COMPACT 5 CNC with New Software Package (CPU-A6C 114 003)

Summary

More G-Functions

G90 Absolute Dimension
Absolute Value Programming

G91 Incremental Dimension
Incremental Value Programming

G92 Preload Registers programmed offset of reference point

G94 Feed per Minute

G95 Feed per Spindle Revolution, in mm/rev (inch/rev)

G26 Tool Offset
Tool compensation and calculation of data in own absolute value system

G23 "DNC Interface"

G24 Radius Programming: X-values are calculated with G24 as radius

More Memory and Computer Functions

+ Number of blocks: 160
N00 ..... up to N159

+ Program and execute any desired angle. Alarm sign A07 (undefined angle programmed) is not given anymore.

+ G00 allows any desired angle to be traversed.

+ Speed of computer calculation is improved.

+ Blocks can be added or deleted at later stage.

Additional Facilities using Turret Board

+ Mounting of turret toolholder possible

+ DNC Interface

+ Switching information exit

General:

The initial status (switch-on-status) remains unchanged, i.e. programming and working is possible the way as up to now.

COMPACT 5 CNC with New Software Package (CPU-A6C 114 003)

Summary

More G-Functions

G90 Absolute Dimension
Absolute Value Programming

G91 Incremental Dimension
Incremental Value Programming

G92 Preload Registers programmed offset of reference point

G94 Feed per Minute

G95 Feed per Spindle Revolution, in mm/rev (inch/rev)

G26 Tool Offset
Tool compensation and calculation of data in own absolute value system

G23 "DNC Interface"

G24 Radius Programming: X-values are calculated with G24 as radius

More Memory and Computer Functions

+ Number of blocks: 160
N00 ..... up to N159

+ Program and execute any desired angle. Alarm sign A07 (undefined angle programmed) is not given anymore.

+ G00 allows any desired angle to be traversed.

+ Speed of computer calculation is improved.

+ Blocks can be added or deleted at later stage.

Additional Facilities using Turret Board

+ Mounting of turret toolholder possible

+ DNC Interface

+ Switching information exit

General:

The initial status (switch-on-status) remains unchanged, i.e. programming and working is possible the way as up to now.

Information for Books "BASIS" and "INSTRUKTOR"

With the expansion of the computer functions, the format for G00 changes. Since angles can be programmed, the Z-coordinate has to be put in. In all programs with G00 and movement in X-direction Z=0 has to be programmed.

If there are no G24/G90/G92 programmed, all values will be taken as incremental ones. If no G95 is programmed, the feed will be in mm/min. So there is no change as compared with up-to-now programs, except that with G00 the Z-value has to be put in.

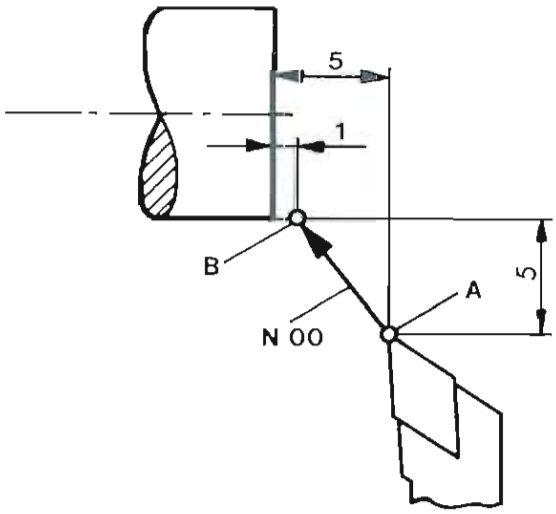
Start of Program

After input of G22 the key "START" has to be pressed twice.

After
- Program interruption (INP + REV)
- Dwell G20/G26

however START has to be pressed just once to carry on with the program.

G00 – Positioning with Rapid Traverse



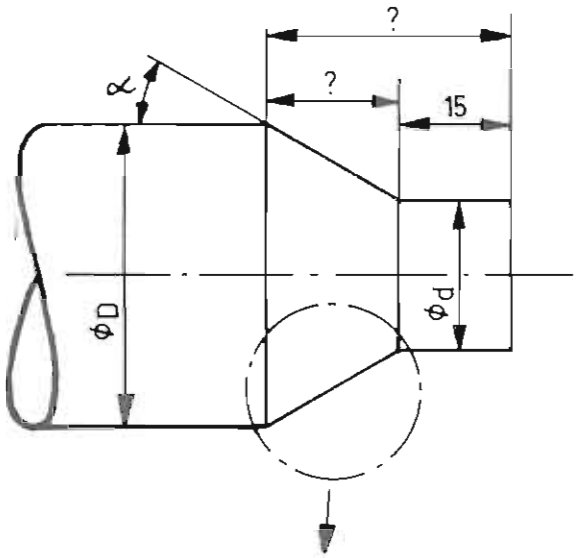
Also with rapid traverse the slides move in any desired angle.

Input format G00
N. . ./G00/X ± . . . ./Z ± . . . . .

Example:

Turning tool moves from point A to point B with rapid traverse.

N	G	X	Z	F
00	00	-500	-400	
01	22			



Angles with G00 and G01

In technical drawings you often do not find the length of the angle but just the angle indicated.

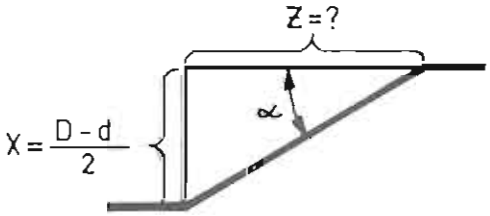
The Z-coordinate has to be calculated via the angular functions.

$$\operatorname{tg}\alpha = \frac{\frac{D-d}{2}}{Z}$$

$$Z = \frac{\frac{D-d}{2}}{\operatorname{tg}\alpha}$$

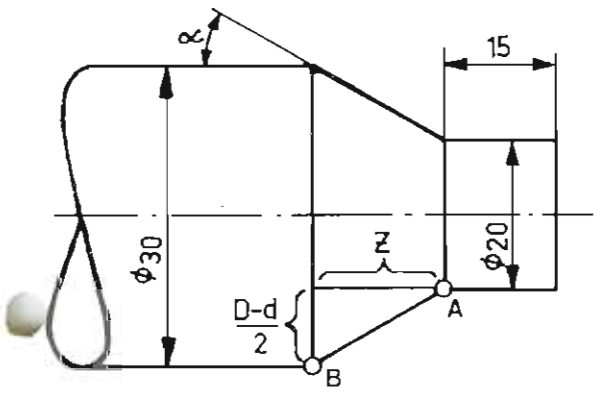
The X-value is known from the difference in diameter. The Z-value has to be calculated.

The tangens values can be read from the charts or slide rules or calculators.



Common angle with turned parts

α	tg α
15°	0,268
30°	0,577
45°	1
60°	1,732
75°	3,732



Example:

$$\operatorname{tg}\ 30^{\circ} = \frac{\frac{D-d}{2}}{Z}$$

$$Z = \frac{\frac{D-d}{2}}{\operatorname{tg}\ 30^{\circ}} = \frac{\frac{30-20}{2}}{\operatorname{tg} 30^{\circ}} = \frac{5}{0,577} = 8,66 \text{ mm}$$

Programming from A to B (incremental)
N.../G01/X=500/Z= -866/F...

G94 – Indication of Freed rate in mm/min (inch/min)

G95 – Indication of Feed size in mm per revolution (inch per revolution)

Switch-on-status (initial status)

If G94 and G95 are not programmed, the feed is put in and calculated in mm/min.

Programming of G94/G95

Format G94	Format G95
N.../G94	N.../G95

G94 and G95 are self-holding functions.
That means: G94 is valid until it is revoked by a G95 instruction and vice-versa.

Feed Sizes, Inputs and Resolution

G94 (mm/min)

Metric inputs

Resolution: mm per minute,
possible inputs: 2 - 499 (mm/min)

Inch inputs

Resolution: 1/10 inch per minute,
possible inputs: 2 - 199 (= o,2 - 19,9 inch/minute)

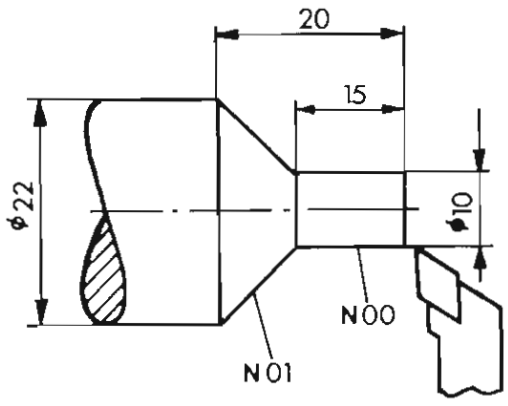
G95 (mm/rev)

Metric inputs

Resolution: 1/1000 mm per revolution,
possible inputs: 2 - 499
(o,oo2 - o,499 mm/rev)

Inch inputs

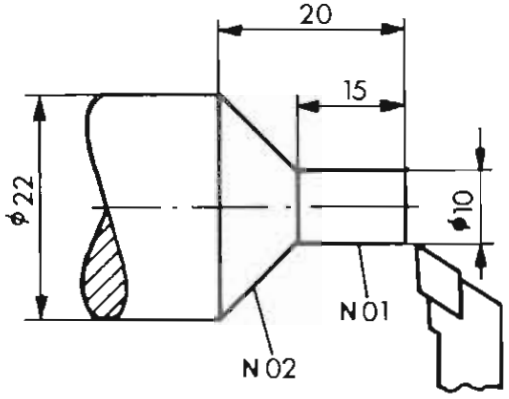
Resolution: 1/10,000 inch per revolution,
possible inputs: 2 - 199 (0,0002 - 0,0199 inch/rev)



Example 1

N	G	X	Z	F
00	01	0	-1600	100
01	01	600	-500	120
02				
03				

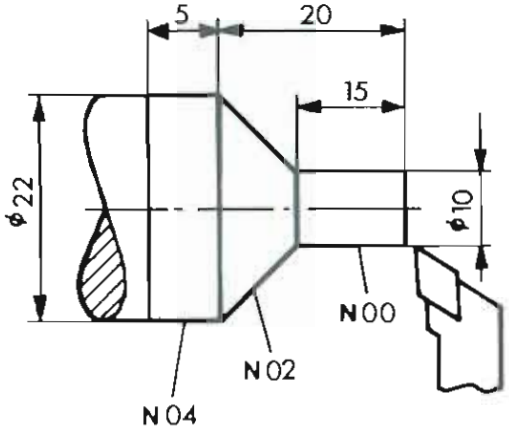
If no G94 or G95 is programmed, then all feed values will be calculated as feed rate.



Example 2

N	G	X	Z	F
00	**95**			
01	01	0	-1600	50
02	01	600	-500	40
03				
04				

In block NO1 the feed is o,o5o mm/rev, in block NO2 the feed is o,o4o mm/rev. The feed indication is taken as "feed per revolution) until a G94 instruction is given.



Example 3

N	G	X	Z	F
00	01	0	-1600	120
01	**95**			
02	01	600	-500	40
03	**94**			
04	01	0	- 500	100

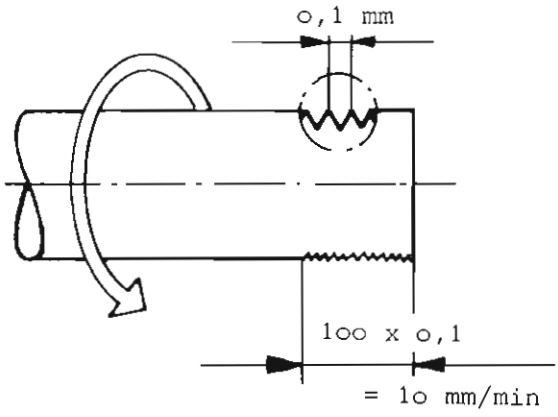
Block NOO: mm/min (12o mm/min)
Block NO2: mm/rev (o,o4 mm/rev)
Block NO4: mm/min (1oo mm/min)

Starting the program if G95 (mm/rev or inch/rev) is programmed

The program runs only if the main spindle is switched on, since the computer has to synchronize main spindle revolutions and feed rate. If there is no information on the main spindle speed, the computer cannot give any information in mm/rev.
The speed of the tool slides depends on the spindle speed if there is constant cutting speed.

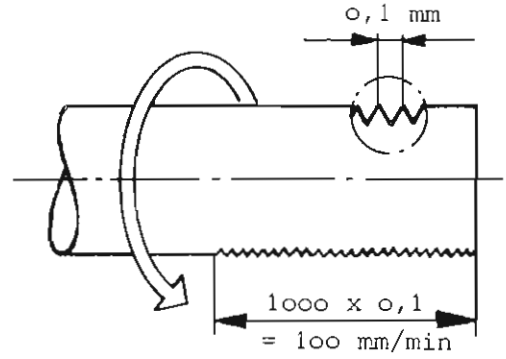
Example 1

Main spindle speed 1oo rev/min
rogrammed feed o,1 mm/rev
The slide moves per minute
1oo x o,1 mm, thus 1o mm.



Example 2

Main spindle speed 1ooo rev/min
Programmed feed o,1 mm/rev
The slide moves per minute
1oo mm, thus 1o times the distance when compared with a spindle speed of 1oo rev/min



Limitation of the Maximum Feed per Revolution by the Main Spindle Speed

The max. feed rate with the feed per revolution is automatically limited with 499 mm/min.

Example:

Main spindle speed is 3000 rev/min.
G95 programmed feed per revolution:
o,499 mm/rev (max. programmable feed per revolution)

F (mm/min) = S (rev/min) x F (mm/rev)
= 3000 x o,499 = 1497 mm/min

The speed of the slide would be 1497 mm/min. With this spindle speed the slide moves at approx. o,17 mm/rev even though o,499 mm/rev are programmed.

$$\frac{499 \text{ mm/min}}{3ooo \text{ U/min}} = o,166 \text{ mm/U}$$

Compare technological data, pages 1.8-1.1o.

G90 – Absolute Dimension
G91 – Incremental Dimension
G92 – Preload Registers
(programmed offset of reference point)

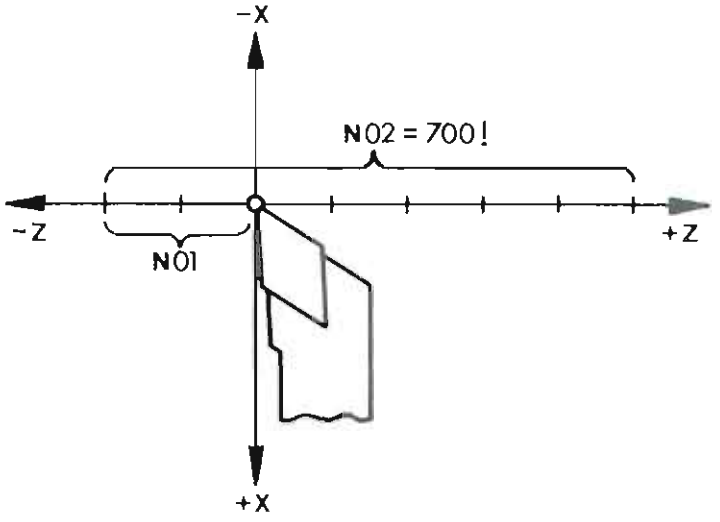
General:

G90/G91/G92 are self-holding instructions and are valid until they are revoked or modified.

Switch-on-status (initial status)

If no G90/G91/G92 is programmed, all dimensions are calculated incrementally.

G90 – Absolute Dimensions
Programming of Absolute Value
Programming of Reference Point



Note:

Coordinate zero point is the position of slides (tool) when G90 is programmed the first time in the manuscript.

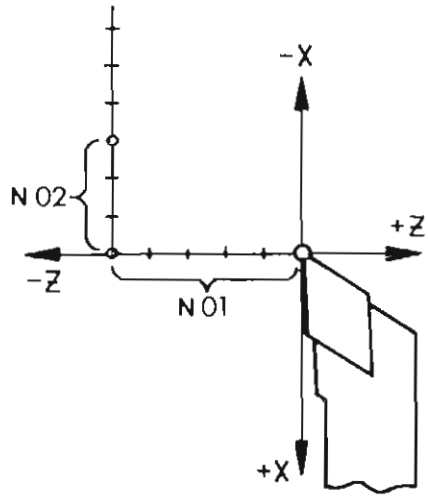
Format G90
N . . ./G90

Example 1: Z-Dimensions

You put in the program:

N	G	X	Z	F
00	90			
01	01	0	-200	100
02	00	0	+500	100
03	22			

Zero point of the coordinate system is the actual position of the slides when programming G90 the first time in a program (if no G92 was programmed before).
In block N02 Z +500 is programmed. The slide, however, moves +700 (+7 mm).



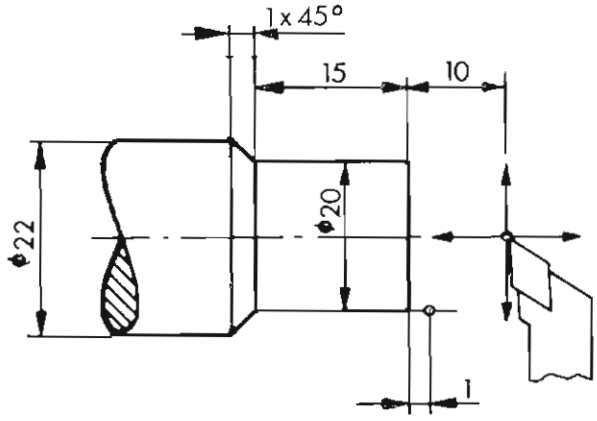
Example 2: X-Dimensions

X-values are calculated as diameter values with G90.

N	G	X	Z	F
00	90			
01	00	0	-500	
02	00	-600	0	
03	22			

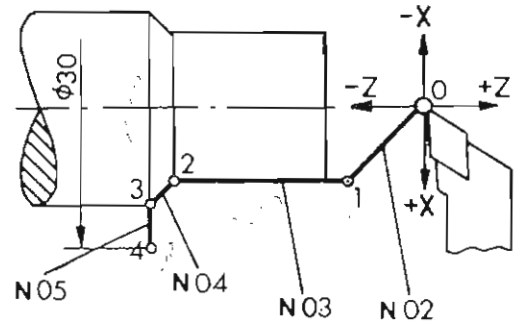
Explanation:

Most dimensions of turning parts are given as diameters. Therefore it is more comfortable to program diameters.



Example 3:

- Position of right-hand side tool is at program start as in drawing.
- The zero-point of the coordinate system is at the top of the tool.
- Programming of points 0 to 4.

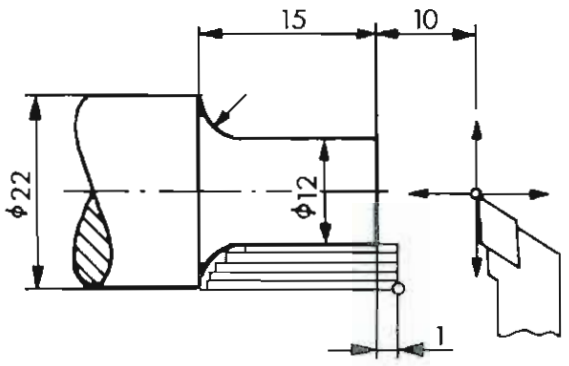


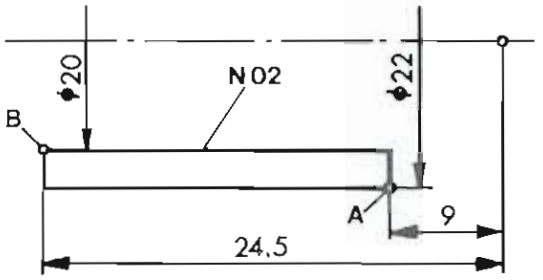
Remember

In block N01 G90 is programmed.
Actual position (slide positions) determines the coordinate zero-point.

PROGRAMMING SHEET EMCO COMPACT 5 CNC

N	G	X	Z	F	Remarks	S(1/mm)
00	95				Feed in mm/rev	
01	90				Absolute dimension	
02	00	2000	-900		X-value = diameter value from point 0 to 1	
03	01	2000	-2500	30	From point 1 to 2	
04	01	2200	-2600	30	From point 2 to 3	
05	00	3000	-2600		From point 3 to 4	
06	22					

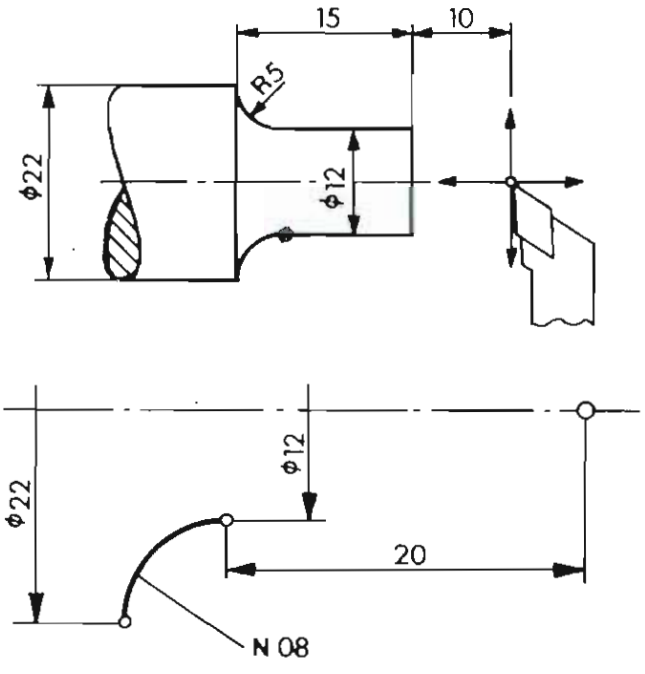




Example 4: G84/G78

Cycle G78/G84 is determined by programming of point B (between second and third step). The cycle starts in point A which was programmed last.

N	G	X	Z	F
00	90			
01	00	2200	-900	
02	84	2000	-2450	120
03	84	1800	-2430	120
..	..	....	....	...

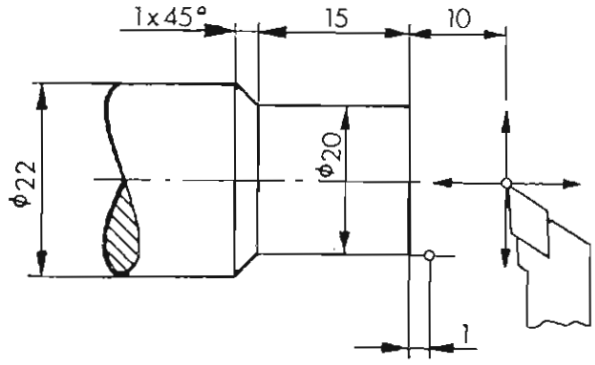


Example 5: G02/G03

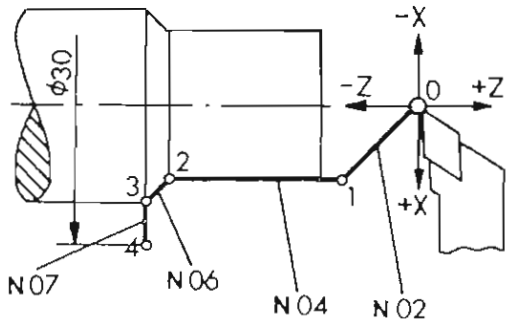
With G02/G03 the end point of the 4th path of circumference is programmed in absolute dimension. The tool moves along the fourth path of circumference between the point programmed last and the point programmed under G02/G03.

N	G	X	Z	F
.	90			
.				
.				
07	01	1200	- 2000	120
08	02	2200		120
.				
.				

Mixed Programming
G90/G91



The instruction G90 is invalidated by G91 until G90 is programmed again. The computer, however, remembers the zero-point of the coordinates once existing within a program, which was fixed with the first programming of G90.



Example:

Points 0 to 4 are being programmed. Position of tool bit when starting as in drawing.

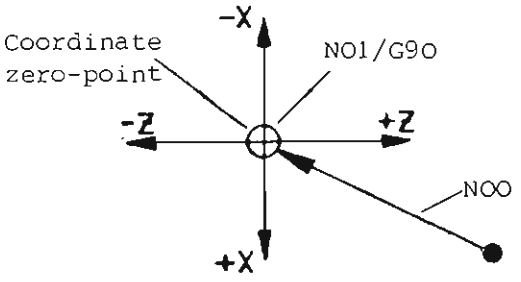
PROGRAMMING SHEET COMPACT 5 CNC

N	G	X	Z	F	Remarks
00	95				
01	**90**				Absolute dimension
02	00	2000	-900		From point 0 to point 1
03	**91**				Incremental dimension
04	01	0	-1600	50	From point 1 to point 2
05	**90**				Absolute dimension
06	01	2200	-2600	30	From point 2 to point 3
07	00	3000	-2600		From point 3 to point 4

Attention:

If you press "START" once again, the coordinates zero point would be point 4, because with the first instruction G90 the tool slides were positioned in point 4.

Attention with Mixed Programming



Demonstrating example:

1. When programming G90 the zero point of the axis system is the slide position with the first G90 programming.

N	G	X	Z	F
00	00	-100	-200	
01	90			

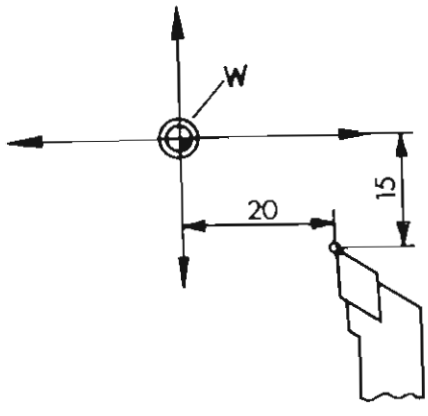
2. If in a program G90 is followed by G91 and then G90 again, the coordinates system is valid which was determined in the first G90 information.

3. Except with inside turning an absolute value programming is not useful or possible without G92, because the tool bit is positioned in the zero point of the workpiece with the first G90 programming.

G92 – Preload Registers
Programmed Offset of Reference Point

In the previous examples we have learned that the zero-point of the coordinates is in the slides (tool) position in which G90 was programmed for the first time.

This would be unpractical because when programming G90 the tool can rarely be positioned on the center line axis in X-direction.
Using G92 you can shift (offset) the zero-point of coordinates on the machine wherever you like it.
Subsequently with G92 all following values will be calculated on absolute basis. Programming of G90 is not necessary.



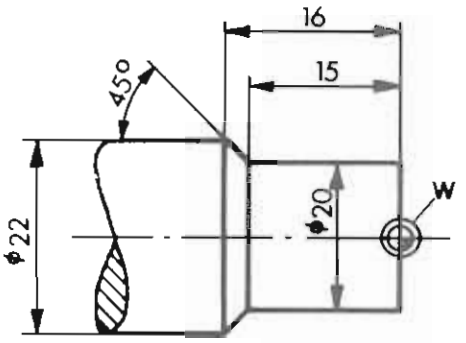
Example:
Zero-point of coordinates has to be offset from indicated top of tool to point W.

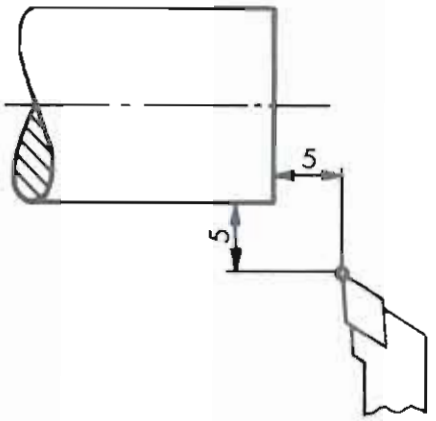
Offset

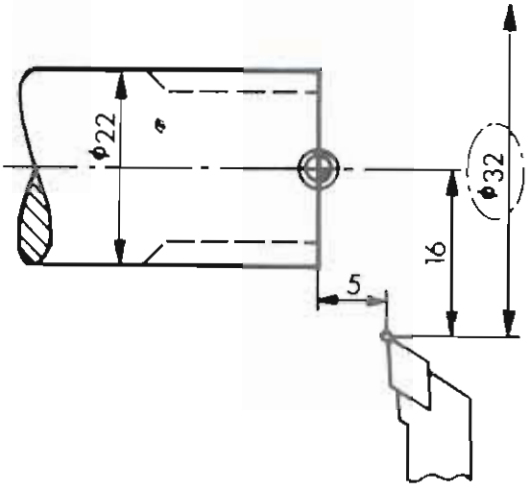
N	G	X	Z	F
..	92	+3000	+2000	

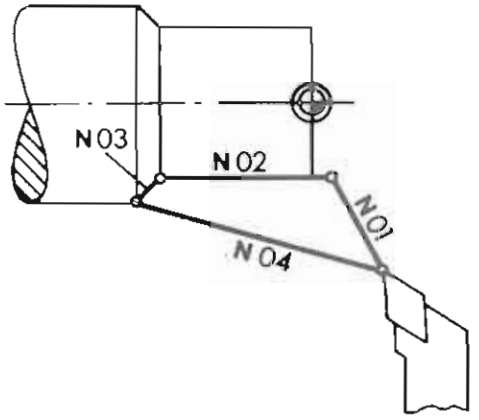
- Take the zero-point of coordinates in point W and describe the top of the tool (slides) starting from this point.

- X-values are indicated as diameter-values.









Example:

Measurements are put on from left hand side with this workpiece. Therefore it is useful to offset the zero-point of coordinates to the indicated position. This zero-point is called workpiece zero-point W (Symbol ⊕ W).

1. Positioning of tool when starting

The tool bit is moved into the position as indicated before the operation starts (via "scratching").
In this example: X +5 mm, Z +5 mm

2. Determination of the zero-point of coordinates

A logic zero-point of coordinates is the workpiece zero-point as indicated.

3. Offset

N	G	X	Z	F
00	92	3200	500	

The programming of the operation starts from the desired zero-point of coordinates.

Programming exercise

Position of tool bit as indicated

N	G	X	Z	F
00	92	3200	500	
01	00	2000	100	
02	01	2000	-1500	120
03	01	2200	-1600	120
04	00	3200	500	
05	22			
06				

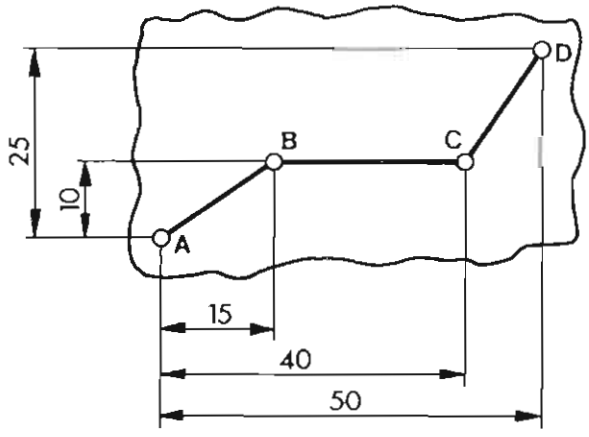
G24 – Radius Input with Absolute Value Programming

When programming G90/G92 X-values will be calculated as diameter values. When milling this would mean calculation work.
G24 makes radius programming possible, i.e. X-coordinates are taken as absolute values but not as diameter values.

Programming

+ G24 has to be programmed in the first block (otherwise alarm A00)
+ G24 is self-holding
+ G24 cannot be revoked within a program
+ In the next block G90 must be programmed otherwise the values are calculated incrementally.

Format G24
N00/G24

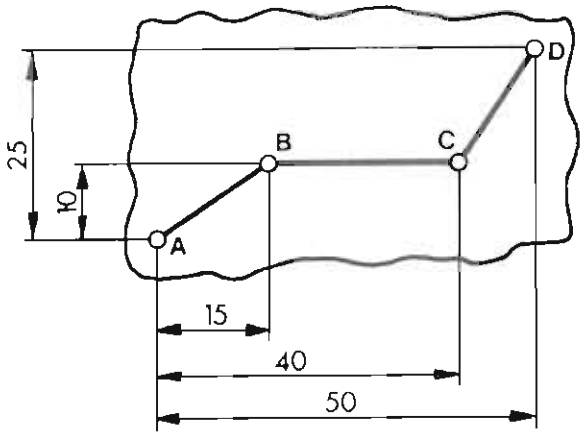


Example: 1

Plotter pen to move from point A to B,C D. Since points on drawing are indicated as absolute values, absolute value programming will be useful.

Program:

N	G	X	Z	F	
00	24				
01	90				
02	00	**-1000**	1500		A→B
03	00	**-1000**	4000		B→C
04	00	**-2500**	5000		C→D
05	22				
06					



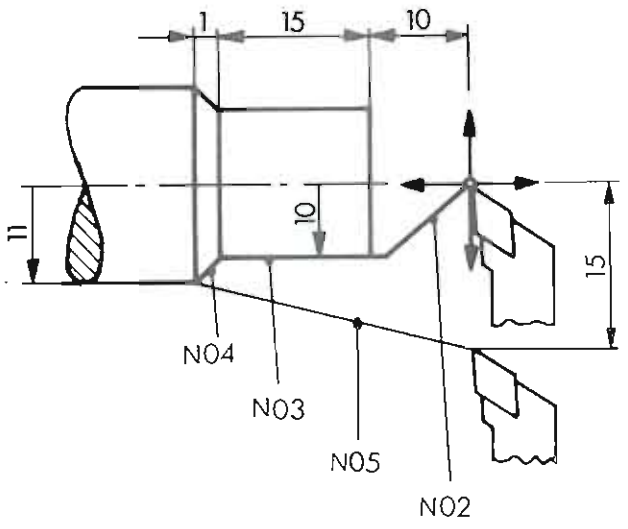
Example: 2

Describe B/C/D starting from point A, if you have programmed G90 in the first block.

N	G	X	Z	F
00	90			

Example:3

Describe the movements of the plotter of written program - Example 2 if in the first block G24 is programmed.



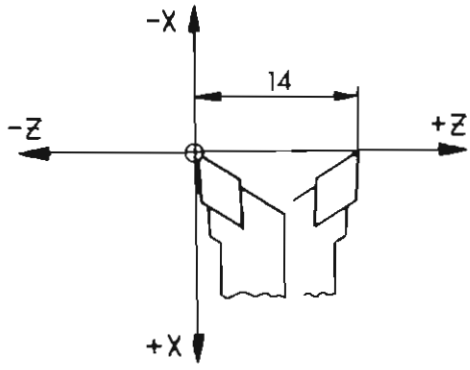
Example:4

N	G	X	Z	F
00	24			
01	90			
02	95			
03	00	1000	-900	
04	01	1000	-2500	30
05	01	1100	-2600	30
06	00	1500	0	
07	22			

G26 – Tool Data Processing
Tool Offset
Tool Wear Compensation

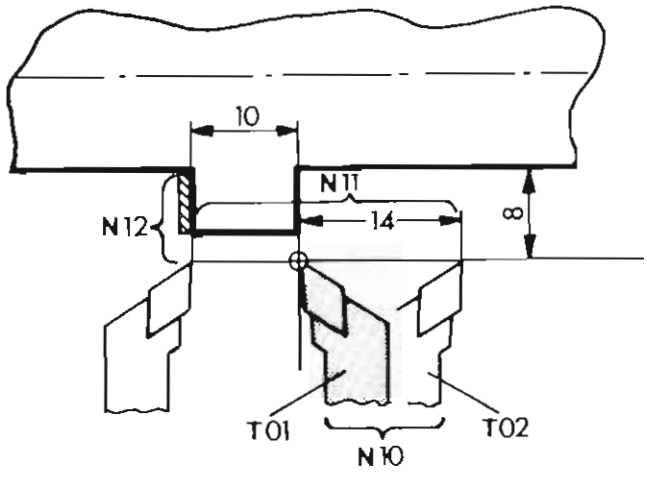
Repeat

The tools are preset using the setting gauge. The position of the tools to each other therefore is known (compare page 8.2o).
In the program the difference-values are calculated.



Example:

- The tool to start with is the right hand side tool.
- The cutting edge of the left hand side tool features the coordinate X=0, Z= +14 mm as seen from the zero-point of the right hand side tool (T01).
- When programming, this measurement has to be considered.

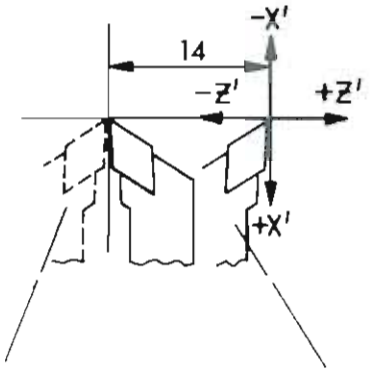


Programming Examples:

N	G	X	Z	F
..	..			
10	20			
11	00	0	-2400	
12	01	-800		

- When turning off the left shoulder, the right hand side tool has to be replaced by the left hand side tool.
- Position of the left hand side tool = Z +14 mm.
- In block N11 the distance -(14+1o) has to be programmed. You had to do this calculation. This calculation work can be taken over by the computer.

G26 – Tool Length Calculation



Desired value Actual value

Under G26 you can put in the coordinates of single tools. These values will be processed automatically.

We take the right hand side tool as reference tool.

We program the desired value of the left hand side tool.

X = 0
Z = −1400

G26 with F=O means automatically dwell.

The data are processed in the following block:

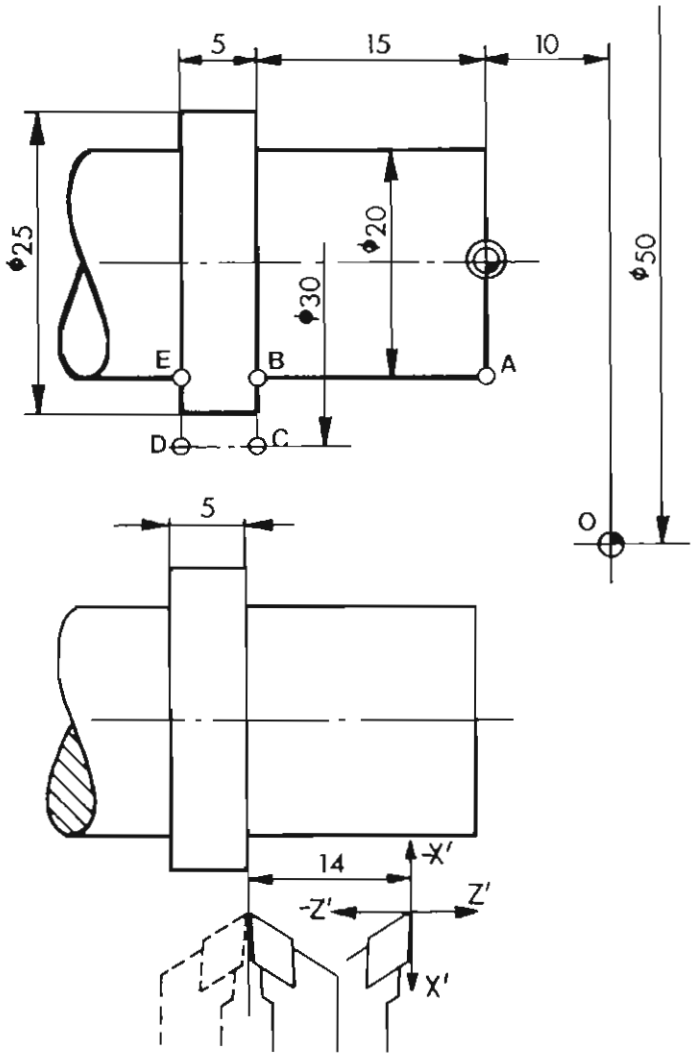
Format G 26
N../X ± .../Z ± .../F = O

Note:

-Program desired position
-F = O means: Program hold

PROGRAMMING SHEET COMPACT 5 CNC .

N	G	X	Z	F	Remarks	S [1/min]
..	26	0	0	0	Right hand side tool T01	
..	26	0	-1400	0	Left hand side tool T02	



Example:

Programming contour ABC with the right hand side tool (TO1); DE with the left hand side tool (TO2).

Block NOO: Setting off the reference point
Block NO1: Coordinates of right hand side tool
Block NO2-NO4: Contour ABC

Block NO5:

- G26 with F = O means automatically "dwell".

- Tool change

- The position of left to right hand side tool is X=O/Z= +14 mm.

Block NO6:

- Traverse of left hand side tool T2 after pressing START in point D.

- The value Z= +14 mm is automatically taken into account.

- Although a slide movement of 5 mm (from Z -1500 to Z -2000) is programmed, the slide moves -19 mm.

Block NO7:

Movement from D to E.

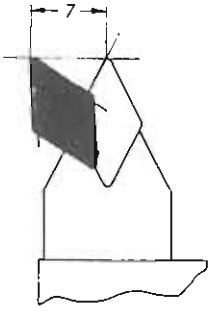
N	G	X	Z	F	Remarks	S [1/min]
00	92	5000	1000		Tool TO 1	
01	26	0	0	0		
02	00	2000	0			
03	01	2000	-1500	100		
04	01	3000	-1500	100		
05	26	0	-1400	0	Tool TO2	
06	00	3000	-2000			
07	01	2000	-2000	100		
08						
09						

Positioning of cutting edges of tools to each other if slide position remains unchanged (Positions are adjusted with mechanical presetting gauge)

Right hand side tool: G26/X = 0/Z = 0/F = 0

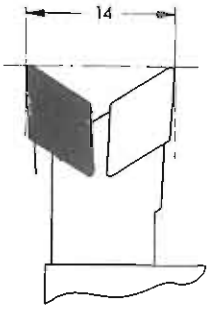
Neutral Side Tool

Desired Position: X=0/Z=-700
G26/X=0/Z=-700/F=0



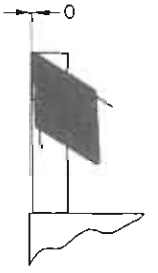
Left Hand Side Tool

Desired Position: X=0/Z=-1400
G26/X=0/Z=-1400/F=0



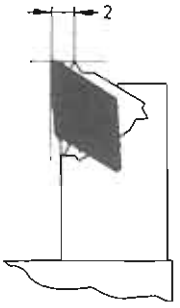
Parting-off Tool

Desired Position: X=0/Z=0
G26/X=0/Z=0/F=0



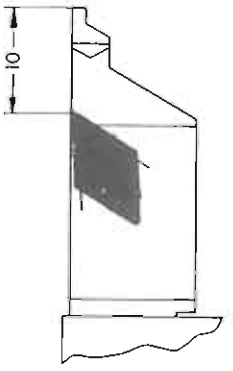
Outside Threading Tool

Desired Position: X=0/Z=-200
G26/X=0/Z=-200/F=0



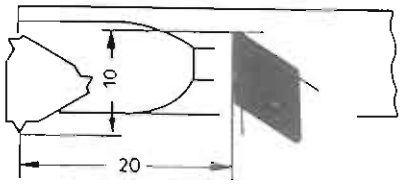
Plunging Tool

Desired Position: X=1000/Z=0
G26/X=1000/Z=0/F=0



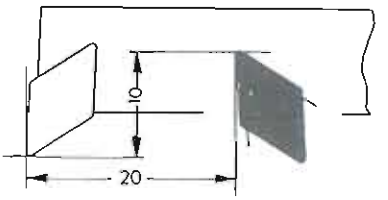
Inside Threading Tool

Desired Position: X=-1000/Z=2000
G26/X=-1000/Z=2000/F=0



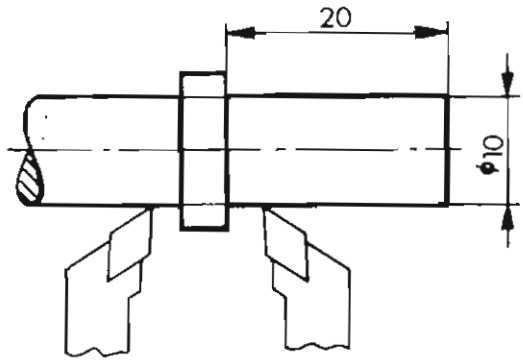
Inside Turning Tool

Desired Position: X=-1000/Z=2000
G26/X=-1000/Z=2000/F=0



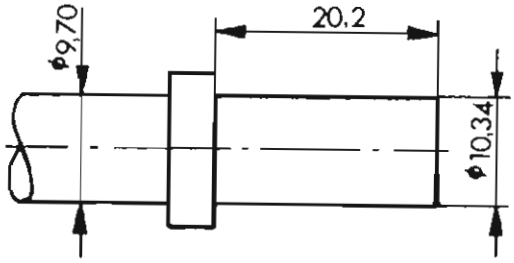
Compensation of Tool Edges

With tools which are worn off or not set accurately, the workpiece is not manufactured to measurement. G26 enables compensation values to be programmed.



Example

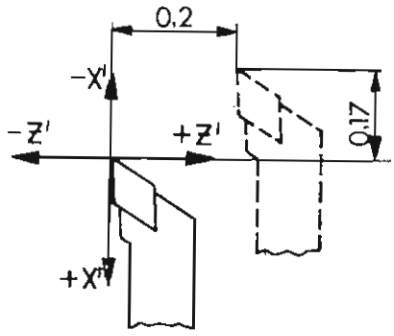
You have programmed and manufactured the given workpiece. Right and left hand side tool were set using the tool presetting device.



Right hand side tool

When measuring you find out that the diameter of the manufactured workpiece (with the right hand side tool) equals 1o,34 mm.

The desired value in X-direction is
X= -O,17 mm
The desired value in Z-direction is
Z= +O,2 mm

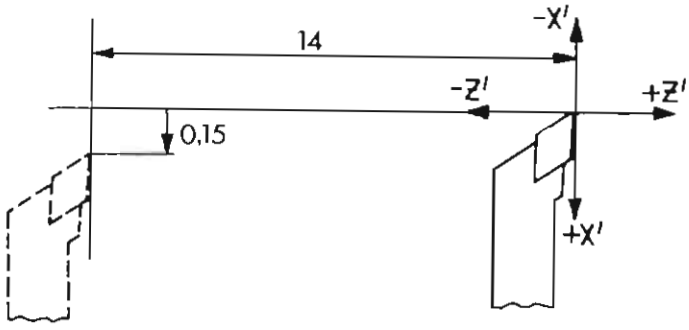


Compensation input:
G26/X= -17/Z= +20/F=0

Left hand side tool

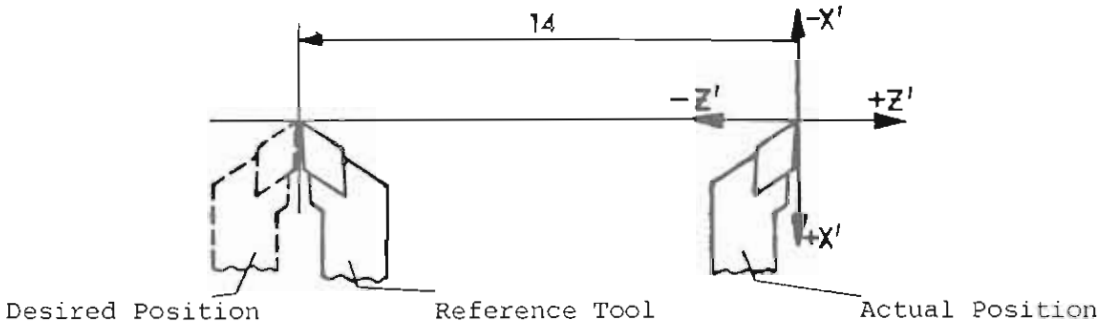
The diameter is too small by o,3 mm.
Desired value of the left hand side tool originally X=O/Z= -1400

Compensated desired value:
X= +O,15 mm
Z= -14,OO mm



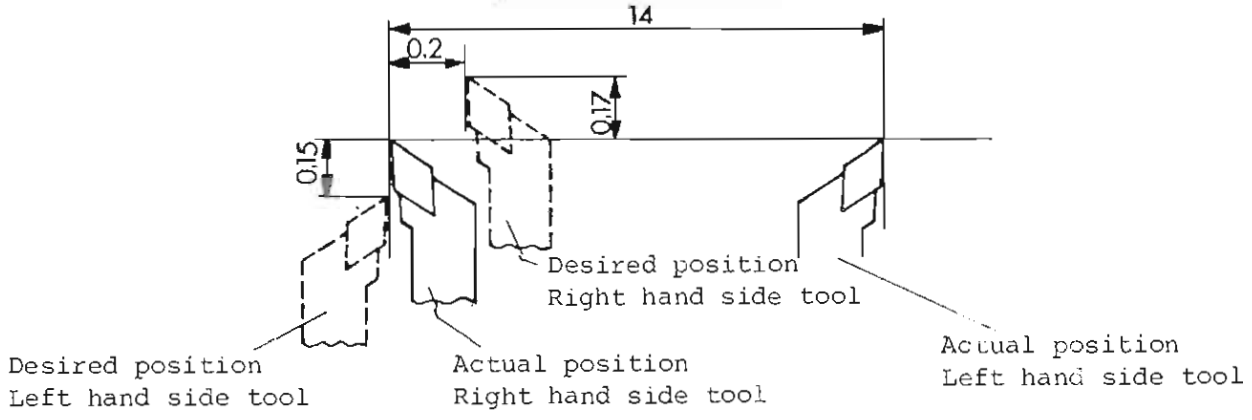
Compensation input:
G26/X=15/Z= -1400/F=0

Original Tool Values



N	G	X	Z	F	Bemerkungen	S [1/min]
..	26	0	0	0	Right Hand Side Tool	
..	26	0	-1400	0	Left Hand Side Tool	

Compensated Tool Values

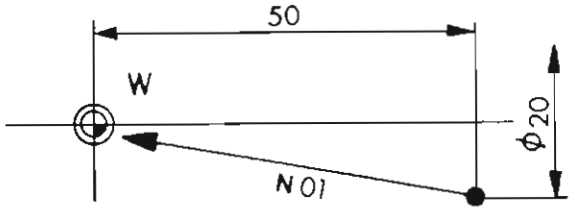


PROGRAMMBLATT EMCO COMPACT 5 CNC

N	G	X	Z	F	Bemerkungen	S [1/min]
..	26	-17	20	0	Right Hand Side Tool	
..	26	15	-1400	0	Left Hand Side Tool	

Exercises G92/G26

For a better understanding of G92/G26 mount the plotter device and execute following exercises:

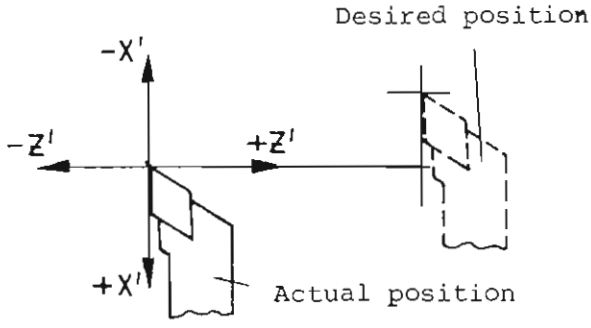


Example G92

N	G	X	Z	F
00	92	2000	5000	
01	01	0	0	400
02	22			

The plotter pen moves into the O-point of the coordinates system (in X-direction -2000:2 = -1000 (-10 mm), in Z-direction -5000 (-50 mm).

Example 1 on G26

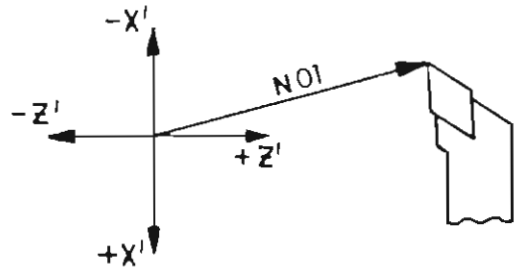


Tool no. 1: Right hand side tool
Actual position: like indicated
X=0/Z=0
Desired position: X= -400
Z= +1500

Program originally:

N	G	X	Z	F
00	26	0	0	0
01	00	0	0	
02	22			

The traverse path in block NO1 is programmed with O.



Program compensated:

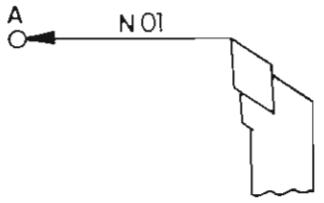
N	G	X	Z	F
00	26	-400	+1500	0
01	00	0	0	
02				

Though the traverse path is programmed with O in block NO1, the slide moves into desired position.

Note:

After G26 "START" has to be pressed.

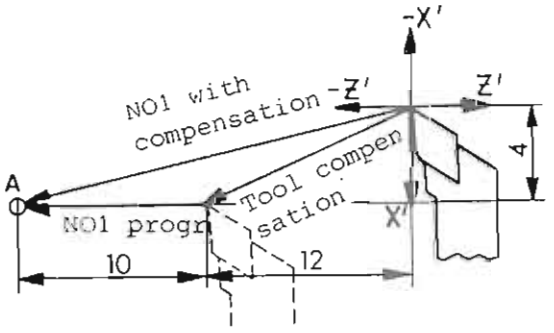
Example 2 on G26



1. No tool compensation:

N	G	X	Z	F
00	26	0	0	0
01	00	0	-1000	
02	22			
03				

The slide moves the distance Z= -1000 to point A.

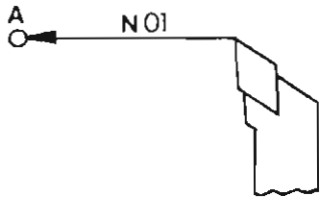


2. Tool compensation:

Actual position: X=0/Z=0
Desired position: X=400/Z= -1200

N	G	X	Z	F
00	26	400	-1200	0
01	00	0	-1000	100
02	22			

The computer calculates the shortest possible way to point A and moves to it diagonally.

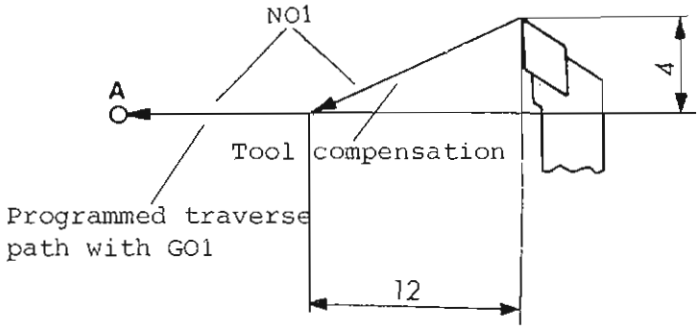


Example 3 on G26

Program:

N	G	X	Z	F
00	26	0	0	0
01	01	0	-1000	100
02	22			
03				

The tool bit or plotter pen moves Z -1000 to point A.



Program with compensated tool values

N	G	X	Z	F
00	26	400	-1200	0
01	01	0	-1000	100
02	22			
03				

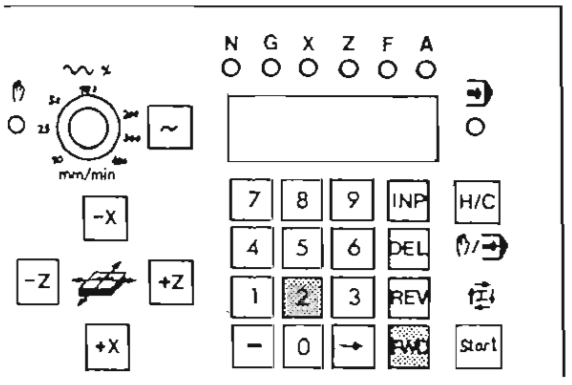
The tool first moves according to the compensation X= 400/Z= -1200 (from actual to desired position) and then the programmed path of block N01 to point A.

Summary

- With all "working functions" (G01/G02/G03/G33/G78/G84) the desired position is headed first.
- If after a tool compensation in the following block G00 is programmed, then compensation and G00 information will be executed simultaneously in the shortest distance possible.
- Attention:

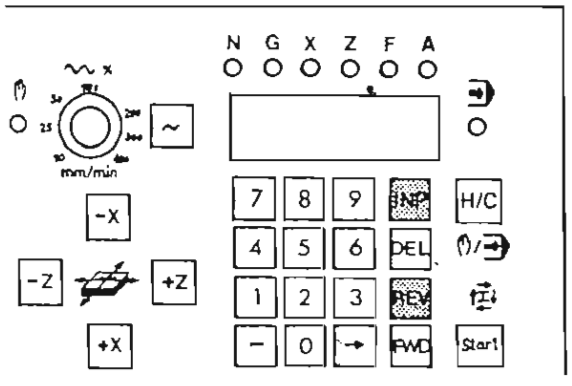
  X-values in connection with G26 will be calculated incrementally.

Operating the Turret Toolholder



1. By hand:

Press key FWD and a number key, the turret will swivel by the number on the number key pressed.
For example: you press FWD and 2: the turret swivels twice.

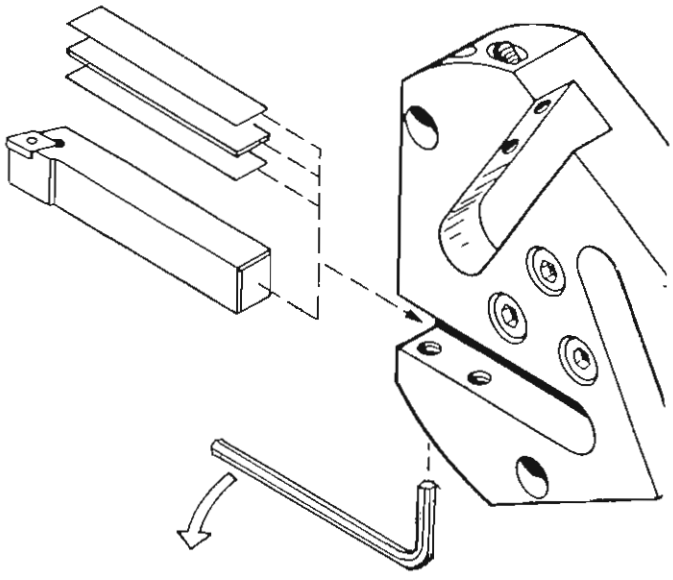


2. Swivelling in CNC-operation:

G23/X=0/Z=0/F..
Put in the number of positions to be swivelled under address F.
For example: G26/X=0/Z=0/F=2: the turret swivels by 2 positions.

Interrupting the swivel operation

Press key INP + REV

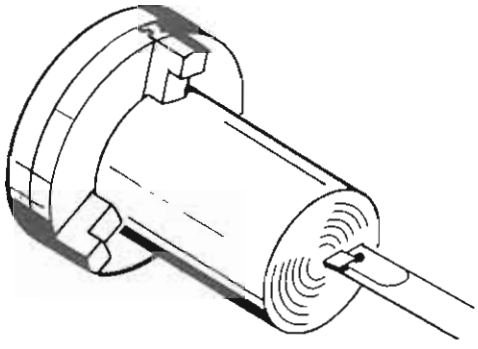


Mounting of tools:

The turret holds 3 outside and 3 inside tool bits.

Mounting of outside tool bits

Adjust tool bits to correct center height by using washers (o,2/o,5/1 mm), whereby tools should not stand out more than 13 mm.



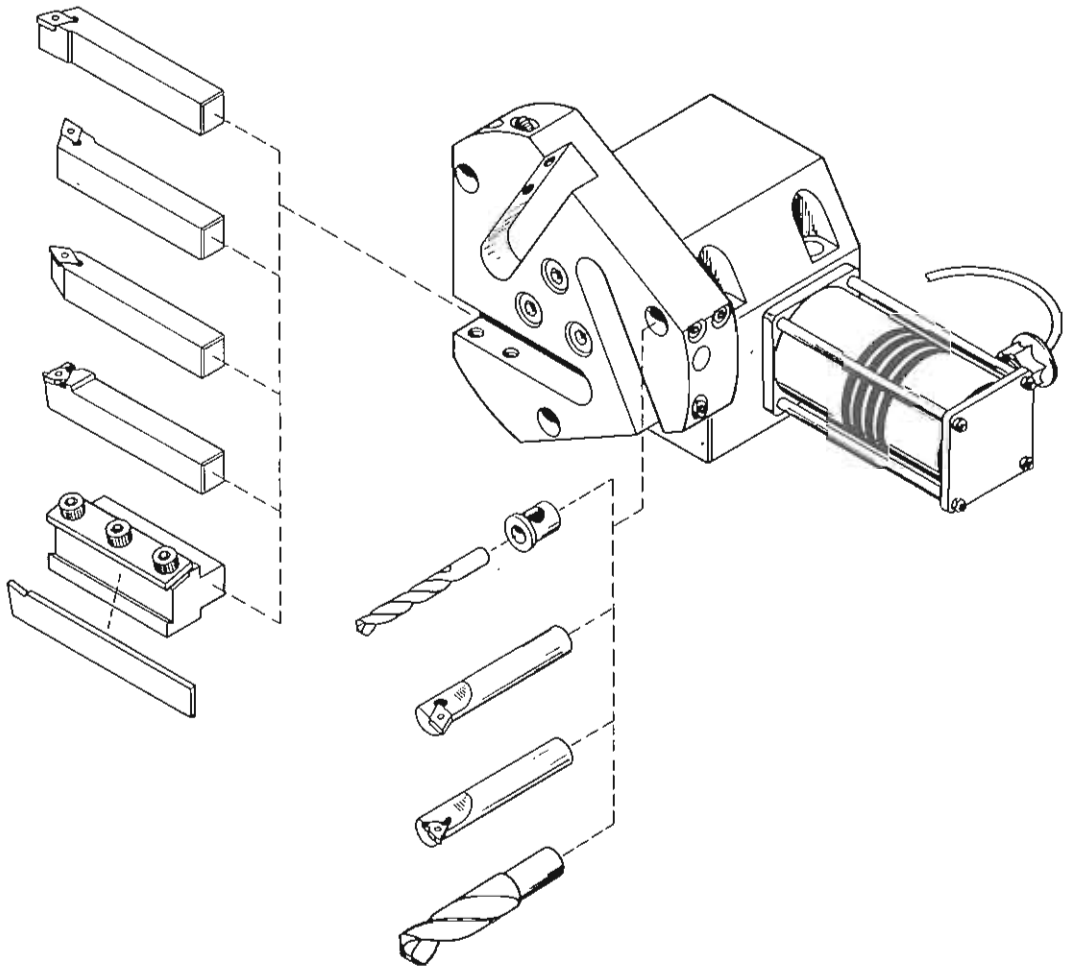
Mounting of inside tool bits

For drills use correct adaptor sleeve. Adjust inside turning tool and inside threading tool such that carbide tip is at center height. The correct adjustment is achieved by mounting a workpiece in the chuck.

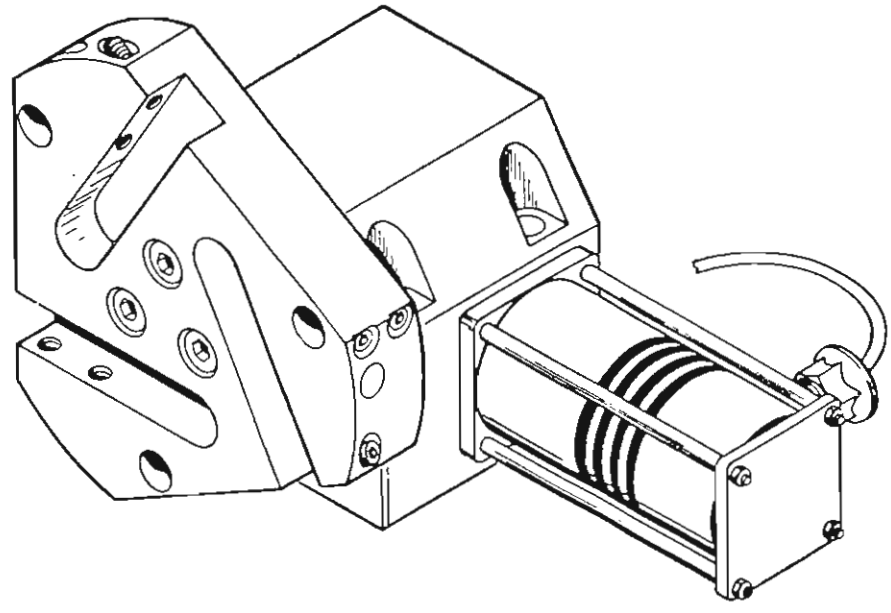
Note:

If center height of the drill is not exact, loosen the socket head screws of the turret plate. Swivel the plate to reach center height and tighten screws again.

Tooling System for Automatic Turret Toolholder



Tool data



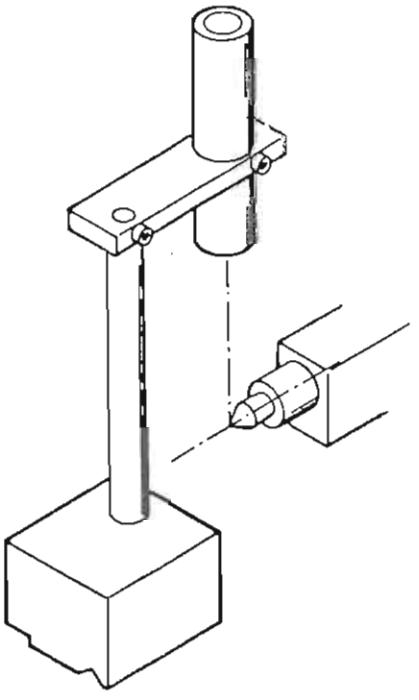
Draw the positions of the different tools so that you can program the respective divisions.

TOOL CHART

Tool	Position in turret	X-value	Z-value

Collecting tool data with optical presetting device.

By "scratching" the workpiece, one could measure the positions of the various tool edges to each other. This system would be very inefficient.



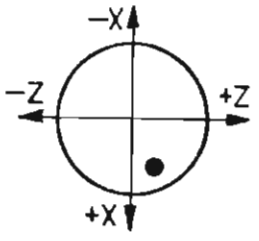
Procedure

1. Optical pre-setting device to be put onto machine bed and to be adjusted at axis center.

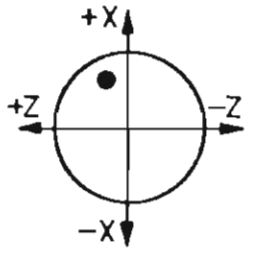
   - Adjust tube in height until tailstock center appears clear and sharp.
   - Clamp tube support such that cross wire is exactly on axis center.

Image inversion

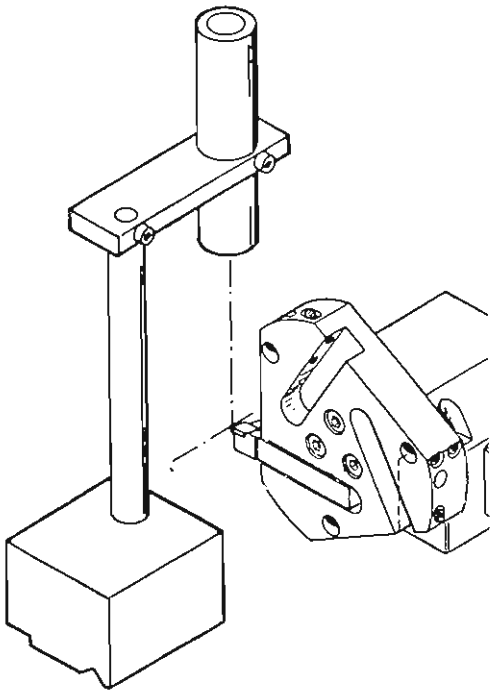
Because of the given optics the image is reflected along the X/Z axis.
A point, seen through the pre-setting device, seems to be reflected along the X/Z axis.



Point position seen without optics.

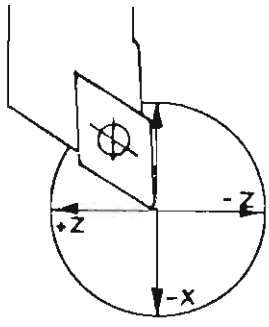


Reflected point position seen through the optical pre-setting device.



2. Collecting of tool data

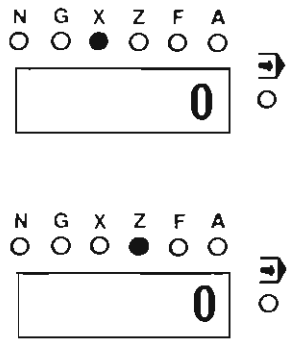
- Place optical pre-setting device in front of turret.
- Move reference tool into cross wire. You can choose anyone tool as reference tool. A common reference tool will be the one with which you start the operation. If the tool is in the cross wire, set X and Z read-outs to zero.



Right hand side tool in cross wire

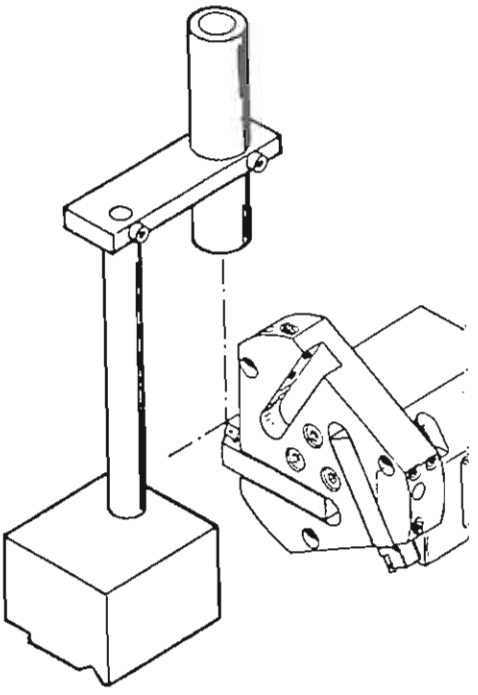
Example: right hand side tool is reference tool.

- X and Z display read-out to be set at zero (press key DEL).
- Enter data into tool chart.

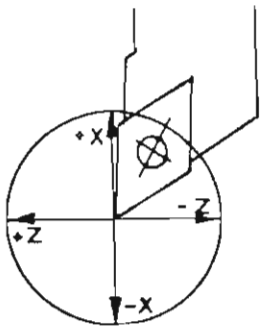


TOOL CHART

Tool	Position in turret	X-value	Z-value
R.H. Tool T01	1	0	0

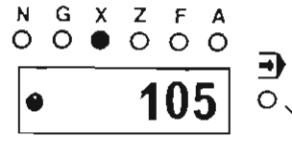


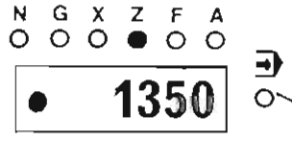
- Swivel following tool into working position.
- Move slides until tool is in cross wire.
- Enter X and Z values (traverse paths)



Left hand side tool
in cross wire

Example: left hand side tool





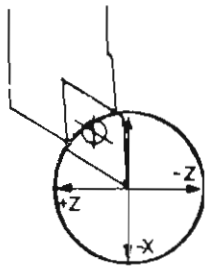
WERKZEUGBLATT

Werkzeug	Position im Werkzeugrevolver	X-Wert	Z-Wert
Right Hand Side Tool	1	0	0
Left Hand Side Tool	3	- 105	-1350

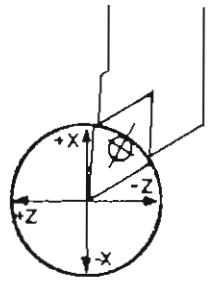
In this way all further tools can be determined. The values of the traverse movements are put into the tool chart so that they are available when programming.

Reflected Images Seen Through the Pre-Setting Device

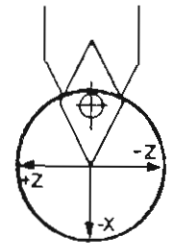
Right hand side tool



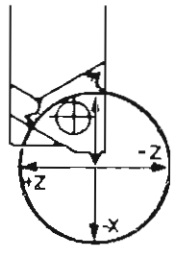
Left hand side tool



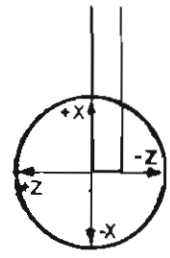
Neutral side tool



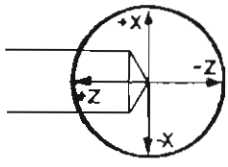
Threading tool outside



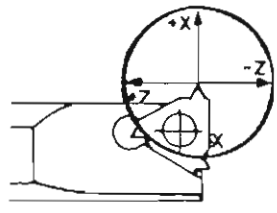
Parting-off tool



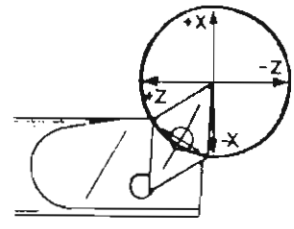
Twist drill
Centering drill

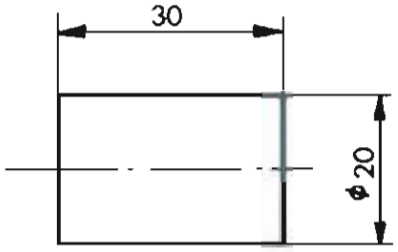


Threading tool inside



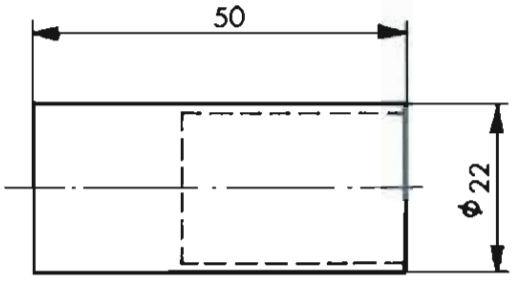
Boring tool inside



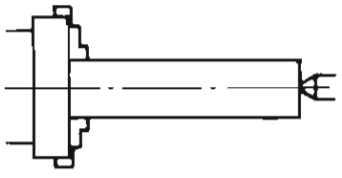


Example:

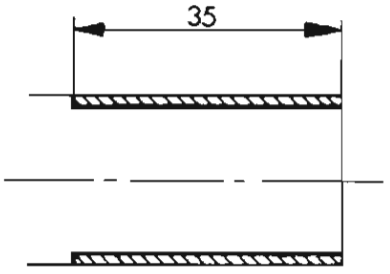
Workpiece drawing



Workpiece raw material



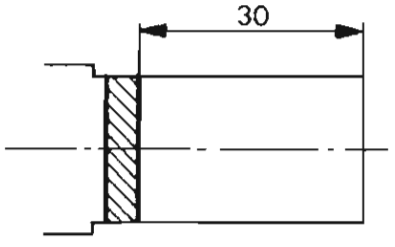
Clamping sketch



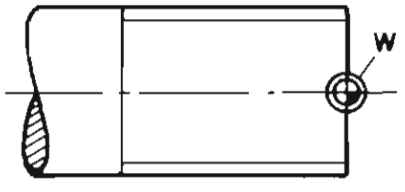
Procedure

1. Determine steps of operation

1.1. Turning off with right hand side tool to dia. 2o mm, length 35 mm.



1.2. Parting-off to a length of 3o mm using parting-off tool.



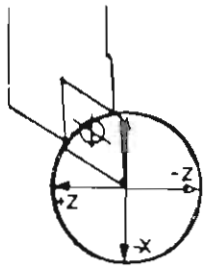
2. Programming: absolute

3. Determination of workpiece zero point

As on drawing.

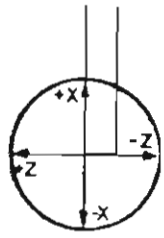
4. Mount right hand side tool and parting-off tool at center height.

5. Adjust optical pre-setting device at center line.



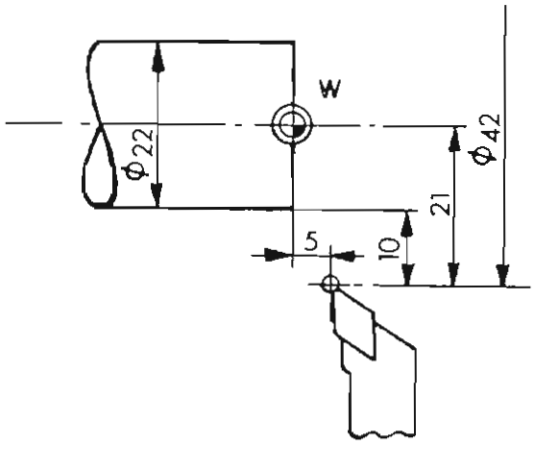
6. Move right hand side tool into cross wire, set digital read-out for X and Z to zero.

7. Swivel turret, move parting-off tool into cross wire; enter X and Z values into tool chart.

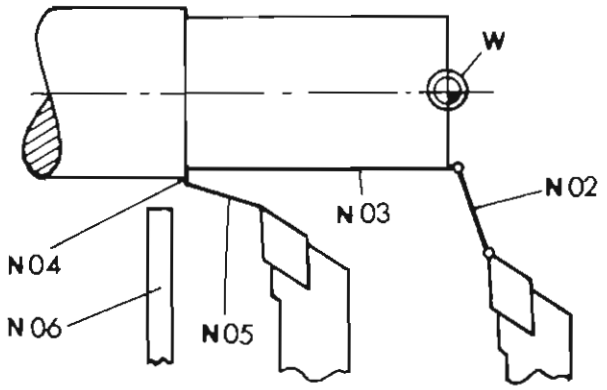


TOOL CHART

Tool	Position in turret	X-value	Z-value
R.H. Tool	1	0	0
Cutting off Tool	3	200	1352

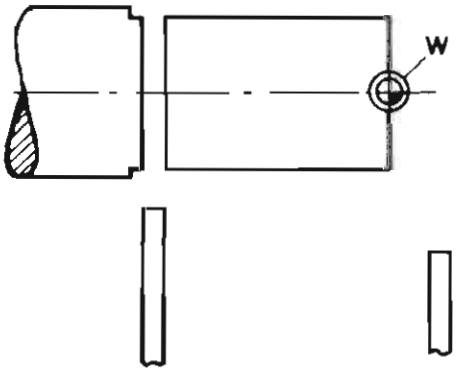


8. Determine point of program start, swivel in right hand side tool.



9. Set up program

N	G	X	Z	T
00	92	4200	500	
01	26	0	0	0
02	00	2000	100	
03	01	2000	-3500	100
04	01	2200	-3500	100
05	00	4200	500	
06	26	+200	1352	2
07	00	2200	-3000	
08	01	0	-3000	30
09	01	2200	-3000	
10	26	0	0	4
11	00	4200	500	
12	22			

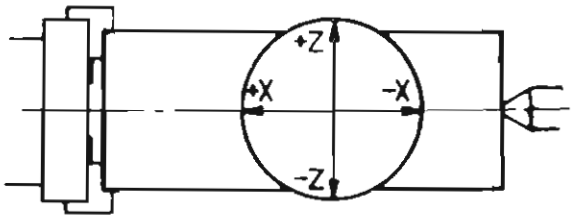


10. Mount workpiece:
Put in program, switch on main spindle drive, start program.

Program starting point

The program starting point should be selected such that the workpiece can easily be mounted and dismounted.

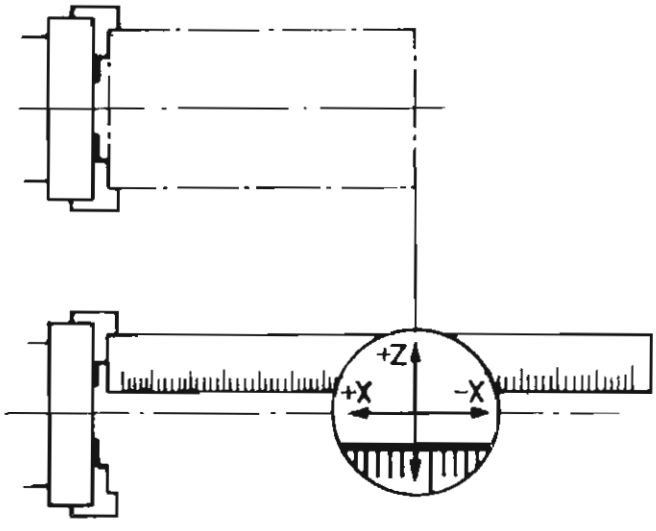
Moving to the Program Starting Point



Possibility 1

Scratching system:

Since the reference tool is already adjusted at center line, a scratching of the workpiece outside is not necessary. Only the face side has to be scratched.



Possibility 2

In many cases the length of the workpiece is known. By means of a ruler the pre-setting device can be positioned such that the cross wire is in line with the workpiece length. In this case no scratching is necessary.

The starting point is approached from the position of the tool in the cross wire.

Possible Collisions using the Automatic Turret Toolholder

To take into account the possibilities of collisions is very important when programming the automatic turret toolholder.

Possibilities of Collisions:

- Collision of tools with workpiece (during machining and swivelling process)
- Collisions with chuck:
  + Collision with jaws extended
  + With headstock (drill 12 mm)

The overhead slides 10.1a - 10.5 serve to take into account the possibilities of collision in connection with the automatic turret toolholder.

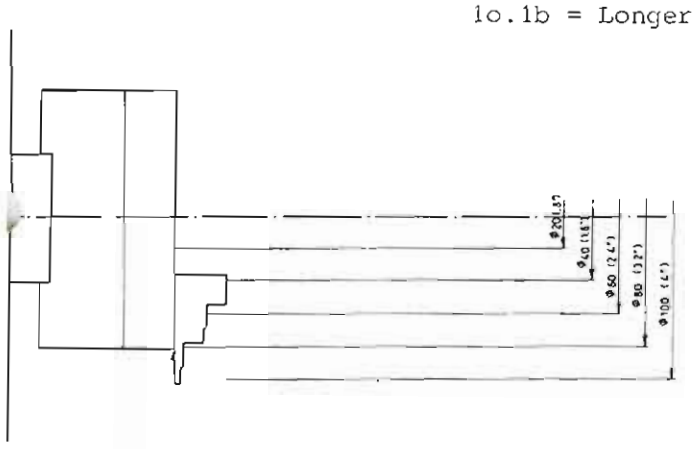
Possibilities to find out about Collisions

1. Dry run

A dry run shows collisions which may occur.
The program has to be stopped, rewritten, tool positions have to be changed.
It is useful to avoid possible collisions already in advance, i.e. when writing the program.

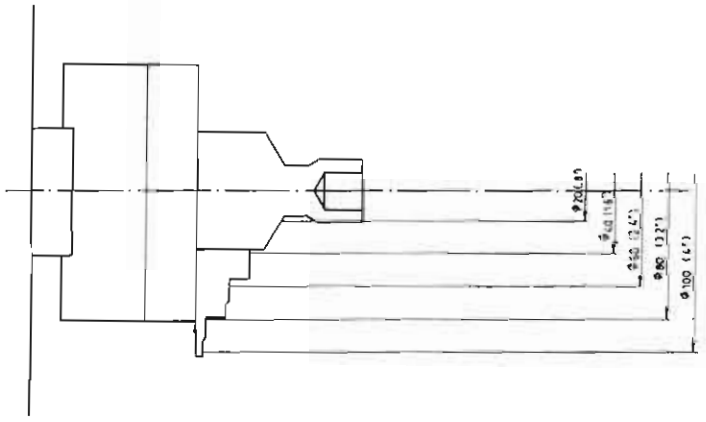
2. 3-D (three-dimensional) view of tool positions using slides 10.1a – 10.5 (projection sheets)

- Cut out the shapes of slide 10.5 and place them on the contours of the resp. slides.
- Use, according to the mounted chuck, slide 10.1a or 10.1b.
  10.1a = Short chuck
  10.1b = Longer chuck

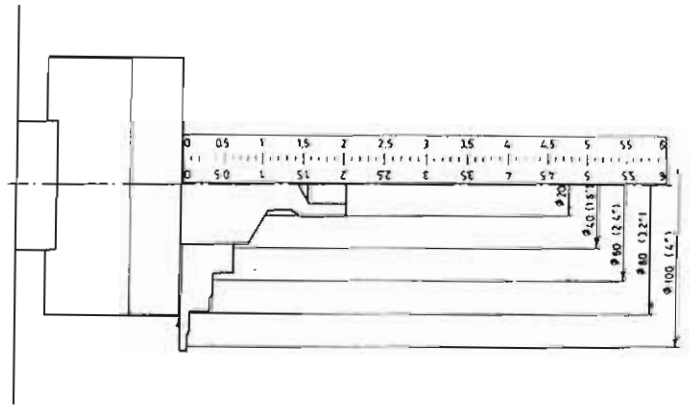


2.1. You know diameter of the raw material:

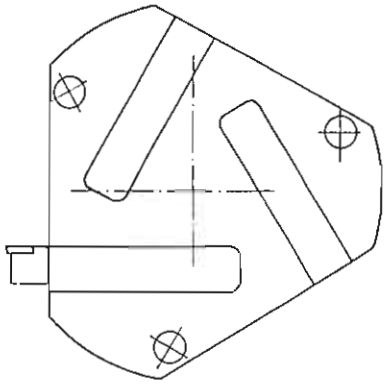
Place (use sticker) the jaws accordingly on slide 10.1a or 10.1b.



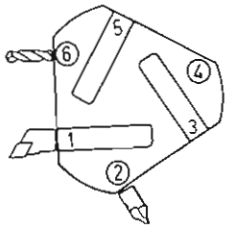
2.2. Place (use sticker) the drawing with the final shape (scale 1:1) on the slide.



2.3. Place the Z-ruler (inch or metric) on the slide, in order to get a better orientation in case you use the turret toolholder.



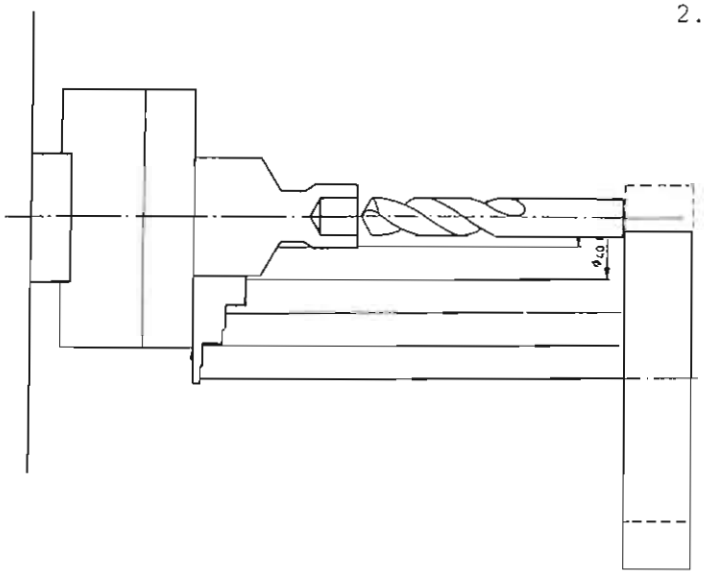
2.4. Place the resp. tools on slide 1o.2 and 1o.4.



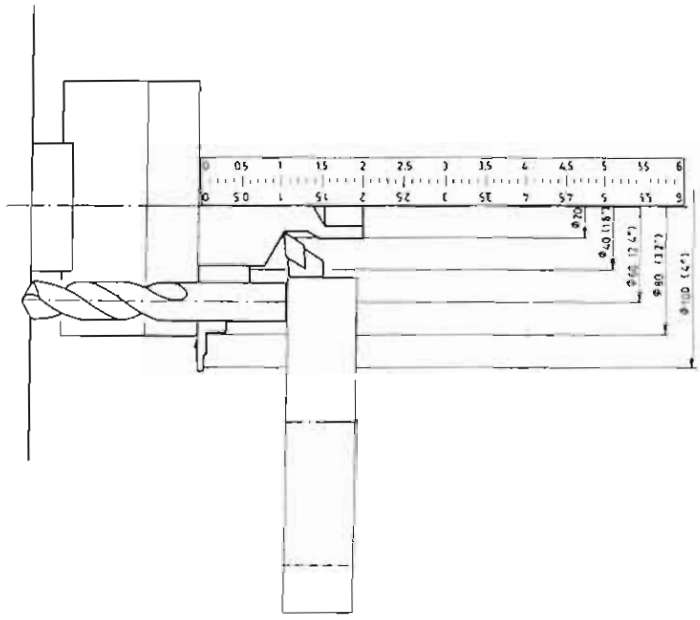
2.5. On slide 1o.4 the tools are to be placed for your understanding and better orientation.

2.6. Simulation of working process:

2.6.1. Centering
2.6.2. Drilling

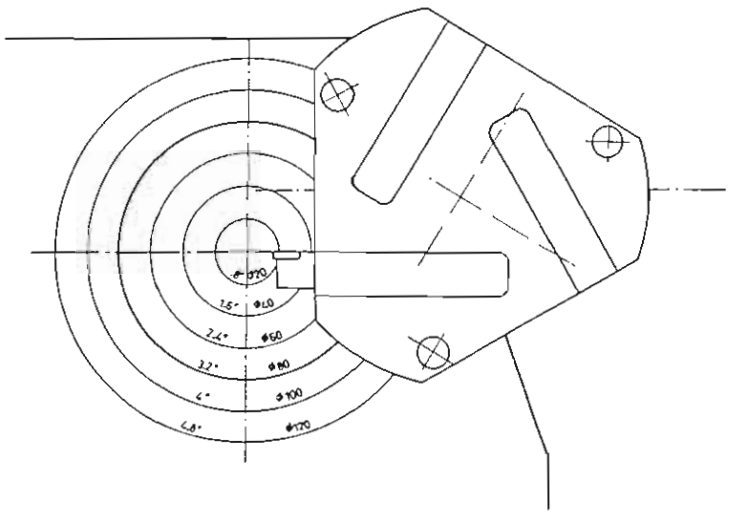


No collision to be expected.

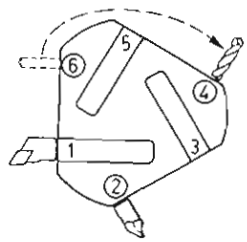


2.6.3. Turning with the right hand side tool:

a) Seen from top view the drill seems to collide with the chuck. As the right hand side tool and the drill are not in the same plane, you have to check the situation applying another view.



b) The front view:
If the right hand side tool is moved to dia. 18 mm, the drill in position 6 collides with the swing of the extended jaws (dia. 1o4 mm)



Therefore mount spiral drill in position 4.

NO COLLISION!

Applying this method all types of collisions can be seen in advance. The swivelling of the turret toolholder can be monitored and possible crashes can be avoided.

Some tips

- The change from absolute to incremental programming often causes a mixing up and confusion. Use the graph paper with the coordinate system.

- It seems to be easier to start with incremental programming. Once the preparatory functions G00/G01/G02/G03/ G33/G78/G84 are known, both systems should be used.
  For absolute programming the same examples can be used as with incremental programming.

Presetting of tools
Collecting the tool data

1. Quick-change toolpost

1.1. Preadjusting tools with mechanical presetting device

The student identifies the different positions of the tools easily.

With tools preadjusted it is easier to control the programs in group instruction. The student can easily calculate the offset data as they are not given in two decimals (three inch).

Tool data calculation with G26

In the beginning the student should calculate the tool offset data because he will get a better understanding, how the computer is calculating in combination with G26.

1.2. Collecting the tool data with the optical presetting device

Tools can be mounted in any position. Calculation without using G26 is difficult because the increments will be in hundredth of mm (thousandth of inch)

2. Automatic turret toolholder

2.1. Preadjusting tools with the mechanical setting gauge

It cannot be practiced because tool bits could brake if they are not positioned exactly in the grooves.

2.2. Collecting the tool data by scratching or measuring on the toolholder

Scratching

Possible, but complicated and is not very exact.

Measuring on the toolholder

Difficult and not very exact.

Corrections of tool data are necessary in both cases.

It is possible that the tools are clamped in any position. By scratching a reference point at the workpiece, the tool data can be collected and programmed (G26). Difficulties arise with the inside turning tools.
If tool data are collected with these methods, the first workpiece has to be measured and the tool data must be corrected according to the measuring results.

2.3. Collecting the tool data with the optical tool presetting device

This method is highly recommended. Tool data can be collected exactly and quickly.