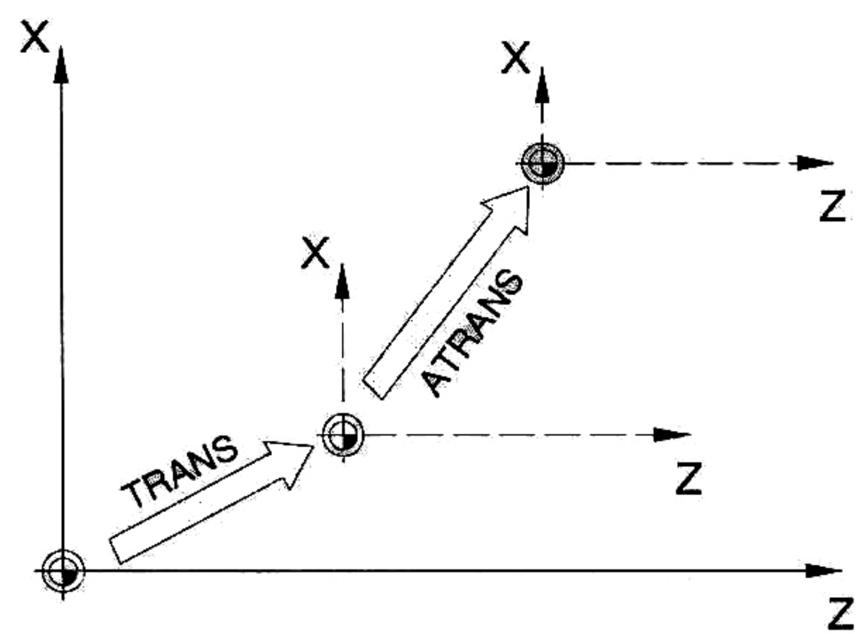
Kornizat (konturat) programohen në fjali të posaçme programuese edhe atë kryhen si:

TRANS – ATRANS – TRANSLACIONI I SISTEMIT KOORDINATIV

TRANS – e zhvendos pikën zero –W, G54, G55, ... (nga baza e të dhënave) në pozicion të ri.

ATRANS – e kalon pikën zero në raport me pozicionin e fundit (G54, G55, ... TRANS).

Formati: TRANS/ATRANS X...Z...



ROT – AROT – RROTULLIMI I SISTEMIT KOORDINATIV

ROT / AROT i rrotullon koordinatat e copës punuese rreth çdo aksi të sistemit koordinativ X dhe Z ose këndin **RPL** në sipërfaqen e zgjedhur punuese.

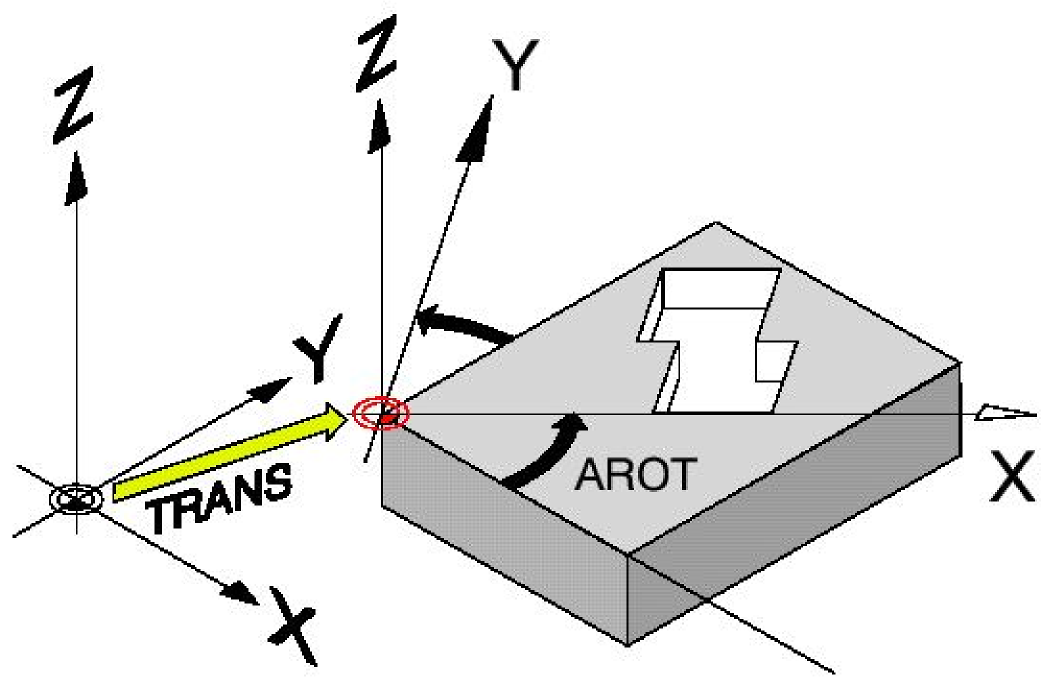
Me këtë është mundësuar programimi nëpër kontura në sistemin kryesor koordinativ gjegjësisht rrotullimi përkatës.

X, Z – rrotullimi në shkallë rreth aksit të zgjedhur,

RPL – **R**otation in the **Pl**ane – rrotullimi nëpër sipërfaqe në shkallë.

Shembull:

ROT X40 Z30 Ose **AROT RPL=45**



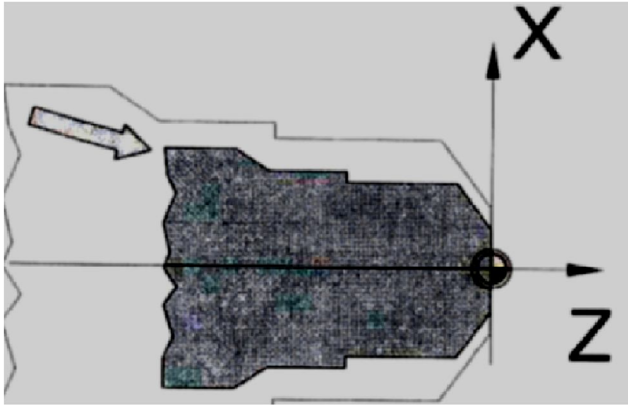
SCALE – ASCALE – MATËSI I PROGRAMUAR

SCALE – ASCALE – mundëson vendosjen e raportit përkatës për çdo aks X, Y, Z.

Me këtë rriten ose zvogëlohen dimensionet e copës punuese – zgjatet ose shkurtohet rruga e instrumentit

SCALE i fshin të gjitha kornizat e programuara paraprakisht.

Shembull: **SCALE/ASCALE X0.8 Z0.6**

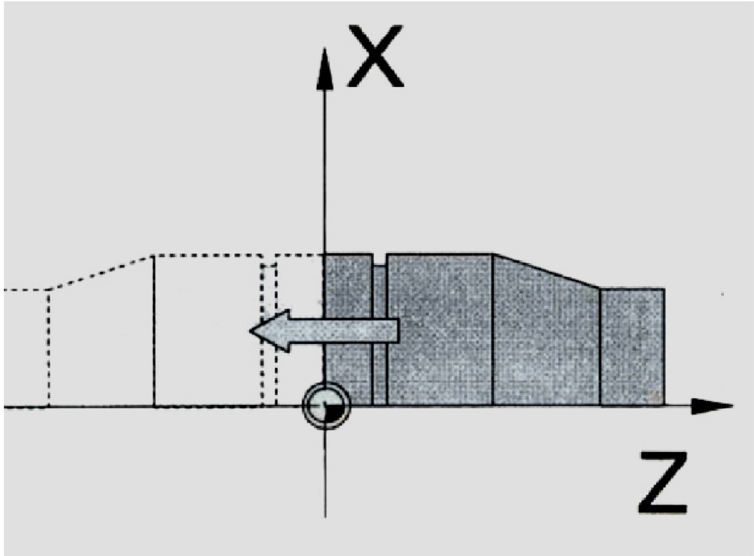


MIRROR – AMIRROR – PROGRAMIMI I PASQYRIMIT

MIRROR / AMIRROR mundëson pasqyrimin e copës punuese rreth akset koordinatave X dhe Z.

Kur pasqyrohet kontura automatikisht ndryshon kahja e rrotullimit G2/G3 dhe funksionet e kompensimit të instrumentit G41/G42.

Shembull: **MIRROR X**



FUNKSIONI I CIKLIT

Cikli është një varg i punimeve të përcaktuara paraprakisht të cilat makina do ti realizonte automatikisht.

Ciklet mund të jenë:

- standarde
- përdoruese

Ndodhen në **Menu \ Programs \ Standard cyles** ose **Menu \ Programs \ User cycles**

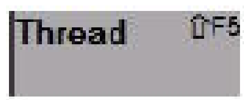
Thirrja e ciklit është me ndihmën e funksionit horizontal të tastit <F4> **Support.** Ciklet gjithashtu mund të thirren edhe me komandën **MCALL**

Ciklet standarde janë:

Cikli për tornim <F2>

Cikli për shpim të thellë <F4>

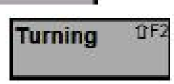
Cikli për prerjen e filetës <F5>



Ndërsa për cikle shfrytëzuese tasti <F6>

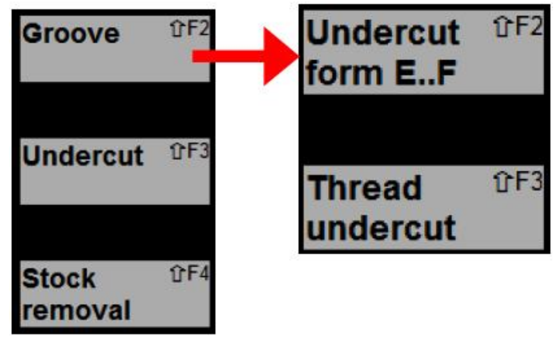
Recompile <F7>

është tasti i cili mundëson drejtimin e parametrave të shkruar të ndonjë cikli përmes menysë së tij.



CIKLET PËR TORNIM

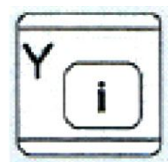
Me aktivizimin e ciklit për tornim <F2> hapet dritare e re e cila ofron lloje të tornimit.



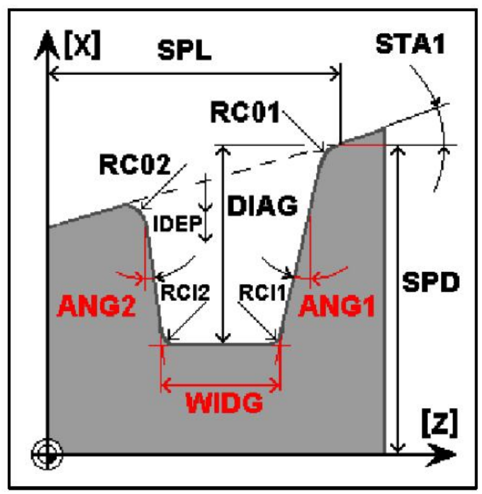
Sinumerik 840D TURN ofron cikle për punimin e kanaleve (**Groove – cyle 93)**, nënprerje (**Undercut – cycle 94, 96**) dhe tornim kontural (**Stock removal – cyle 95**).

GROVE – CYCLE 93 – Cikli për punimin e kanaleve (gropëzimeve) te tornimi <F2>

Paraqitja grafike e të gjitha parametrave aktivizohet me tastin



Cycle params:	CYCLE93	
Start. point	SPD	0.
Start. point	SPL	0.
Width	WIDG	0.
Groove depth	DIAG	0.
Angle	STA1	0.
Flank angle 1	ANG1	0.
Flank angle 2	ANG2	0.
Rad./chamfer	RCO1	0.
Rad./chamfer	RCO2	0.
Rad./chamfer	RCI1	0.
Rad./chamfer	RCI2	0.
Fin. allow.	FAL1	0.
Fin. allow.	FAL2	0.
Infeed depth	IDEPL	0.
Dwell time	DTB	0.
Operation	VARI	5



Star Point → **SPD** – pozicioni fillestar sipas aksit X (pa parashenjë)

Star Point → **SPL** – pozicioni fillestar sipas aksit Z

Width → **WIDG –** gjerësia e kanalit në fund

Groove depth → **DIAG** – thellësia e kanalit relativisht nga pozicioni fillestar (pa parashenjë)

Angle → **STA1** – këndi i konicitetit të aksit të rrotullimit dhe koturës së copës punuese

Flank angle 1 → **ANG1** – këndi anësor në anën e pozicionit shartues (pa parashenjë)

Flank angle 2 → **ANG2** – këndi anësor në anën e kundërt (pa parashenjë)

+Rad./ -chamfer → **RCO1** – rrezja+ / pjerrësimi – i këndit të jashtëm në anën e pikës startuese

+Rad./ -chamfer → **RCO2** – rrezja+ / pjerrësimi – i këndit të jashtëm në anën e kundërt

+Rad./ -chamfer → **RCI1** – rrezja+ / pjerrësimi – i këndit të brendshëm në anën e pikës startuese

+Rad./ -chamfer → **RCI2** – rrezja+ / pjerrësimi – i këndit të jashtëm në anën e kundërt

Fin. Allow → **FAL1** – trashësia përfundimtare e prerjes te punimi i fundit të kanalit

Fin. Allow → **FAL2** – trashësia përfundimtare e prerjes te punimi i fundit të kanalit

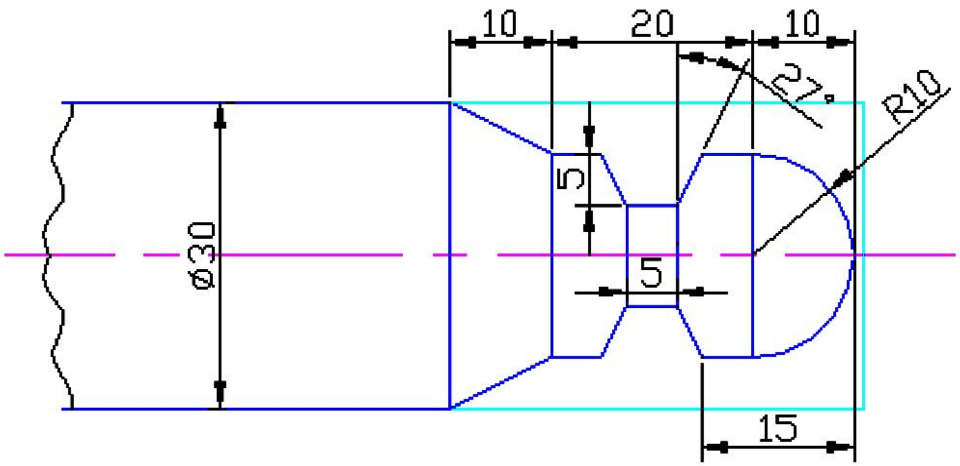
Infeed depth → **IDEPL** – trashësia maksimale e prerjes sipas hapit

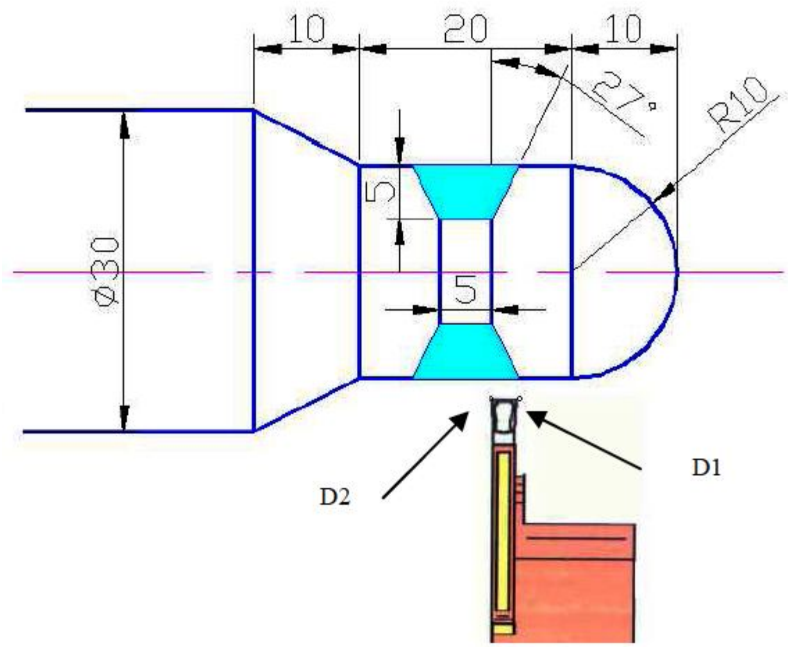
Dwell time **DTB** – koha e pritjes

Operation → **VARI** – varianta të punimit të kanaleve

Ushtrimi 3. Punimi i programit për BOSHTIN 3 (Cikli i punimit të kanaleve – GROOVE)

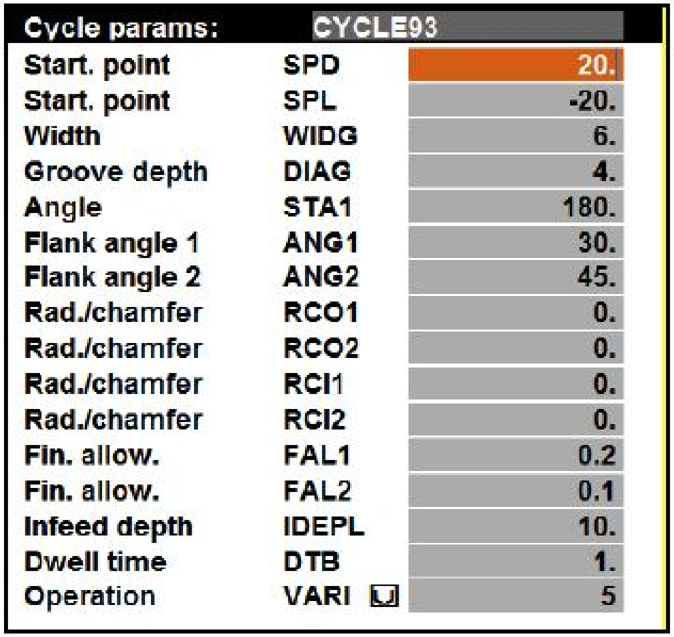
- kanali punohet me thikë për prerje (parting tool) i cili ka numër T6
- regjimet e prerjes për këtë thikë janë v = 80 m/min dhe F 0.2 mm/rr
- duhet të definohen dy korrektime të instrumentit D1 dhe D2, d.m.th. të dy skajet e instrumentit duhet patjetër të maten dhe të shkruhen në bazën e të dhënave (d.m.th. thika mund të punojë kanal nga ana e përparme ose e prapme)
- ana tjetër matëse duhet patjetër të shkruhet si D2 për instrumentin e njëjtë (*New Tool edge*)
- thika duhet të sjellet në pozitën fillestare në koordinatën X25 Z-15
- pamja e ciklit të shkruar CYCLE 93 (20,-15,5,5,0,27,27,0,0,0,0,0,2,0,1,3,1,05)



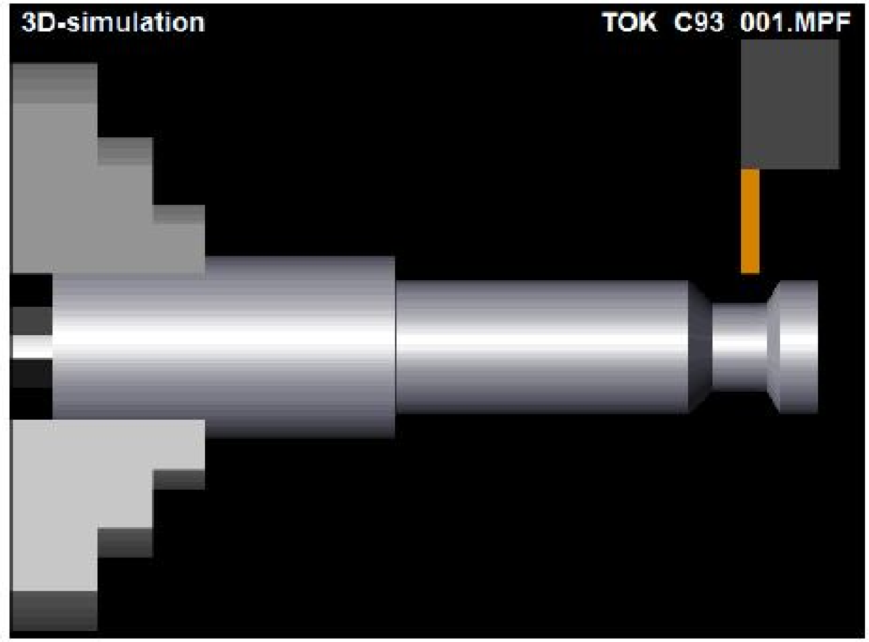


Nr. Rend.	Funksionet e programit												Vërejtje
%MP N9000	G	X	Z	F	S	M	T	D	CR	AR	I	J	
N1	18												
N2	54												
N3	23												
N4	90	0	2	0.05	1500	3	1	1					
N5	1		0										
N6	3	20	-10						10				
N7	1		-30										
N8		30	-40										
N9			-100										
N10	0	32											
N11			100										
N12	90	22	-17.5	0.05	1500	3	2	1					
N13	1	20											
N14		10	-20										
N15	0	22											
N16			-22.5										
N17	1	20											
N19		10	-20										
N20	0	22											
N21			100										
N22						2							

Shembull i plotësimit të tabelës është treguar në figurë



Cycle params:	CYCLE93	
Start. point	SPD	20.
Start. point	SPL	-20.
Width	WIDG	6.
Groove depth	DIAG	4.
Angle	STA1	180.
Flank angle 1	ANG1	30.
Flank angle 2	ANG2	45.
Rad./chamfer	RCO1	0.
Rad./chamfer	RCO2	0.
Rad./chamfer	RCI1	0.
Rad./chamfer	RCI2	0.
Fin. allow.	FAL1	0.2
Fin. allow.	FAL2	0.1
Infeed depth	IDEPL	10.
Dwell time	DTB	1.
Operation	VARI	5



Përshkrimi në NC programin duket kështu:

```
CYCLE93(22,-10,6,4,180,30,45,0,0,0,0,0.2,0.1,10,1,5)
```

UNDERCUT – CIKLI PËR NËNPREREJE (THELLIM) TE TORNIMI <F3>

CYCLE 94 – prodhojnë thellime në përputhje me **DIN 509** të formës **E** dhe **F** për prodhime të gatshme me përmasa më të mëdha se 3 mm

Sipas **DIN509** forma **E** shfrytëzohet vetëm te një sipërfaqe e përpunuar – sipërfaqe periferike.

Forma **F** shfrytëzohet kur janë përpunuar dy sipërfaqe – sipërfaqja periferike dhe mbështetësi.

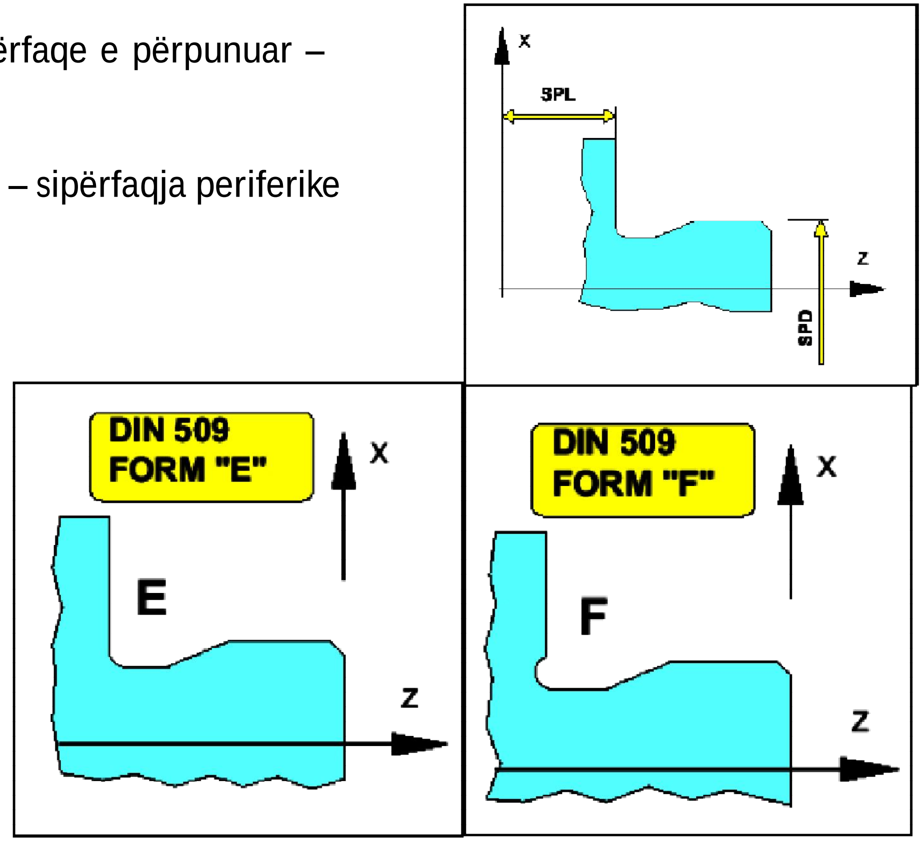
Paraqitja grafike e të gjitha parametrave aktivizohet me tastin 

Star Point **SPD** – pozicioni fillestar sipas aksit X (pa parashenjë)

Star Point **SPL** – pozicioni fillestar sipas aksit Z

From **FROM** – forma e thellimit

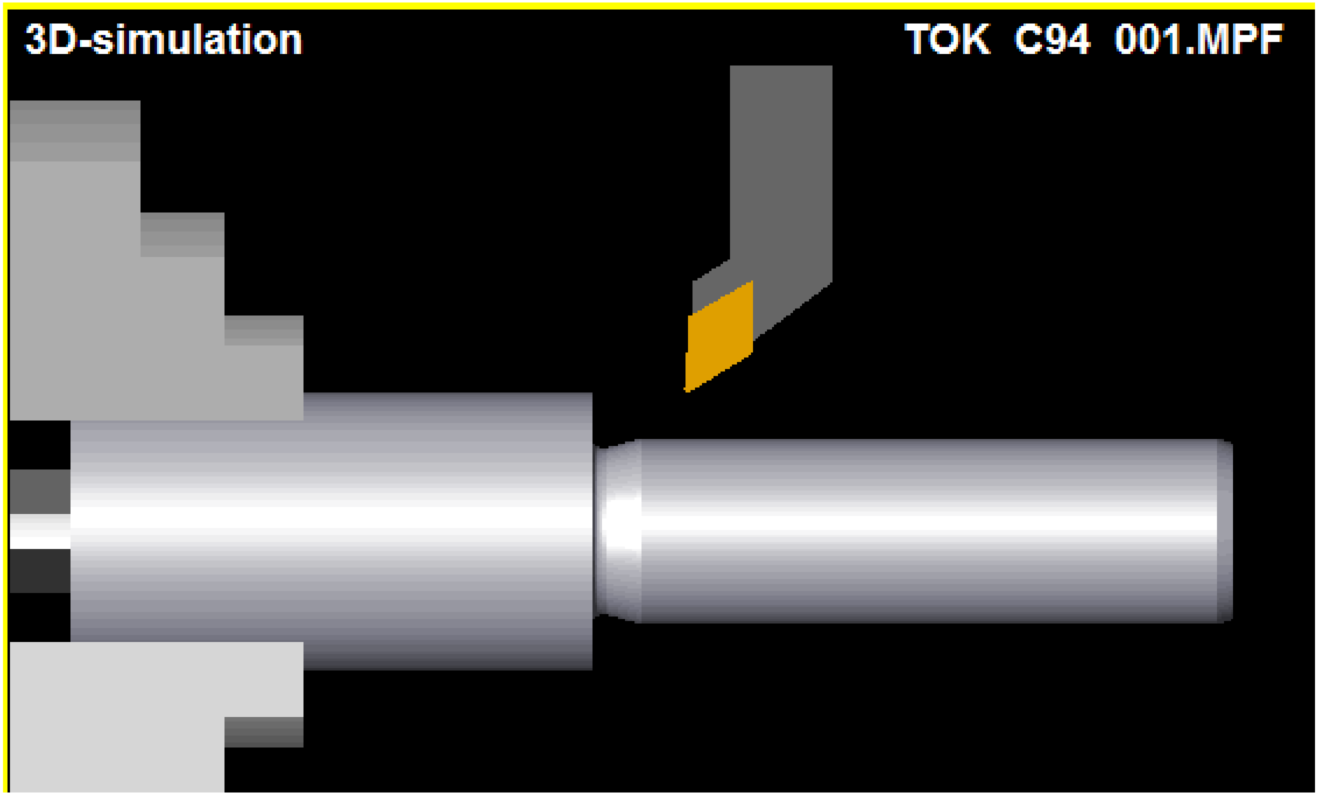
Cycle params:	CYCLE94	
Start. point	SPD	18.
Start. point	SPL	-70.
Form	FORM	E



Shkrimi në programin NC duket kështu:

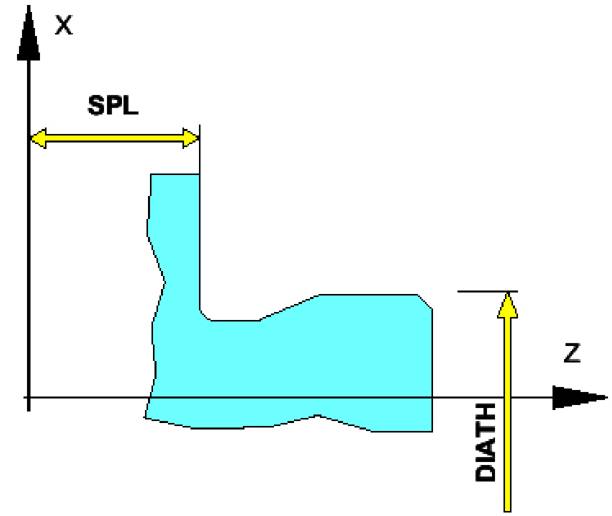
```
CYCLE94(18,-70,"E")
```

Shembulli i figurës



Cikli për prerje te punimi i filetave

Ky cikël – CYCLE 96 – prodhon thellime në përputhje me **DIN 96** të formës **A, B, C** dhe **D** për punimin e ISO filetave metrike me dimensione M3 deri M68.



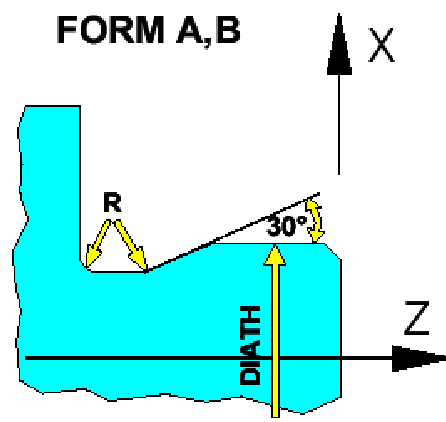
Forma:

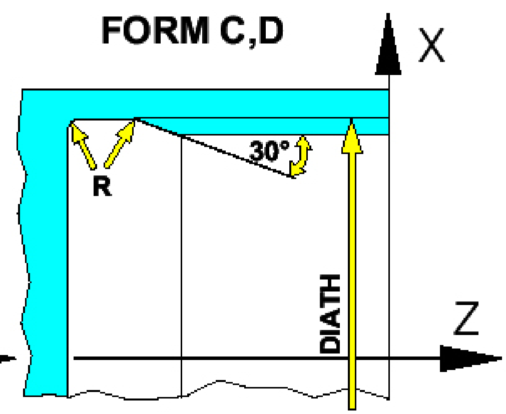
A – fileta të jashtme

B – fileta të jashtme version i shkurtër

C – fileta të brendshme

D – fileta të brendshme versioni i shkurtër





Përshkrimi i ciklit fillon me tabelën kontrolluese e cila përmban emërtimin e ciklit dhe parametrat e tij.

Cycle params:	CYCLE96	
Nominal diam.	DIATH	0.
Start. point	SPL	0.
Form	FORM	A

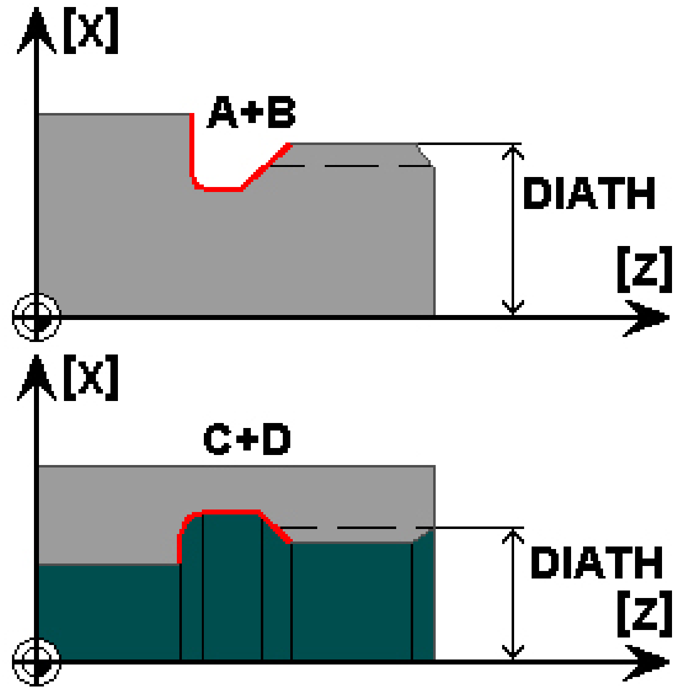
Paraqitja grafike e të gjitha parametrave aktivizohet me tastin

Start.Point **SPL** – pozicioni fillestar sipas aksit Z

From **FROM** – forma e thellimit

Tabela e përshkruar është treguar në figurë:

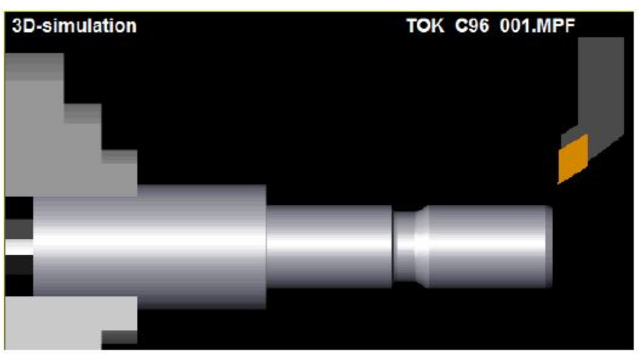
Cycle params:	CYCLE96	
Nominal diam.	DIATH	20.
Start. point	SPL	-40.
Form	FORM	A

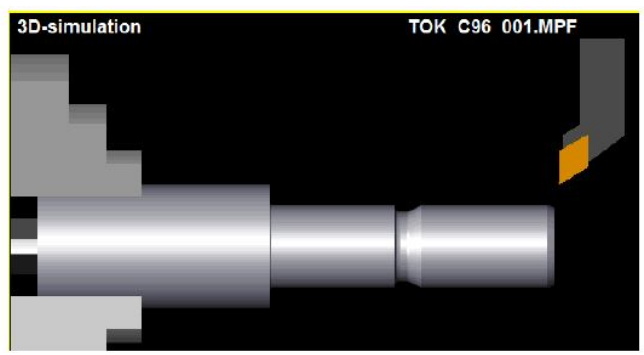


Shkrimi në programin NC duket kështu:

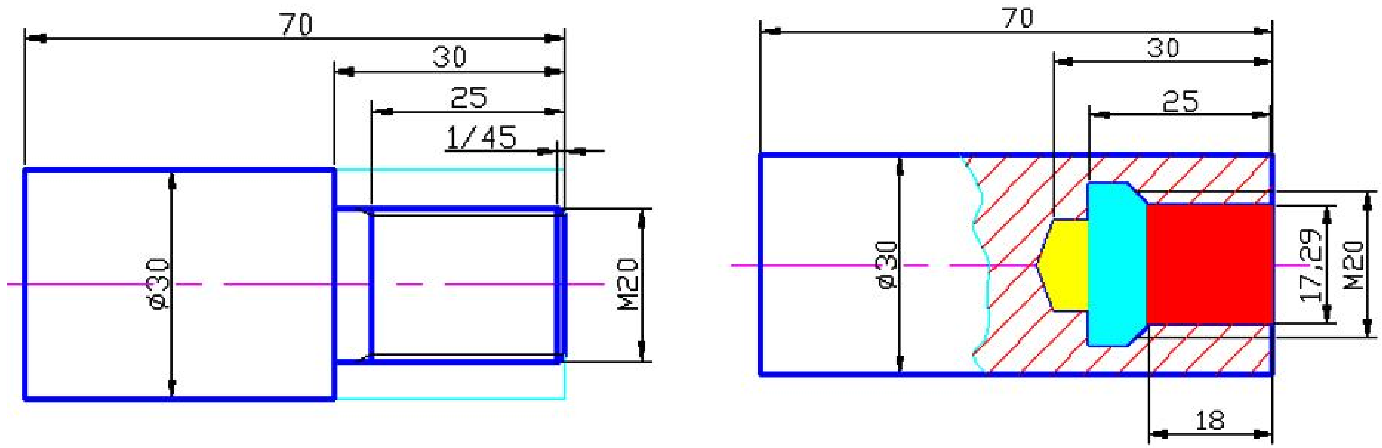
```
CYCLE96(20,-40,"A")
```

Shembulli I simulimit A dhe B





Punimi i filetës së jashtme dhe të brendshme Cycle 97



Për punimin e boshtit me filetë të jashtme dhe të brendshme nevojitet të shfrytëzohen 5 instrumente të ndryshme

- thika e djathtë për tornim të jashtëm për heqje të shtresave prej Ø20x30 dhe rrëzimin e teheve gjegjësisht paraprerje
- thika për punimin e filetës së jashtme M20x25
- punto spirale Ø12 për hapjen e vrimës Ø12x30
- alezatori për zgjërimin e vrimës Ø12 në Ø17.29x18 dhe prerje të brendshme
- thika për prerjen e filetës së brendshme M20

Heqja e shtresave të materialit në kuotën Ø20x30 të bëhet me 4 prerje me trashësi 2x1.2 mm dhe 2x1 mm

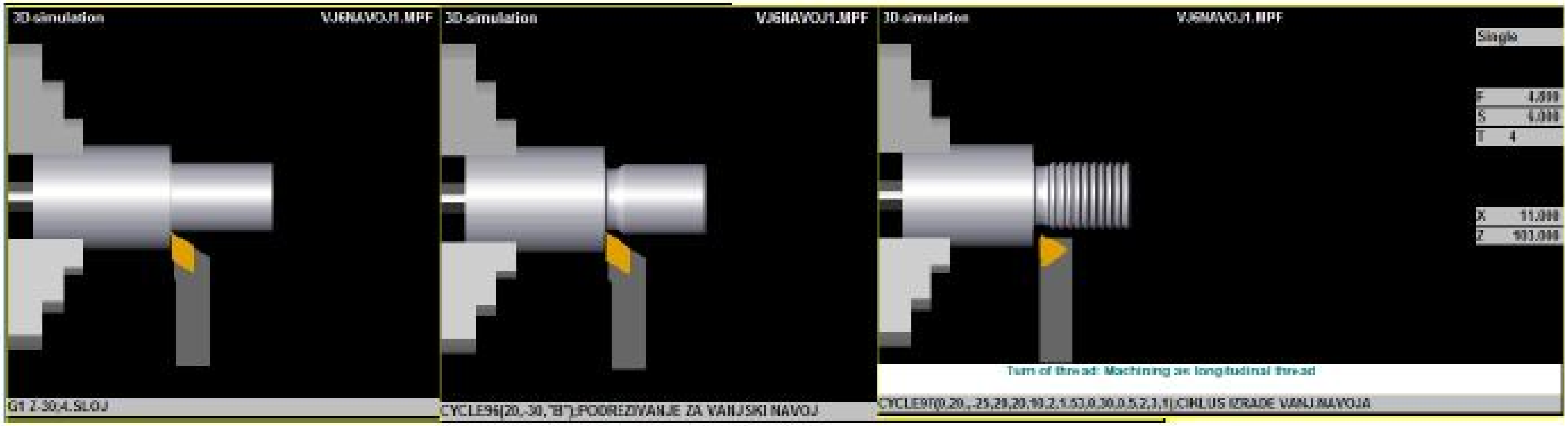
Zgjërimi i vrimës në diametër nominal Ø17.29 (diametri standard për M20) të sjellet me 3 prerje me trashësi 2x1 mm dhe prerja e fundit deri te diametri nominal

Për operacionet 1, 2, 3, 6, 7 të shfrytëzohet funksioni për shpejtësi konstante të prerjes (G97)

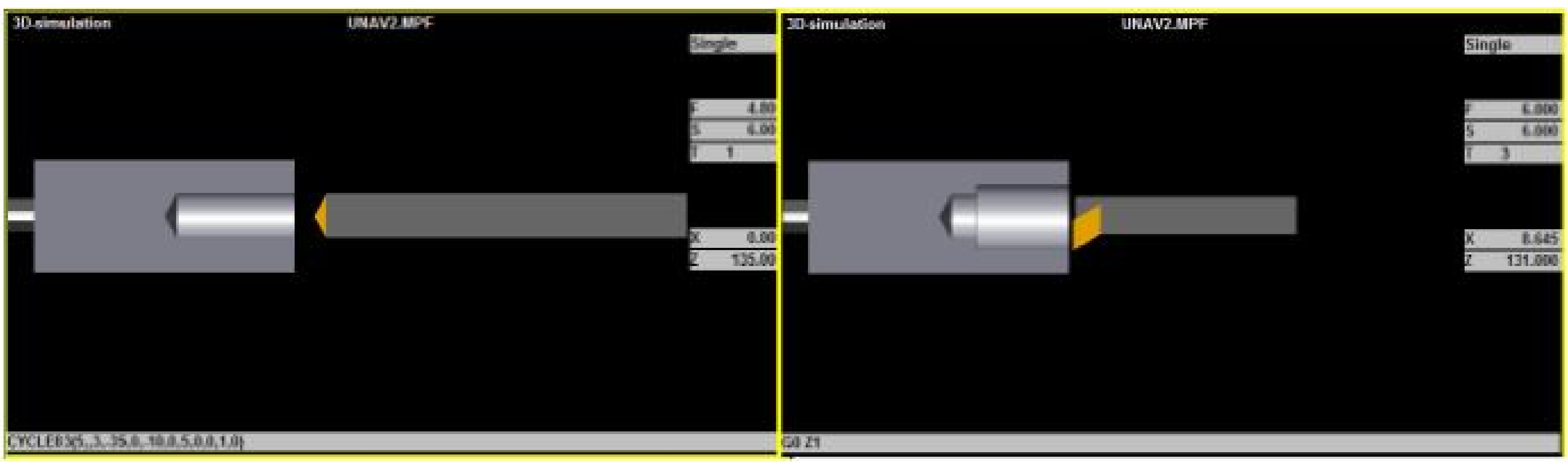
Te thirrja e ciklit për filetim të shqyrtohen parametrat e nevojshëm dhe të shfrytëzohen tabelat për fileta standarde në shtojcë.

Nr. rend	Përshkrimi i kalimeve - operacioneve	Instrumenti	Hapi mm/rr	Numri i rrot. rr/min
1.	Tornimi i jashtëm në Ø20x30	T2	0.08	v=150 m/min
2.	Rrëzimi i tehut 1/45°	T2	0.08	v=150 m/min
3.	Paraprerja e jashtme, forma B	T2	0.08	v=150 m/min
4.	Punimi i filetës së jashtme M20x25	T4		500
5.	Shpimi i vrimës Ø12x30	T1	0.08	1100
6.	Zgjërimi i vrimës Ø17.29x22	T3	0.1	v=150 m/min
7.	Prerja e brendshme, forma D	T3	0.1	v=150 m/min
8.	Punimi i filetës së brendshme M20	T5		500

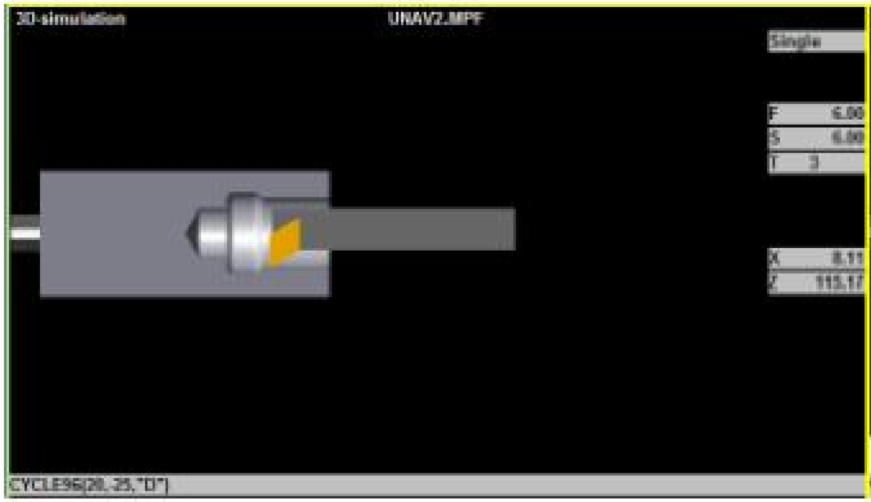
Pamja e simulimeve për filetë të jashtme

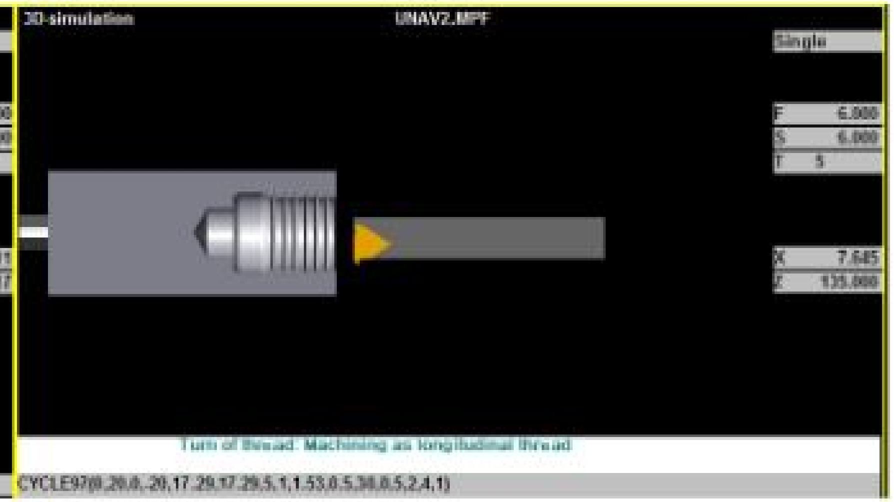


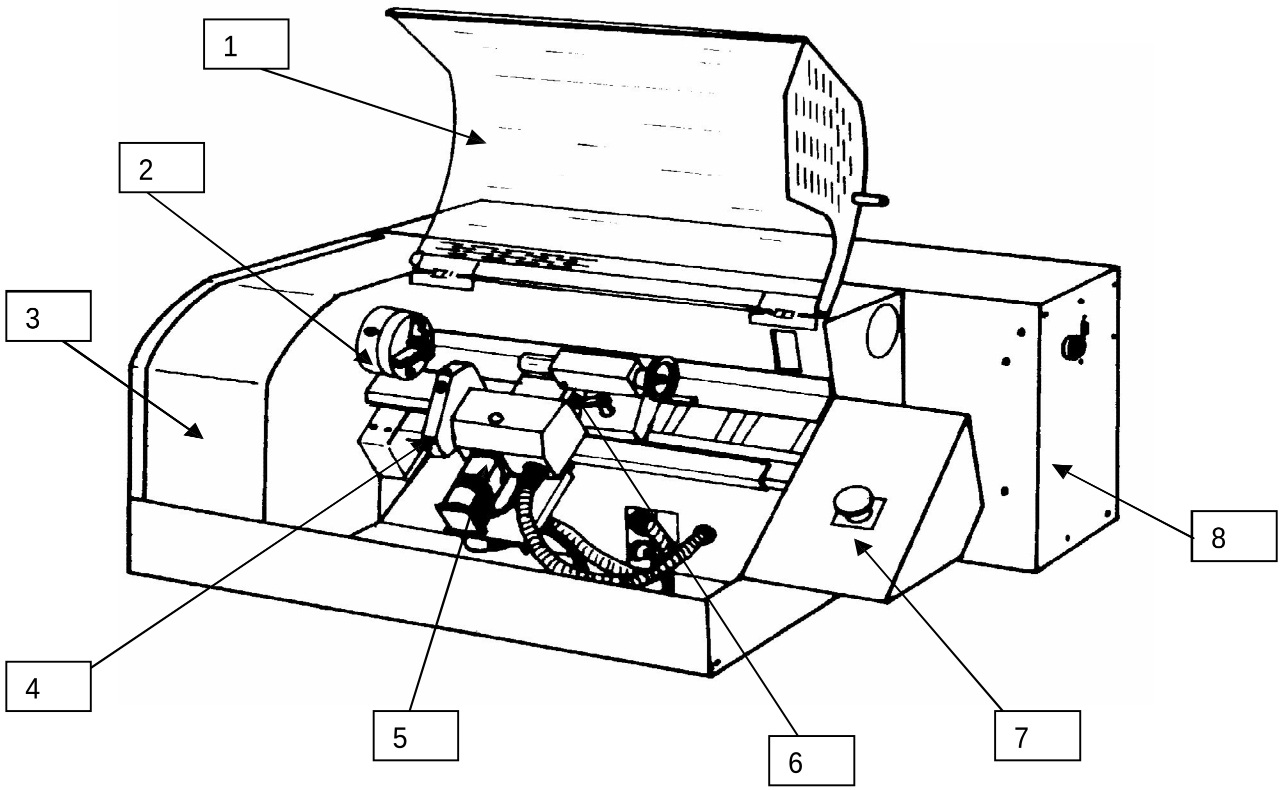
Pamja e simulimeve për filetë të brendshme



Pamja e simulimeve për filetë të brendshme







1. Kapaku mbrojtës
2. Koka për shtrëngim
3. Motori kryesor
4. Bartësi i veglave-koka revolverike
5. DC- Motorët ndihmës
6. Kaluçi
7. Siguresa
8. Pjesa elektrike e pajisjes

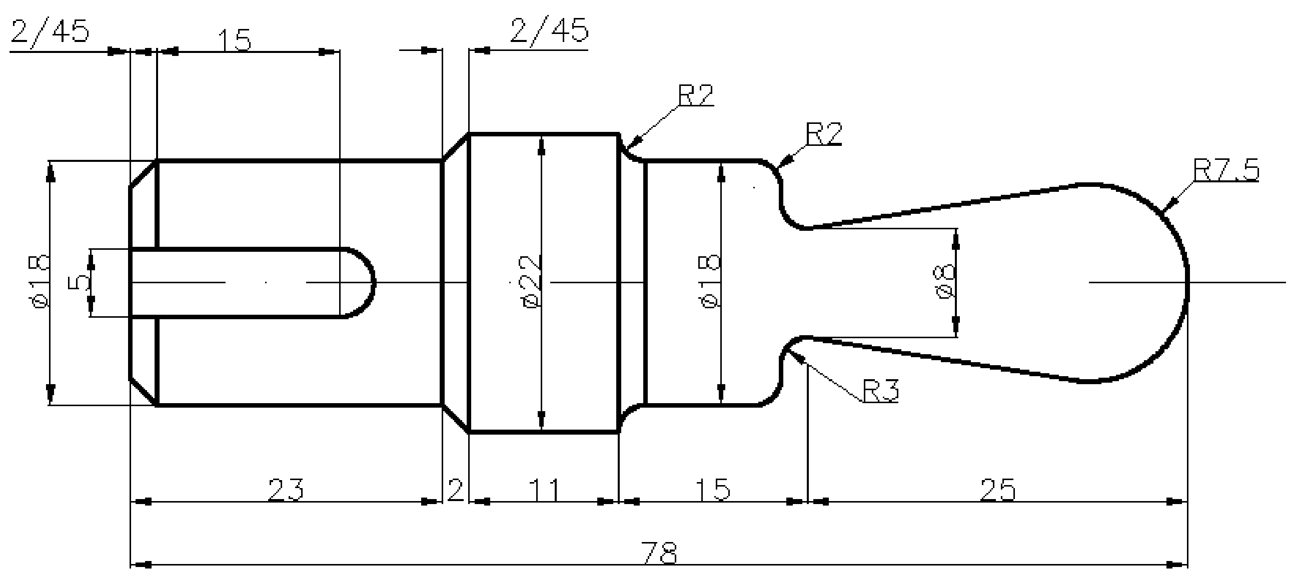
Emco Turn 55

PICOMILL CNC

Makina frezuese **CNC – x.mill 900**

Detyrë:

1.Për detalin e dhënë, të hartohet lista programuese (programi burimor) për makinën tornuese



Nr. Rend.	Përshkrimi i vendosjes - operacionit	Instrumenti prerës	Hapi mm/min	Numri rrotullimeve rr/min
1	Shtrëngimi	Koka shtr.	0.05	1500
2	Tornim I ashpër gjatësor	ISO 6	0.05	1200
3	Përpunim I radiusit	ISO 6	0.05	1500

%MP	G		X	Z	F	S	M	T	D	CR	AR	I	J
N9000													
N1	G55												
N2								T1					
N3									D1				
N4	G0	G90	X0	Z2	F0.05	S1500	M3						
N5	G1		X0	Z0									
N6	G3		X14.5	Z-7.25						CR=7.25			
N7	G1		X8	Z-25									
N8	G2		X14	Z-28						CR=3			
N9	G3		X18	Z-30						CR=2			
N10	G1		X18	Z-40									
N11	G2		X22	Z-42						CR=2			
N12	G1		X22	Z-53									
N13	G0		X24	Z-53									
N14			X24	Z100									
N15								T2					
N16									D1				
N17	G0	G90	X14	Z2	F0.05	S1200	M3						
N19	G1		X14	Z0									
N20	G1		X18	Z-2									
N21	G1		X18	Z-24									
N22	G1		X22	Z-26									
N22	G1		X24	Z100									
N24							M2						

Përshkrimi i vendosjes - operacionit

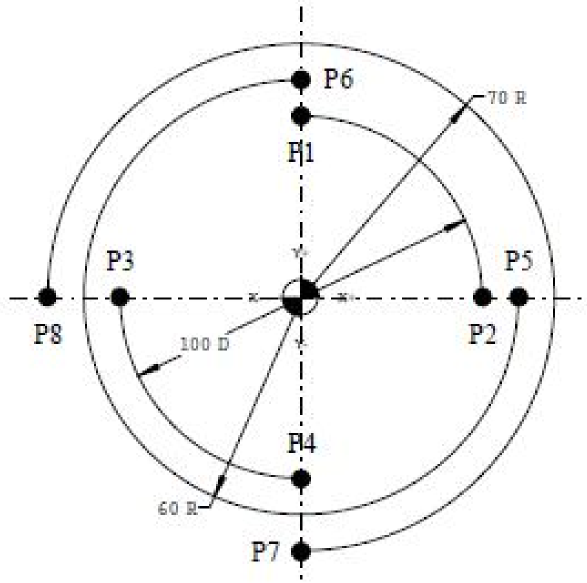
Nr. Rend.	**Përshkrimi i vendosjes - operacionit**	**Instrumenti prerës**	**Hapi** mm/min	**Numri i rrotullimeve** rr/min
1	Shtrëngimi	Koka shtr.		
2	Hapja e kanalit të pykës	Freza Ø5	F=80 [mm/min]	S=1500 [min -1]

Programimi

%MP N9000	G		X	Y	Z	F	S	M	T	D	CR	AR	I	J
N1	G54													
N2									T1					
N3										LM06				
N4	G0	G90	X0	Y0	Z5	F80	1500	M3						
N5	G1		X0	Y0	Z-2									
N6	G1		X15	Y0	Z-2									
N7	G0		X15	Y0	Z20									
N8								M2						

Circular Programmed Movements – e.g. 1

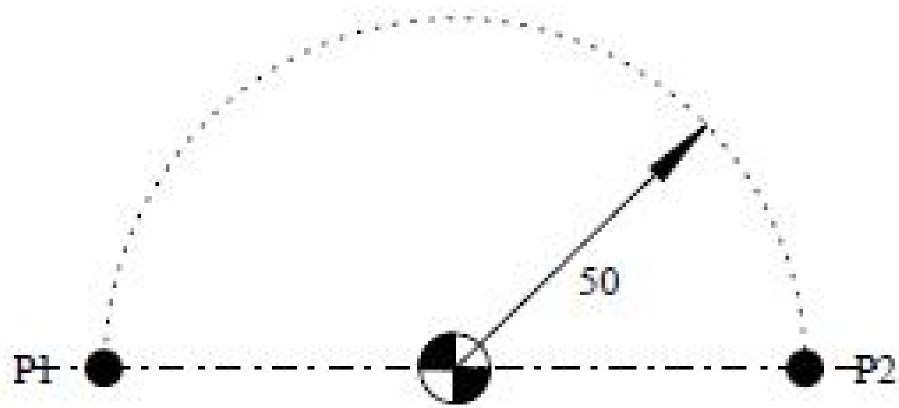
(Absolute)



1	G90 G0 X0 Y50 G2 X50 Y0 P50
2	G90 G0 X-50 Y0 G3 X0 Y-50 P50
3	G90 G0 X60 Y0 G2 X0 Y60 P-60
4	G90 G0 X0 Y-70 G3 X-70 Y0 P-70

Circular Programmed Movements – e.g. 2

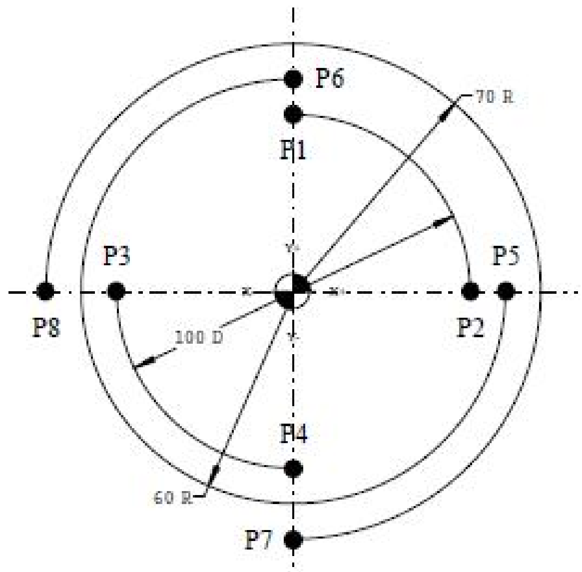
(Absolute)



1	G90 G0 X-50 Y0
	G2 X50 P50
2	G90 G0 X50 Y0
	G3 X-50 P50

Circular Programmed Movements – e.g. 4

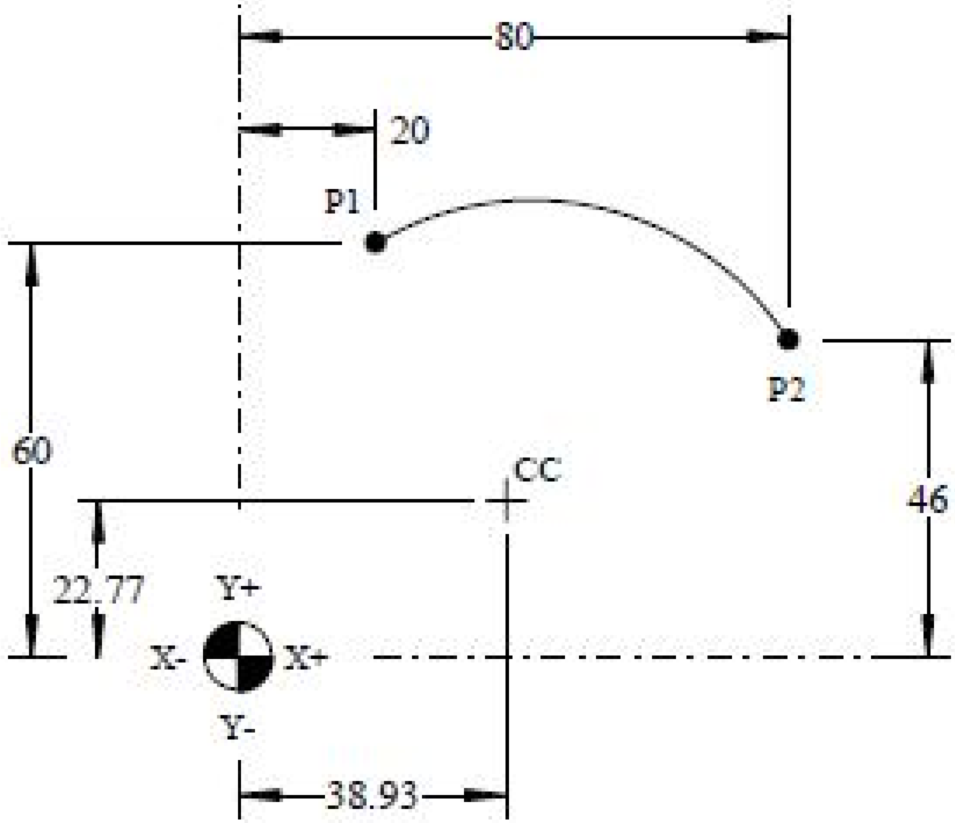
(Incremental)



1	G90 G0 X50 Y0 G91 G2 X50 Y-50 P50	G90 G0 X0 Y50
2	G90 G0 X-50 Y0 G3 X-50 Y-50 P50	G91 G3 X50 Y-50 P50
3	G90 G0 X60 Y0 G2 X-60 Y60 P-60	Nese: P60 ??
4	G90 G0 X0 Y-70 G3 X-70 Y70 P-70	Nese: P70 ??

Circular Programmed Movements – e.g. 5

X, Y, I & J (Absolute)



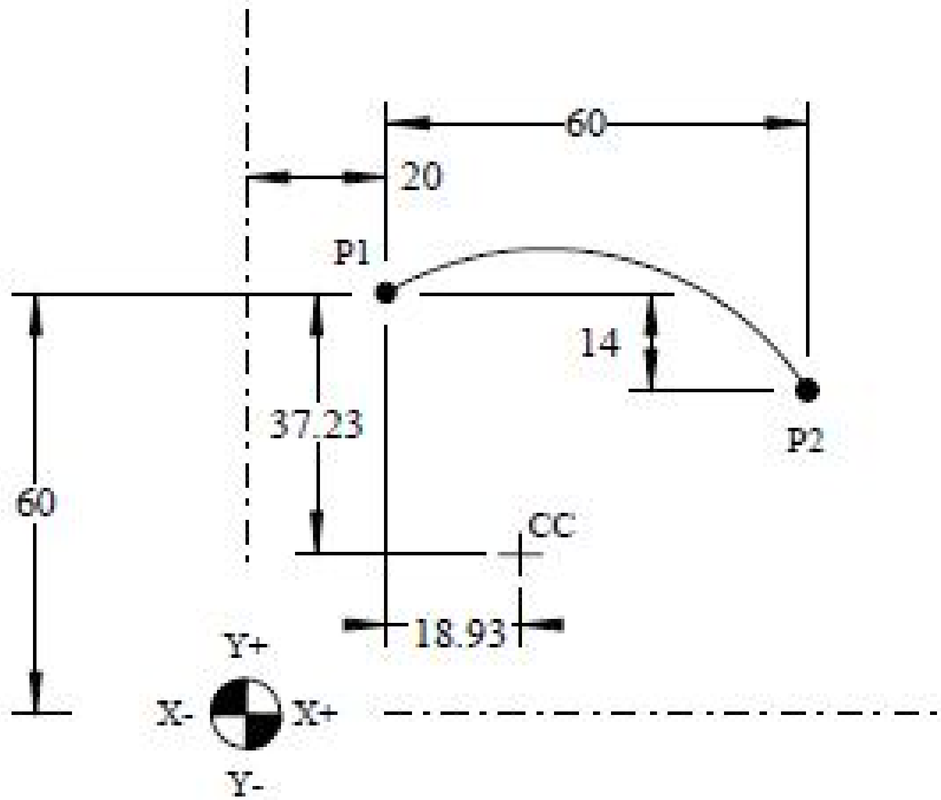
G0 G90 X20 Y60
G2 X80 Y46 I38.93 J22.77

****

G0 G90 X20 Y60
G3 X20 Y60 I38.93 J22.77

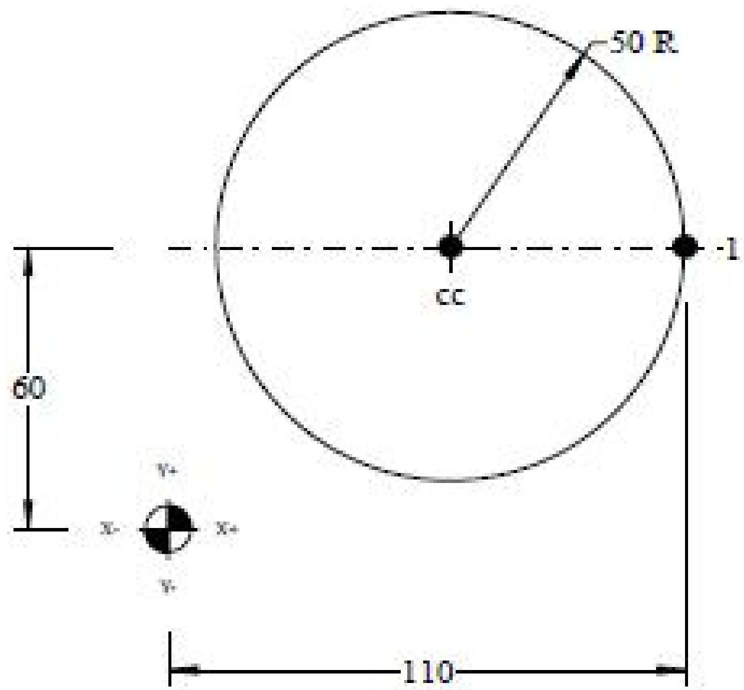
Circular Programmed Movements – e.g. 6

X, Y, I & J (Incremental)



G0 G90 X20 Y60
G2 G91 X60 Y-14 I18.93 J-37.23

Full Circular Movements – e.g.8



G0 G90 X110 Y60
G2 X110 Y60 I60 J60

Incremental

Note:

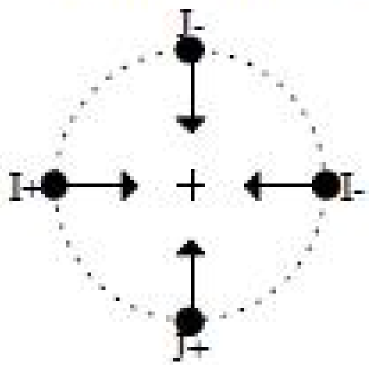
The end points and circle centre positions are taken from the "Start point"

G0 G90 X110 Y60
G2 G91 X0 Y0 I-50 J0

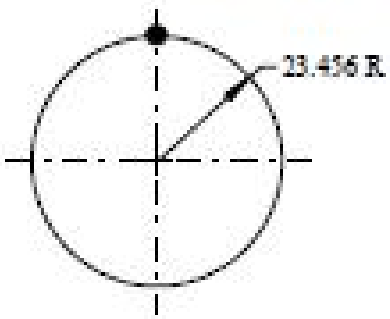
Full Circular Movements – e.g.9

By selecting a pole point of a circle (12, 3, 6 or 9 o'clock position) and using an "Incremental line of program" to create a full circle, all values on this line of program will have a zero value except for the I or J axis on the appropriate pole axis which will represent the radius to be produced:

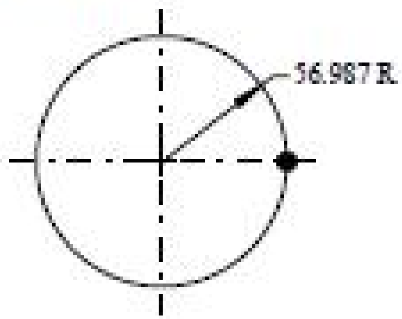
i.e. G91 G2 X0 Y0 I0 J-50



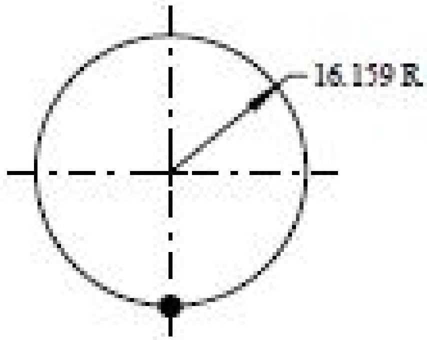
Create the line of circle program for each of the following quadrant points in the diagrams below using "Clockwise".



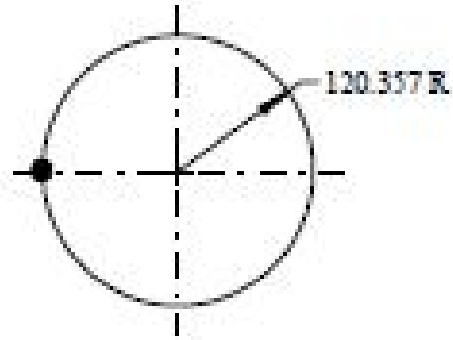
G91 X0 Y0 I0 J-23.456



G91 X0 Y0 I-56.987 J0

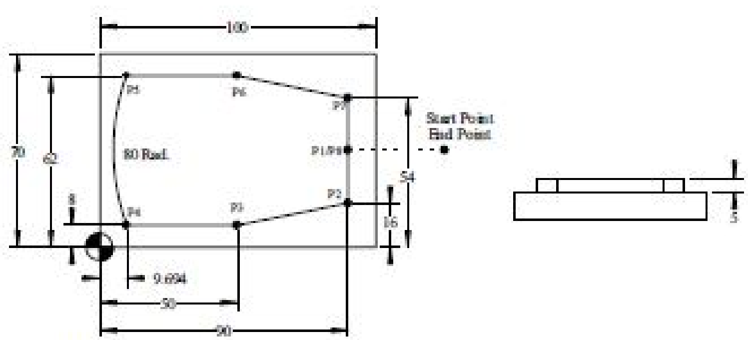


G91 X0 Y0 I0 J16.159



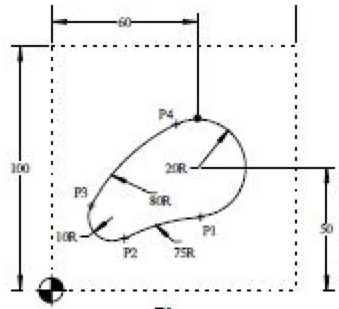
G91 X0 Y0 I120.357 J0

Compensation e.g. 1

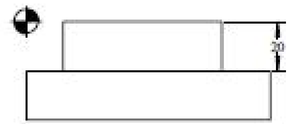



```
: T1 M6
G0 G90 G40 G71 G17 G94
X125 Y35 Z100 H1 S? M3
Z5
G1 Z-5 F?
G41 X90 M8
Y16
X50 Y8
X9.694
G2 Y62 P80
G1 X50
X90 Y54
Y35
G40 X125
G0 G90 Z100 M30
```

Compensation e.g. 2



Plan



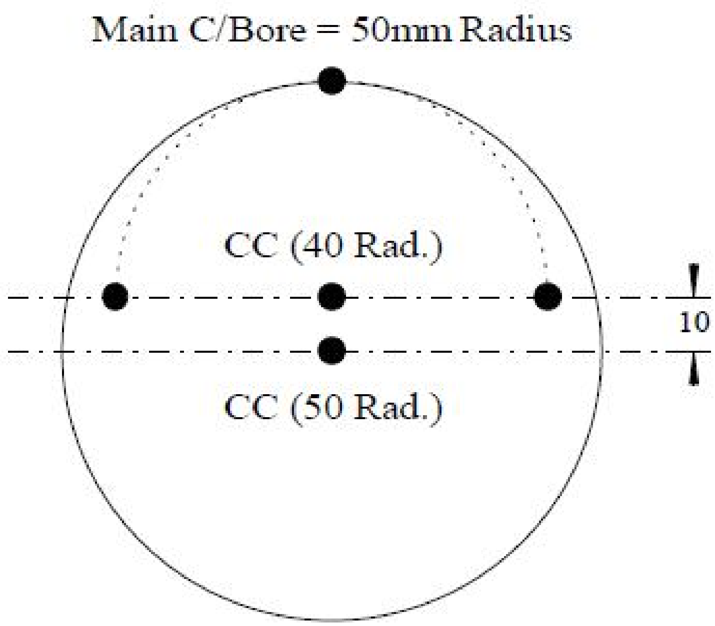
Front

	P1	P2	P3	P4
X	61.11	29.738	16.257	51.266
Y	30.031	21.194	34.854	67.992

```
: T? M6
G0 G90 G40 G71 G17 G94
X-50 Y-120 Z100 H1 S? M3
Z5
G1 Z-20 F?
G41 Y70
X60 M8
G2 X61.11 Y30.031 P20
G3 X29.738 Y21.194 P75
G2 X16.257 Y34.854 P10
X51.266 Y67.992 P80
X60 Y70 P20
G1 X150
G40 Y120
G0 G90 Z100
M30
```

Programmable Cutter Radius Compensation

Circle Tangent inside a full Circle



Main Radius = AR
Arc ON/OFF Radius = SR
CC Difference = YD

Note:

Make the **Approach & Departure Arc** a value less than the original radius to be produced, greater than the cutter radius being used, a radius value which can be subtracted from the original arc to leave a whole number for the "CC Difference" and attached to one of the pole points as the example above.

i.e.

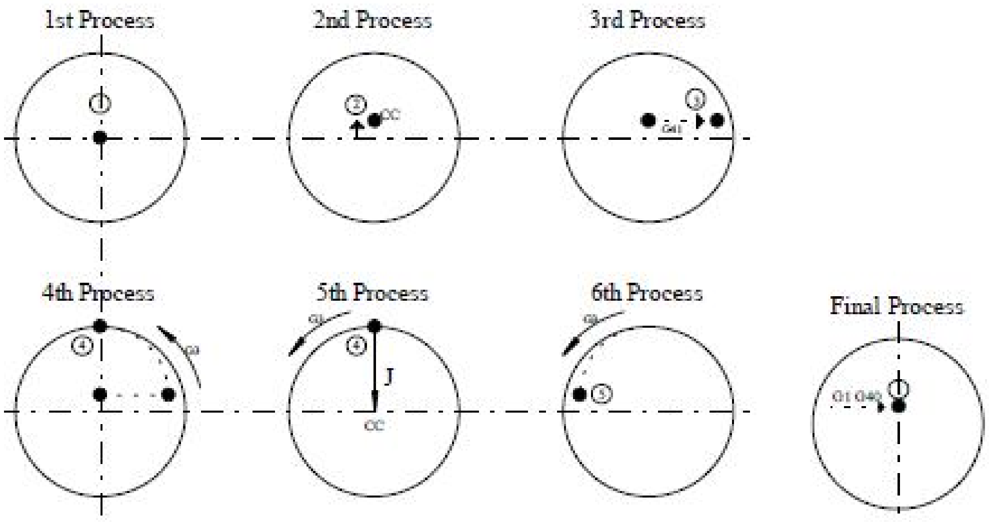
SR = Radius less than AR

YD = AR – SR (i.e. 121.946 – 101.946 = 20)

As above graphical example
AR = 50
SR = 40
YD = 50 - 40 = 10

Programmable Cutter Radius Compensation

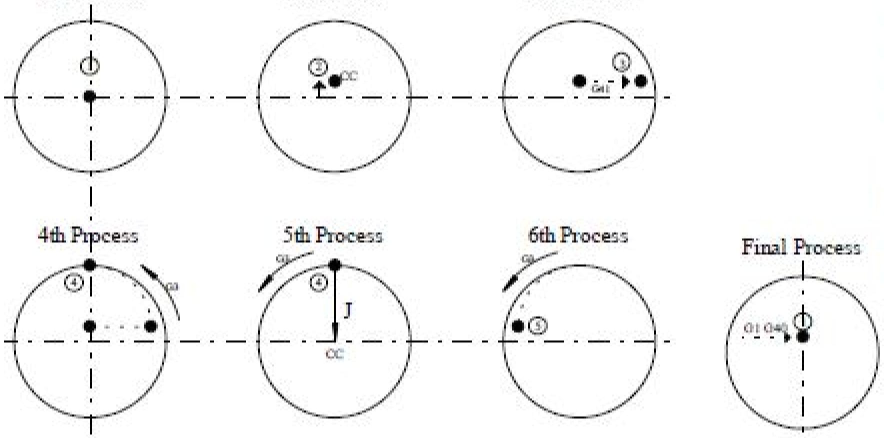
Circle Tangent



: T? M6 ;Toolchange line - 25mm Endmill cutter
G0 G90 G40 G71 G17 G94 ;Safety default line
X0 Y0 Z100 H? S? M3 ;Absolute Start Point – Centre of Actual radius – position 1
Z5 ;Rapid to a position above material
G1 Z-? F? ;Feed to required cut depth before compensation has been applied
G91 Y(YD) ;Incremental move to centre of Arc on/off as calculated – position 2
G41 X(SR) M8 ;Move to point 3 - Apply compensation incrementally
G3 X-(SR) Y(SR) P(SR) ;Move to position 4 - Arc On.as SR Rad.
X0 Y0 I0 J-(AR) ;Move 360 Degrees back to position 4 by Radius of AR
X-(SR) Y-(SR) P(SR) ;Move to position 5 - Arc Off as SR Rad.
G1 G40 X(SR) ;Move to start position cancelling compensation
G0 G90 Z100 ;Move to Absolute safe height above material after comp is cancelled
M30 ;End program

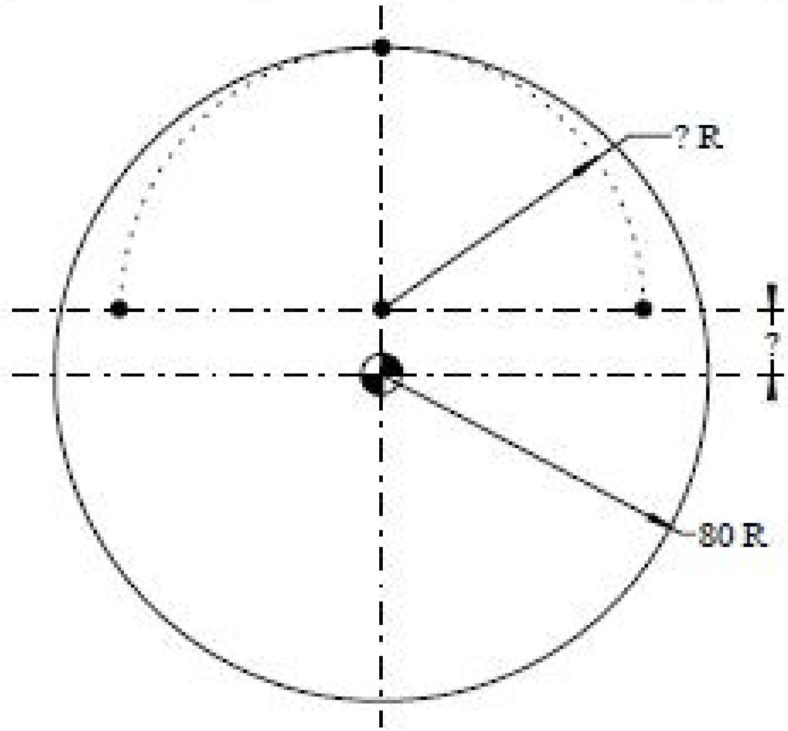
Programmable Cutter Radius Compensation

Circle Tangent



: T? M6 ;Toolchange line - 25mm Endmill cutter
G0 G90 G40 G71 G17 G94 ;Safety default line
X0 Y0 Z100 H? S? M3 ;Absolute Start Point – Centre of Actual radius – position 1
Z5 ;Rapid to a position above material setting length offset
G1 Z-? F? ;Feed to required cut depth before compensation has been applied
G91 Y10 ;Incremental move to centre of Arc on/off as calculated – position 2
G41 X40 M8 ;Move to point 3 - Apply compensation incrementally
G3 X-40 Y40 P40 ;Move to position 4 - Arc On.as SR Rad.
X0 Y0 I0 J-50 ;Move 360 Degrees back to position 4 by Radius of AR
X-40 Y-40 P40 ;Move to position 5 - Arc Off as SR Rad
G1 G40 X40 ;Move to start position cancelling compensation
G0 G90 Z100 ;Move to Absolute safe height above material after comp is cancelled
M30 ;End program

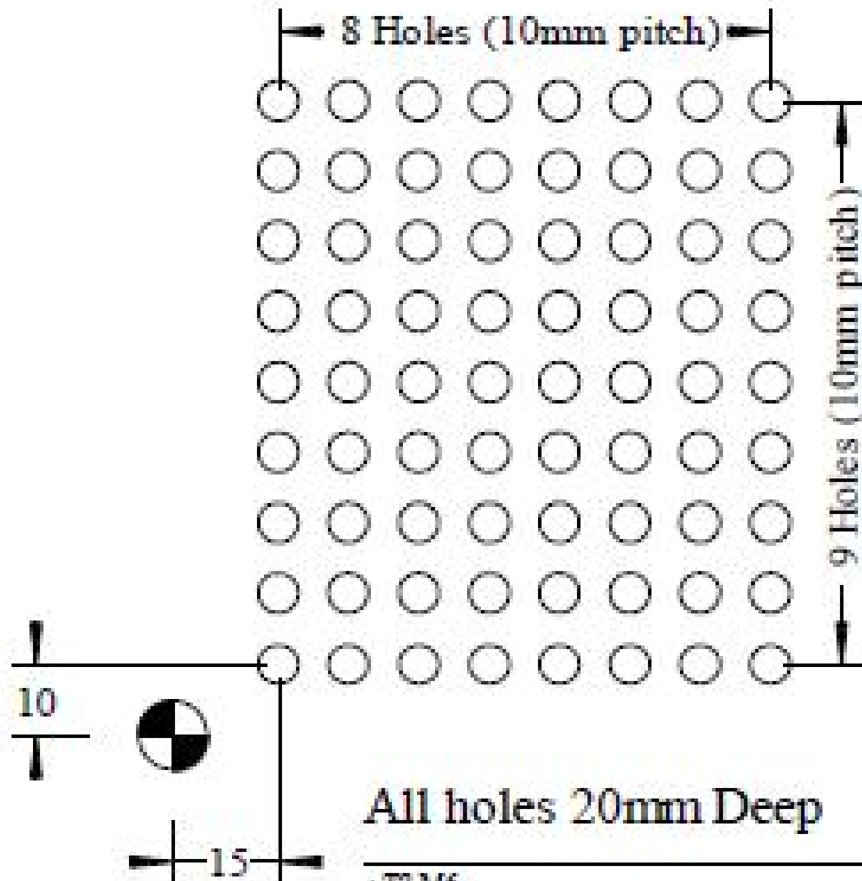
Circle Tangent Compensation e.g. 1




```
: T? M6
G0 G90 G40 G71 G17 G94
X0 Y0 Z100 H1 S? M3
Z5
G1 Z-? F?
G91 Y20
G41 X60 M8
G3 X-60 Y60 P60
X0 Y0 I0 J-80
X-60 Y-60 P60
G1 G40 X60
G0 G90 Z100
M30
```

Grid Pattern Cycle

G38



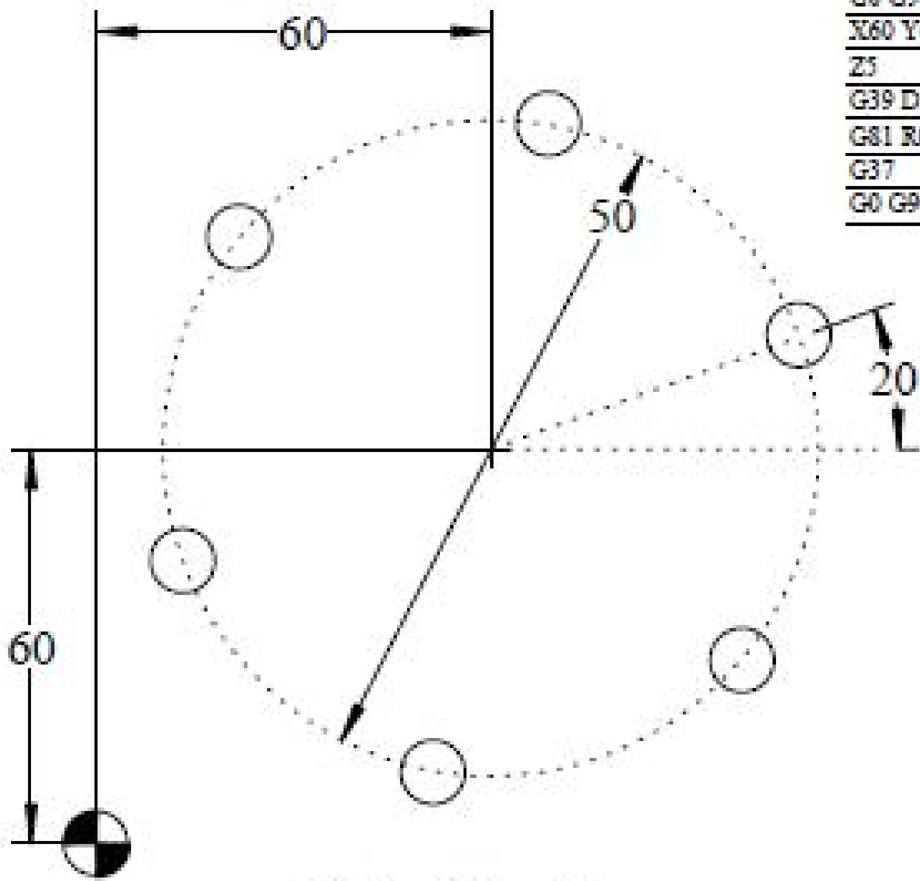
All holes 20mm Deep

```
: T? M6
G0 G90 G40 G71 G17 G94
X15 Y10 Z100 H? S? M3
Z5
G38 U10 I8 J9 V10
G81 R0 Z-20 F? M8
G37
G0 G90 Z100 M30
```

P.C.D. Pattern Cycle

G39

```
: T7 M6
G0 G90 G40 G71 G17 G94
X60 Y60 Z100 H7 S7 M3
Z5
G39 D50 P20 K6
G81 R0 Z-20 F7 M8
G37
G0 G90 Z100 M30
```



Holes 20mm Deep

Linear Motion systems

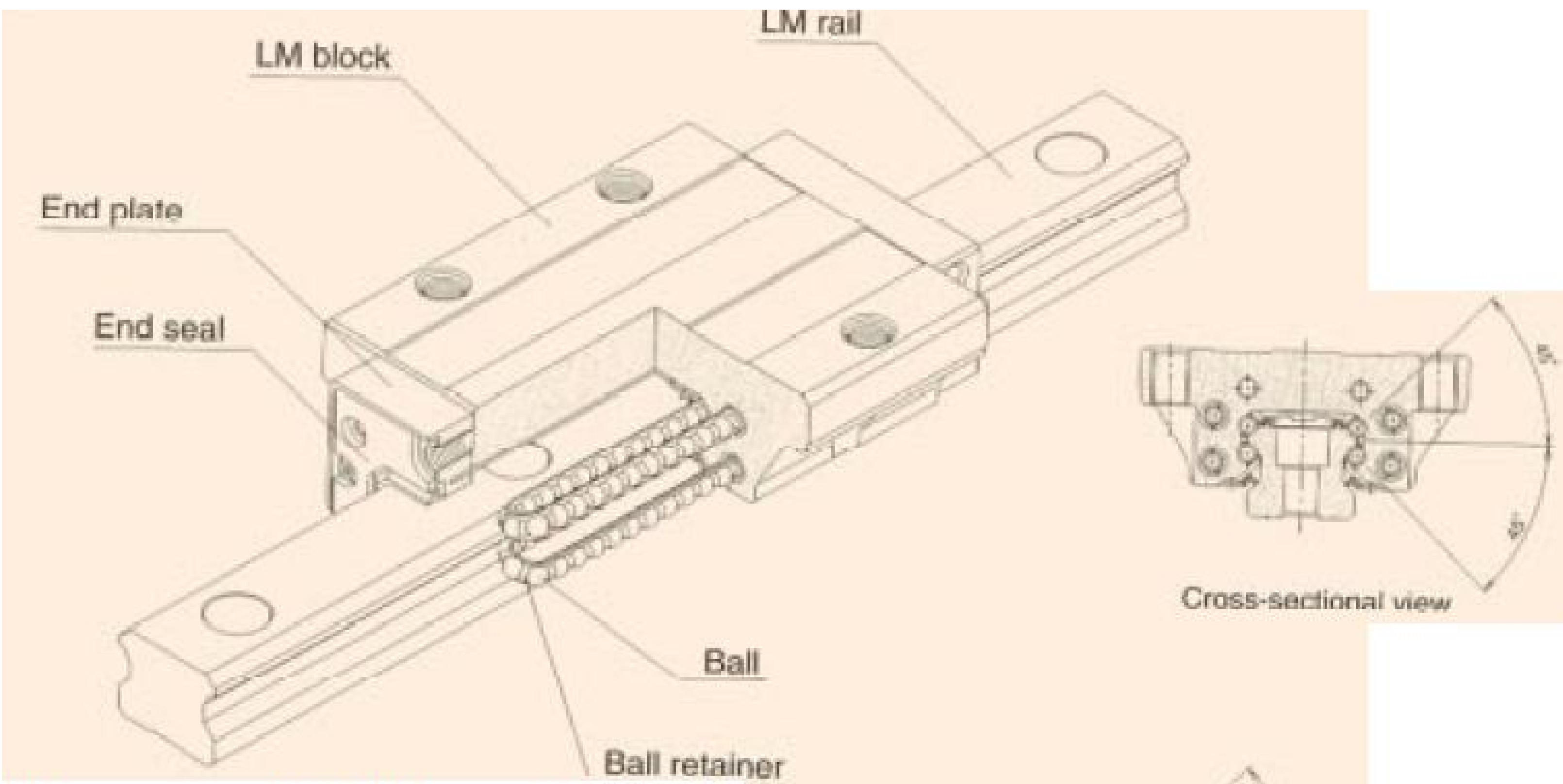


Fig. 10.16 — Antifriction guideways used in CNC machine tools (Courtesy THK Co. Ltd., Japan).

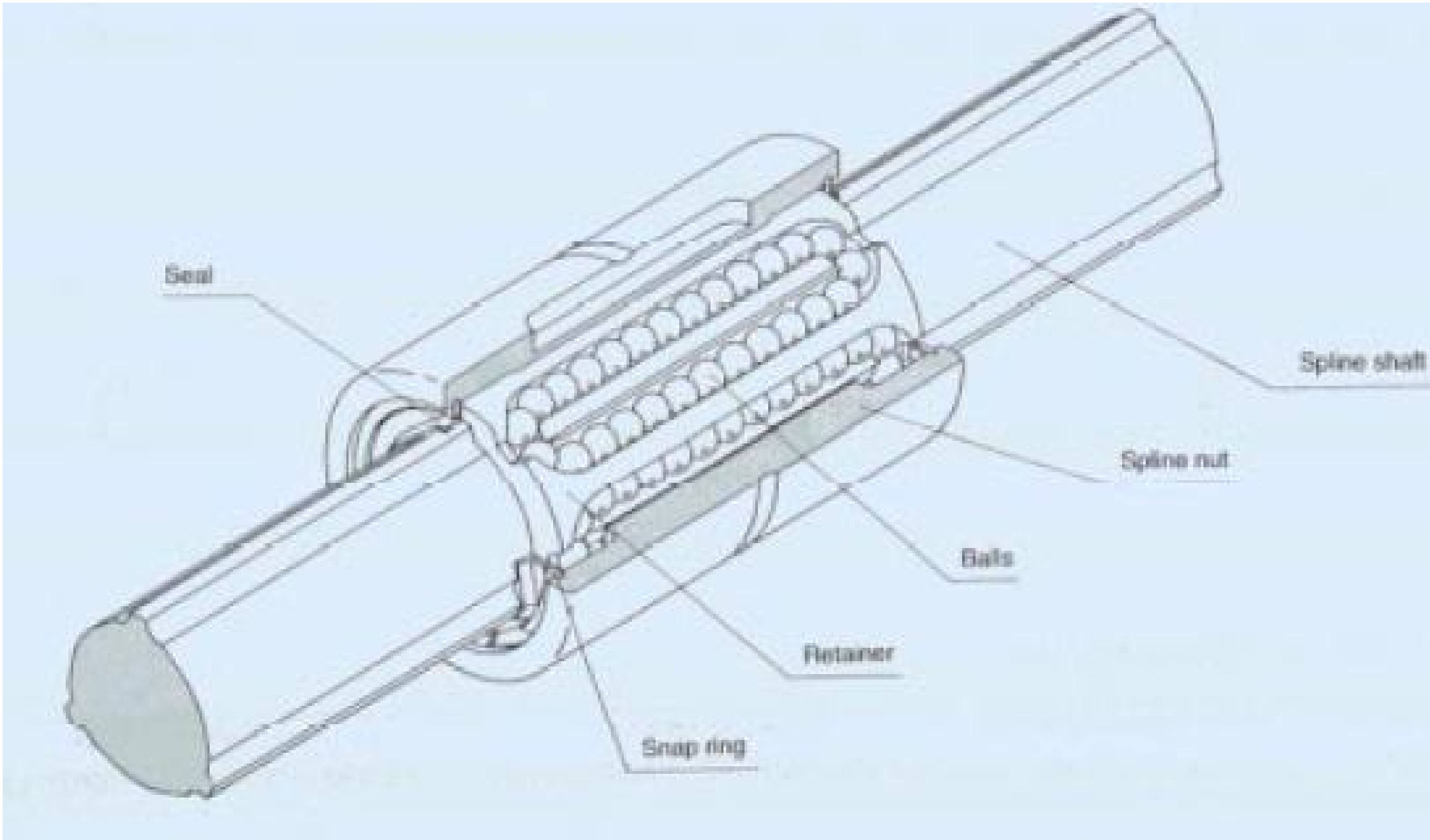


Fig. 10.17 — Ball bush used for linear movement in CNC machine tools (Courtesy THK Co. Ltd., Japan).

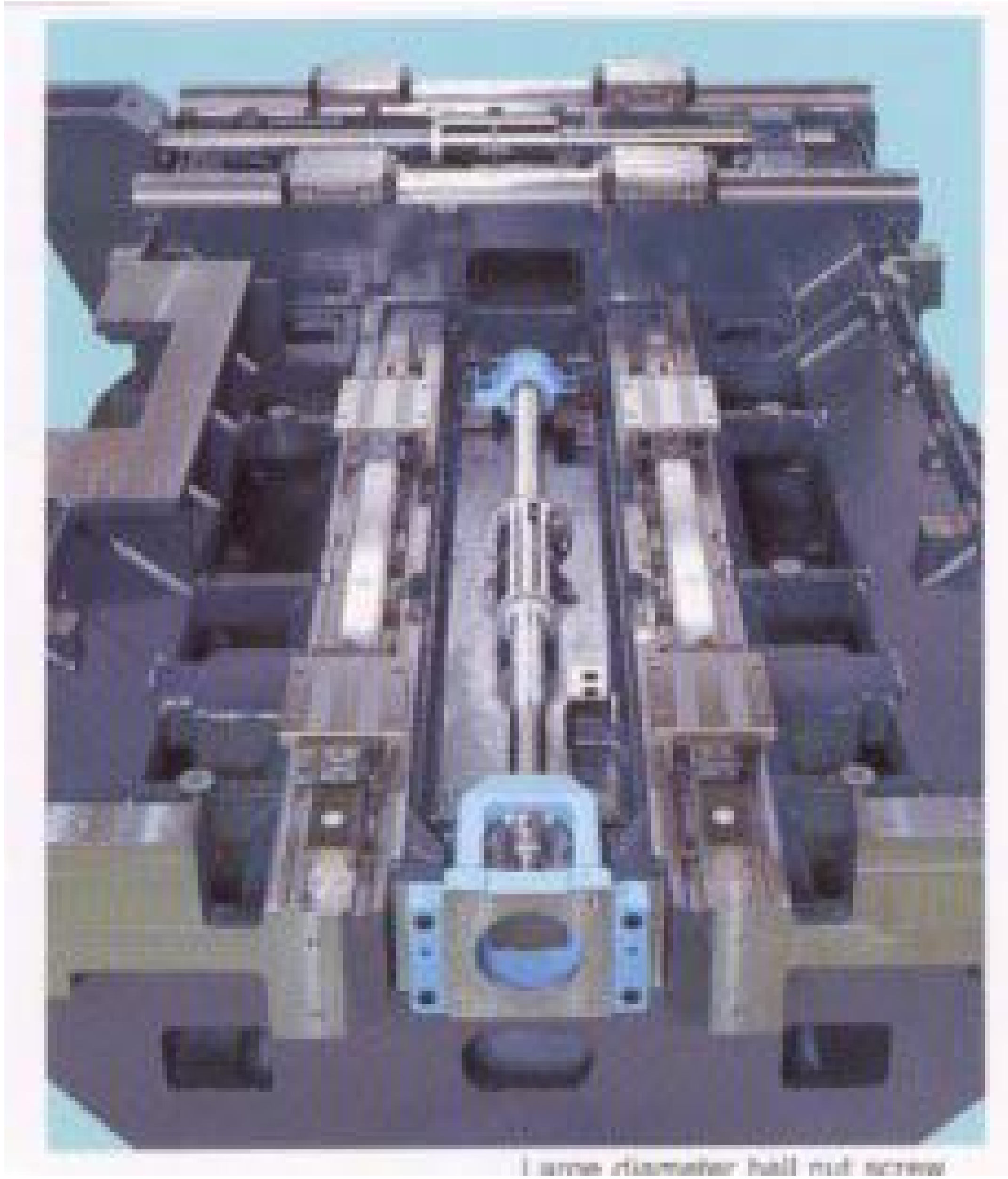
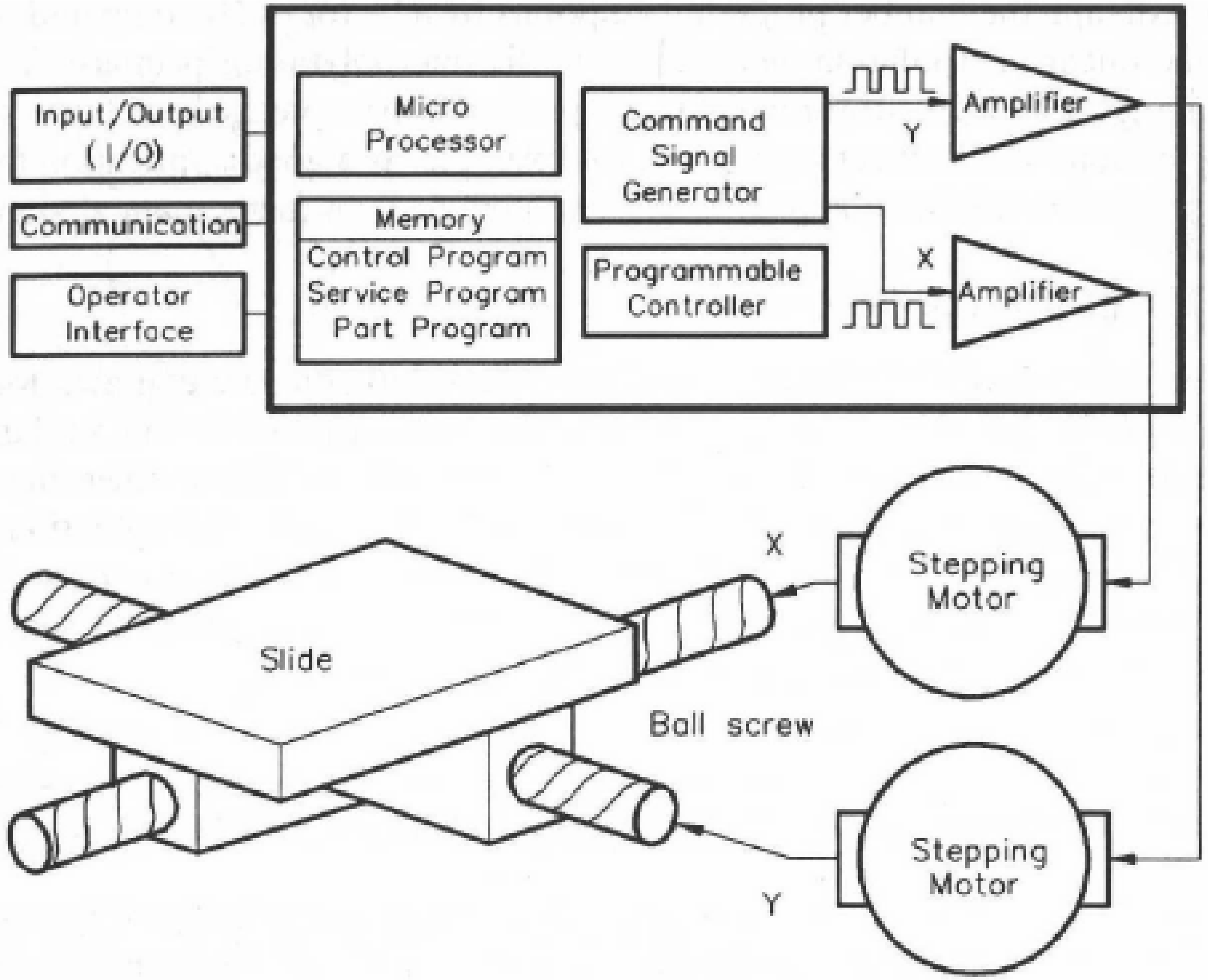


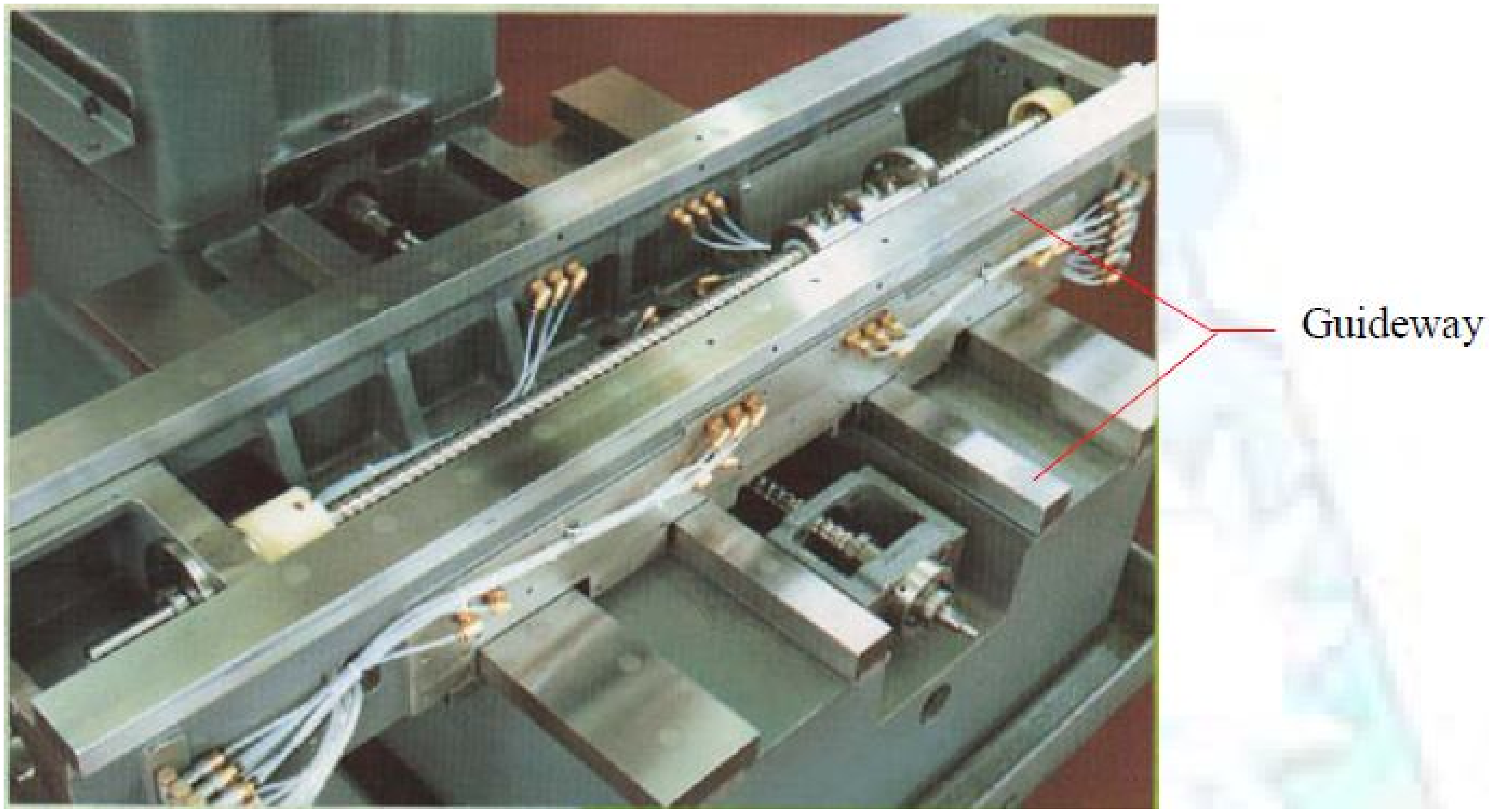
Fig. 10.18 Use of recirculating ball screw and the LM device for axis movement in the bed of a CNC machine tool (Courtesy Makino Milling Machines Co. Ltd., Tokyo, Japan)

Open-loop Control Systems

Open-loop systems normally use stepping motors as the drive devices to move the machine slide. Due to the advent of precision ball screw and stepping motor control technology, open-loop control can be refined to 0.001 in. resolution, which is accurate enough to be used in many precision positioning and light-load contouring applications.



Input/Output
(I/O)
Communication
Operator
Interface
Micro
Processor
Memory
Control Program
Service Program
Part Program
Command
Signal
Generator
Programmable
Controller
Y
X
Amplifier
Amplifier
Stepping
Motor
Stepping
Motor
X
Y
Slide
Ball screw


Guideway

Closed-loop Control Systems

A feedback loop is implemented to monitor the actual output and correct any discrepancy from desired output. Both analog-type and digital-type can be applied.

Most modern closed-loop NC systems are able to provide very fine resolution of 0.0001 in.

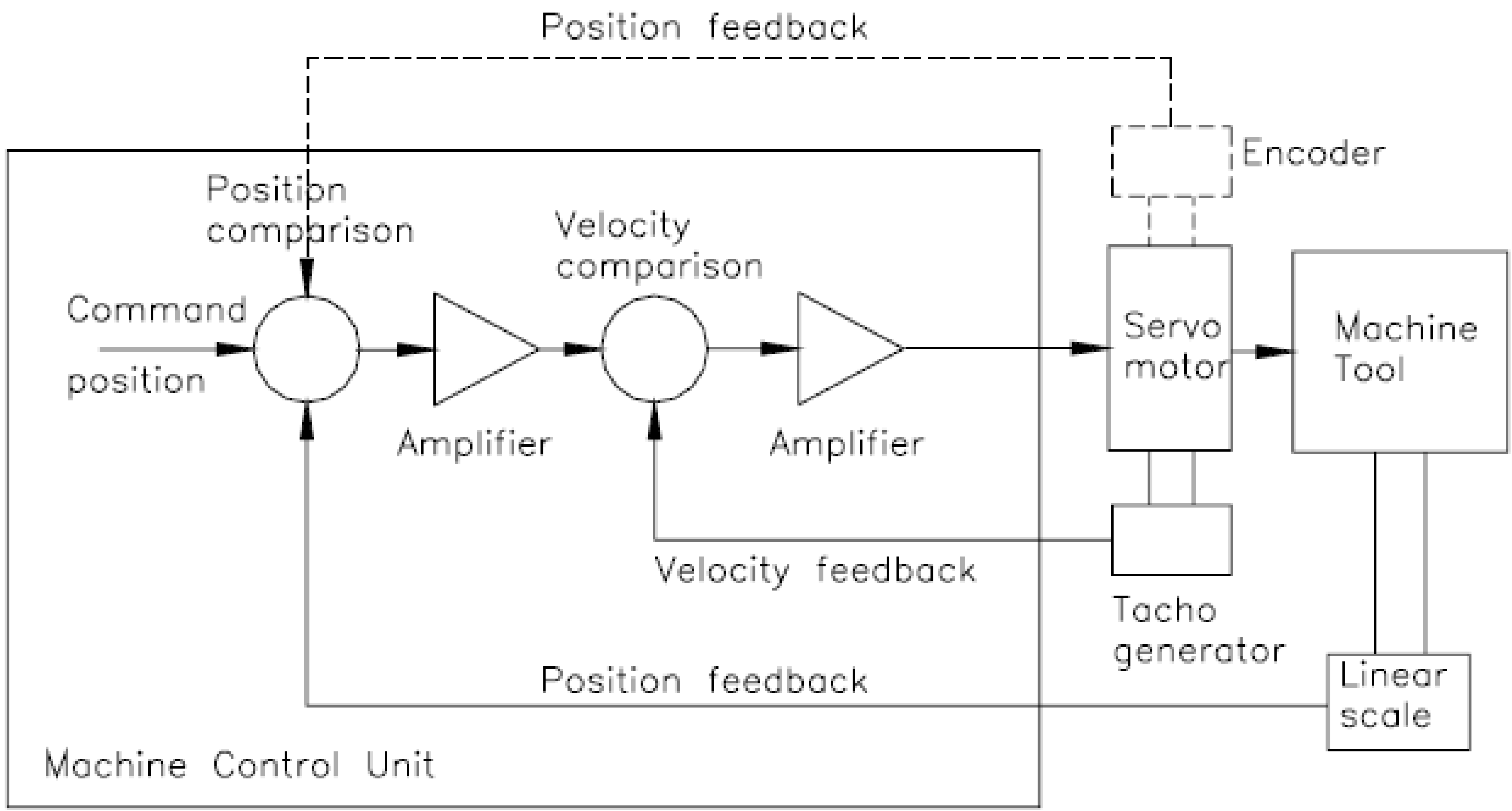


Fig. 10.19 . The closed loop control system used for the control in a CNC machine tool

Optical rotary encoder

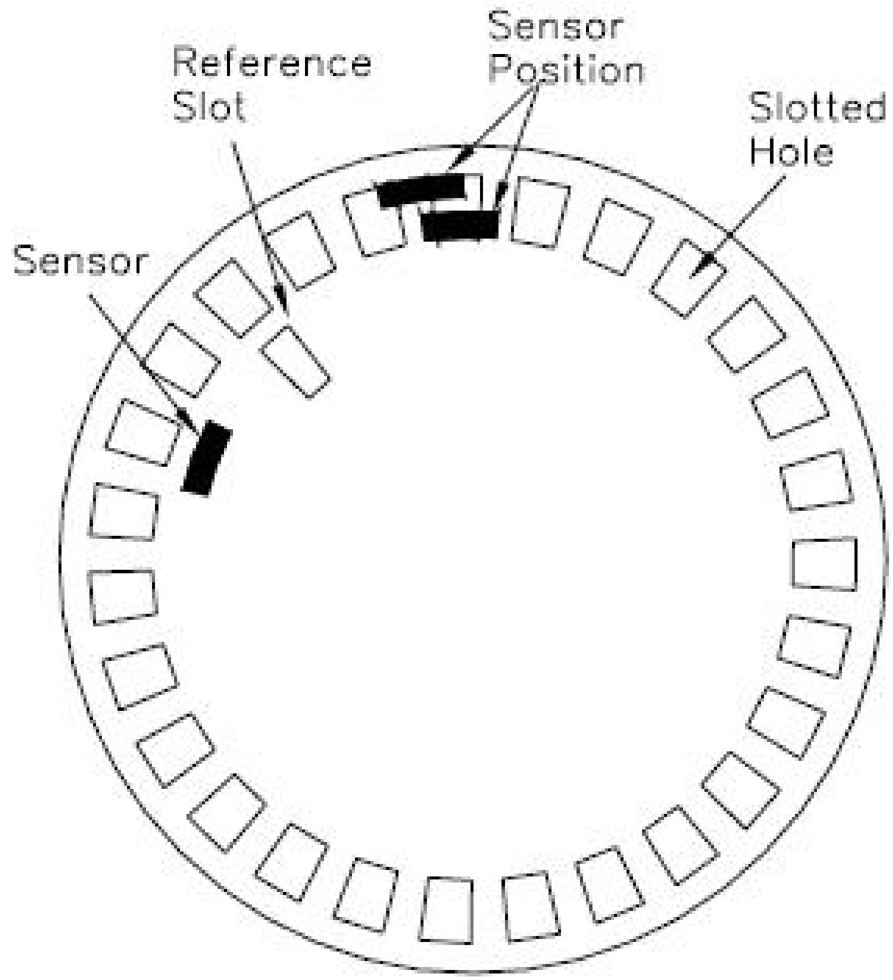


Fig. 10.20 . The encoder disc for rotary position measurement.

Incremental encoder

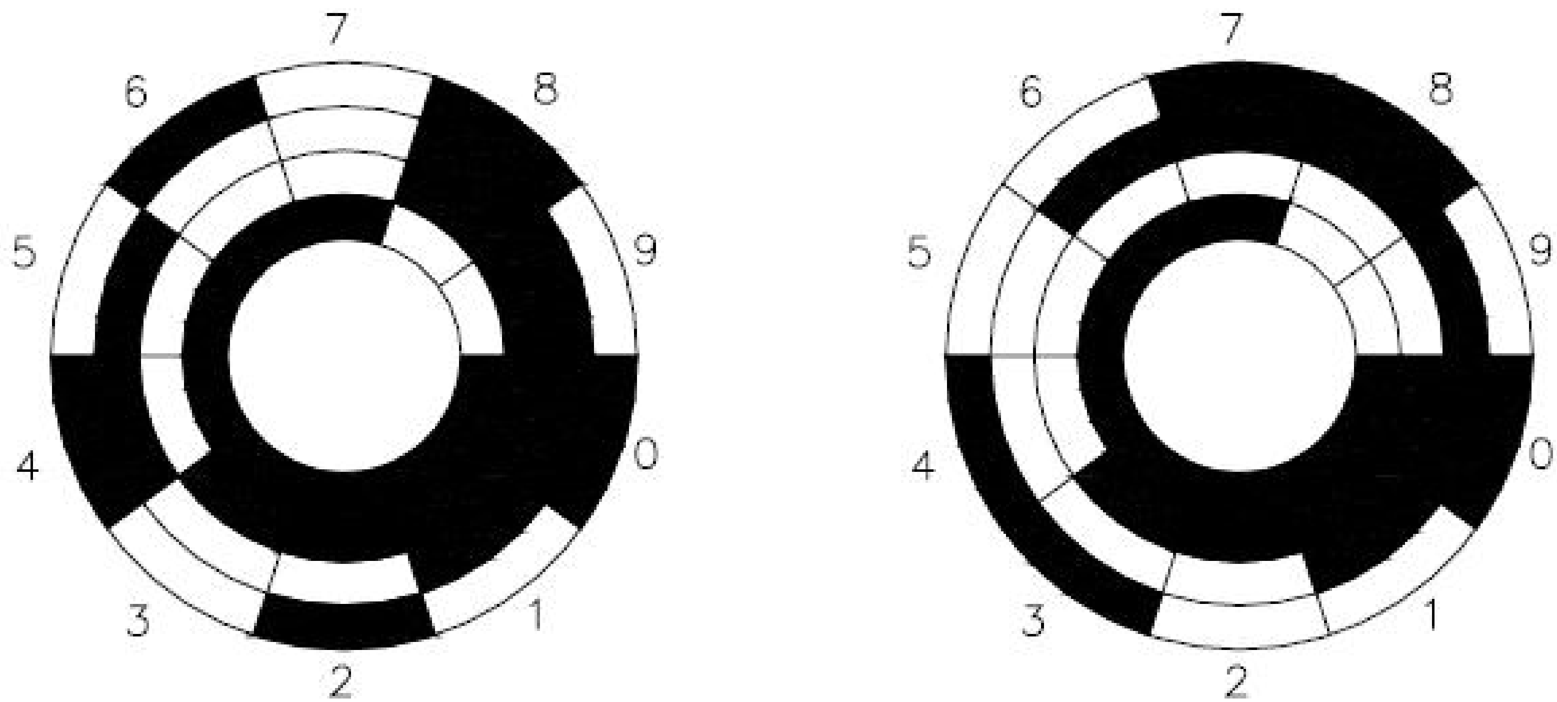


Fig. 10.21 . The absolute encoder disc for rotary position measurement.

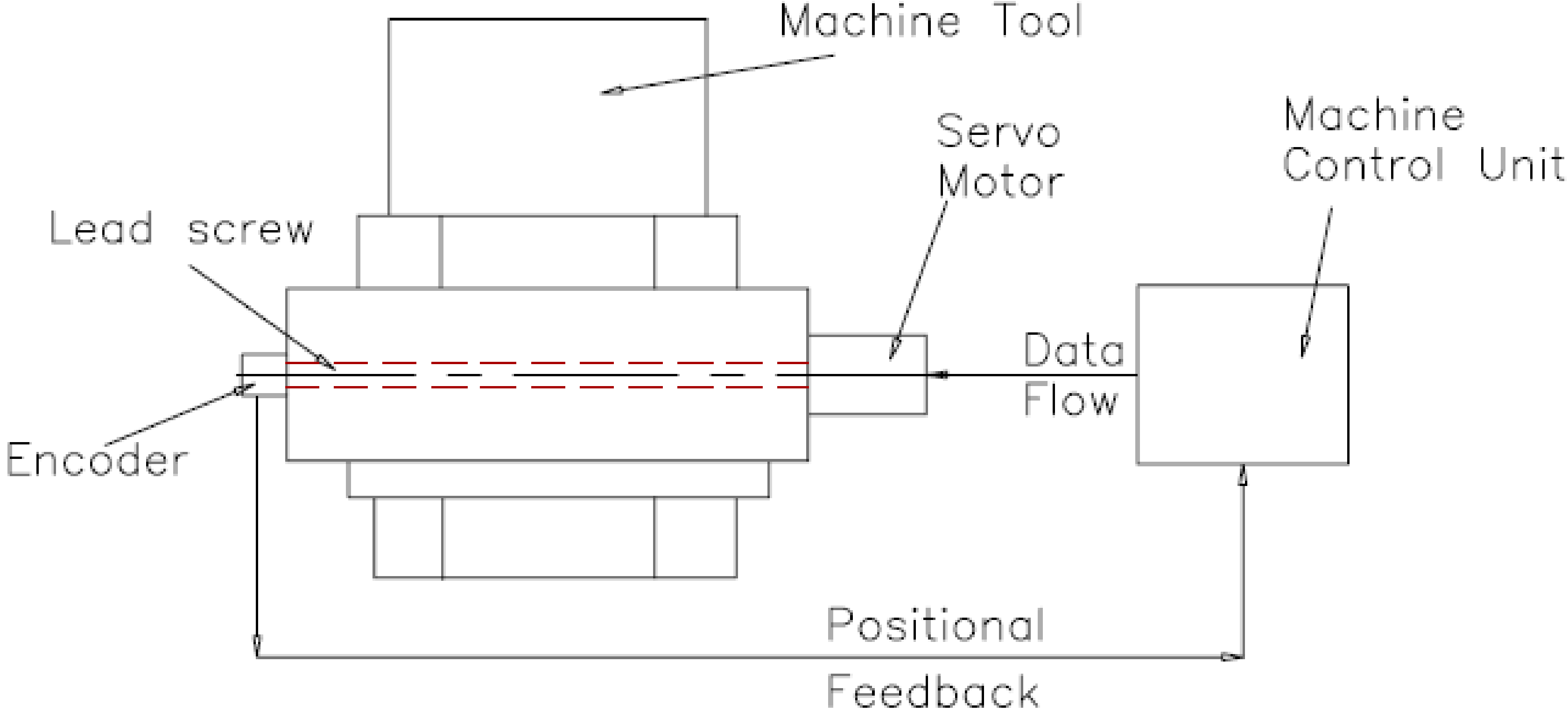


Fig. 10.23 . The encoder disc for rotary position measurement.

Linear scale

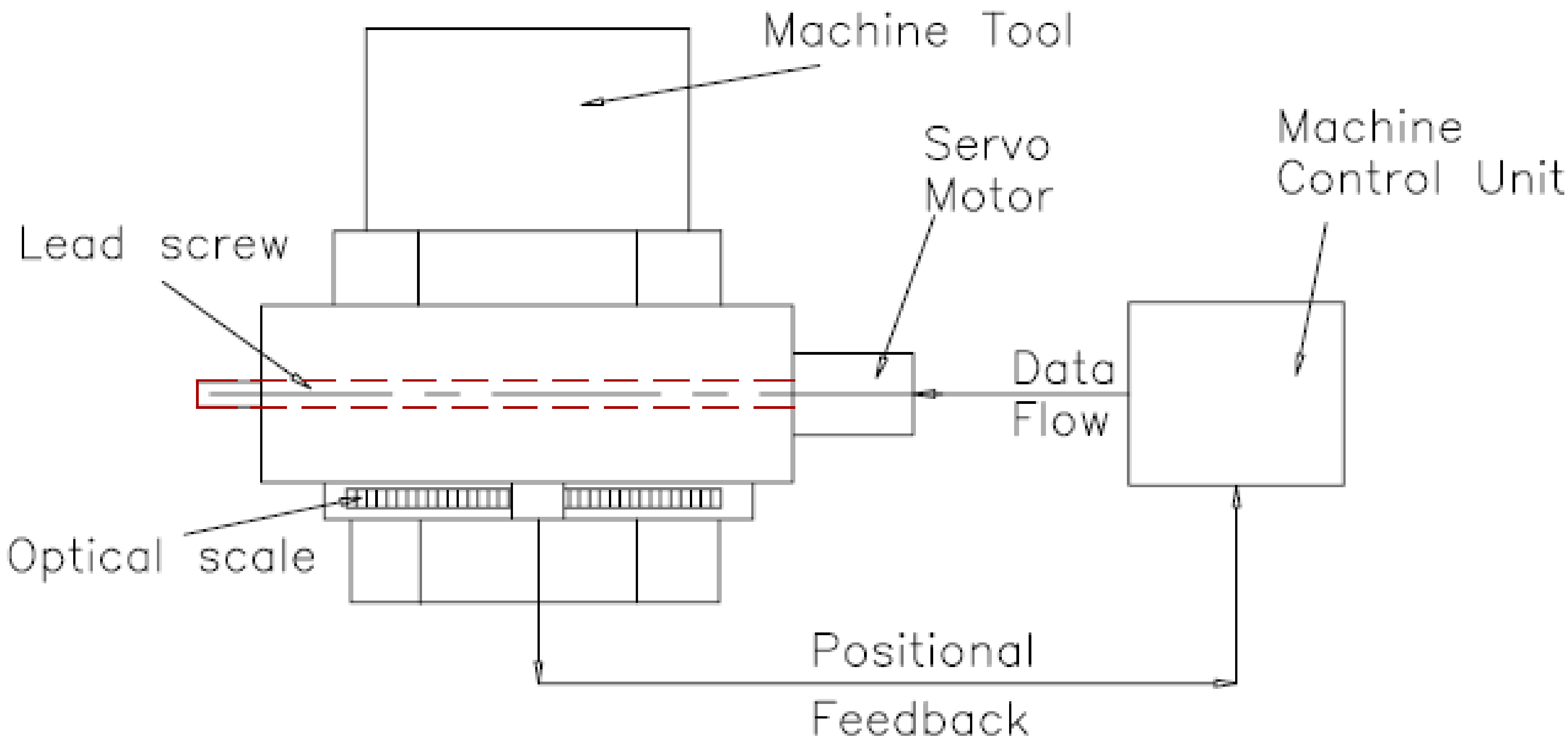
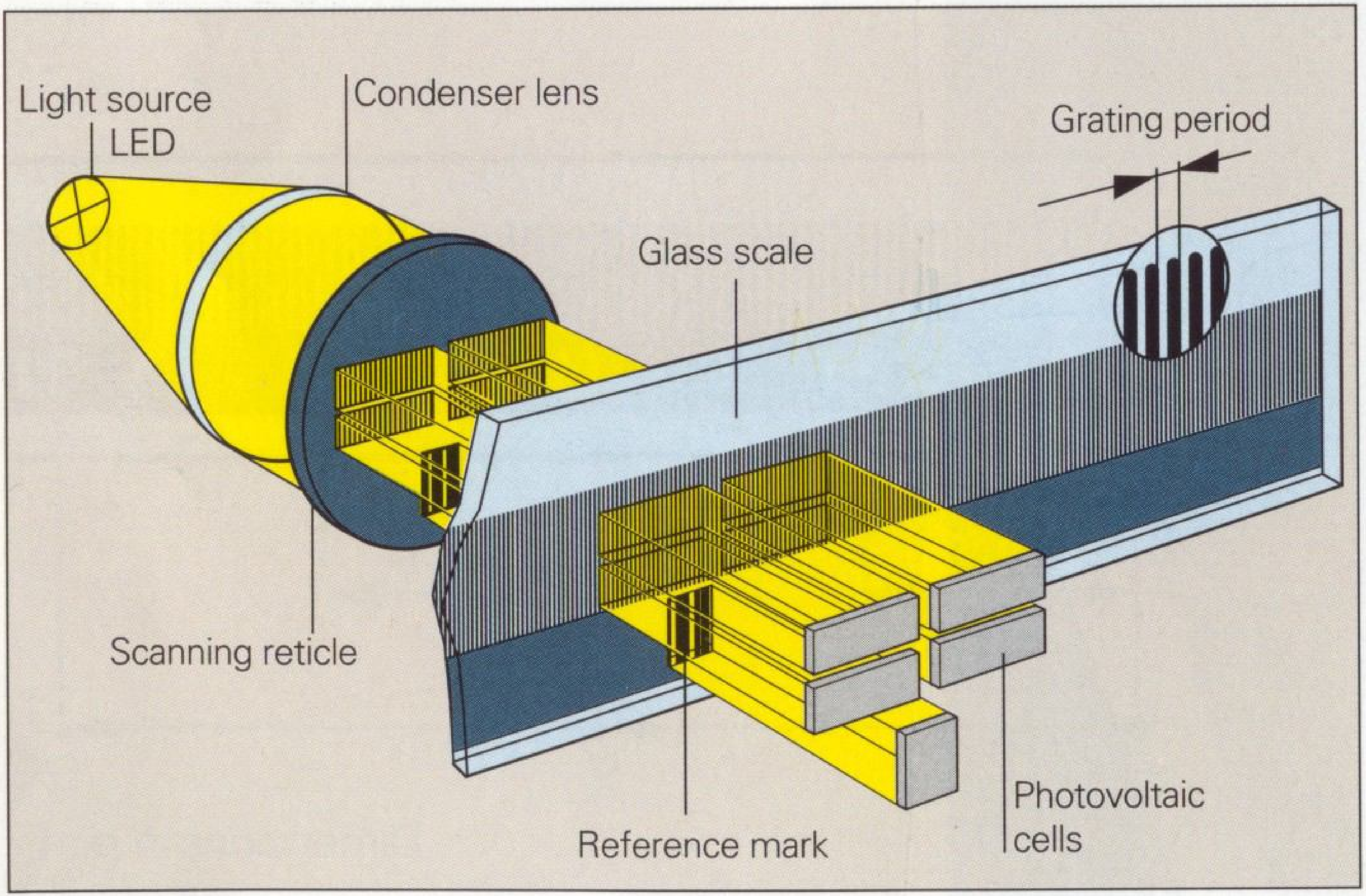
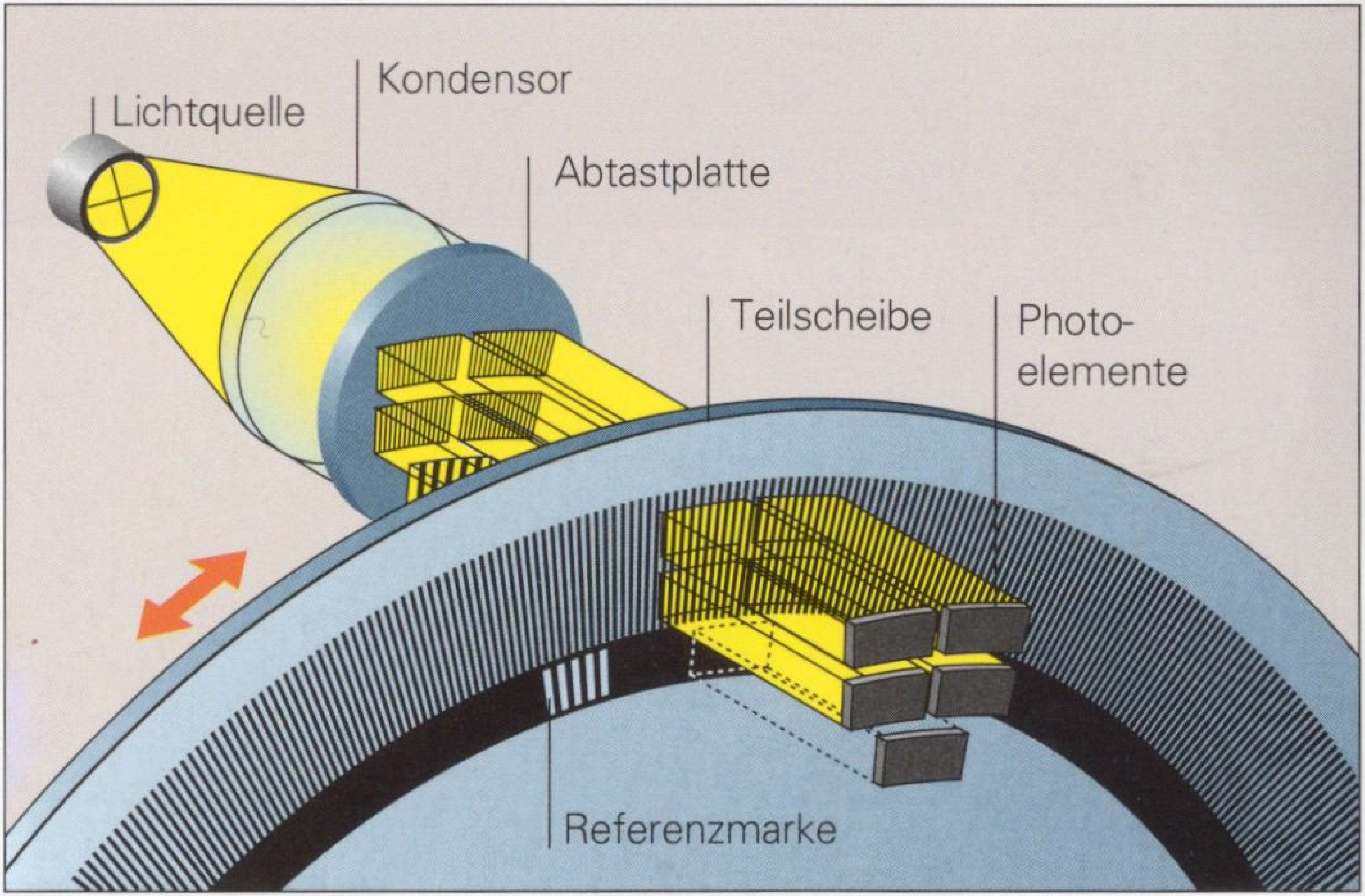


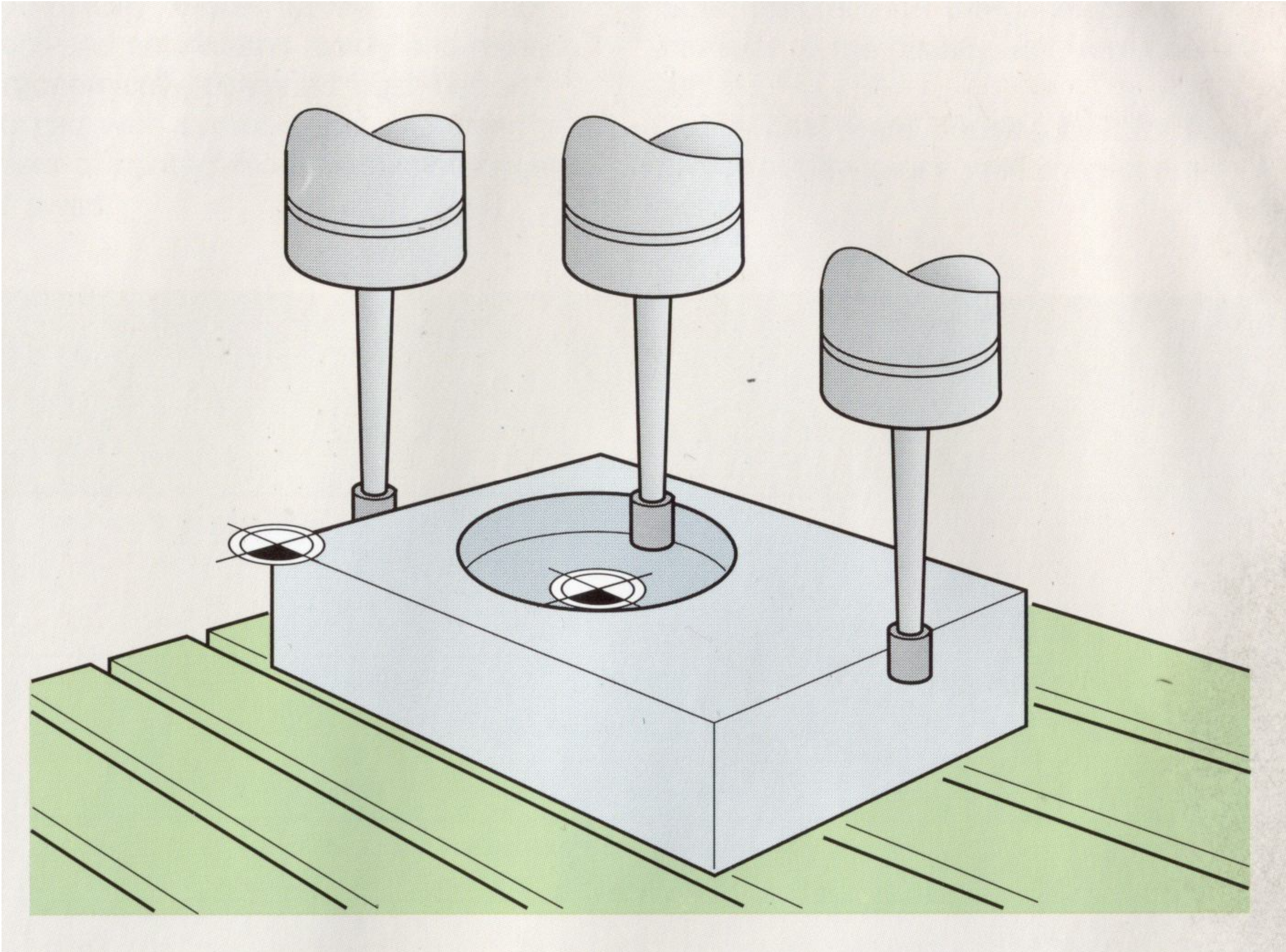
Fig. 10.24 . The linear scale fixed to the machine tool structure or direct position measurement.



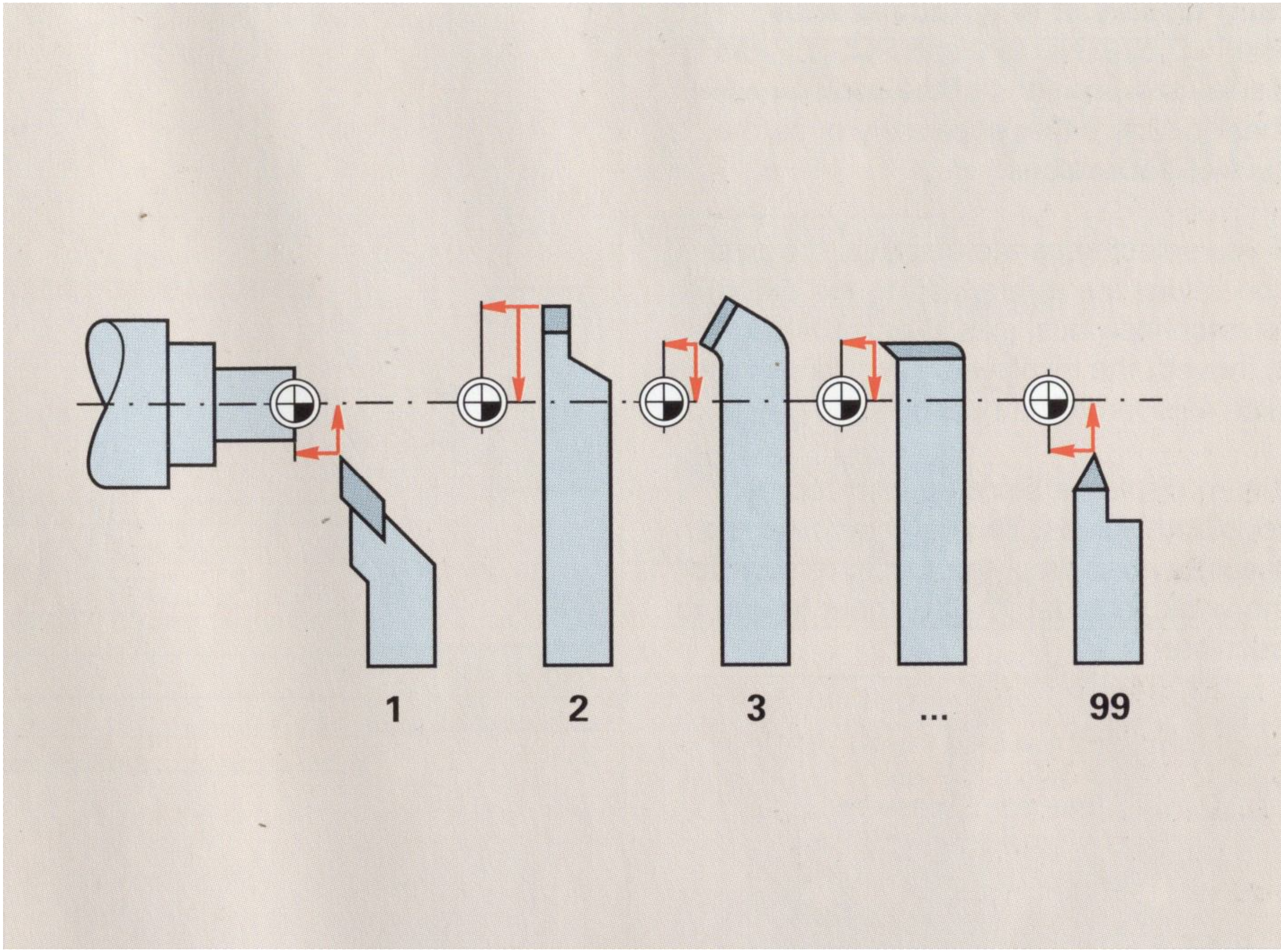
Signal generation using the imaging principle with glass scale and four-field scanning (transmitted-light method)



Photoelektrische Abtastung von Radialgitterteilungen auf Glas
(Durchlicht-Verfahren)



Aplikimi i vizoreve matese optike te makinat shpuese



Aplikimi i vizoreve matese optike tek makina tornuese

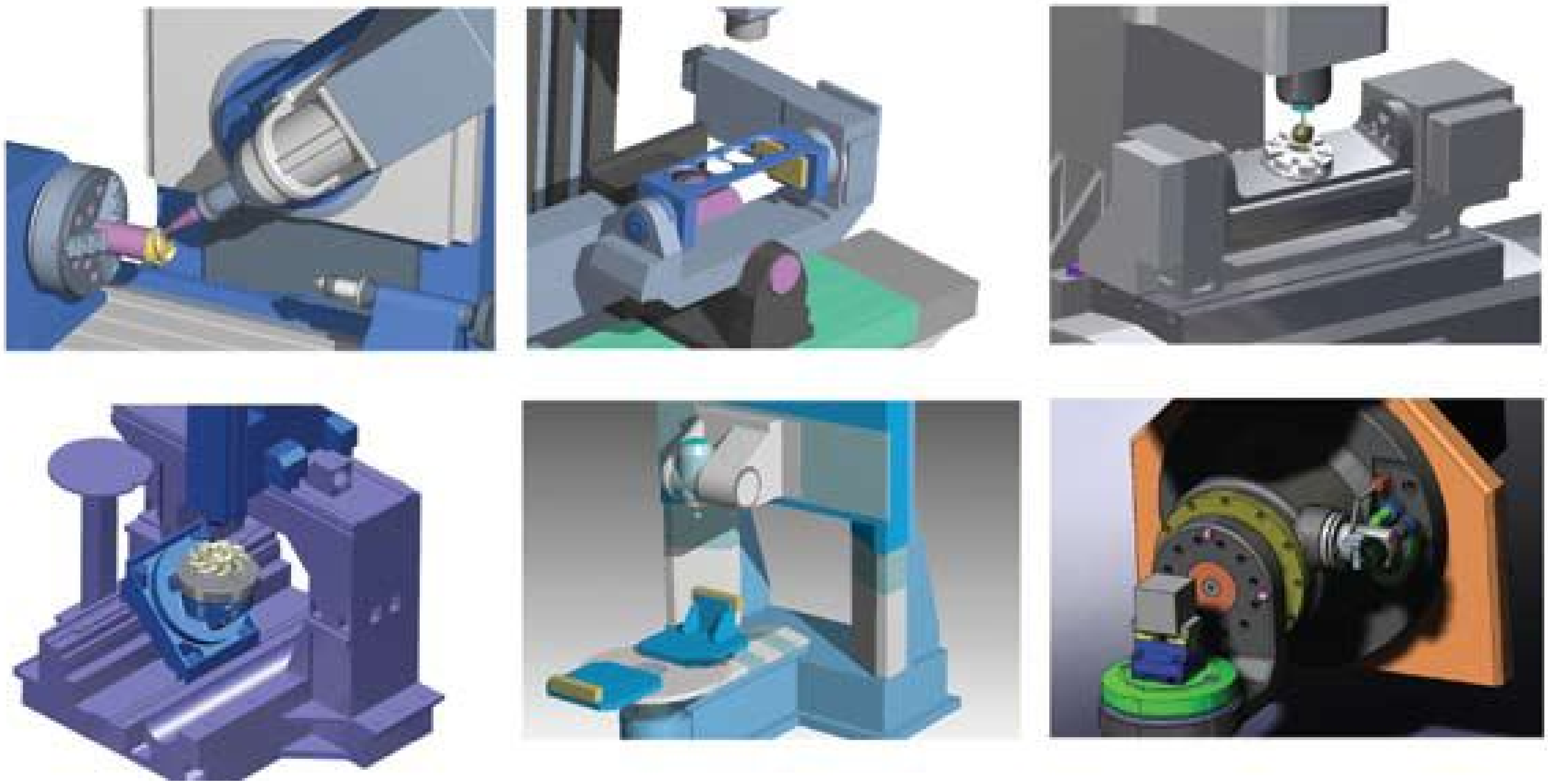
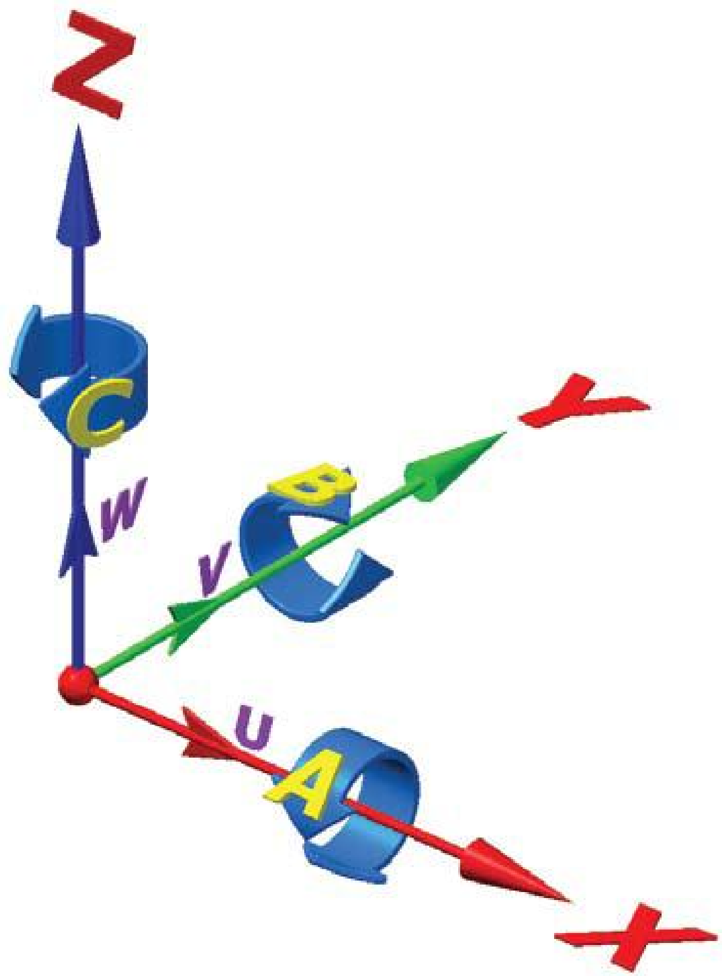


Figure 2-1 Typical arrangements of multiaxis CNC machines.



XYZ - are linear axes where Z is aligned with the spindle of the machine

.

ABC - are rotary axes rotating around XYZ respectively.

UVW - are parallel linear axes along XYZ respectively.

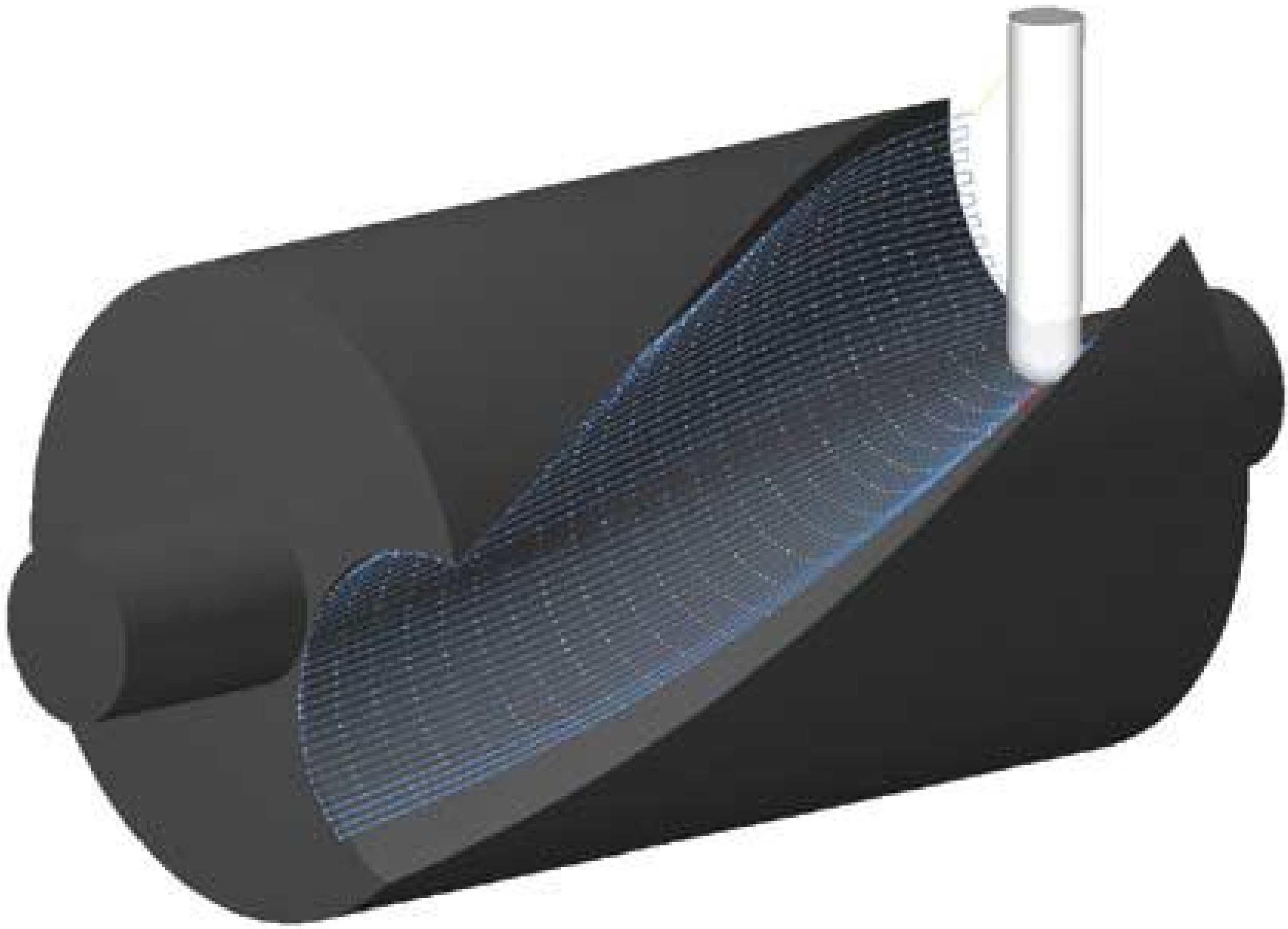


Figure 5-3 Sketch of simultaneous cutting on a 4-axis machine -XYZA.

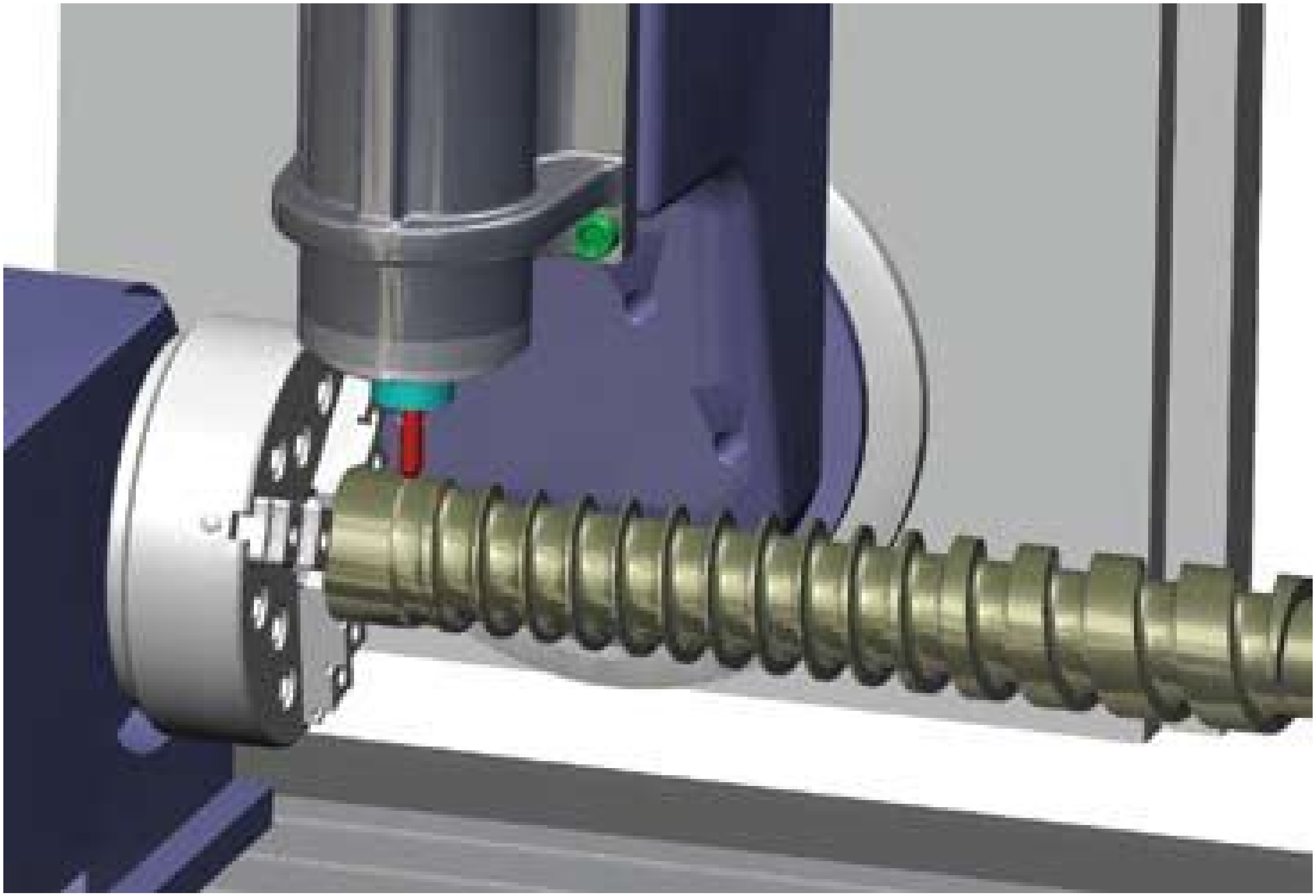


Figure 5-4 A 4-axis machine set-up for cutting a variable-pitch thread on an *auger using motions on XYZ and A axes.*

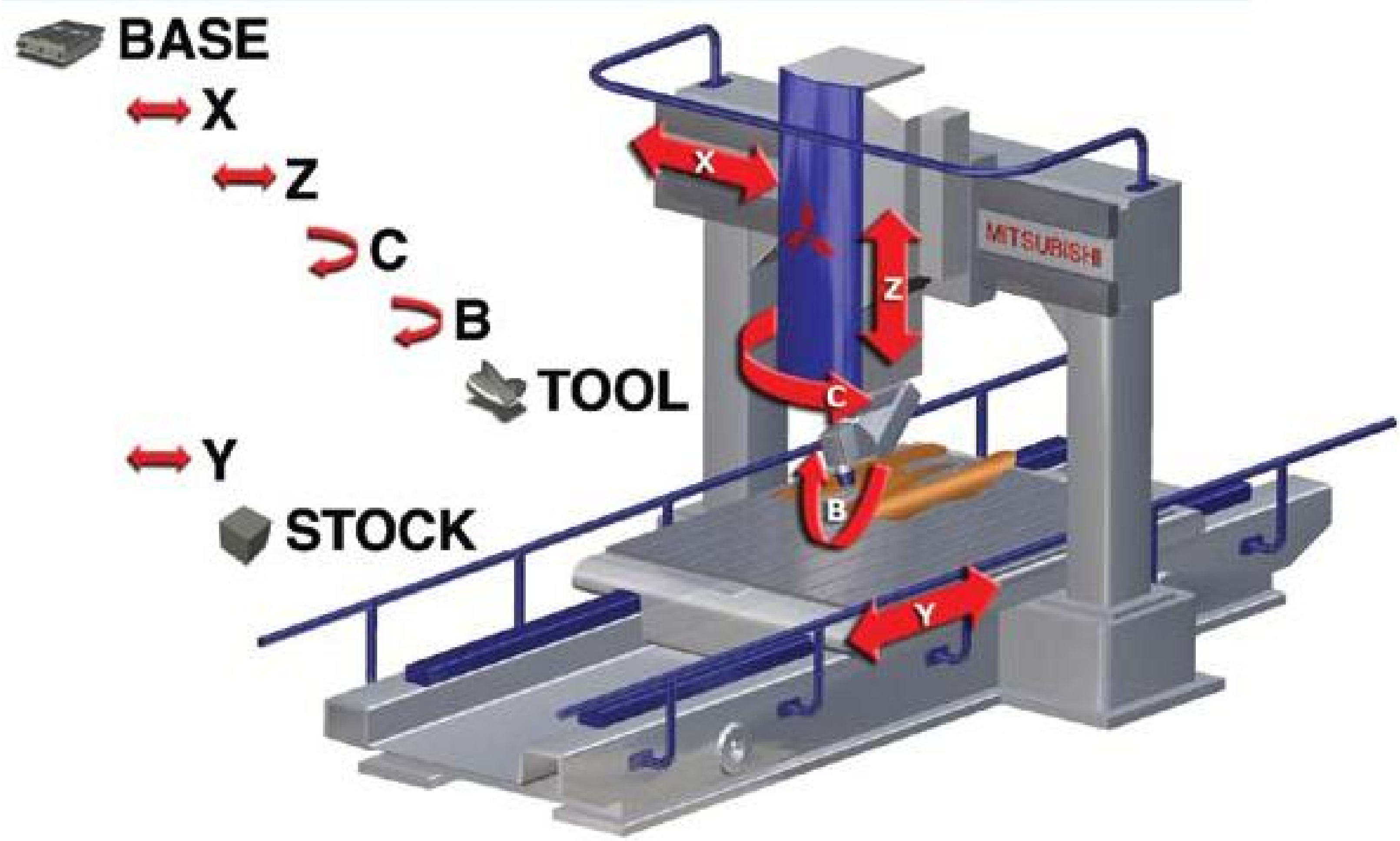
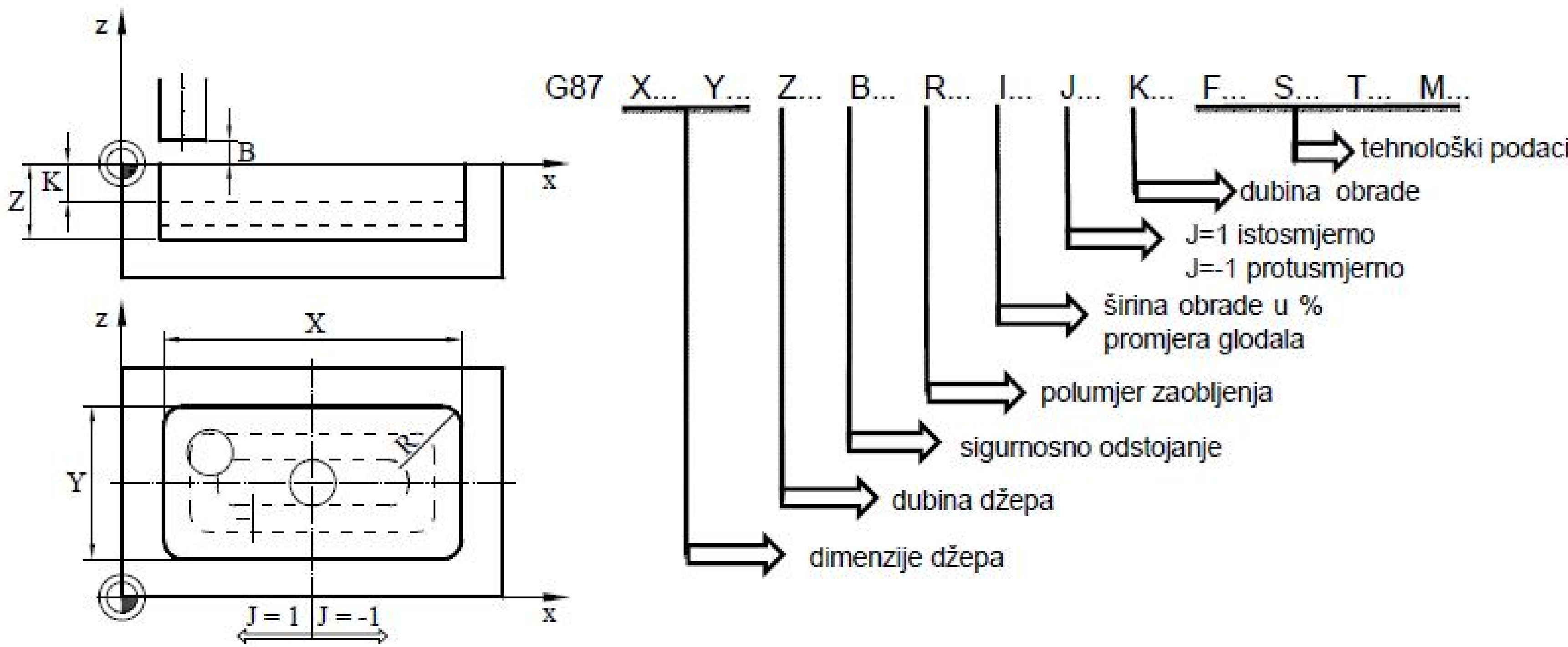
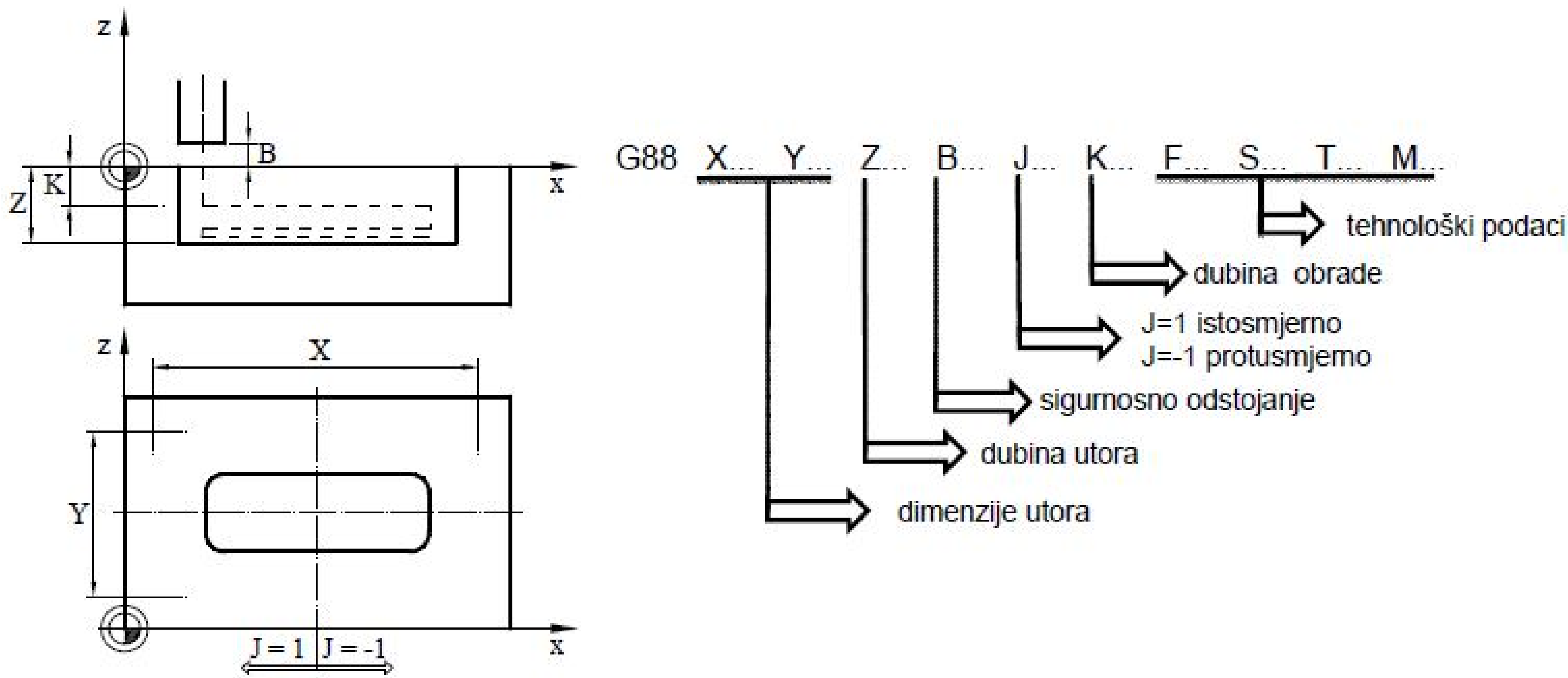


Figure 7-10 Vertical 5-axis laser machine, with a dual rotary head.

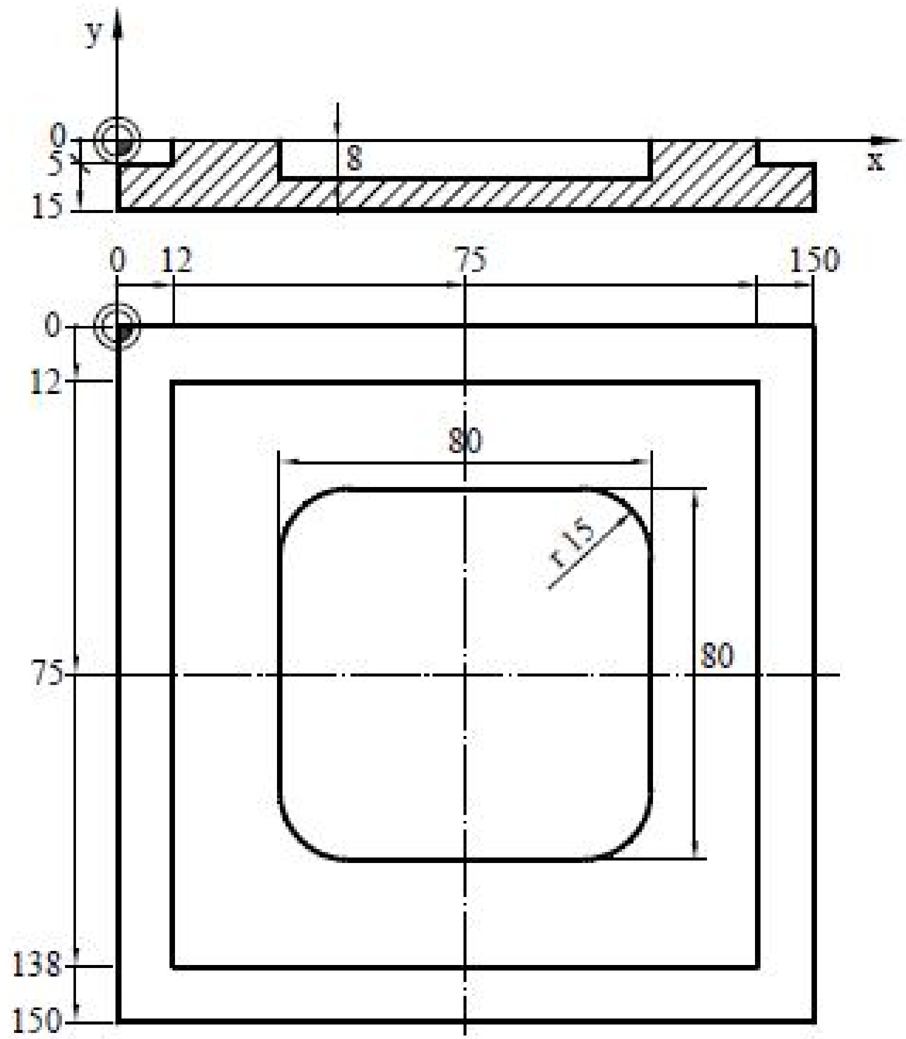
CIKLI I FREZIMIT – G87



CILKI I FREZIMIT – G88



SHEMBULL



Plan obrade:

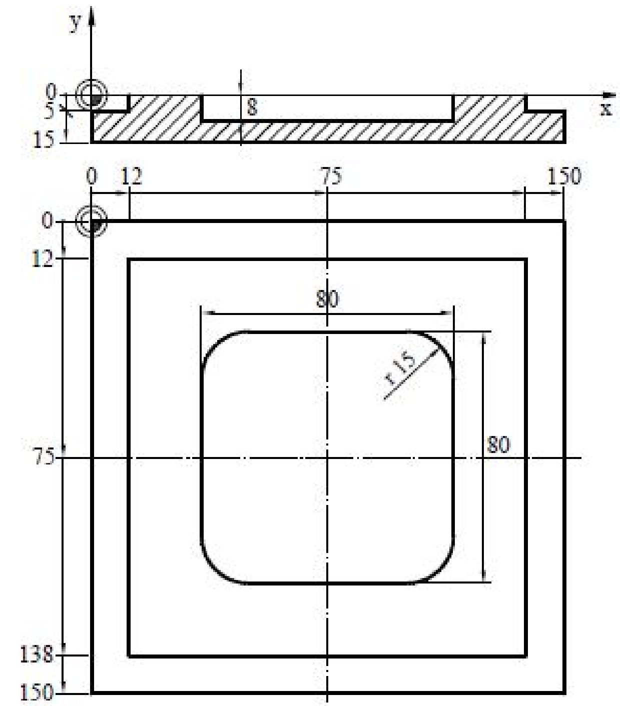
Operacija	Opis	Broj alata	Opis alata
1	Obrada okvira i obrada “džepa”	1	Glodalo za utore 20mm

Podaci o alatu:

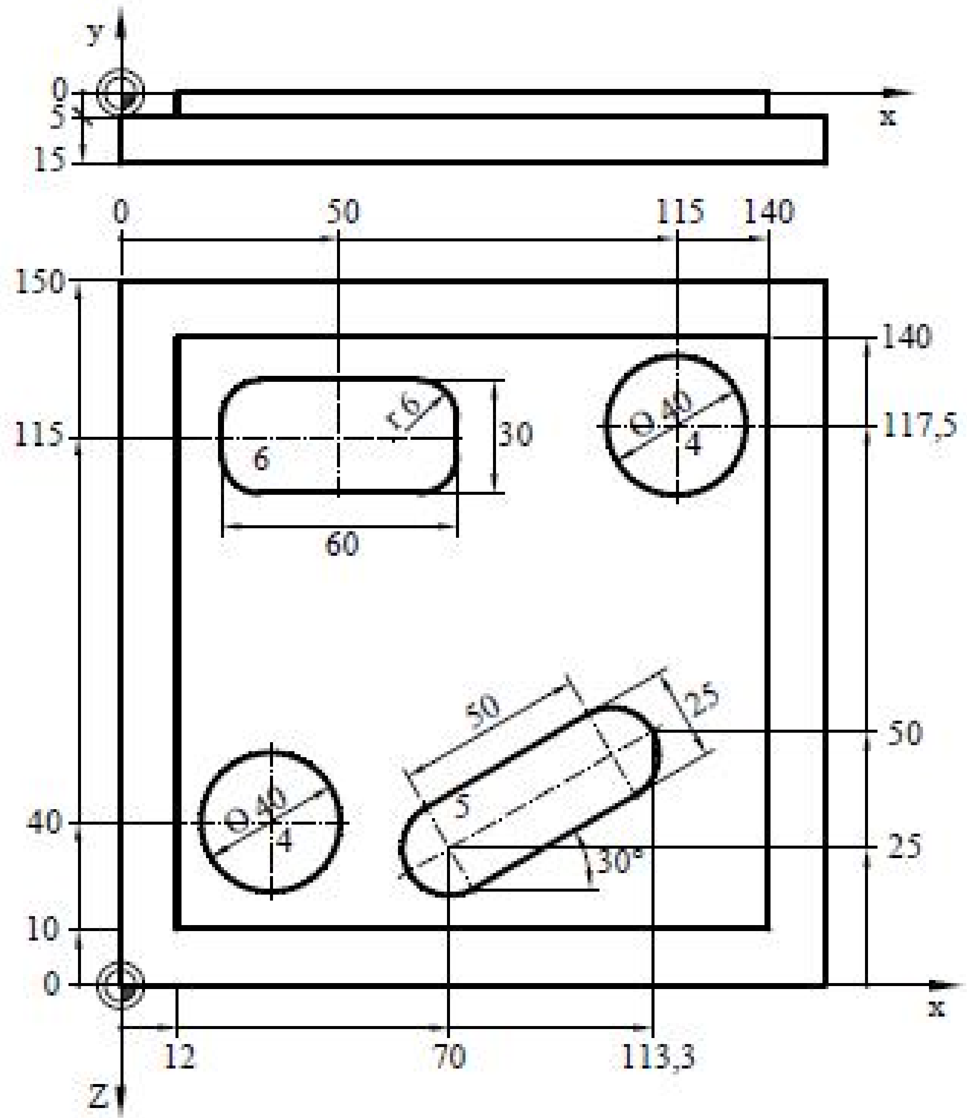
Broj	Opis	Materijal	Promjer t.	Promjer s.	Brzina	Br. okr.	Posmak
1	Glodalo za utore	HSS	20		80	1250	300

Program:

```
%PM
N900100                                   - broj programa
N5      G55                               - nul-točka
N10     G18  S1250  T1  M66               - ravnina obrade i poziv alata
N15     G0  X-12  Y-5  Z-12  F500  M3     - početna točka za glodanje okvira
N20     G43                               - korekcija "do"
N25     G1  Z12                           - pravocrtna interpolacija
N30     G41  X138                         - kompenzacija polumjera alata
N35     Z138
N40     X12
N45     Z-12
N50     G40                               - ukidanje kompenzacija
N55     G0  Y5                            - "dizanje" alata iznad obratka
N60     G87  X80  Y80  Z-5  B5  R15  I75  K4  - definicija ciklusa
N65     G79  X75  Z75  F400  M3           - poziv ciklusa i pozicioniranje
N70     Y125  M30
```



SHEMBULL : Ciklet e frezimit te xhepave G87, G88 dhe xhepit rrethor G89



Plan obrade:

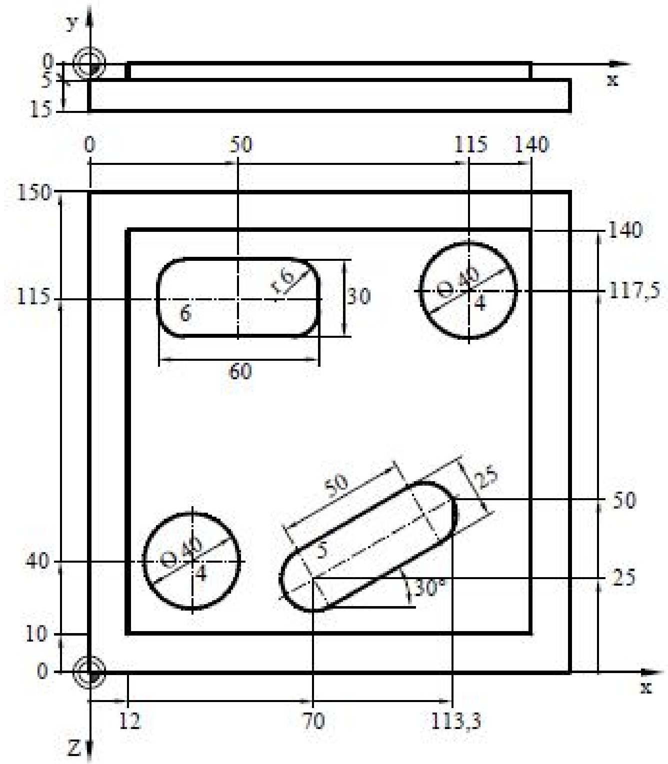
Operacija	Opis	Broj alata	Opis alata
1	Obrada okvira i obrada "džepa"	1	Glodalo za utore 20mm
2	Obrada 1 utora i 2 kružna utora	4	Glodalo za utore 10 mm

Podaci o alatu:

Broj	Opis	Materijal	Promjer t.	Promjer s.	Brzina	Br. okr.	Posmak
1	Glodalo za utore	HSS	20		80	1250	300
2	Glodalo za utore	HSS	10		50	1600	100

%PM

N900200		- broj programa
N5	G55	- nul-točka
N10	G18 S1250 T1 M66	- ravnina obrade i poziv alata
N15	G0 X-15 Y-5 Z15 F500 M3	- početna točka za glodanje okvira
N20	G43	- korekcija "do"
N25	G1 X10	- pravocrtna interpolacija
N30	G41 Z-140	- kompenzacija polumjera alata
N35	X140	
N40	Z-10	
N45	X-15	
N50	G40	- ukidanje kompenzacija
N55	G0 Y2	- "dizanje" alata iznad obratka
N60	G88 X75 Y25 Z-5 B2 J1 K5	- definicija ciklusa
N65	G79 X50 Z-25 B1=30	- poziv ciklusa i pozicioniranje
N70	G0 Y125	
N75	S1600 T4 M66	- izmjena alata
N80	F100 M3	
N85	G87 X60 Y30 Z-6 B2 I75 K3 R6	- definicija ciklusa glodanja "džepa"
N90	G78 X50 Y0 Z-115	- poziv ciklusa i pozicioniranje
N95	G89 Z-4 B2 I75 K4 R20	- definicija ciklusa kružnog utora
N100	G79 X35 Y0 Z-40	- poziv ciklusa i definiranje
N105	G79 X115 Z-117,5	- poziv ciklusa i definiranje
N120	Y125 M30	



CIKLI I SHPIMIT G81

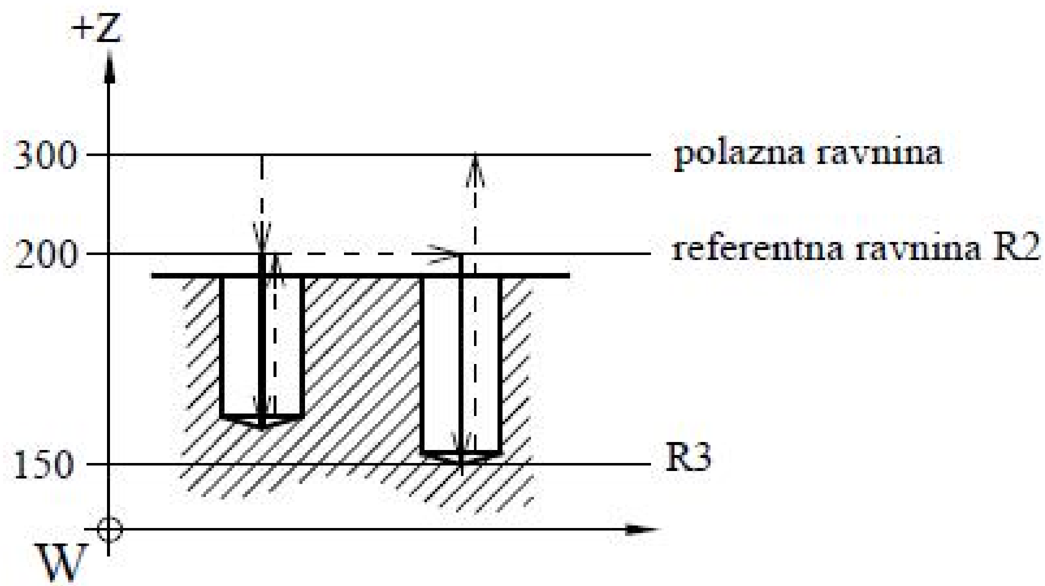
Potprogrami → dijelovi programa koji se često ponavljaju → adresa L
Mogućnost pisanja programa i potprograma u parametarskom obliku za izvođenje tipskih obrada → kinematika koja se ponavlja ali s primjenom različitih dimenzija

N100 G81 X120 Y200 R2 200 R3 150 F03 S1000

N120 X200 Y300

:

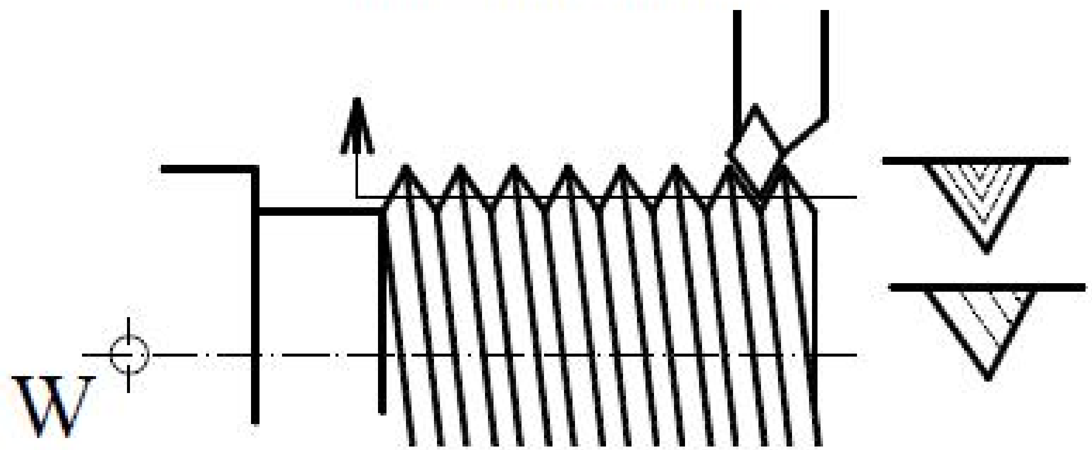
N150 G80 Z300



FORMA PARAMETRIKE- R -URDHERESAT

R20 – veličina koraka
R21 – početna vrijednost X
R22 – početna vrijednost Z
R23 – broj praznih prolaza
R24 – dubina korjena zuba
R25 – dodatak završni rez.
R26 – zalet (2 – 3 koraka)
R27 – izlaz
R28 – broj prolaza
R29 – ½ kut navoja
R31 – završni X
R32 – završni Z

- Narezivanje navoja



N200 G00 X... Z... S80 T04 D04 M08
N210 R20=6 R21=80 R22=380
R24=-2 R26=12 R27=6
R28=14 R29=29 R31=70 R32= RIJEČ
N220 L97 → potprogram za narez
N220 G00 X... Z..... M09

M - funkcije

M0 – bezuvjetno zaustavljanje programa

M1 – uvjetno zaustavljanje programa

M2 - naredba završetak glavnog programa i "povratak" na početak

M3 – rotacija gl. vretena u smjeru kazaljke na satu

M4 – rotacija gl. vretena u smjeru obrnutom od kazaljke na satu

M5 – zaustavljanje rotacije glavnog vretena

M6 – naredba za izmjenu alata

M8 – uključivanje SHIP-a (emulzije)

M9 – isključivanje SHIP-a (emulzije)

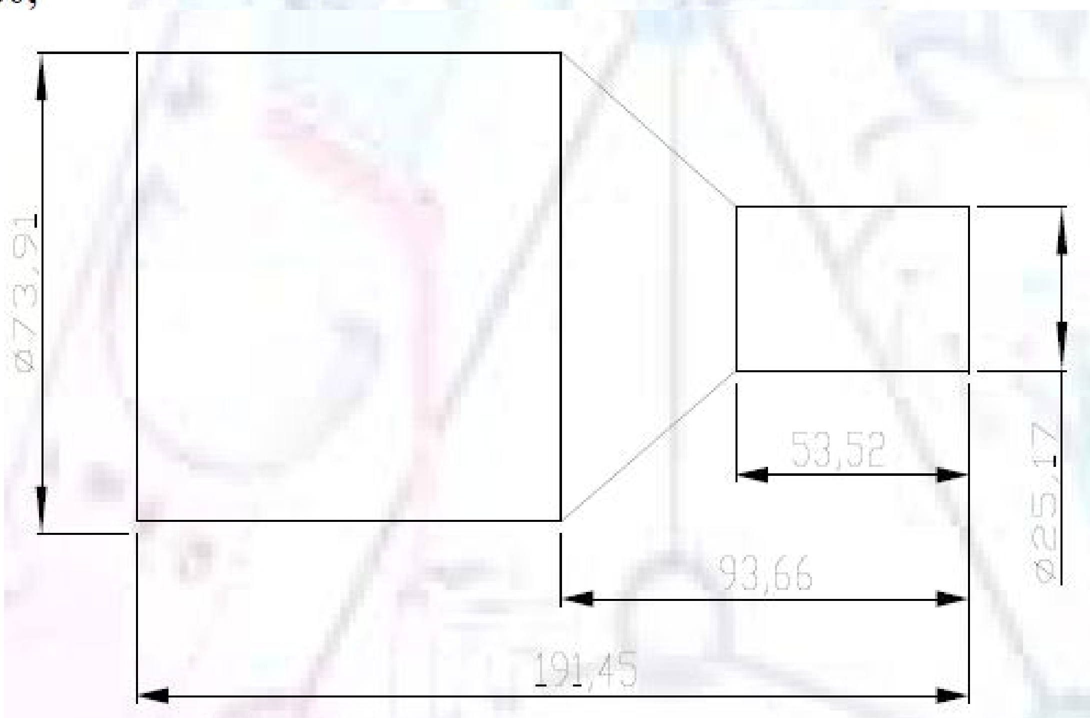
M19 - orjentirano (pod određenim kutem) zaustavljanje glavnoga vretena

M30 - naredba završetak glavnog programa

M66 – automatska izmjena izradaka

مثال على استخدام الدالة **G71** :

```
T0100   M48;
G0 X26.   Z2.  M08 T0101;
G90 G95;
G97 S500 M03;
G71 P1 Q2 U0.3 W0.0 D500 F0.1;
N1 G0 G42 X0.0;
        G01            Z0.0;
        G01   X25.17;
        G01                   Z-53.52;
        G01   X73.91          Z-93.66;
        G01            Z-191.45;
N2;
G28 M30;
```



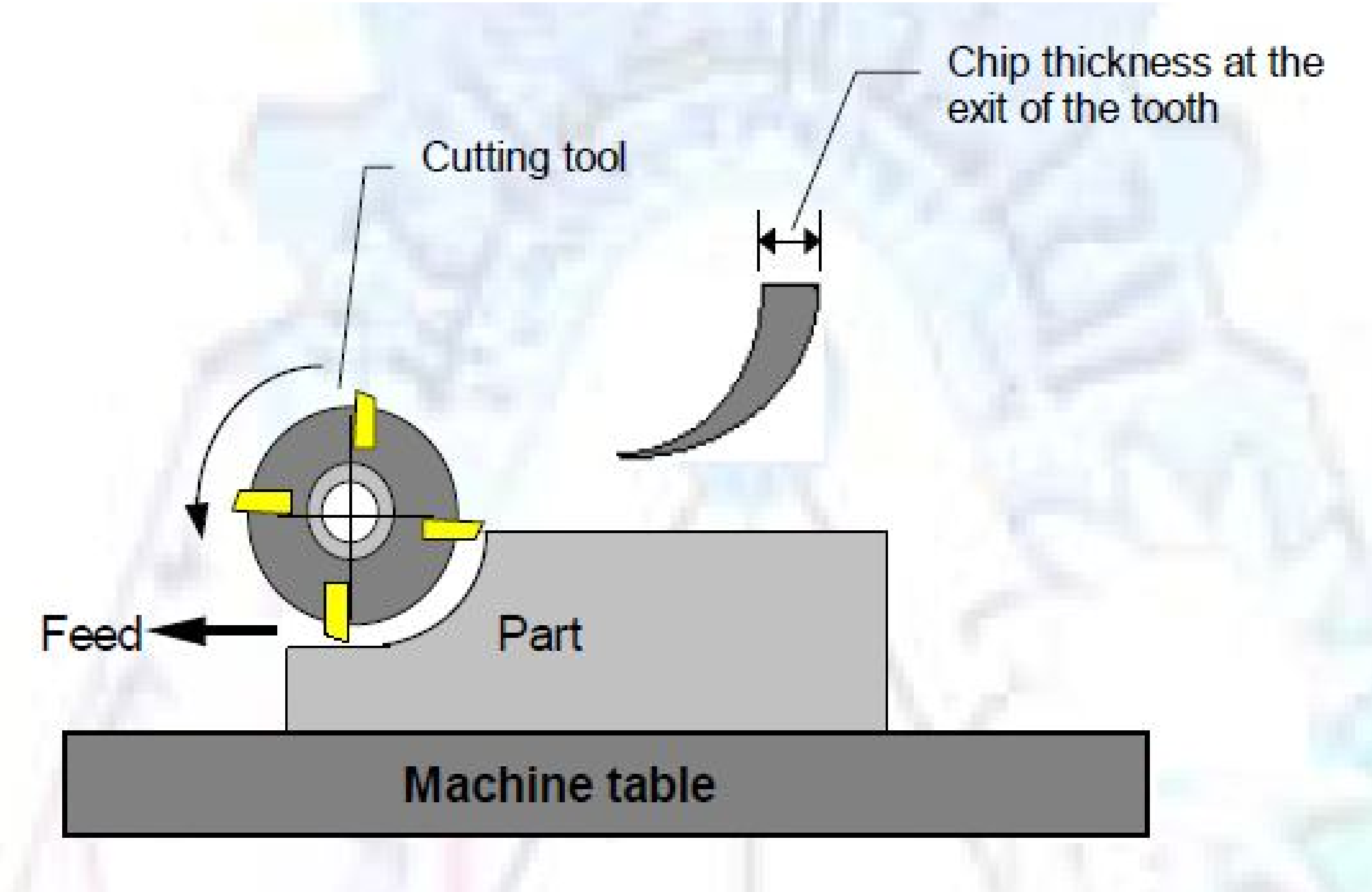
الجانب العملي :

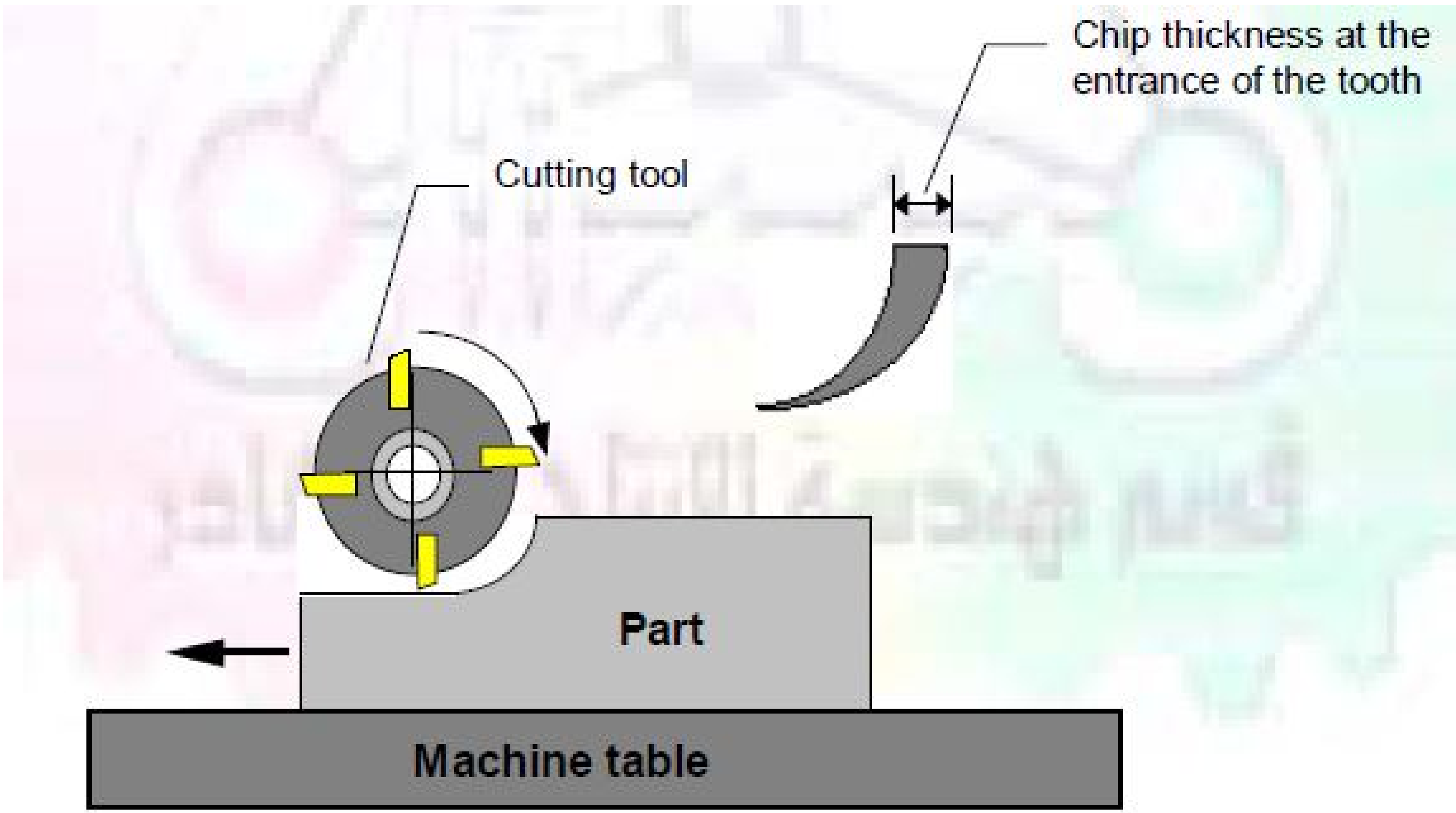
الاجهزة والادوات المستخدمة :

1. ماكنة خراطة مبرمجة CNC
2. قضيب من الالمنيوم قطر 80 ملم
3. فيرنيةٍ
4. هولدر كاربيدي ذواتجاه أيمن

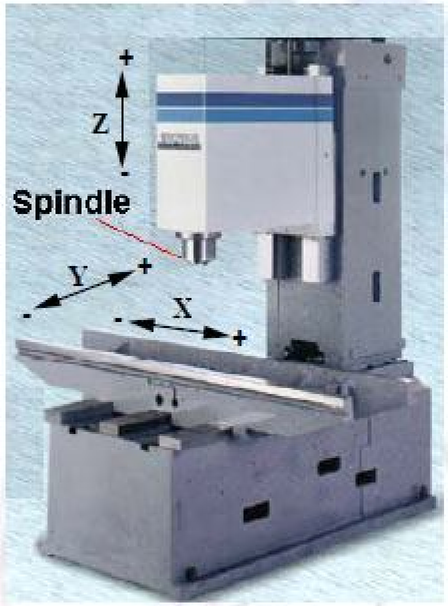
طريقة العمل :

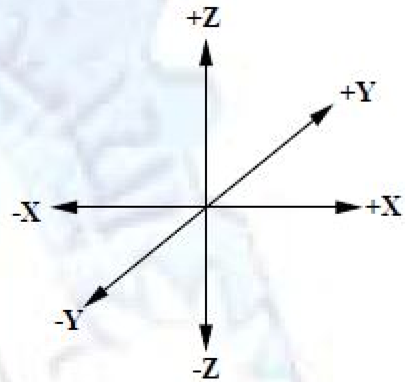
1. يتم تثبيت قضيب من الالمنيوم على العينة التابعة للماكنة مع مراعاة الطول المراد استعماله في عملية الخراطة .
2. تثبيت عدة القطع في المكان الصحيح في (turret) .

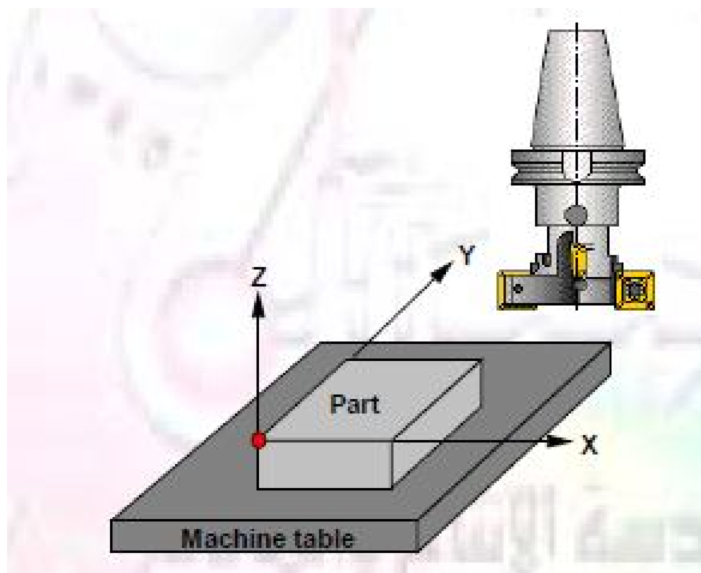

Chip thickness at the
exit of the tooth
Cutting tool
Feed
Part
Machine table



Chip thickness at the
entrance of the tooth
Cutting tool
Part
Machine table


+
Z
-
Spindle
Y
+
-
X
-
+

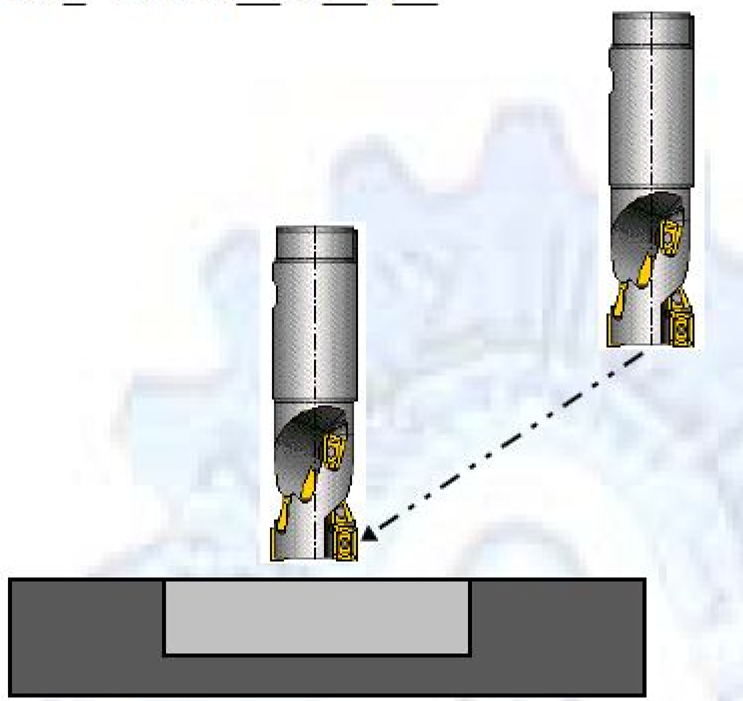

+Z
+Y
-X
+X
-Y
-Z


Y
Z
Part
X
Machine table

G00 Rapid traverse

When the tool being positioned at a point preparatory to a cutting motion, to save time it is moved along a straight line at Rapid traverse, at a fixed traverse rate which is pre-programmed into the machine's control system. Typical rapid traverse rates are 10 to 25 m /min., but can be as high as 80 m/min.

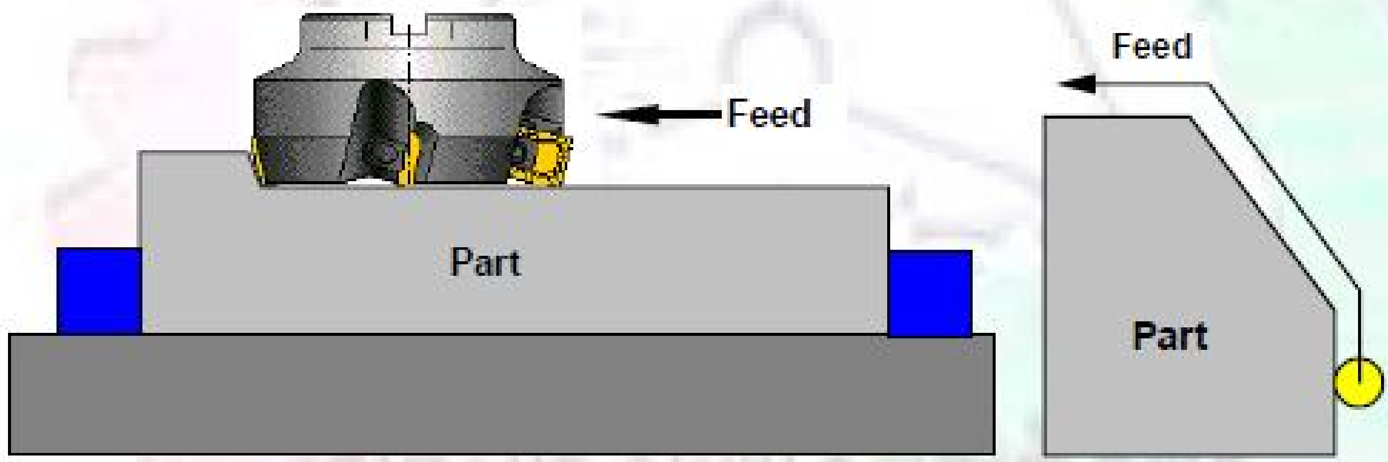
Format : N_ G00 X__ Y__Z__



G01 Linear interpolation (feed traverse)

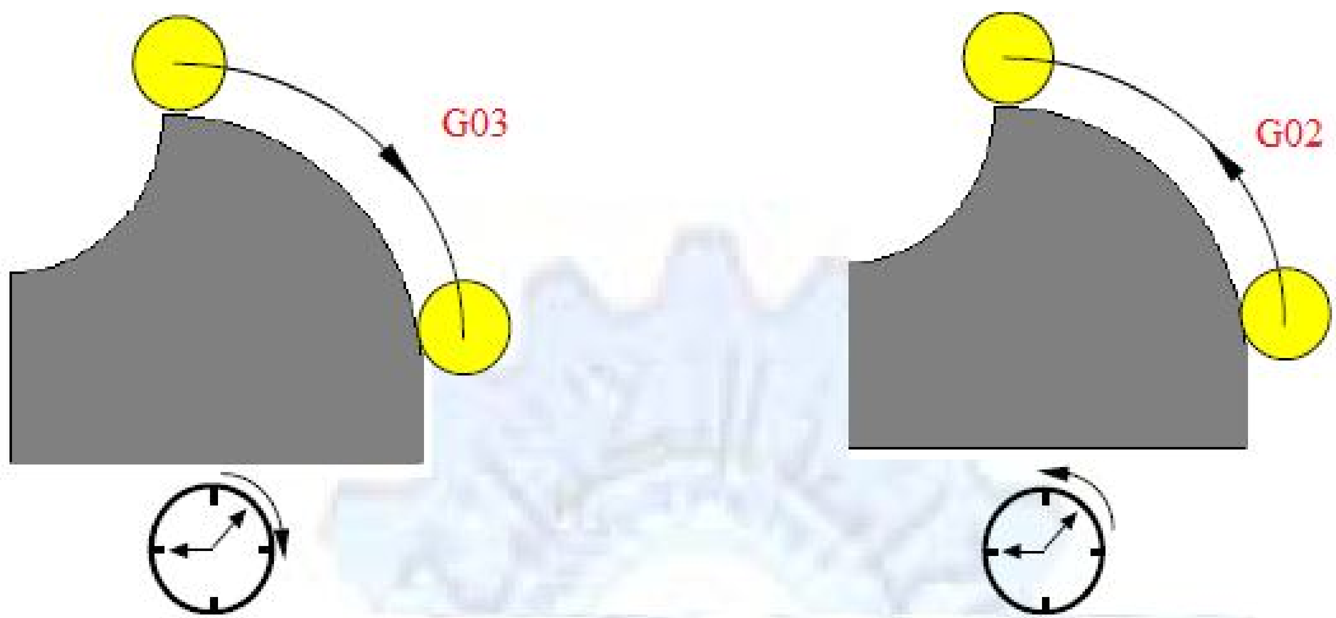
The tool moves along a straight line in one or two axis simultaneously at a programmed linear speed, the *feed rate.*

Format : N__ G01 X__ Y__Z__ F__



G02/G03 Circular interpolation

The tool moves along an arc in two axes simultaneously at a programmed linear speed, the *feed rate.*



Format

N__ G02/03 X__ Y__Z__ I__ J__K__ F__ using the arc center

OR

N__ G02/03 X__ Y__Z__ R__ F__ using the arc radius

G02 moves along a CW arc

G03 moves along a CCW arc

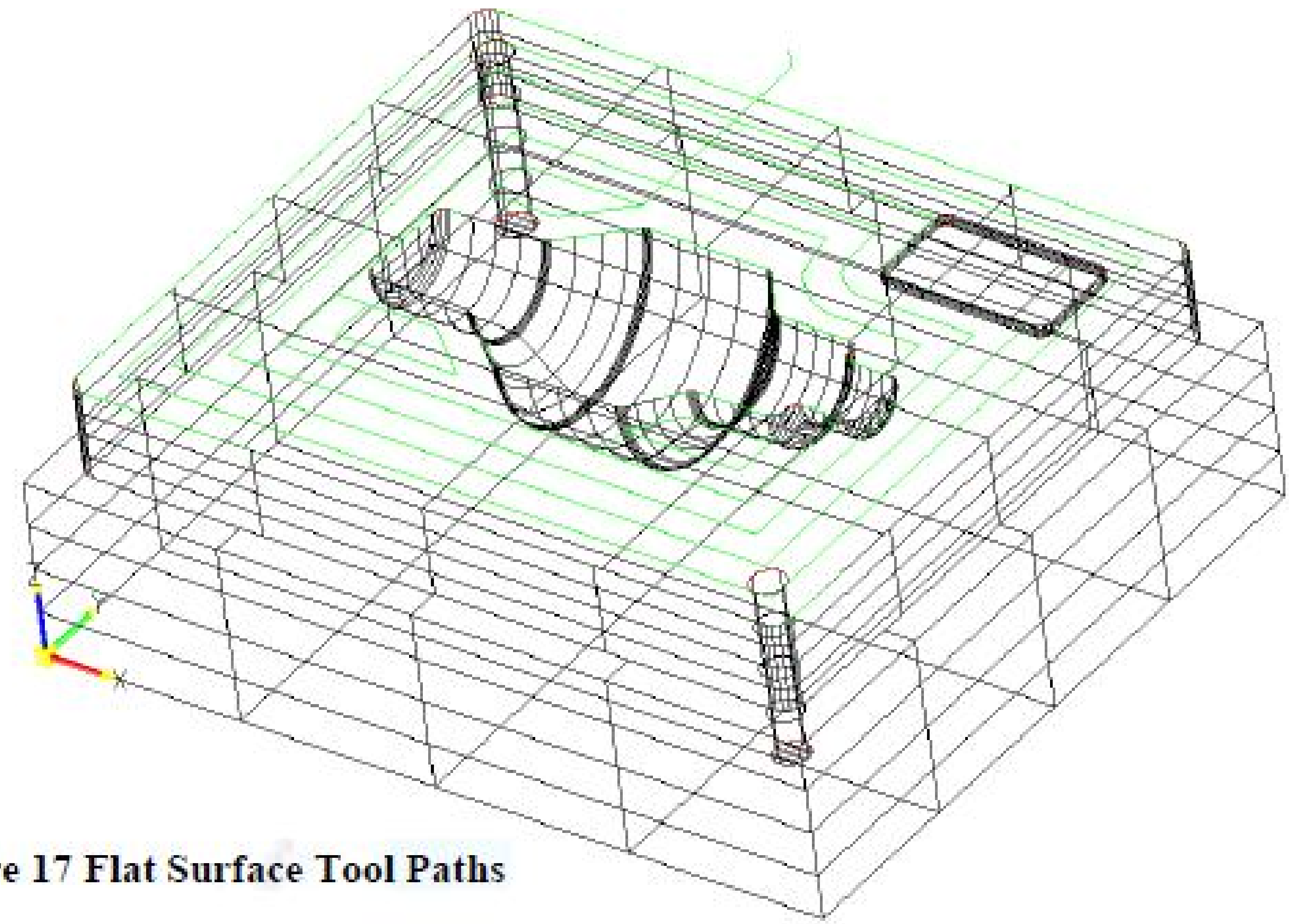


Figure 17 Flat Surface Tool Paths

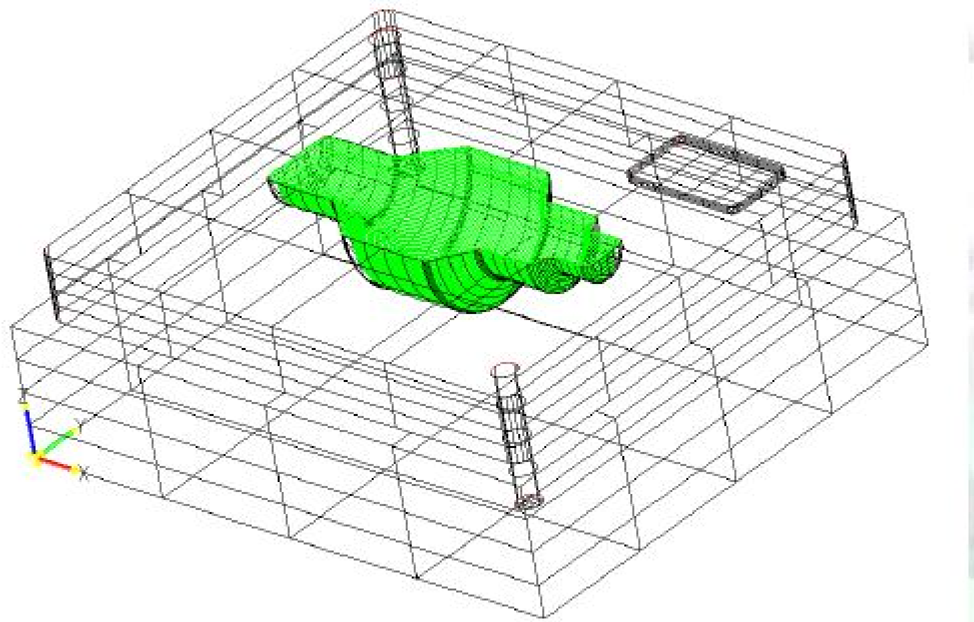


Figure 18 3D Offset Paths

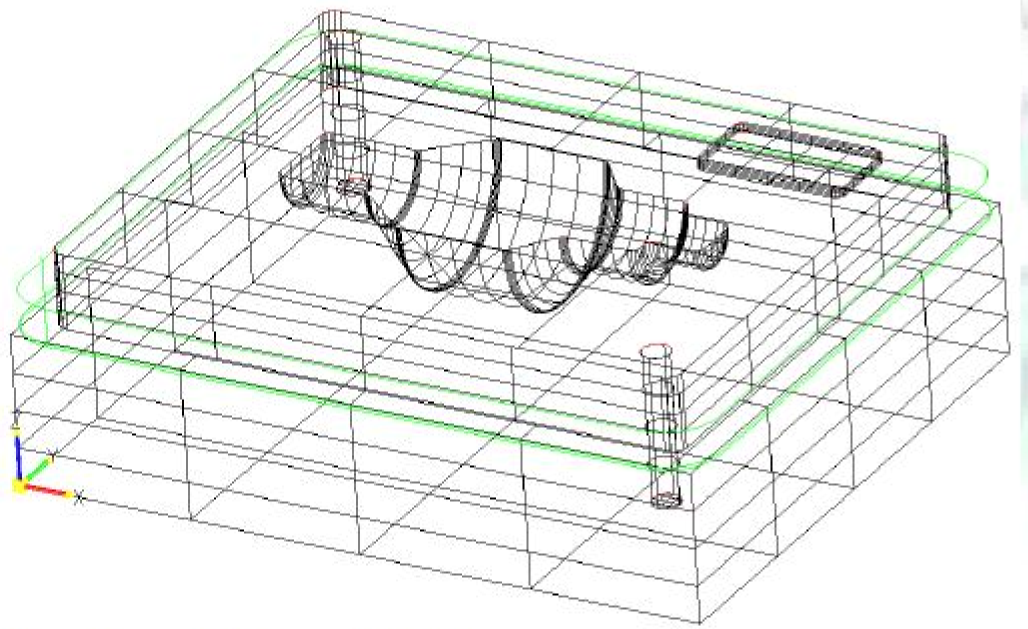
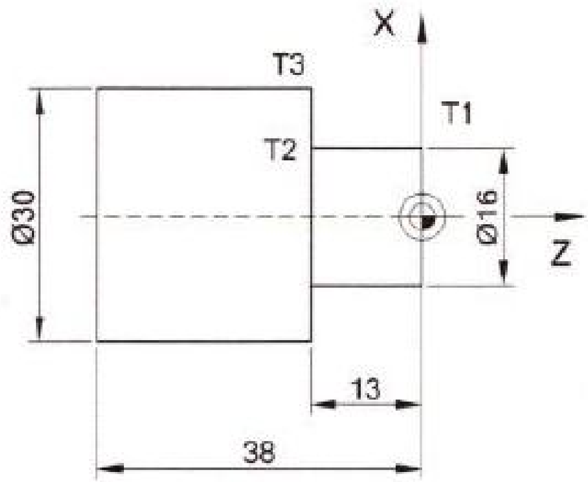


Figure 20 Contouring operation paths

Puna ne diameter dhe rreze



G90	
...	...
DIAMON	DIAMOF
...	...
G1 Z0 X16	G1 Z0 X8
G1 Z-13	G1 Z-13
G1 X30	G1 X15

Nëse programimi bëhet në sistemin matës inkremental (G91) dhe është i kycur opsioni DIAMON, të gjitha lëvizjet e instrumentit në drejtim të aksit X duhet të shprehen në diametër (duhet të shumëzohen me dy)

9. CNC tokarenje

Tokarenje je obradba rotacijskih dijelova na tokarilicama. Ovisno o načinu stezanja mogu se obrađivati i dijelovi koji nisu simetrični kao što je npr. koljenasto vratilo.

Glavno gibanje je kružno i ostvaruje ga obradak stegnut u steznu glavu. Pomoćno gibanje je translacijsko i izvodi ga alat.

9.1. CNC tokarilice

Među prvim konstruiranim strojevima bile su i tokarilice. S razvojem NC-upravljanja tokarilice su se razvijale u konstrukcijskom i upravljačkom dijelu pa ih ima različitih vrsta. Prema položaju radnog vretena tokarilice se mogu podijeliti na:

- *horizontalne (horizontalno radno vreteno) i*
- *vertikalne (karusel tokarilice).*

Horizontalnu tokarilicu susrećemo gotovo u svakoj strojarskoj radionici, a vertikalne su rijetke i uglavnom se primjenjuju za obradbu predmeta većih dimenzija.

Pravac i smjer glavnih osi određuje se na temelju pravila „desne ruke". Vertikalne tokarilice imaju uglavnom dvije osi koje se označuju sa *X* i *Z*. Horizontalne tokarilice mogu imati dvije, tri, četiri i šest osi. Pozitivni smjer osi *Z* usmjeren je od radnog vretena prema van i poklapa se s osi radnoga vretena, a pozitivni smjer osi *X* ovisi o smještaju posača alata (s prednje ili stražnje strane). Bez znanja o smjerovima osi na stroju nije moguće programirati stroj.

Slika 9.1. Horizontalna CNC tokarilica: a) smjer osi na školskoj tokarilici, b) industrijska