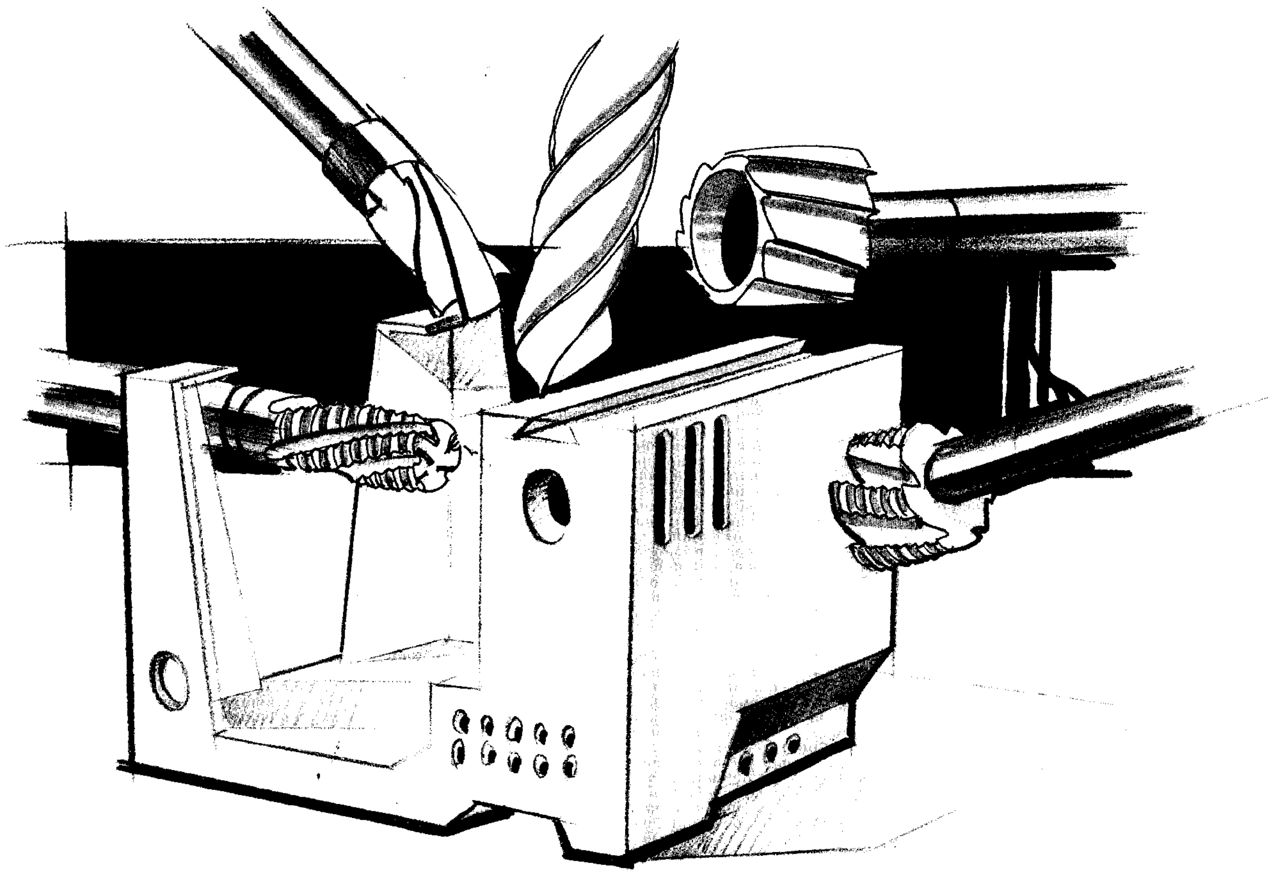
Chapter 6

Tools, tool lengths compensation, radius compensation of milling cutter

The Programming of the Tools



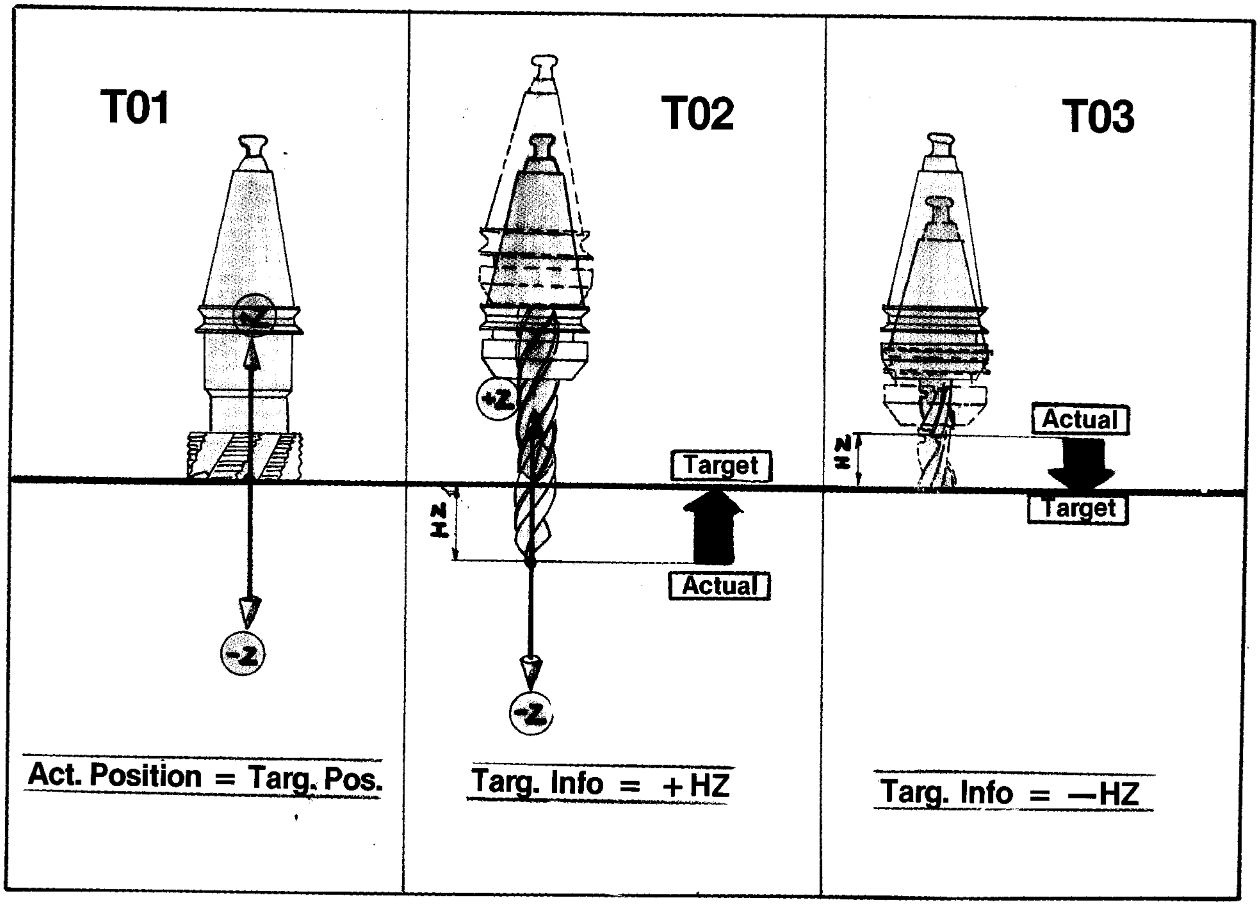
Tool magazines of industrial NC-machines are equipped with up to 50 or more tools.

The sequence is programmed.
Technological data and dimensions have to be programmed for each individual tool bit.

Tools are programmed using the T-address.
T stands for tool.

page 6.2 is blank

Tool Lengths Compensation



T01

| M06/D. . . ./S. . . ./Hz=O/T01 |

T02

| M06/D. . . ./S. . . ./Hz= + .../T02 |

T03

| M06/D. . . ./S. . . ./Hz=— . . . ./T03 |

The computer is given information on the target position or desired position.

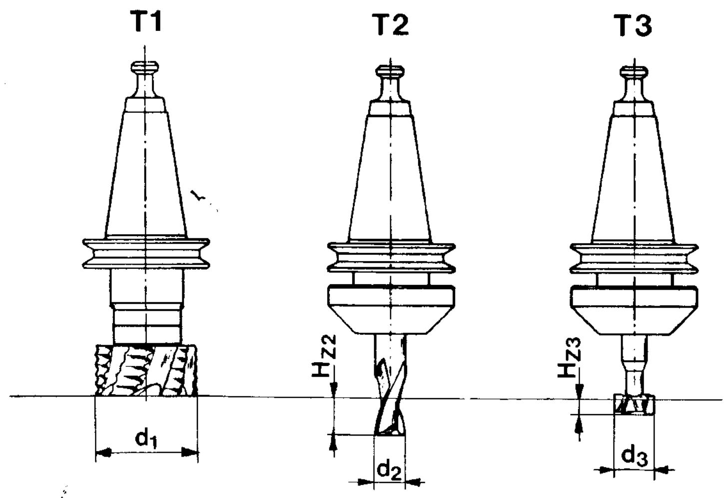
Imagine the coordinate system transferred into the reference plane of the tool.

The target position is described starting from the actual position.

page 6.4 is blank

Working with various Tools

Determining the tool sequence
Detecting the tool data
Compensation of tool lengths



For the manufacture of a workpiece you often need different tools: drills, various milling cutters etc.

The programmer needs to know various data such as

- kinds of tools
- application of different tools,
- position of tools to each other

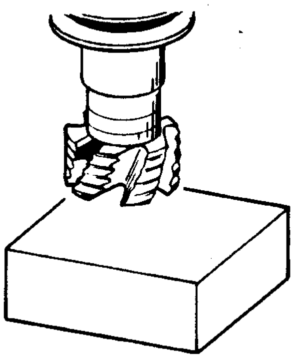
1. The milling cutters are of different diameters. These are known to you.

2. The tools are of different lengths. These are not known to you. You have to measure the lengths and take them into consideration when programming. Otherwise you move the cutter in the air without chip removal or you run it into a workpiece (crash).

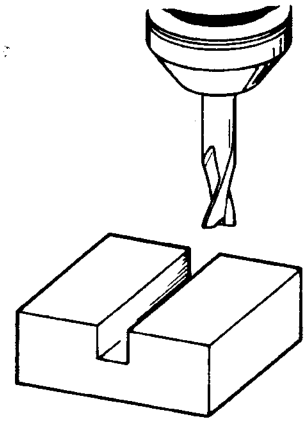
page 6.6 is blank

Procedure

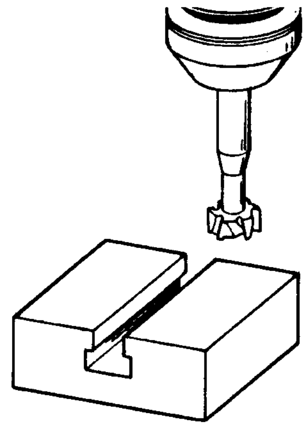
1. Determining the tool sequence



Facing with T1

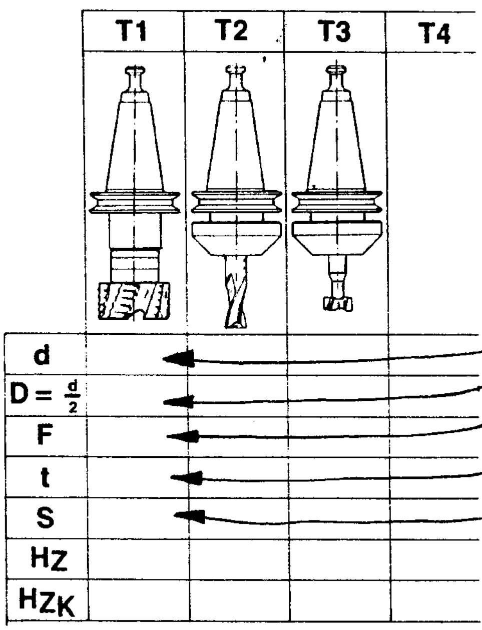


Milling a slot with T2



Milling a T-slot with T3

2. Determination of tool data
2.1. Diameter, technological data



Entering the data

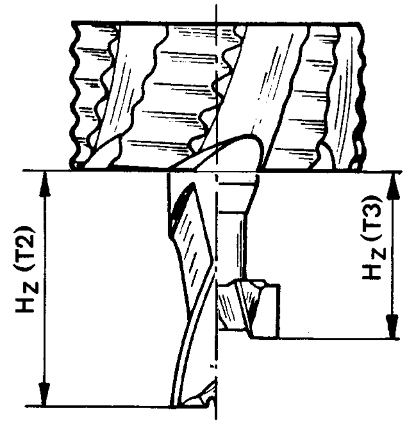
1. Stick the tools into the correspon-
   ding column.

2. Enter the technological data:
   d = Cutter diameter
   D = Cutter radius
   F = Speed of feed
   t = Maximum depth of cut
   S = Speed

These data will make the programming easier.

page 6.8 is blank

2.2. Detecting the Tool Length Differences (H_z)



The differences in tool lengths have to be measured. The measurements can be taken using an external presetting device. In many cases the measuring system within the CNC-machine is taken use of.

You can scratch with all tools a reference surface or measure the data using a dial gauge.

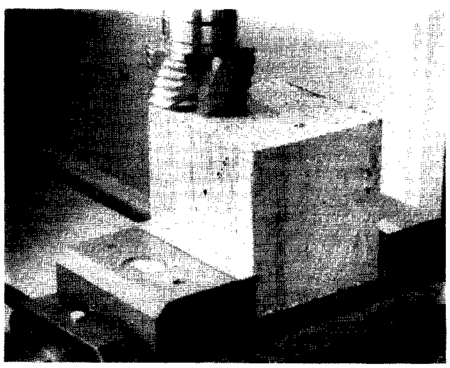
The difference is called H_z.

Procedure

Mount T1 (reference tool) and scratch reference surface, set dial gauge respectively.

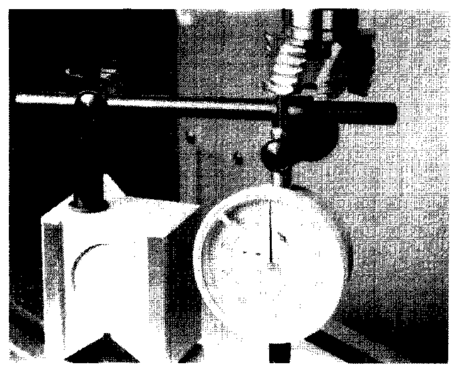
Detection of data by scratching

Scratching only when cutter is turning

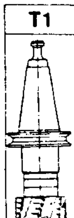


Detection of data with dial gauge.

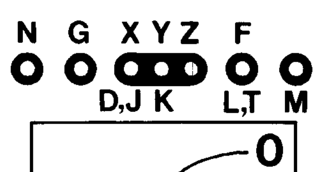
Set dial gauge when machine is at stand-still.



Set dial gauge to 0.

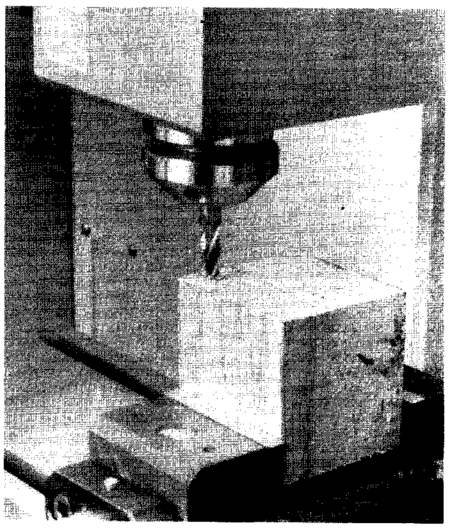
	T1	T2	T3	T4
				
d				
$D = \frac{d}{2}$				
F				
t				
S				
Hz	0			
HzK				

Press key $\boxed{\text{DEL}}$, the Z-value display is set to 0.

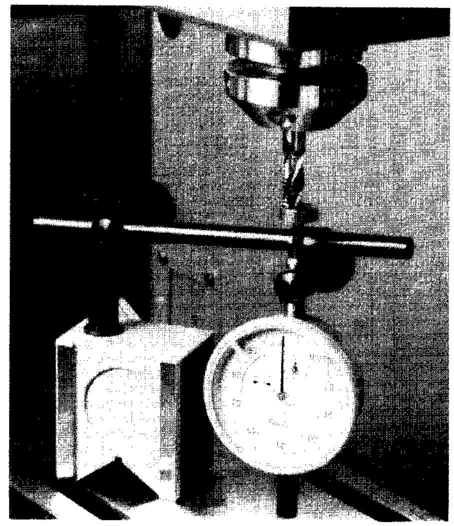


page 6.10 is blank

Mount T2



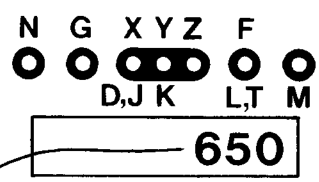
Scratch surface



Touch dial gauge with cutter until it shows 0.

	T1	T2	T3	T4
				
d				
$D = \frac{d}{2}$				
F				
t				
s				
Hz	0	650		
Hz_K				

Read value from display.



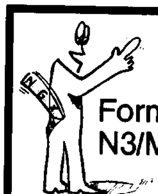
Enter value into tool data sheet. In this way you determine all tool lengths.

Pay attention to the signs!

3. Calculation of Tool Lengths (Tool lengths compensation)

Since these data are known you could take the various lengths into consideration. This would, however, be quite confusing calculation work and will often lead to mistakes.

Calculation of tool length M06
(Tool lengths compensation)
(Programming)

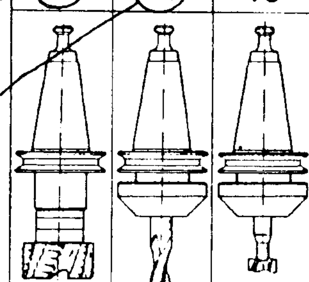


Format M6
N3/M06/D(X)4/S4/Z(Hz) ± 4/T(F)3

The data are entered into the programming sheet.

T = tool number
D = milling cutter radius
S = spindle speed (only for your information)
H_Z = difference in tool length

N	G (M)	X (J) (D)	Y (K) (S)	Z	F (L) (T) (H)
N...	M06	1000	1100	0	1
N...	M06	125	2000	650	2



	T1	T2	T3	T4
d	2"	1/4"	3/8"	
$D = \frac{d}{2}$	1"	.125"	.188"	
F	40	80	20	
t	.1"	.2"	.3"	
S	1100	2000	2000	
HZ	0	.65	-.32	
HZ_K				

Note:

If you write a number 1,2,3,4 under the F(T) address when programming M06, this automatically means program hold.
If there is a 0 under the F(T) address, there will be no hold.

3. Calculation of Tool Lengths (Tool lengths compensation)

Since these data are known you could take the various lengths into consideration. This would, however, be quite confusing calculation work and will often lead to mistakes.

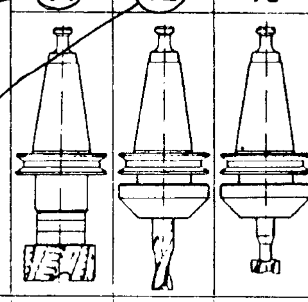
Calculation of tool length M06 (Tool lengths compensation) (Programming)

Format M6
N3/M06/D(X)5/S4/Z(Hz) ± 5/T(F)3

The data are entered into the programming sheet.

N	G (M)	X (J) (D)	Y (K) (S)	Z	F (L) (T) (H)
N...	M06	2000	1100	0	1
N...	M06	500	2000	650	2

T = tool number
D = milling cutter radius
S = spindle speed (only for your information)
H_Z = difference in tool length



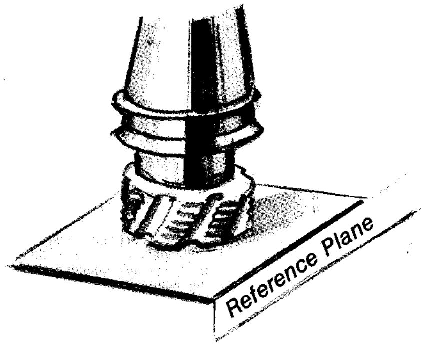
	T1	T2	T3	T4
d	40	10	16	
$D = \frac{d}{2}$	20	5	8	
F	80	160	40	
t	0,7	5	8	
S	1100	2000	2000	
HZ	0	650	-320	
HZ_K				

Note:

If you write a number 1,2,3,4 under the F(T) address when programming M06, this automatically means program hold.
If there is a 0 under the F(T) address, there will be no hold.

page 6.14 is blank

Tool Lengths Compensation in the Program Sequence



The first tool (T01) has a H_Z value = 0.

N.../M06/D2000/S1300/Z(H_Z) = 0/T01

⇩

Manufacture

⇩

Tool change T02

⇩

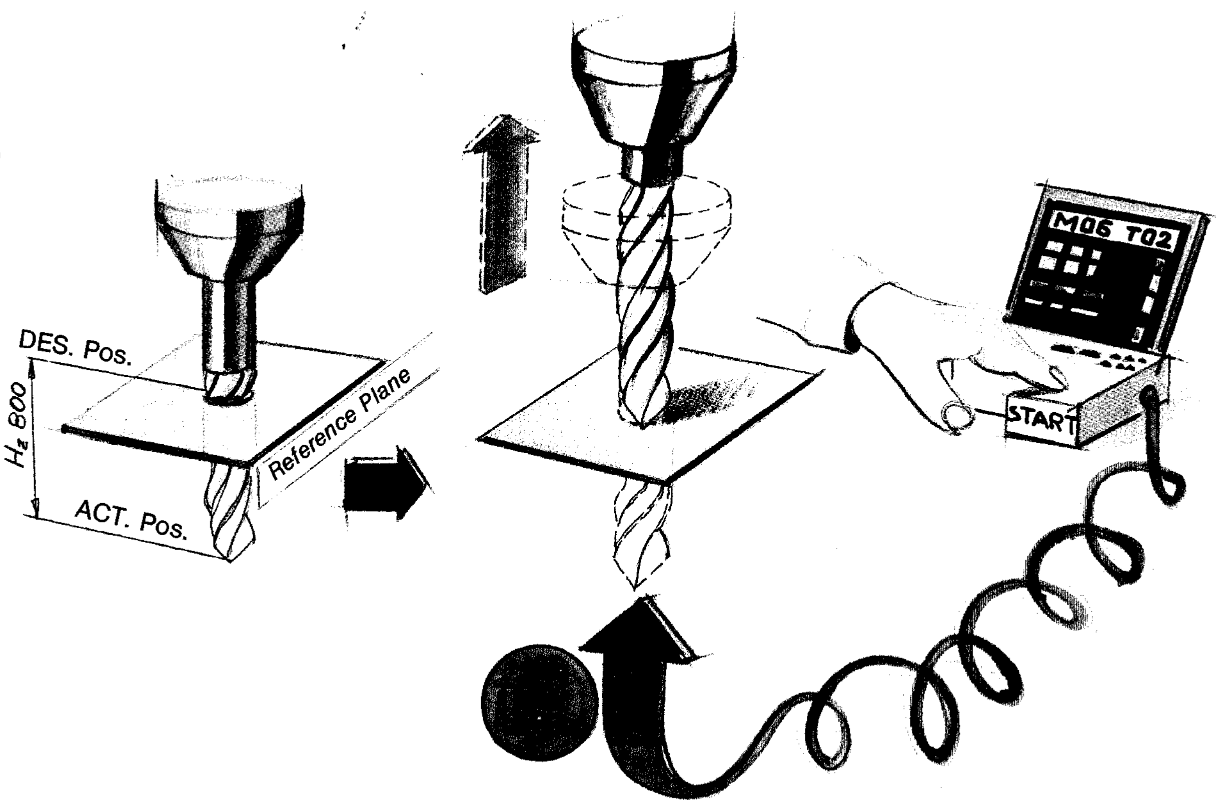
N.../M06/D500/S2000/Z(H_Z) = 800/T02

⇩

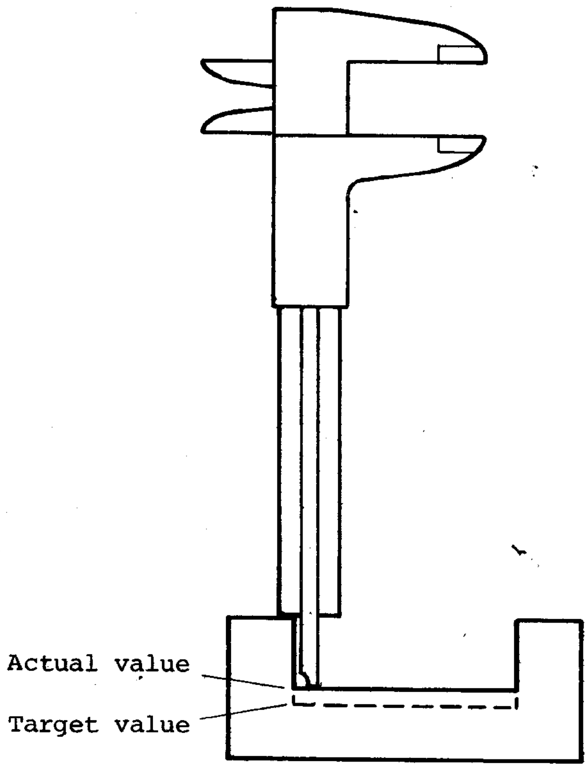
Start

First the tool T02 moves from the actual position to the target position.

Then the manufacture itself starts.



Tool Lengths Corrections



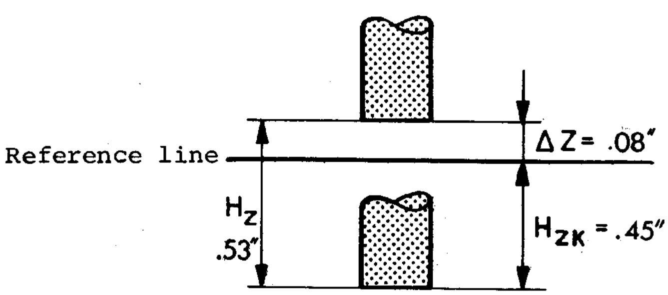
You have finished the manufacture of a workpiece and find out that the Z-measurement is not correct.

- The program is correct

- The starting position of the cutter is correct.

What is the reason?

The target value information (H_Z value) was not correct (wrong, inaccurate measurements, cutter not resharpened).



TARGET INFORMATION Hz wrong

M06/D.../S.../Z+ **530** /T02

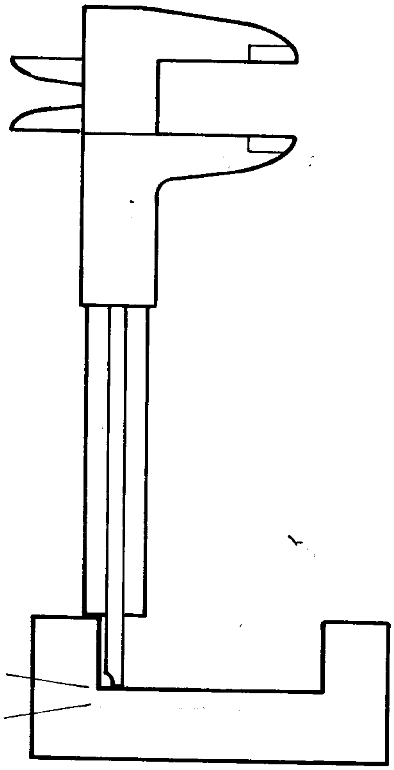
The target information Hz has to be corrected.

<u>Hzk = Corrected target information</u>

Hzk = Hz + (± correction value △ Z)

M06/D.../S.../Z+ **450** /T02

Tool Lengths Corrections



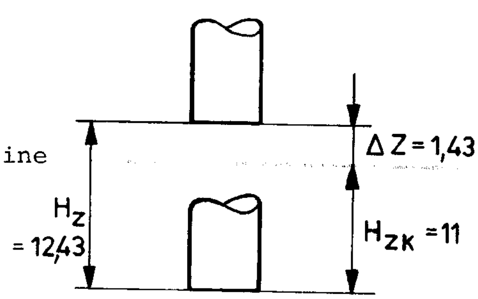
You have finished the manufacture of a workpiece and find out that the Z-measurement is not correct.

- The program is correct

- The starting position of the cutter is correct.

What is the reason?

The target value information (H_Z value) was not correct (wrong, inaccurate measurements, cutter not resharpened).



TARGET INFORMATION Hz wrong

M06/D.../S.../Z+ 12.43/T02

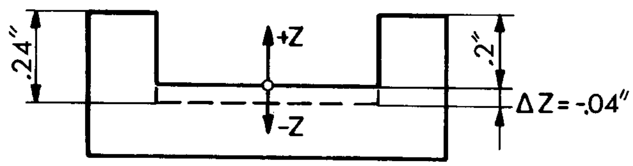
The target information Hz has to be corrected.

Hzk = Corrected target information

Hzk = Hz + (± correction value △ Z)

M06/D.../S.../Z+ 1100/T02

Example of a Correction of the Hz-value

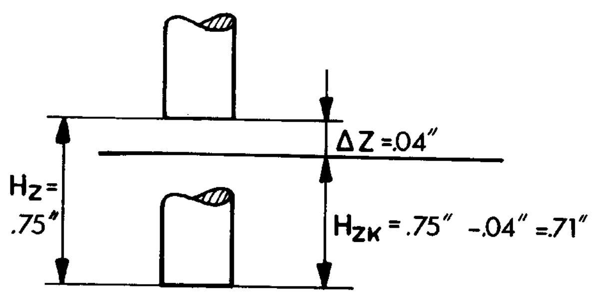


You may

1. Measure tool once again

2. Detect the correction value by measuring the workpiece.

The Hz information has to be corrected by the Δ Z value.



- Imagine the coordinate system transferred to the Z-actual position of the workpiece.

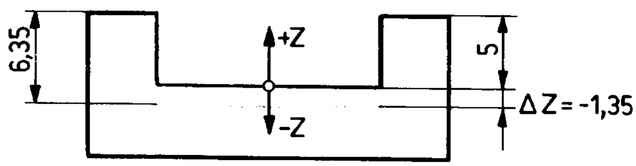
- Add the correction value Δ Z to the target information Hz of the tool bit.

<u>Pay attention:</u> Δ Z may have $\pm$ sign.

$$
\begin{aligned}
Hzk &= Hz + (\pm \Delta Z) \\
&= .75'' + (- \Delta Z) \\
&= .75'' - .04'' \\
&= .71''
\end{aligned}
$$

The value Hzk = .71" is corrected in the programming sheet, tool data sheet and in the memory.

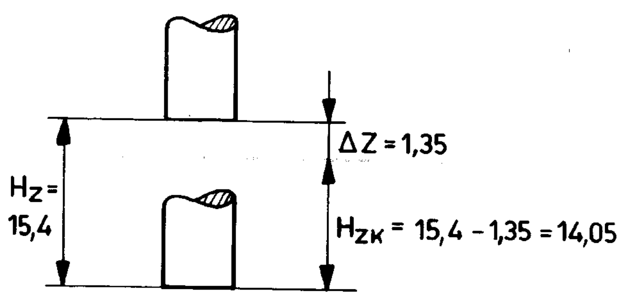
Example of a Correction of the Hz-value



You may

1. Measure tool once again
2. Detect the correction value by measuring the workpiece.

The Hz information has to be corrected by the $\triangle$ Z value.



- Imagine the coordinate system transferred to the Z-actual position of the workpiece.
- Add the correction value $\triangle$ Z to the target information Hz of the tool bit.

Pay attention: $\triangle$ Z may have $\pm$ sign.

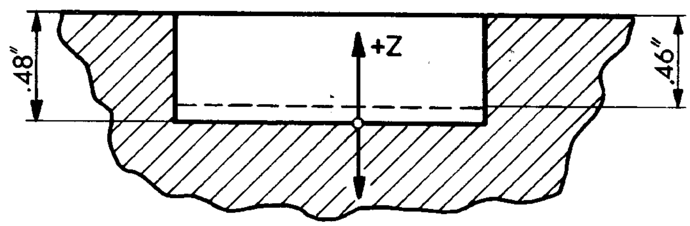
$$
\begin{aligned}
Hzk &= Hz + (\pm \; \Delta Z) \\
&= 15.4 + (- \; \Delta Z) \\
&= 15.4 - 1.35 \\
&= 14.05
\end{aligned}
$$

The value Hzk = 14,05 is corrected in the programming sheet, tool data sheet and in the memory.

Example

Programmed Hz-value (actual information): -.23″

Workpiece measurements: Actual and target, compare drawing.



Correct the Hz-value

Hzk = Hz + (± Δ Z)

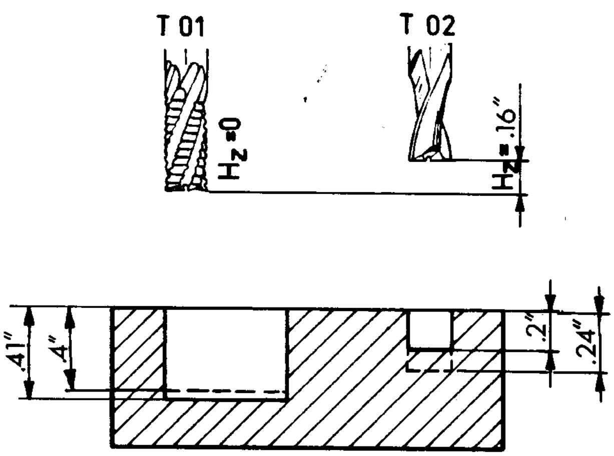
Pay attention to the sign of Δ Z.

Hzk = ...

Example

Hz of TO1 = 0

Hz of TO2 = .16″



Workpiece:

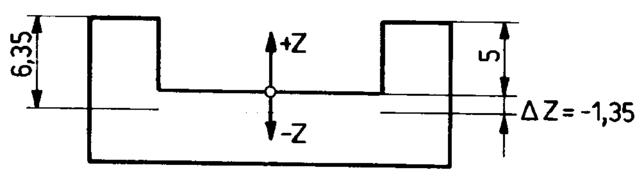
Actual value TO1 = .41″
Actual value TO2 = .2″

Target value TO1 = .4″
Target value TO2 = .24″

Correct the Hz-values of TO1 and TO2.

	TO1	TO2
Hzk		

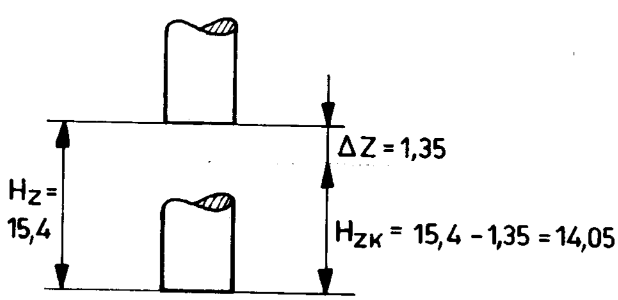
Example of a Correction of the Hz-value



You may

1. Measure tool once again
2. Detect the correction value by measuring the workpiece.

The Hz information has to be corrected by the $\triangle$ Z value.

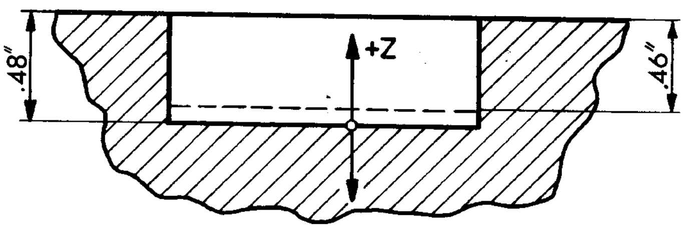


- Imagine the coordinate system transferred to the Z-actual position of the workpiece.
- Add the correction value $\triangle$ Z to the target information Hz of the tool bit.

<u>Pay attention:</u> $\triangle$ Z may have $\pm$ sign.

$$
\begin{aligned}
Hzk &= Hz + (\pm \Delta Z) \\
&= 15.4 + (- \Delta Z) \\
&= 15.4 - 1.35 \\
&= 14.05
\end{aligned}
$$

The value Hzk = 14,05 is corrected in the programming sheet, tool data sheet and in the memory.



Example

Programmed Hz-value (actual information): -.23″

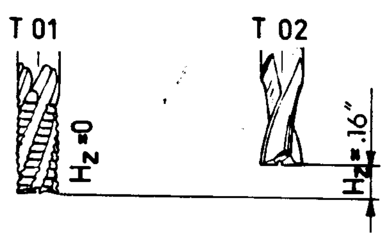
Workpiece measurements: Actual and target, compare drawing.

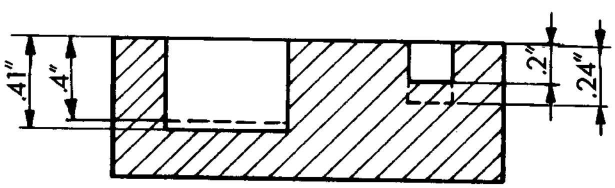
Correct the Hz-value

$Hzk = Hz + (\pm \Delta Z)$

Pay attention to the sign of ΔZ.

Hzk = ...





Example

Hz of TO1 = 0

Hz of TO2 = .16″

Workpiece:

Actual value TO1 = .41″
Actual value TO2 = .2″

Target value TO1 = .4″
Target value TO2 = .24″

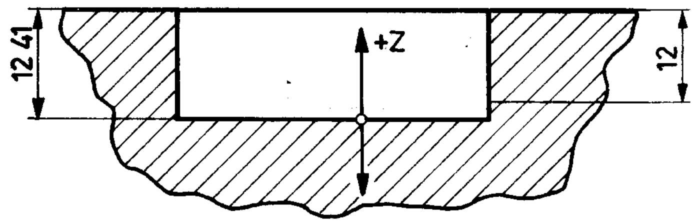
Correct the Hz-values of TO1 and TO2.

	TO1	TO2
Hzk		

Example

Programmed Hz-value (actual information): - 6,25 mm

Workpiece measurements: Actual and target, compare drawing.



Correct the Hz-value

$Hzk = Hz + (\pm \Delta Z)$

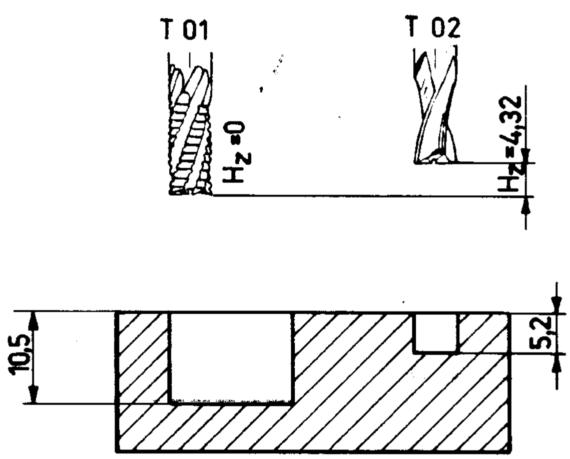
Pay attention to the sign of Δ Z.

Hzk = ...

Example

Hz of TO1 = 0

Hz of TO2 = -4,32



Workpiece:

Actual value TO1 = 10,5 mm
Actual value TO2 = 5,2 mm

Target value TO1 = 10 mm
Target value TO2 = 6 mm

Correct the Hz-values of TO1 and TO2.

	TO1	TO2
Hzk		

page 6.22 is blank

Other Cases for Programming M06

N	G (M)	X (J) (D)	Y (K) (S)	Z	F (L) (T) (H)
	M06	D 500			
	G46				

If a G45, G46, G47, G48 or a G72 command (cutter radius compensation) is programmed, in one of the previous blocks a M06 has to be put in, otherwise the alarm sign will appear.
A16: Cutter radius information missing

The computer needs the cutter radius information D in order to calculate the compensated paths (G45,G46,G47,G48).

The same applies with the pocket milling cycle G72.

Alarm A16

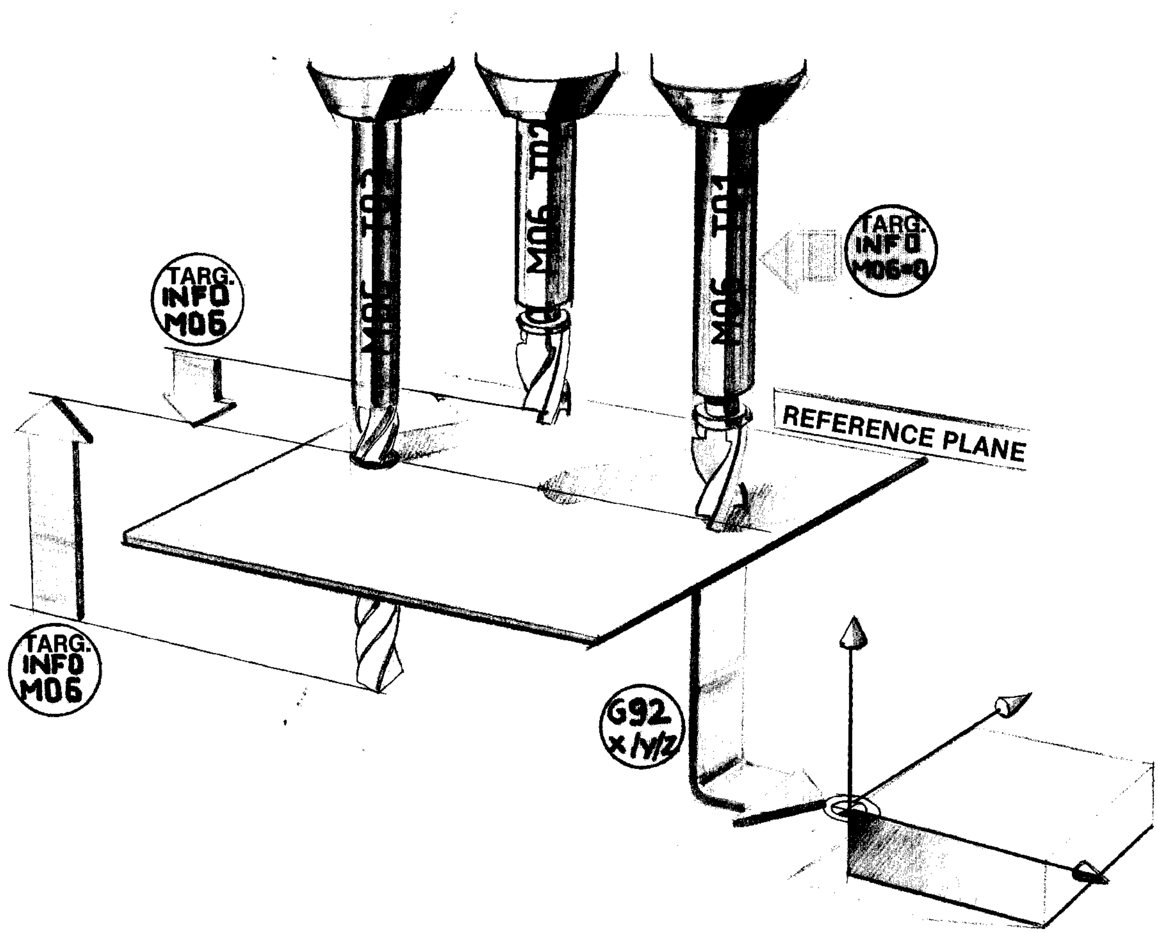
Cutter radius
information missing.

<u>Clearing offsets at end of program</u>

Since tool length offsets remain in effect until they are replaced, they must be cancelled out before ending the program. This will return the Z axis to the proper relative position for the next running of the program.

Connection:

G92 Zero-point offset
M06 Tool lengths compensation



M06

The Hz-information is an incremental target information within an independent coordinate system.

G92

The origin of the coordinate system is determined with G92.

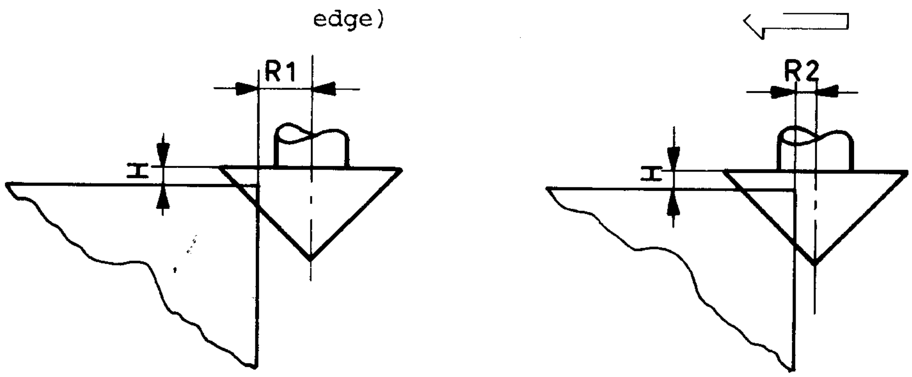
page 6.26 is blank

Milling of Chamfers

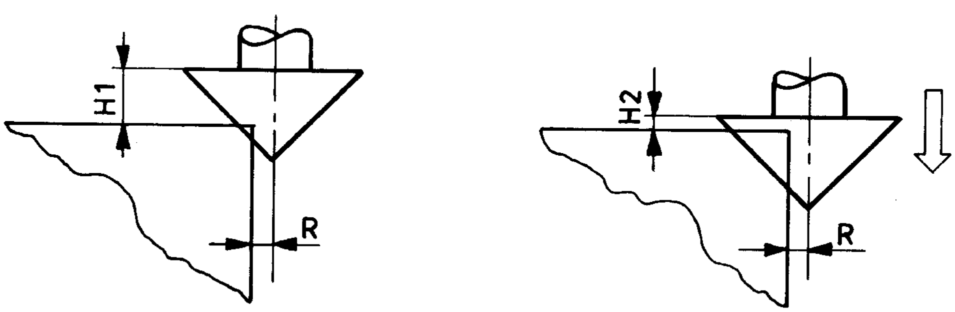
Chamfers are usually milled at an angle of 45°.

The size of the chamfer is determined by the programmed path and/or by the cutting contour.

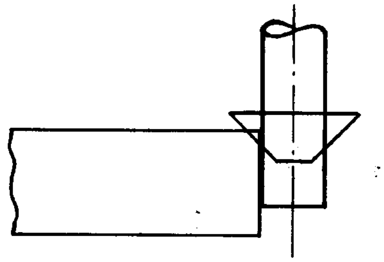
1. Chamfer size determined by different cutter paths (different distances between cutter axis and workpiece edge)



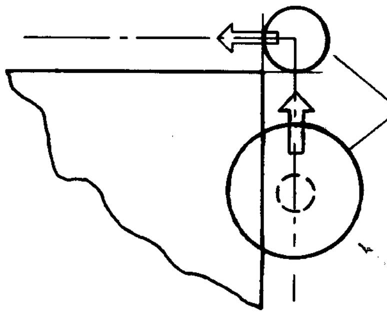
2. Chamfer size determined by different infeed and Z-direction. The cutter path remains unchanged.



Programming a Chamfer with Cutter Path unchanged

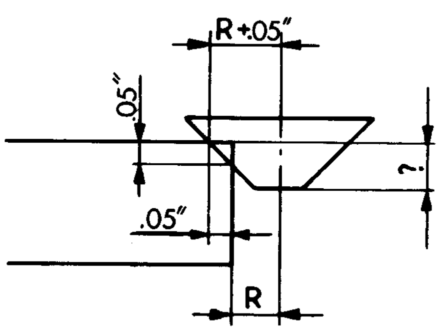


The contour is milled with a cutter of .500″ dia.
To avoid the necessity to program a new cutter path for chamferring, the angle cutter shall be programmed in Z-direction such that a chamfer .05″x.05″ is reached.



Cutter path - end mill
Cutter path - angle cutter

How deep has the Angle Cutter to be fed in?



The radius of the angle cutter which mills the inside contour of the chamfer:

| Radius end mill |

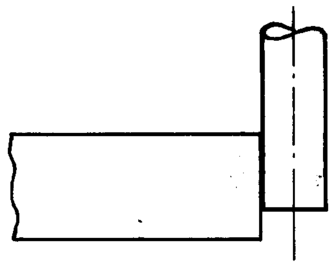
+

| Width of chamfer |

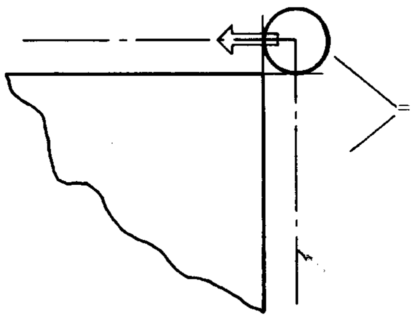
With a mill path using a .5″ shank, dia. .6″ , the radius of the angle cutter produces the chamfer .05″x45°.

R	=	.25″	+	.05″
Angle cutter		Mill path		Width of chamfer

Programming a Chamfer with Cutter Path unchanged



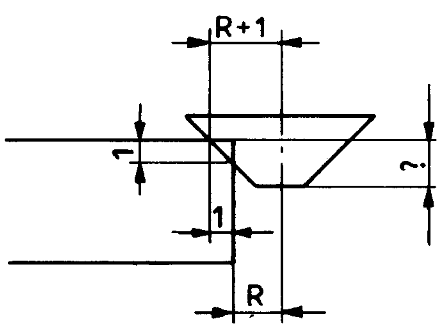
The contour is milled with a cutter of 10 mm dia.
To avoid the necessity to program a new cutter path for chamferring, the angle cutter shall be programmed in Z-direction such that a chamfer 1x1 mm is reached.



Cutter path - end mill
= Cutter path - angle cutter

How deep has the Angle Cutter to be fed in?

The radius of the angle cutter which mills the inside contour of the chamfer:



Radius end mill

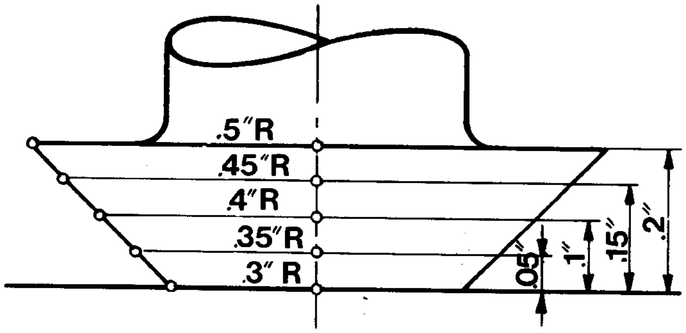
+

Width of chamfer

With a mill path using a 10 mm shank, dia. 12 mm, the radius of the angle cutter produces the chamfer 1x45°.

R	=	5	+	1
Angle cutter		Mill path		Width of chamfer

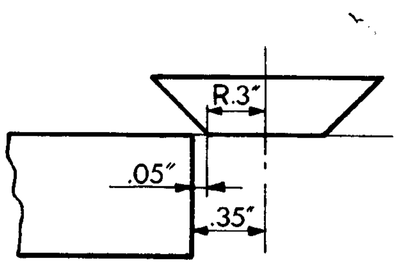
Angle cutter, dia. 1″ × .2″



With a 45° angle cutter, the cutting radius changes by .1″ if the cutter is fed in by .1″.

Example

Radius of mill path .35″

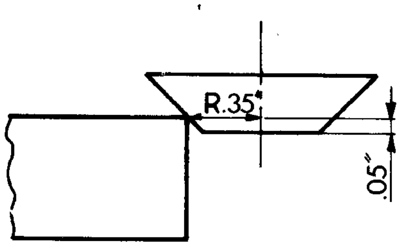


1. Cutter at height 0

Distance to workpiece =.05″

2. Cutter fed in by .05″

Radius .35″ touches edge.



3. Cutter fed in by .05″

Chamfer .05″ × 45° is produced.

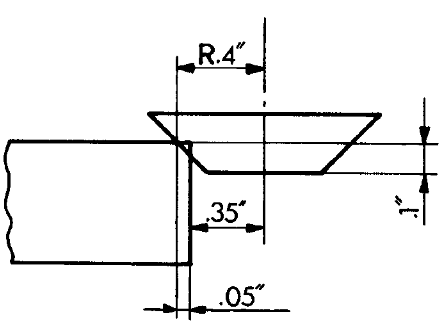
Measure of total depth:

Measure until radius mill path (.05″)

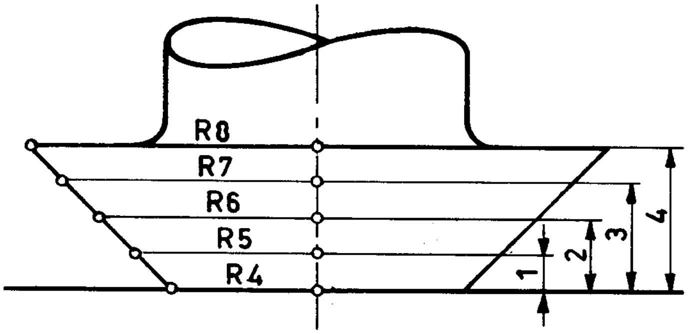
\+

Width of chamfer (.05″)

= .1″



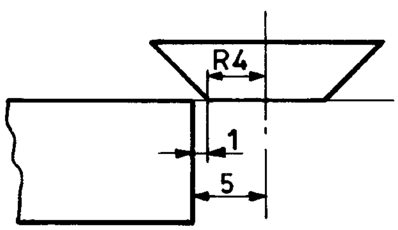
Angle cutter, dia. 16 × 4 mm



With a 45° angle cutter, the cutting radius changes by one mm if the cutter is fed in by 1 mm.

Example

Radius of mill path 5 mm

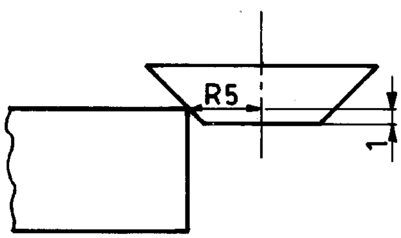


<u>1. Cutter at height 0</u>

Distance to workpiece = 1 mm

<u>2. Cutter fed in by 1 mm</u>

Radius 5 mm touches edge.



<u>3. Cutter fed in by 2 mm</u>

Chamfer 1x45° is produced.

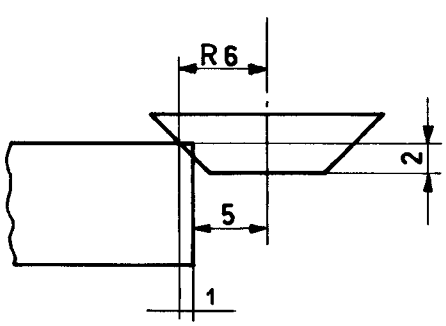
<u>Measure of total depth:</u>

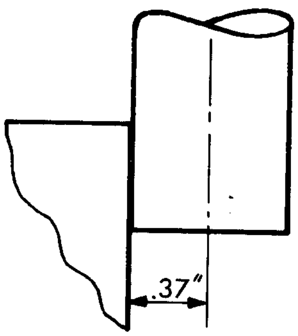
Measure until radius mill path (1 mm)

\+

Width of chamfer (1 mm)

= 2 mm

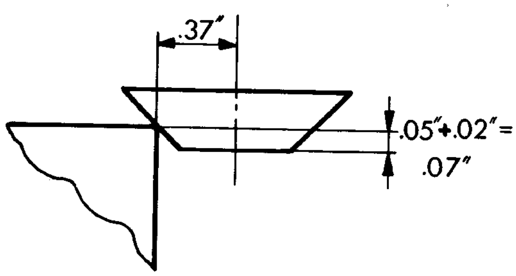




Example

Unchanged mill path

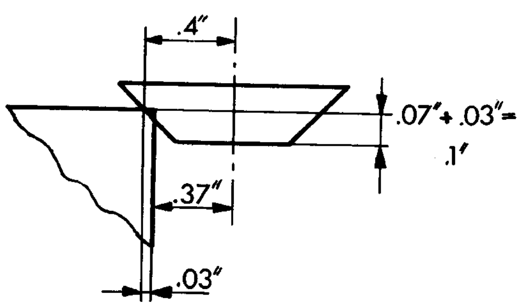
- Radius end mill: .37″
- Chamfer .03″×.03″



With an infeed of .07″ the angle cutter touches the contour.

Infeed .05″ R.35″
Infeed .07″ R.37″

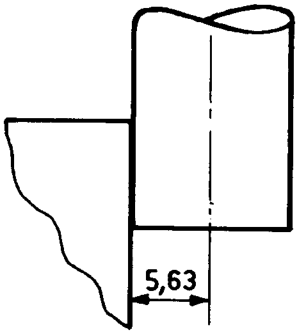
Radius .4″ produces the chamfer contour.



	.37″	radius cutter path
+	.03″	width of chamfer
	.4″	

Cutter infeed

.07″	(radius touches contour)
.03″	(width of chamfer)
.1″	total infeed

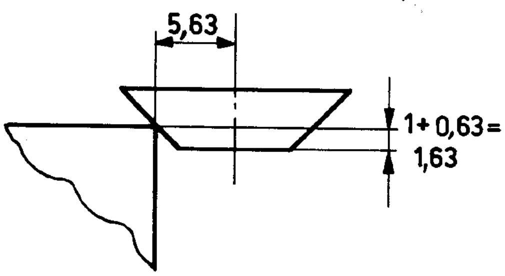


Example

Unchanged mill path

- Radius end mill: 5,63 mm

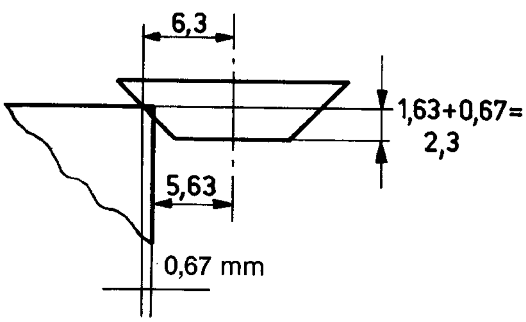
- Chamfer 0,67 x 0,67 mm



With an infeed of 1,63 mm the angle cutter touches the contour.

Infeed 1 mm	R5
Infeed 1,63 mm	R5,63

Radius 6,3 mm produces the chamfer contour.



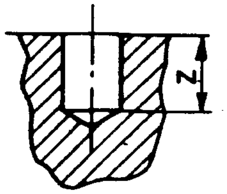
	5,63 mm	radius cutter path
+	0,67 mm	width of chamfer
	6,30 mm	

Cutter infeed

1,63 mm	(radius touches contour)
0,67 mm	(width of chamfer)
2,30 mm	total infeed

====================

The Depth of Bore with Spiral Drill



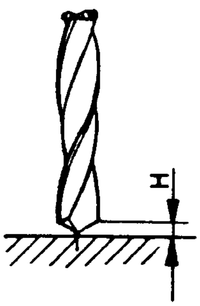
Blind holes are dimensioned down to the flat ground of the bore.

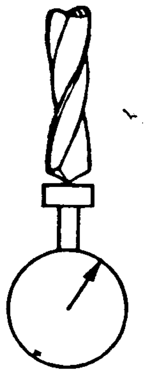
If you want to calculate the tool length you either scratch the surface with the point of the drill bit or you take measurement of the length of the tool.

In order to program the indicated depth of bore you have to add the length of the tool point.

$$tg30^{o} = \frac{H}{\frac{d}{2}}$$

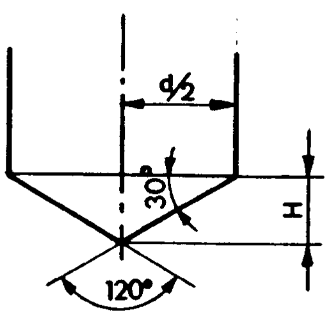
$$H = (tg30^{o}) \times (\frac{d}{2})$$

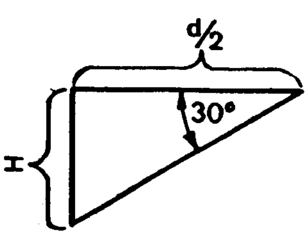




Chart

Drill dia. in		H (in)
1/8	.125	.036
3/16	.188	.054
1/4	.250	.072
5/16	.313	.090
3/8	.375	.108
7/16	.438	.126
1/2	.500	.144

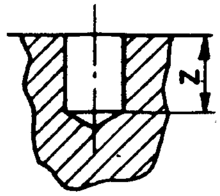




Drill Data for the Tool Sheet

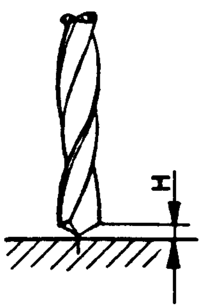
Always deduct value H from the measured data when you enter it.
You need not to calculate anymore and can program the dimensions of the drawing directly.

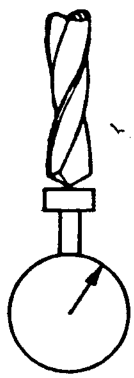
The Depth of Bore with Spiral Drill



Blind holes are dimensioned down to the flat ground of the bore.

If you want to calculate the tool length you either scratch the surface with the point of the drill bit or you take measurement of the length of the tool.





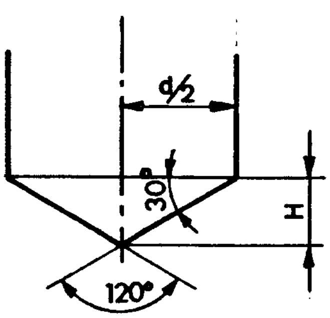
In order to program the indicated depth of bore you have to add the length of the tool point.

$$tg30^{o} = \frac{H}{\frac{d}{2}}$$

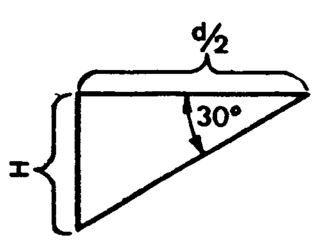
$$H = tg30^{o} \times \frac{d}{2}$$

Chart

Drill dia. in mm	H (mm)
2	0.57
4	1.15
6	1.73
8	2.30
10	2.89
12	3.46
14	4.04
16	4.61



Drill Data for the Tool Sheet



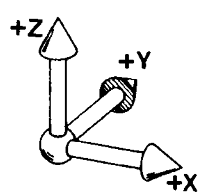
Always deduct value H from the measured data when you enter it.
You need not to calculate anymore and can program the dimensions of the drawing directly.

Tool Data Sheet (inch)

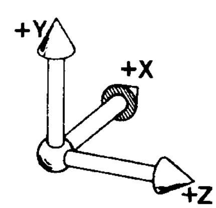
	T1	T2	T3	T4	T5	T6	T7	T8
d								
$D = \frac{d}{2}$								
F								
t								
S								
H_Z								
H_{Z_K}								

d............................ Cutter dia.
D............................ Cutter radius
F............................ Feed speed
t .... Max. milling depth
S .... Spindle speed
H_Z .... Difference measure
H_{Z_K} .... Corrected difference measure

Vertical axis system



Horizontal axis system



Zero-point of workpiece
Start position
Tool change position

Zero-point offset (G92)

X ____________

Y ____________

Z ____________

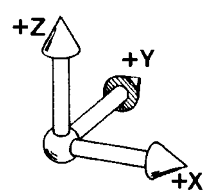
Drawing no.:
Denomination:
Workpiece material:
Program no.:
Name:
Date:

Tool Data Sheet

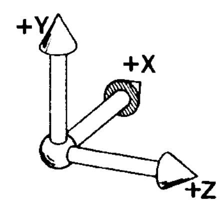
	T1	T2	T3	T4	T5	T6	T7	T8
d								
$D = \frac{d}{2}$								
F								
t								
S								
H_Z								
H_{Z_K}								

d .... (mm) .... Cutter dia.
D .... (mm) .... Cutter radius
F .... (mm/min) .... Feed speed
t .... (mm) .... Max. milling depth
S .... (U/min) .... Spindle speed
H_Z .... (mm) .... Difference measure
H_{Z_K} .... (mm) .... Corrected difference measure

Vertical axis system



Horizontal axis system



Zero-point of workpiece
Start position
Tool change position

Zero-point offset (G92)

X ____________ mm

Y ____________ mm

Z ____________ mm

Drawing no.:
Denomination:
Workpiece material:
Program no.
Name:
Date: