1. General

— Technological data
— Finding the Chip Removal Values, Speeds
— Mounting the Tools
— Clamping the Workpieces

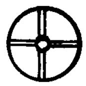
back of tab 1 is blank

Technical Data

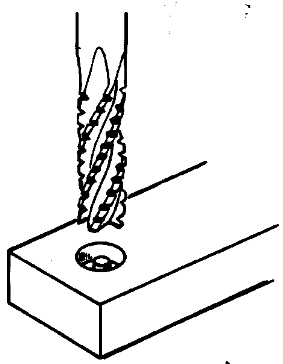
Working area:	
X	7.87" (200 mm)
Y	3.94" (100 mm)
Z	7.87" (200 mm)
Milling head swivel	90°
Milling table:	
Size	16.54" x 4.92" (420 x 125 mm)
Number of T-slots	2
Width/distance of T-slots	.43"/3.54" (11/90 mm)
Distance table - milling head:	
Vertically	.98"-8.86" (25-225 mm)
Horizontally	3.15"-11.02" (80-280 mm)
Throat	5.51" (140 mm)
Milling spindle:	
Tool mounting ANSI B5-50-1978	# 30 Taper
Tool clamping	Quick action system
Drive of milling spindle:	
DC motor	.6 hp 440 W (S1 - 100% ID)
Speed range infinitely variable	300-2000 rpm
Drive of feed motors:	
Step motor - single drive + ball bearing screw	
Feed - manual operation	.20" - 15.75"/min (5-400 mm/min)
Feed - CNC-operation	.08 - 19.65"/min (2-499 mm/min)
Rapid feed	23.62"/min (600 mm/min)
Single step (tip operation)	0004" (0.01 mm)
Feed power	225 lbs
Weight of milling machine	264 lbs (120 kg)
Weight of control unit	88 lbs (40 kg)
Size of machine *(width x depth x height)*	33"/30"/28" (840/750/720 mm)
Size of control unit	28"/18"/21" (710/450/540 mm)
Electrical connection	115 Vac

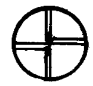
End mills shown are calle "roughing" end mills. Most end mills will have smooth sides. The rough sides act as chip breakers. This is only important for high speed/ high volume milling.

General tips on milling

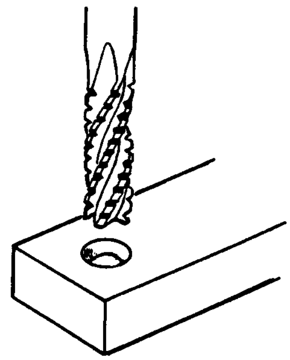


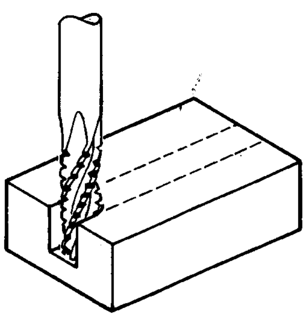
Regular end mill

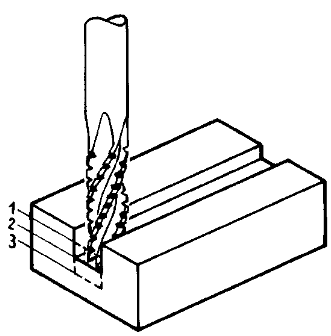




Center cut end mill







Most milling cutters are designed for peripheral work or slot cutting where the cutter is fed in from outside the workpiece edge. They are not designed for feeding into the workpiece from above to make a starting hole. When feeding into a workpiece from above you should always use a center cutting end mill. **Feedrate for this operation should be no more than half that used for normal milling.**

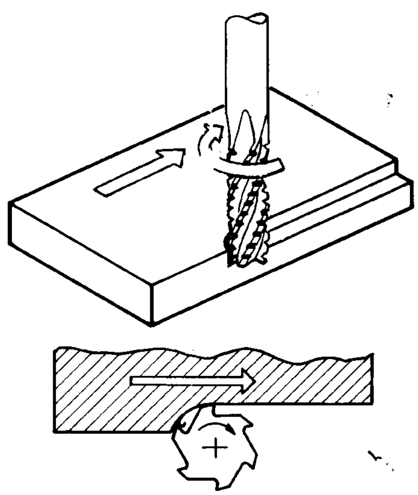
When depth of cut and feedrate are too great the milling cutter may bend, causing a danger of breakage. This is especially true with small diameter end mills and when slotting.

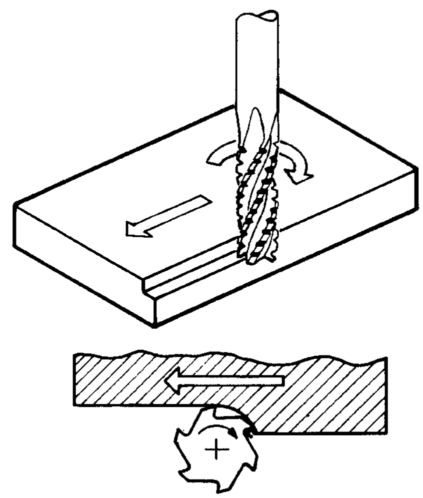
Deep slots should be finished with a series of light cuts with progressively greater depths. Because of the greater force required when slotting, feedrates and speeds should be reduced by 30 % from those used for normal edge milling.

When using end mills of 1/4" diameter or smaller it is a good idea to start with light cuts and conservative feedrates to minimize the hazard of tool breakage. This is especially important when working with an unknown material.

page 1.2 is blank

Climb Milling (downcut milling) vs. Conventional Milling (up-milling)





For many years it was common practice to do up-milling (known as conventional milling) in which the cutting direction goes against the direction of feed. This was popular because most older machines were not fitted with ballscrews and backlash eliminating devices. On more modern NC-machines many people prefer downcut or "climb" milling, where the cutting direction is the same as the feed direction. In climb milling the cutter enters the workpiece with a full chip load that thins out toward the end of the cut. This tends to dissipate heat into the chips rather than the workpiece, and also tends to push the piece into the table creating a greater load. Generally, it makes little difference which method is used on the F1-CNC, except with very small cutters or this sections where conventional up-milling may be preferred.

page 1.4 is blank

Calculations used in milling

1. Cutting speed:

V = .262 x D x RPM where

v = cutting speed in ft/min.

D = cutter diameter in inches

The optimum cutting speed depends on the tool and material being cut. Some suggested speeds with HSS:

Aluminium (2011,2024,7075)	-	450-500 ft/min
Free machining brass	-	250-300 ft/min
Plastics	-	300-400 ft/min
Low carbon steel	-	115 ft/min
Med. carbon & alloy steel	-	80 ft/min

2. Spindle speed:

$$RPM = \frac{3.82 \times V}{D}$$ where

V = desired cutting speed in ft/min

D = cutter diameter in inches

3. Feed per tooth:

$$f_t = \frac{fm}{n \times RPM}$$ where

f_t = feed per tooth

f_n = feedrate in inches/min

n = number of teeth on cutter

Some suggested feeds per tooth, taken from various reference manuals (3/8-3/4" dia. cutters):

Aluminium	-	.003"-.004"
Free machining brass	-	.001"-.002"
Plastics	-	.003"-.004"
Steel	-	.001"-.002"

These feeds are suggested optimum rates. However, they may not be attainable with small diameter cutters because of their limited strength. The higher the feedrate the more force is exerted against the cutter, causing it to bend. With small end mills it is better to start slow and gradually increase feeds till you find a safe limit.

4. Cutting horsepower at spindle:

HP = d x D x fm x P where

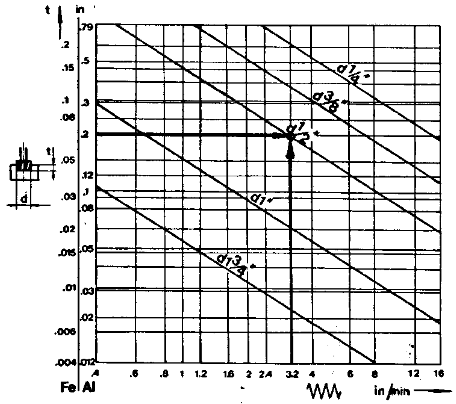
HP = required horsepower
d = depth of cut in inches
D = cutter diameter
fm = feed per minute
P = unit power factor. $HP/In^3/min$

The power required to cut various types of material is often expressed as a Unit Power Factor, in terms of horsepower, required to remove material at the rate of one cubic inch per minute. Unit power factors for milling some common material are:

Aluminium alloys	-	.32-.40
Brass	-	.64-.80
Low carbon steel	-	1.1-1.4
Medium carbon & alloy steel	-	1.5-2.0
Cast iron	-	.6-1.4

Unit power factor can vary depending on the hardness of the material. There are also many other factors which can effect the power and performance in machining operations. Among these are cutter sharpness, lubricants, various material properties, machine drive efficiency, motor torque characteristics, and machine rigidity to mention a few.
While it may be impossible to find the perfect combination without trial and error experiments, the above formulas and figures can be used for estimating starting values.

The following charts can also be used as a guide to starting feeds and speeds for the F1-CNC. To truly optimize programs you will have to experiment a little with your own tools and material.



Procedure

The technological data are written into the tool specification sheet.

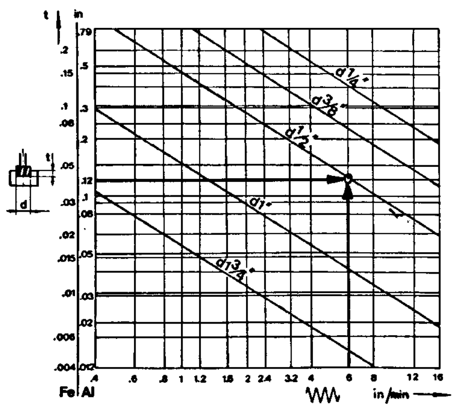
Finding the feed rate and the depth of cut:

Material: aluminium

① Depth of cut (t = .2")

② Dia. of milling cutter (d= 1/2")

→ F = 3.2 in/min

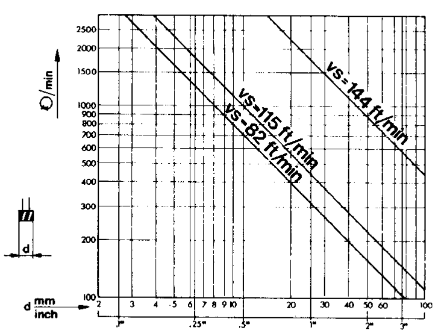


You can also proceed in a different way:

① Feed rate F =6 in/min

② Dia. of milling cutter (1/2")

→ Depth of cut .12"



Finding the speed of rotations:

d — Diameter of milling cutter

Vs — Correct cutting speed for the specific material

S — Spindle speed

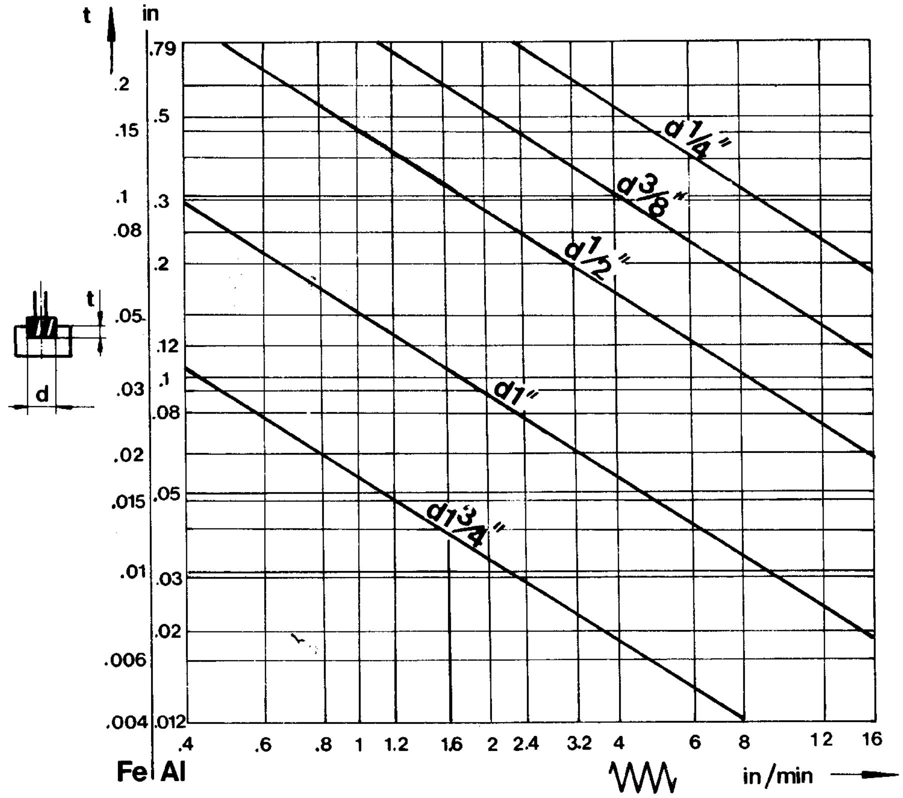
The same procedure applies for drilling.

PS: Downcut milling - Conventional Milling

The specific knowledge is presupposed. However, with the F1-CNC the differences may be neglected.

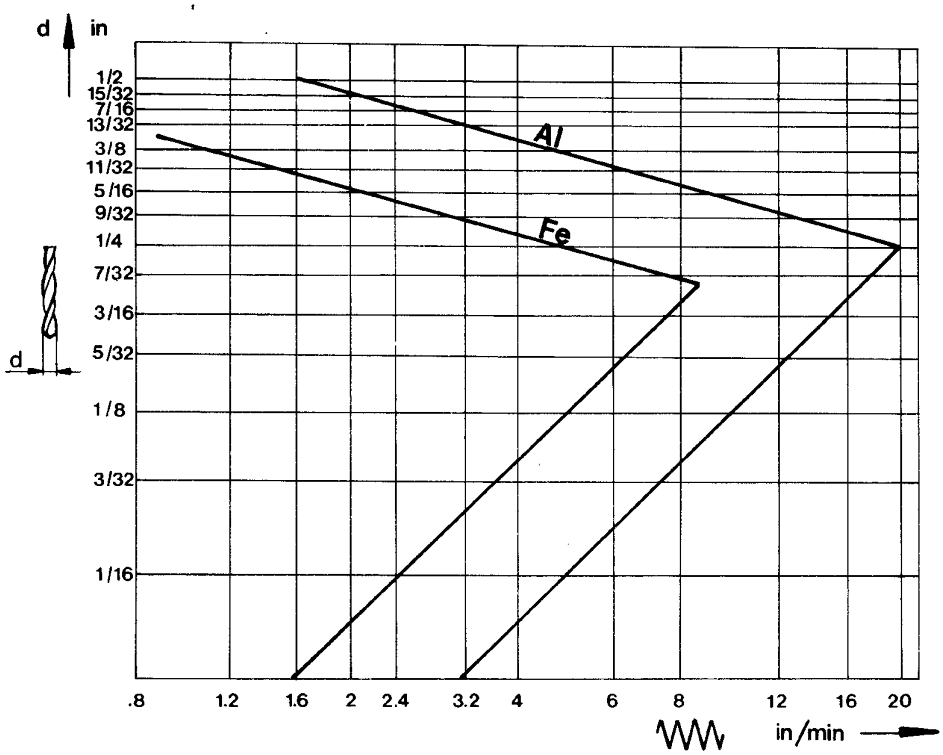
Milling

Depth of cut · Cutter diameter · Feed

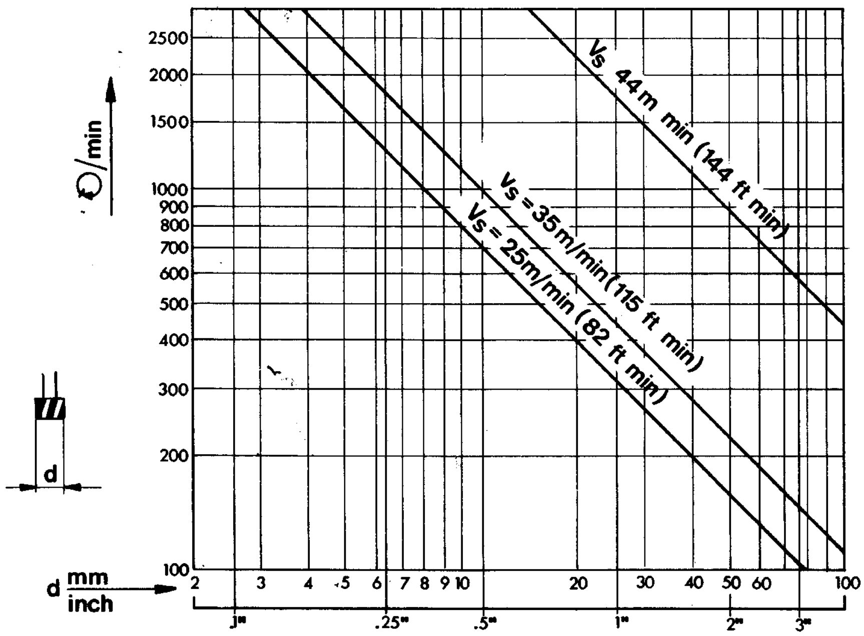


Drilling

Diameter of drill bit · Feed



Speed (of rotation) — Cutting speed — Feed



Attention:

When plunging in with cutter, halve feed values of mill chart.

Service and Maintenance of Machine

Lubrication:

Lubricate guideways of longitudinal, cross and vertical slide daily using oil gun (1 nipple on vertical slide, 2 nipples left side underneath longitudinal slide).

Pressure resistant, corrosion-protective oil with slip-stick reducing characteristics.
2.87"/sec (cSt) reference temperature 40° C.

E.g. CASTROL MAGNA BD 68
This corresponds to the CINCINNATI Specification P47.

Spindle taper for tool mounting

Interior taper of main spindle and tool taper have to be free of grease and dust. They should also be wiped down regularly with a light oil to prevent surface rust.
When not in use toolholders should be removed from the spindle and stored in the proper storage slots.

Safety measures

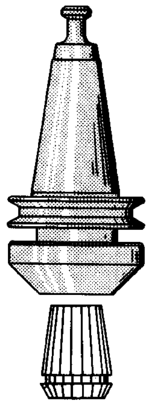
Pay attention to all rules of general shop safety and particularly those for milling. Remember, you are the most important ingredient in safe machining.

Raw material

When using aluminium, choose free machining grades such as 2011, 2017, 2024 or 7075. Generally, the harder tempers will produce finer chips and better finish quality than soft annealled varieties.

Tools

Use of good quality high speed steel cutting tools is recommended on the F1-CNC. Regular end mills, drills, etc. may be used, also special cutters may provide better results on certain non-ferrous materials. In any case, well sharpened cutting tools will give the best results.



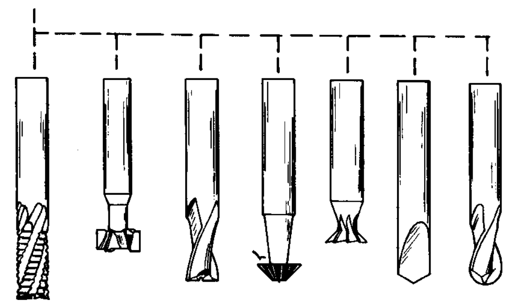
Clamping of Tools

Attention:

Spindle taper and tool taper must be free of all dirt or chips that would interfere with proper seating.

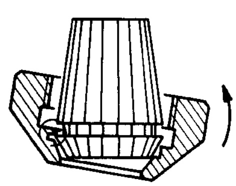
Clamping with collet chuck

Tools with cylindrical shaft are clamped with the collet chuck.



Note:

- Put collet into nut inclined so that the eccentric ring grips the groove of the collet. Screw nut with collet onto collet chuck.



Clamping of tools

Put tool into collet and tighten nut with cylindrical pin in clockwise direction. For counter-holding of main spindle put cylindrical pin into collet holder.

Taking out the collet:

Unscrew nut. The eccentric ring in the nut presses the collet out when unscrewing.

Maintenance

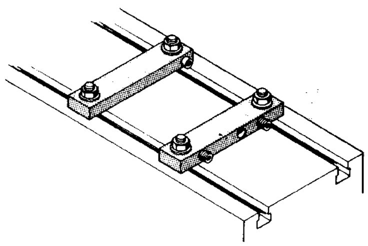
Use oil and clean collet and collet chuck after use. Chips and dirt can damage the tapers and influence the precision.

Collets

You find the clamping capacity in inch and metric engraved on the collets. Diameters smaller or larger than indicated must not be clamped.

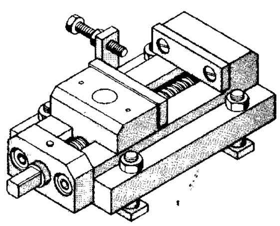
FPC has these items and a small milling vise for the EMCO F1

Clamping Possibilities for Workpieces



Clamping bars

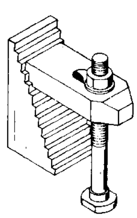
The clamping bars are mounted directly onto the slide depending on the relative workpiece.



Machine vice with stop

Width of jaw: 2.36″

Clamping capacity: 2.36″



Stepped clamping shoe

Height: 2.36″

For clamping a workpiece you need at least two clamping shoes.

page 1.13 is blank

FPC does not have the adapters shown

3-jaw chuck (2 × 3 Jaws)

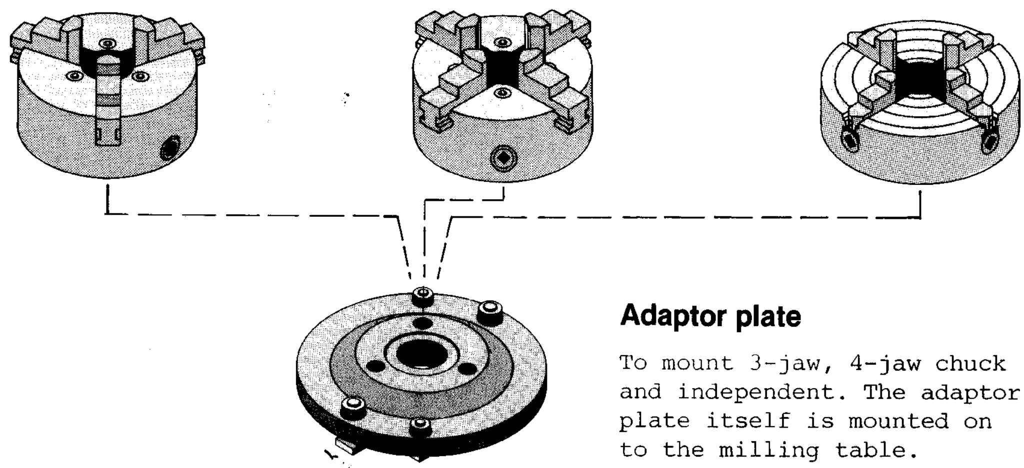
For holding of round, triangular and hexagonal workpieces centrically.

4-jaw chuck (2 × 4 jaws)

For holding of round, square and octogonal workpieces centrically.

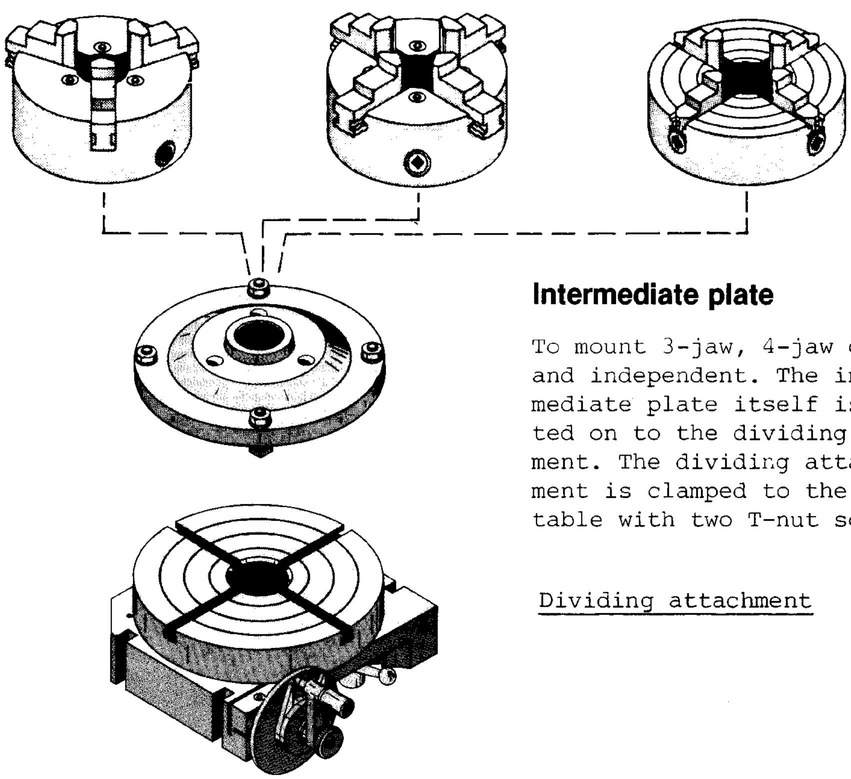
4-jaw independed chuck

For holding of workpieces centrically and eccentrically.



Adaptor plate

To mount 3-jaw, 4-jaw chuck and independent. The adaptor plate itself is mounted on to the milling table.



Intermediate plate

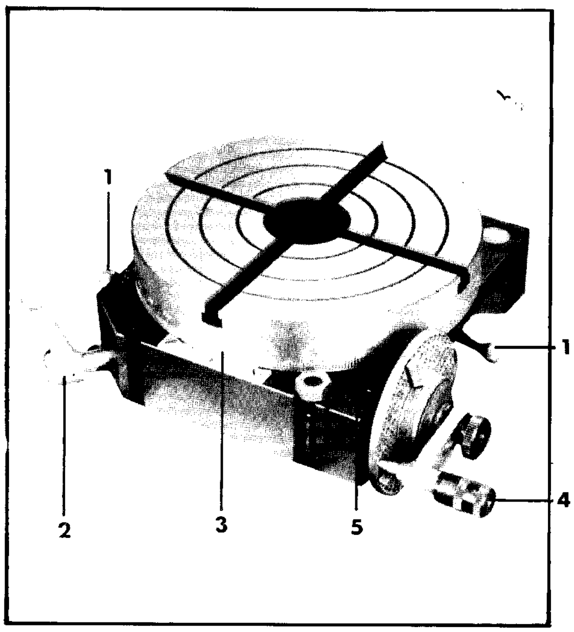
To mount 3-jaw, 4-jaw chuck and independent. The intermediate plate itself is mounted on to the dividing átachment. The dividing attachment is clamped to the milling table with two T-nut screws.

Dividing attachment

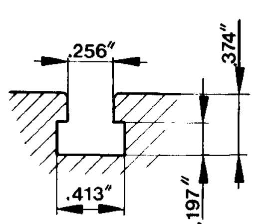
The Dividing Attachment

Operating tips

FPC does not have the dividing head shown. A small PRC unit is available from Harbor Freight for about 100$. Larger American European made units suitable for the Bridgeport can cost as much as 5-6,000$



T-slots of the dividing attachment



TECHNICAL DATA

Diameter of rotary table: 6"

Worm reduction: 1:40

T-slots according to factory standard

Number of holes in dividing plates: 27,33,34,36,38,39,40,42

OPERATING ELEMENTS

Clamping levers for rotary table (1):

Clamping levers are loosened during the dividing operation itself, but must be clamped before every machining operation.

Indexing pin with handle (2):

During direct dividing from 15° to 15°, the pin rests into the parameter notches of the rotary table. During indirect dividing (worm dividing) or free dividing by means of the graduated scale, the indexing pin must be pulled out and swivelled to the left.

The graduated scale (3) is for controlling the divisions.

Crank handle with index plunger (4) moves the worm which is engaged with the wormwheel of the rotary table during indirect dividing.

The shears serve to facilitate adding the number of holes when a fraction of a turn is to be added.

Disengaging and engaging the worm:

The allen head screw (5) is loosened. When the dividing plate is turned counterclockwise, the worm and wormwheel are disengaged. The rotary table can be turned by hand for direct indexing. By turning the dividing plate clockwise, worm and wormwheel are engaged. To facilitate engagement of worm and wormwheel, the rotary table should be moved slightly by hand.
The allen head screw (5) must again be retightened.

Types of Dividing

Indirect dividing:

Indirect dividing offers many more dividing possibilities and is more accurate because of the worm reduction of 1:40.

Indirect dividing method:

If the crank handle is turned 40 times, the rotary table makes 1 revolution (360°). With help of the dividing plates, exact fractions of turns can be executed.

Direct dividing:

Worm and wormwheel are disengaged.

Possibility 1:

Dividing by means of the indexing pin. Dividing possibility from 15° to 15° (i.e., maximum of 24 divisions within 360°).

Possibility 2:

The dividing can be done freely with the aid of the graduated scale on the rotary table.

Note

With indirect dividing the indexing pin is always disengaged. For manufacturing a workpiece the rotary table has to be fixed.

The indexing chart:

1st column: indicates number of divisions per 360°

2nd column: shows the corresponding angle of the division

3rd column: shows the number of 360° crank handle revolutions which are necessary

4th column: shows the number of holes to be added for each index plate

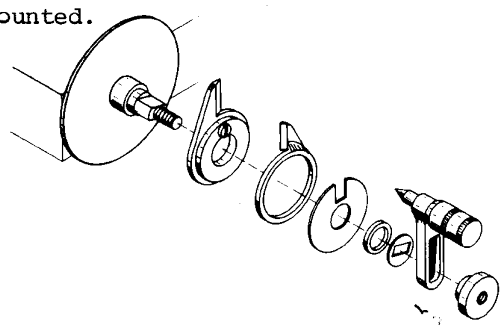
Example of an indirect dividing operation:

Desired division: 13 divisions in 360°

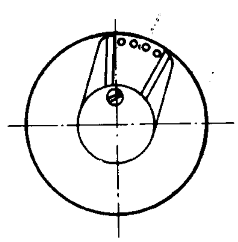
From the indexing table it can be seen that at the desired division 13, 3 full crank turns must be made plus a fraction turn of 3 additionsl holes on the indexing plate 39.

Practical execution:

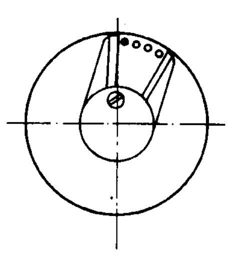
1. The indexing plate with 39 holes is mounted.



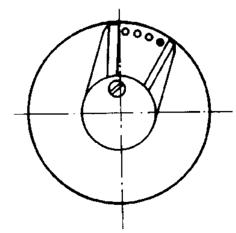
2. in the indexing table one sees that at the division 13, 3 full turns plus 3 holes on the 39 plate have to be added.. Therefore, the shears are fixed so that they include 4 holes.



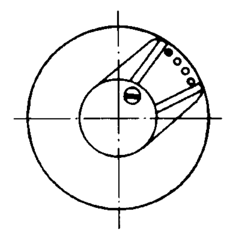
3. The indexing plunger is placed in a hole of the 39 plate (marked black on the drawing) and the left shear arm moved until it touches the pin of the plunger.



4. Execution of the dividing operation: 3 full turns plus the fractional turn of the 3 added holes are made; that means that the plunger is placed in the black hole. One dividing operation is completed.



5. Next dividing operation: The shears are turned until the left arm touches the pin again; the next dividing operation follows as described in 4. above.



NOTE: The shears may not be moved during the dividing operation, otherwise they do not serve their purpose as an orientation aid.

NOTE: If a larger number of holes has to be reached than the maximum opening of the shears allow, you have to set the difference of holes between the shears.

Example

21 divisions per 360° have to be carried through. From the chart one can see that one full turn plus the fractional turn of 38 holes on the disc 42 have to be carried through. 38 holes cannot be set.

Thus: 42-38=4 holes. When dividing you make one additional turn (2 turn alltogether) and turn back the difference of 4 holes (the shears comprise 5 holes).

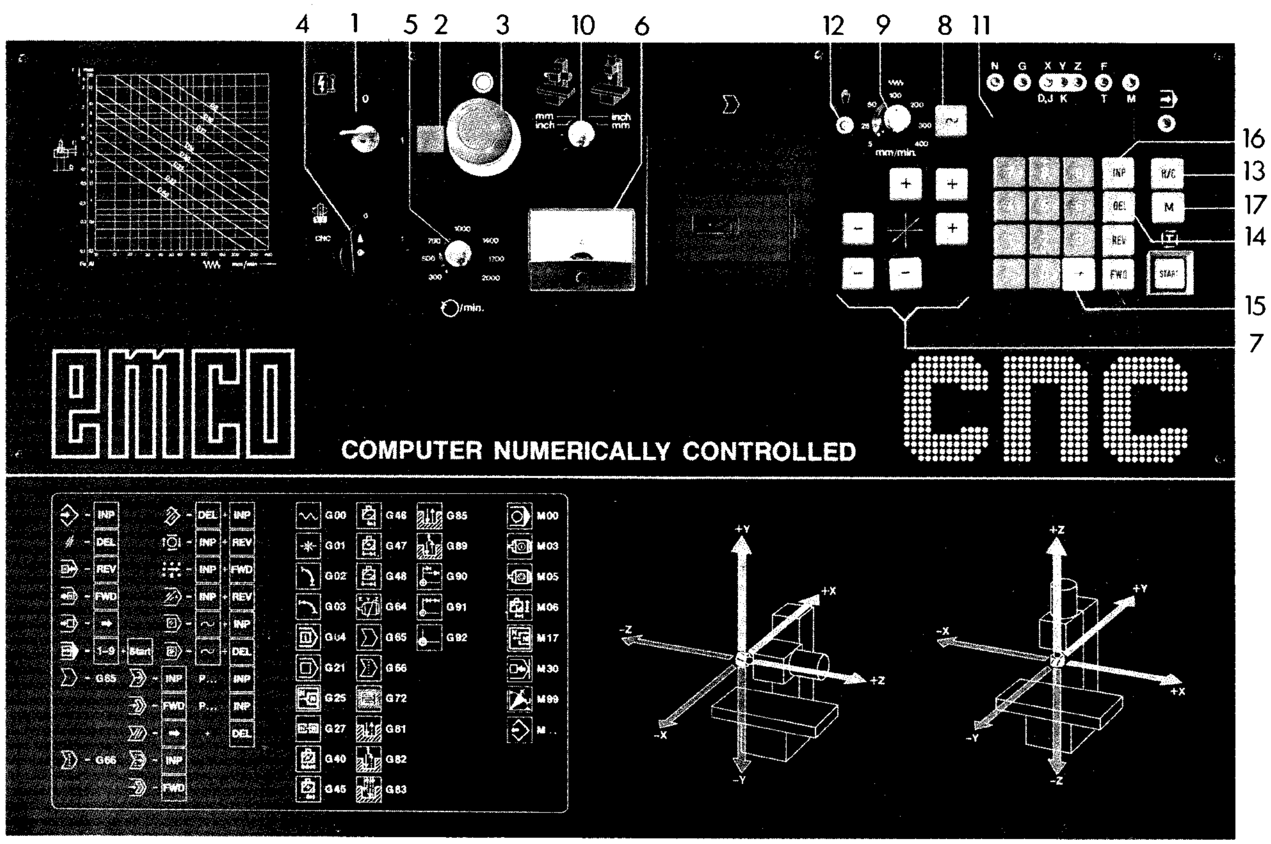
INDEX TABLE	Formula for the Calculation of the Hole Numbers Required
	z = No. of divisions required for one revolution of the workpiece.
	K = No. of revolutions of handle for a complete revolution of the workpiece.
	n = No. of revolutions of handle for one dividing move: $n = \frac{K}{z}$
	Worm reduction of dividing head 1:40; i. e. K = 40.

Division Desired	Degrees	No. of crank turns req'd	Amount of holes to be added for each index plate 27	33	34	36	38	39	40	42
2	180°	20								
	175°	19	12							
	170°	18	24							
	160°	17	21							
	150°	16	18							
	140°	15	15							
	130°	14	12							
	125°	13	24							
3	120°	13	9	11		12		13		14
	110°	12	6							
	100°	11	3							
4	90°	10								
	80°	8	24							
	75°	8	9	11		12		13		14
5	72°	8								
	70°	7	21							
	65°	7	6							
6	60°	6	18	22		24		26		28
	55°	6	3							
7		5								30
	50°	5	15							
8	45°	5								
9	40°	4	12			16				
10	36°	4								
11		3		21						
12	30°	3	9	11		12		13		14
13		3						3		
14		2								36
	25°	2	21							
15	24°	2	18	22		24		26		28
16		2			17	18	19		20	21
17		2			12					
18	20°	2	6			8				
19		2					4			
20	18°	2								
	16°	1	21							
21		1								38
22		1		27						
24	15°	1	18	22		24		26		28
25		1							24	
26		1						21		
27		1	13							
28		1								18
30	12°	1	9	11		12		13		14

Division Desired	Degrees	No. of crank turns req'd	Amount of holes to be added for each index plate 27	33	34	36	38	39	40	42
32		1				9			10	
33		1		7						
34		1			6					
35		1								6
36	10°	1	3			4				
38		1					2			
39		1						1		
40	9°	1								
42										40
44				30						
45	8°		24			32				
48						30				35
50									32	
	7°		21			28				
52								30		
54			20							
55				24						
56										30
60	6°		18							
64									25	
65								24		
66				20						
68					20					
70										24
72	5°		15			20				
76							20			
78								20		
80					17	18	19		20	21
84										20
85					16					
88				15						
90	4°		12			16				
95							16			
96						15				
100									16	
120	3°		9	11		12		13		14
180	2°		6			8				
200									8	
240						6				7
270			4							
360	1°		3							
	40′		2							
	30′					2				
	20′		1							

page 1.20 is blank

Operating Elements
Control Elements
Hand Operation



1. Main switch

Turn key to the right. Machine and control are under power unless emergency stop button is pressed.

2. Indicator lamp main power switch

When main switch is on, lamp is on.

3. Emergency stop button

Control, feed motors and main motors are cut off from power by pressing emergency stop button: turn button to the left - it will jump back to original position. Main motor switch has to be switched on again.

4. Switch for main spindle

Turn switch to the right.

5. Adjusting knob for speed control of main spindle - calibrated in RPM

6. Ammeter

Shows power consumption of main spindle motor. In order to protect motor against overload, the power consumption should not surpass 4 A with 110 V.

7. Feed keys for longitudinal, cross and vertical slide

8. Rapid traverse key

If keys for feed and rapid traverse are pressed together, then the relative slide will move with rapid traverse speed.

9. Adjusting knob for setting the feed rate

10. Inch/metric switch and switch for changing the axis system

11. Digital read-out for slide movement

± X, ± Y, ± Z are shown in 1/100 mm or 1/1000 inch.

Plus movement without sign
Minus movement by a light to left of numbers

- 125

X -1,25 mm or - .125 inch

12. Indicator lamp for hand operation

13. [H/C] switch key: hand operation/CNC operation

If you press the [H/C] key the light of the control lamp hand operation will jump to CNC operation (operation mode: CNC). By pressing the key once again the light will jump back (operation mode: hand operation).

14. [DEL] key

The X,Y,Z values are set to zero when this key is pressed.

15. The [→] key

With the [→] key you can switch from X to Y to Z without movement of slides.

16. The [INP] key

With the [INP] key you enter the values for slide movements.

17. M-key

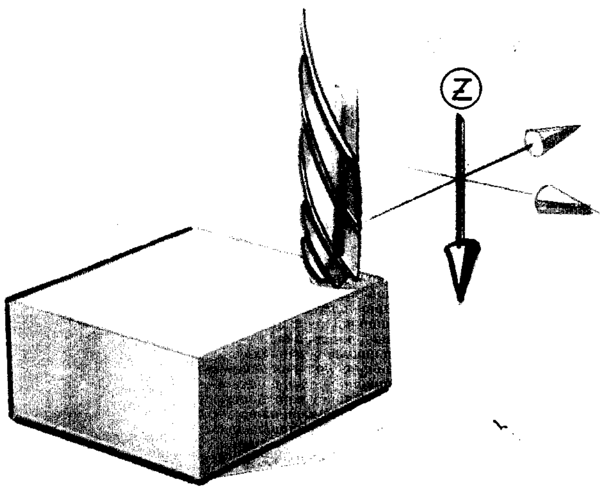
Activates switching exits.

Hand Operation F1-CNC

Positioning of the Milling Cutter

1. Scratching front sides and top side

With milling most measurements refer to outer edges. In order to use the measurements of the technical drawing you have to "zero-set" the display and use as reference/starting point the outer edges.

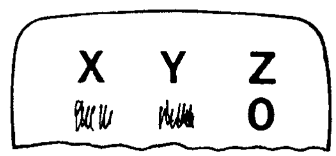


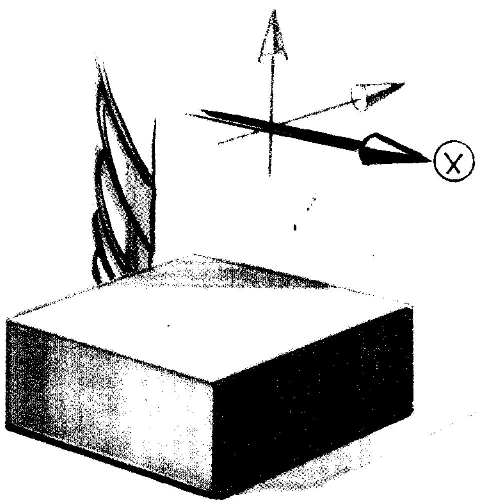
Example

Milling cutter with dia. 1/4".

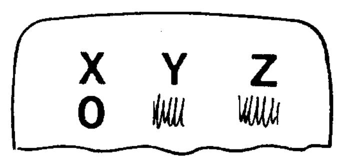
Move milling cutter in Z-direction until you scratch surface slightly.

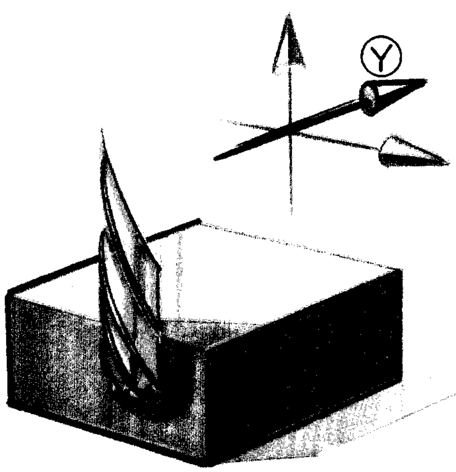
Set Z-display to zero (press key DEL).



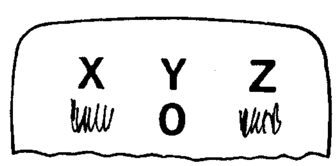


- Scratch front side in X-direction.
- Set X-display to zero (press key DEL)

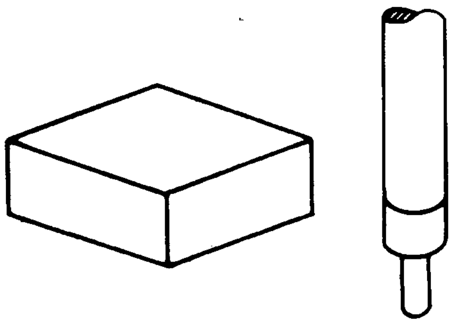




- Scratch front side in Y-direction.
- Set Y-display to zero (press key DEL)

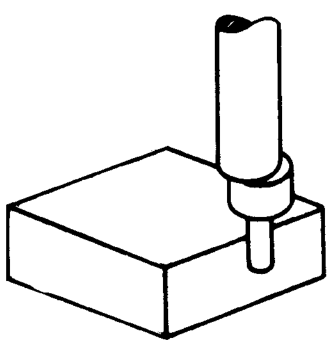


Positioning spindle using edge finder



The edge finder can be used to position the machine relative to the workpiece edge without actually marking the workpiece.

Mount the edge finder in the spindle with the small .200" diameter downward. Turn main spindle on and adjust speed to 1000 rpm. The end of the finder will center itself.

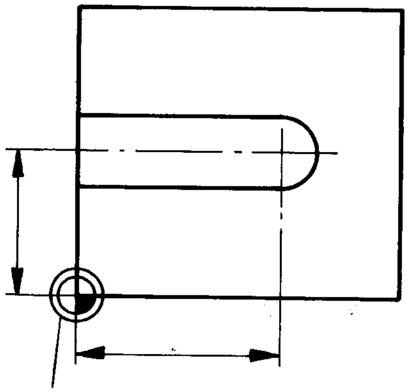


Now slowly jog the machine to move the edge finder near to the desired edge of the workpiece. When the rotating edge finder contacts the workpiece you will see the tip "jump" off-center about 1/32". At this point the centerline of the spindle is exactly .100" away from the edge of the workpiece.

The edge finder is a precision ground instrument that will accurately read position within .001". Like any other instrument it should be treated with care. Approach the work only in slow jog speed and "step" into the piece slowly, never in rapid traverse.

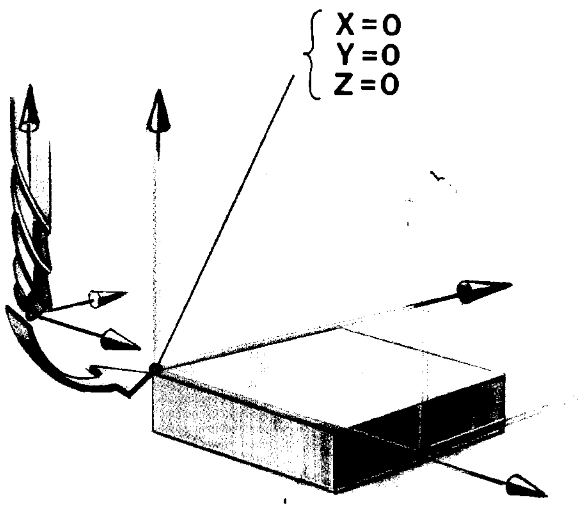
page 2.6 is blank

2. Zero-setting of Display to Zero Point of Dimensioning (Example: Milling)



Example: Milling of groove

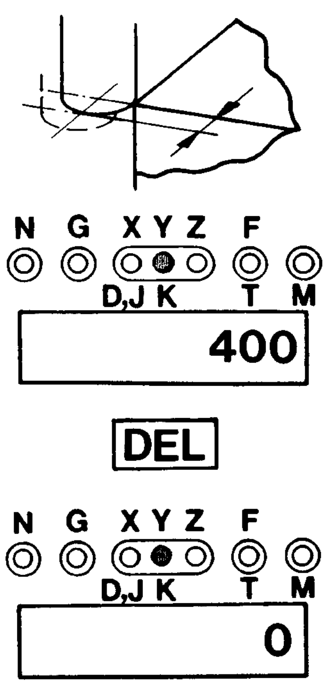
- The groove is milled using a .250" cutter.
- Zero point for the dimensioning is the workpiece edge and surface.
- The measures refer to the center of the milling cutter.



Consequence

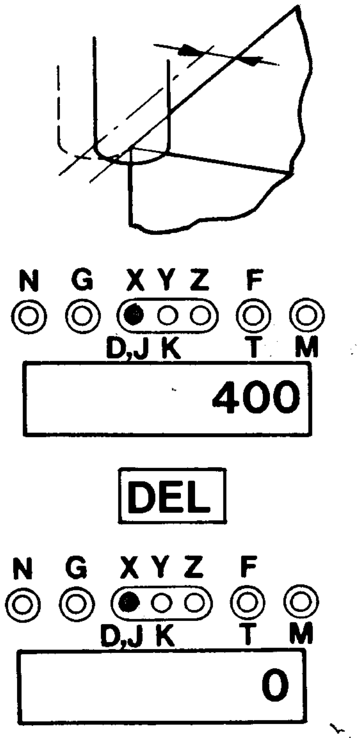
Move axis of milling cutter to edge of workpiece.

a) Scratching of all 3 surfaces and zero-setting of X,Y,Z.



b) Touch milling cutter on left edge to scratch workpiece. Move by value of milling cutter radius (one half the diameter) in the +Y direction. Set Y to "zero" by pressing DEL key.

page 2.6 is blank



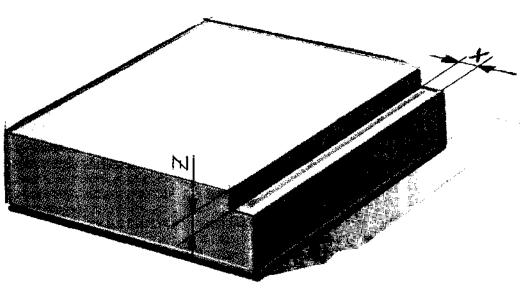
c) Touch milling cutter on front edge to scratch workpiece. Move milling cutter by value of milling cutter radius into X-direction. Set display to "zero".

Exercise

Move milling cutter such that all display values are at "zero".

Exercise

Mill a recess as in drawing. Enter the following values:



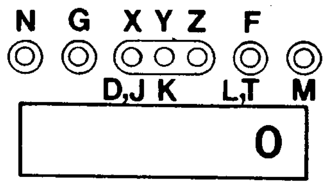
Spindle speed S (rpm)	
Feed mm/min	
Infeed in X (mm)	
Infeed in Z (mm)	

Pay attention to set correct feed.

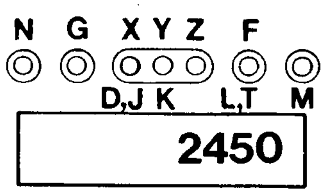
Traverse — Hand Operation

Display

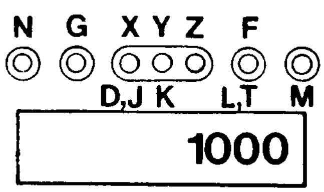
After switching on the machine, the figure 0 appears. Lamps X,Y or Z are on.



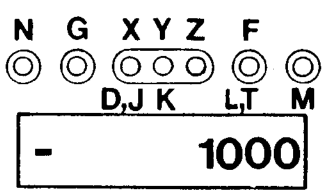
If you traverse in ±X, the X lamp lights up. When you take your finger from the key, the traverse distance is shown in 1/1000" on the display. With a distance of 2.45″ the display indicates 2450.



If you press the Z key, the light jumps to the Z lamp. After you lift your finger from the key, the traverse distance appears (with 1.000" 1000 will appear).

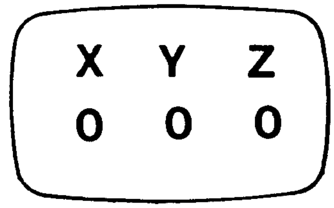


Minus sign on display appears to the left of the numbers.



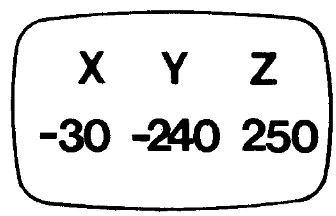
Monitor

The screen shows zero for X,Y,Z when you switch it on.



If you move in X direction and you take your finger from the key, the traverse distance is shown in 1/1000" or 1/100 mm.

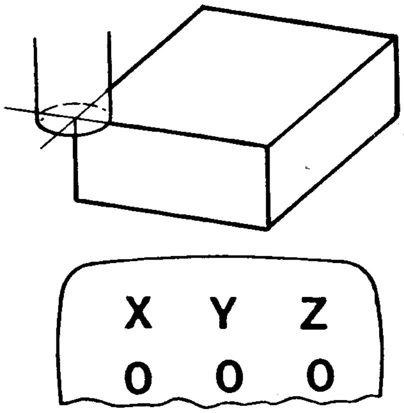
The same happens with Y,Z:
All 3 traverses are shown on the screen.



Minus sign is shown on screen for each axis.

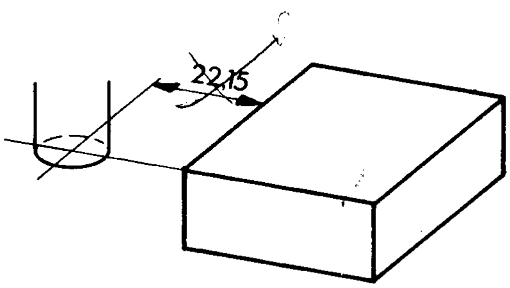
When moving at preset feedrate the display will change every 0.020" or 0.5 mm to update the position shown. In rapid traverse the display will not change until you lift your finger from the key.

Input of X, Y, Z Zero-Values from any chosen Milling Position



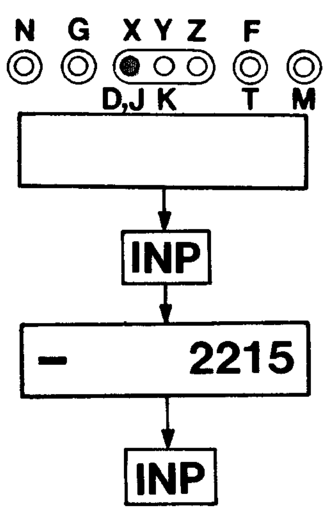
The display should indicate zero, when the milling cutter stands at a given point (X=0, Y=0, Z=0).

You can program the X,Y,Z displays to indicate zero by entering the value of the current position in relation to the desired zero-point.



The center of the milling cutter is at a distance of 2.215" to the workpiece edge in X. The display indicates whatever value.

When the milling cutter traverses in +X direction by 2.215", then the display should indicate the value X=0.

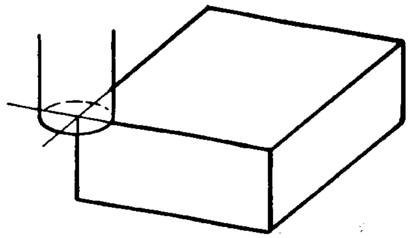


Procedure:

1. The lamp X on the display lights up
2. Press INP - the lamp X flashes
3. Put in the value [2][2][1][5] and minus sign, because the milling cutter should indicate with plus "traverse direction 0".
4. Press key INP. The flashing of the X-lamp stops.

You can enter the Y,Z values in the same way.

When programming minus-values first put in the figures, then press key minus.

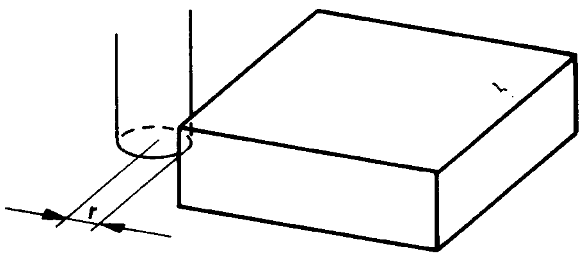


Application of Path Programming in Hand Operation Mode

Zero point for the dimensioning is the workpiece edge. The milling cutter shall move to this point. The displays shall be set zero.

Procedure:

1. Scratch surface, set Z-display zero.

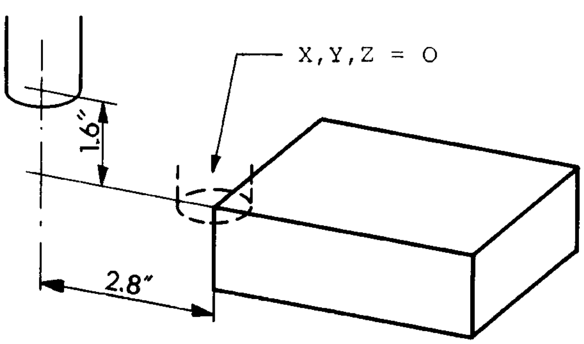


2. Scratch surface in X-direction. Put in value of milling cutter radius r.

3. Scratch surface in Y-direction. Put in value of milling cutter radius r.

Note:

You can traverse after scratching as you like. If you program the zero-point, you have to add to the X,Y display the radius value and put it in.



Exercise:

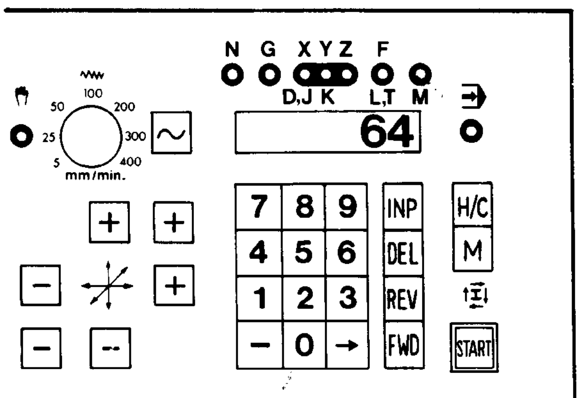
1. Program the display X,Y,Z=O if the milling cutter is positioned onto the edge.

2. Move the milling cutter to the indicated position.

page 2.12 is blank

Switching Feed Motors off

When switching on the machine, the feed motors are not under power.
If you have - in hand or CNC-operation mode - moved the slides, the feed motors stay under power. The video monitor shows a small-[M] symbol in the center of the top line.



Switching power off - with no program being stored

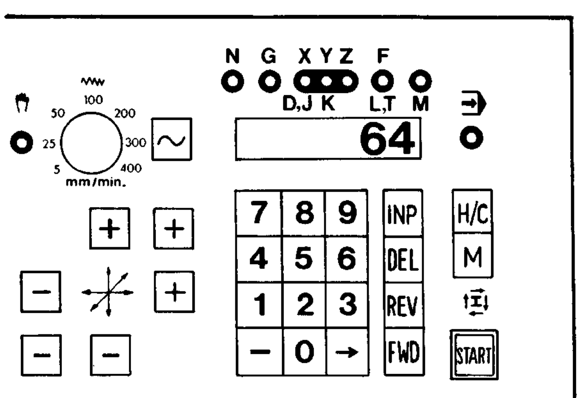
1. Switch to CNC-operation mode: Press H/C key.

2. Press key . The light jumps to G.

3. Key in 6 4. The number appears on the display.

4. Press INP key. Now the feed motors are switched off.

5. The -M symbol disappears from the top line of the video monitor.



Switching power off - with a program being stored

G64 is a pure switching function. It is not stored in memory.

1. Press key so that G light goes on.

2. When a number appears on the display, press DEL.

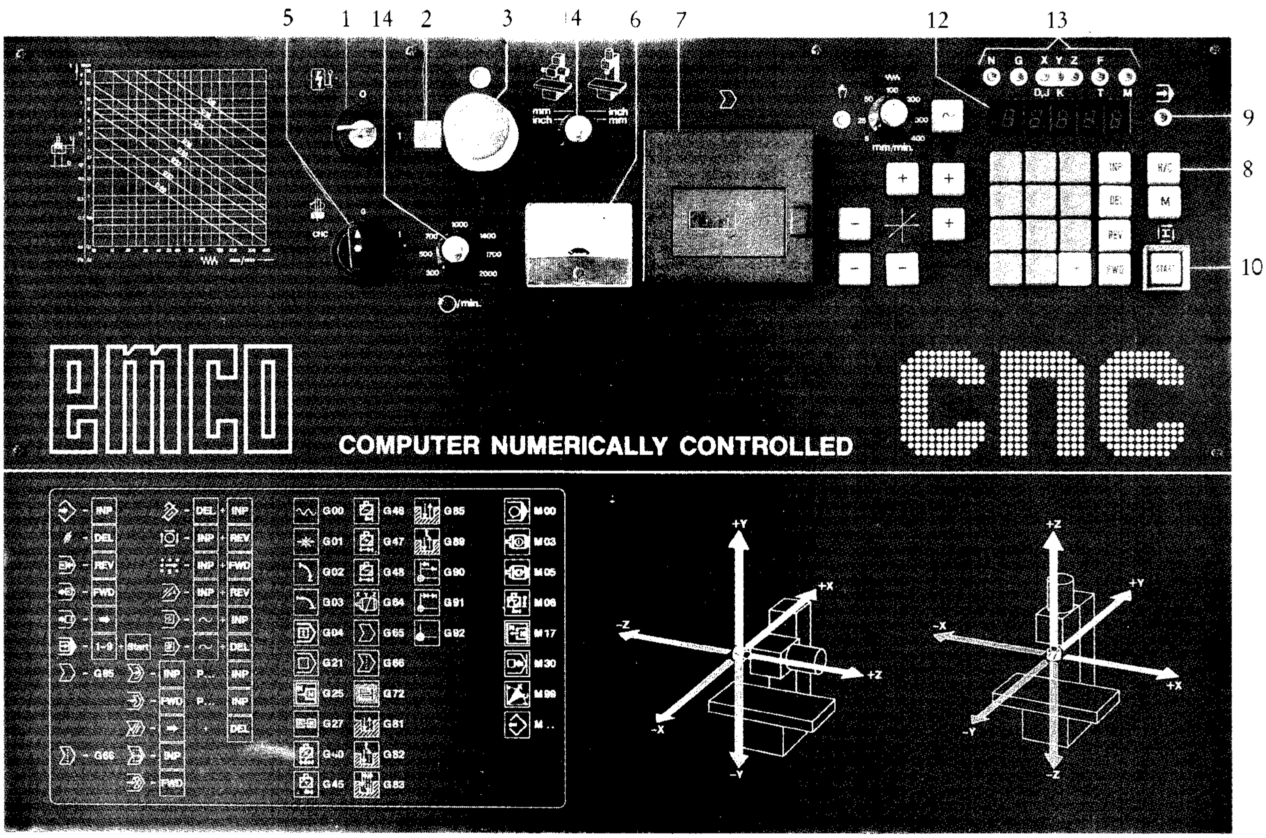
3. Key in 6 4.

4. Press INP key. Now the feed motors are switched off.

5. The video monitor shows that the previous G-value has been replaced in its proper position and the -M symbol disappears.

pages 2.12 & 2.14 are blank

Operating Elements
Control Elements
CNC-Operation

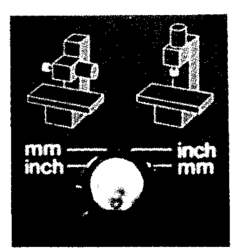


1. Main switch with removable key. Memory is being cleared when switching off.

2. Control lamp shows the power supply of machine and control unit.

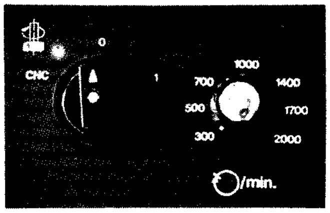
3. Emergency stop button with interlock. Unlocking of button: turn button to the left. To switch on machine, turn main switch to zero and to 1 again. When switching off also memory will be cleared.

4. Optional switch for axis system and for metric or inch mode of operation.



page 3.1 is blank

5. Switch for main spindle



Position 1 (main spindle ON, without M03)

Position CNC: main spindle is switched on by programming M03 and switched off by M05, M06 (with F≠0) and M30.

6. Ammeter

7. Magnetic tape

8. H/C switch key
Manual/CNC operation

9. Control lamp CNC operation

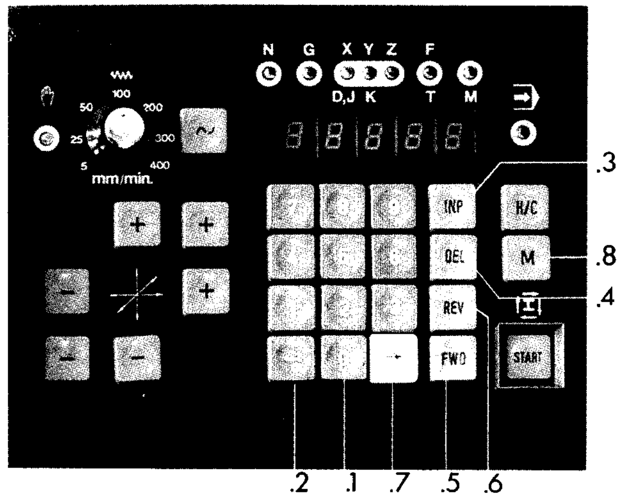
10. START key
The program is being worked off

12. VDU (display):
Indicates values for address letters and modes of operation

13. Control lamp address letters

14. Control of milling spindle speed

11. Keys for program input, correction, storing of program on tape, V24 operation etc. (see detailed explanations)



11.1. Number keys 0 - 9

11.2. – The minus sign key
To enter minus values the minus sign – has to be pressed after input of numbers.

11.3. INP key (INPUT = storing)
Storing key

11.4. DEL key (DELETE = erase)
Erasing key

11.5. FWD key (FORWARD)
Program jumps forward block by block

11.6. REV key (REVERSE)
Program jumps backwards block by block

11.7. → Arrow key
Display jumps word by word

11.8. M key: key for entering of miscellaneous functions.

Survey

Preparatory Functions, G-Codes (metric)

Note: Formats inch: all X,Y,Z values have 4 digits (No differences in horizontal and vertical Axis system)

G 00 Rapid traverse

V: N3/G00/X±5/Y±4/Z±5

H: N3/G00/X±4/Y±5/Z±5

G 01 Linear interpolation

V: N3/G01/X±5/Y±4/Z±5/F3

H: N3/G01/X±4/Y±5/Z±5/F3

G 02 Circular interpolation clockwise

G 03 Circular interpolation counterclockwise

Quadrants:

V: N3/G02 G03/X±5/Y±4/Z±5/F3

H: N3/G02 G03/X±4/Y±5/Z±5/F3

N3/M99/J2/K2 (Partial circles)

G 04 Dwell

N3/G04

G 21 Empty block

N3/G21

G 25 Sub-routine program call

N3/G25/L(F)3

G 27 Jump instruction

N3/G27/L(F)3

G 40 Tool radius compensation cancelled

N3/G40

G 45 Add tool radius

N3/G45

G 46 Subtract tool radius

N3/G46

G 47 Add tool radius twice

N3/G47

G 48 Subtract tool radius twice

N3/G48

G 64 Feed motors without current (switching function)

N3/G64

G 65 Magnetic tape operation (switching function)

N3/G65

G 66 Activating RS 232 Interface

N3/G66

G 72 Pocket milling cycle

V: N3/G72/X±5/Y±4/Z±5/F3

H: N3/G72/X±4/Y±5

G 74 Thread-cutting cycle (left-hand)

N3/G74/K3/Z±5/F3

G 81 Fixed boring cycle

N3/G81/Z±5/F3

G 82 Fixed boring cycle with dwell

N3/G82/Z±5/F3

G 83 Fixed boring cycle with chip removal

N3/G83/Z±5/F3

G 84 Thread-cutting cycle

N3/G84/K3/Z$\pm$5/F3

G85 Fixed reaming cycle

N3/G85/Z$\pm$5/F3

G89 Fixed reaming cycle with dwell

N3/G89/Z$\pm$5/F3

G90 Absolute value programming

N3/G90

G91 Incremental value programming

N3/G91

G92 Offset of reference point

V: N3/G92/X$\pm$5/Y$\pm$4/Z$\pm$5

H: N3/G92/X$\pm$4/Y$\pm$5/Z$\pm$5

V = Vertical

H = Horizontal

Miscellaneous or Switching Functions

M00 – Dwell
N3/M00

M03 – Milling spindle ON, clockwise
N3/M03

M05 – Milling spindle OFF
N3/M05

M06 – Tool offset, milling cutter radius input
N3/M06/D5/S4/Z ± 5/T3

M17 – Return to main program
N3/M17

M08, M09, M20, M21, M22, M23 – Switching exits
N3/M2

M26 – Switching exit – impulse
N3/M26/H3

M30 – Program end
N3/M30

M99 – Parameters circular interpolation
(in connection with G02/03)
N3/M99/J2/K2

Alarm Signs

A00: Wrong G/M code

A01: Wrong radius / M99

A02: Wrong X-value

A03: Wrong F-value

A04: Wrong Z-value

A05: M30 code missing

A06: M03 code missing

A07: No significance

A08: Tape end with cassette operation SAVE

A09: Program not found

A10: Writing protection

A11: Loading mistake

A12: Checking mistake

A13: Inch/mm switching with full program memory

A14: Wrong mill head position/path increment with LOAD ⊥ /M or ⊣ /M

A15: Wrong Y-value

A16: Value of milling cutter radius missing

A17: Wrong sub-routine

A18: Path milling cutter compensation smaller zero

Possible Inputs
(Otherwise alarm signs)

	Metric		Inch	
	Values	Unit (mm)	Values	Unit (inch)
X_V	0-19999	1/100 mm	0-7999	1/1000"
X_H	0-9999	1/100 mm	0-3999	1/1000"
Y_V	0-9999	1/100 mm	0-3999	1/1000"
Y_H	0-19999	1/100 mm	0-7999	1/1000"
Z_{VH}	0-19999	1/100 mm	0-7999	1/1000"
Radii	0-9999	1/100 mm	0-3999	1/1000"
D(X) milling cutter radius with M06	0-9999	1/100 mm	0-3999	1/1000"
F	2-499	mm/min	2-199	1/10"/min
T(F) tool address M06	0-499	1	0-199	1
L(F) jump instructions	0-221			
H(F) exit signs M26	0-299			
J/K circular parameter	0-90			

Adresses

N, G, X, Y, Z, F, D, J, K, L, M, T, S, H

page 3.8 is blank

Operation CNC

[INP]	Storing of word contents
[DEL]	Deleting of word contents
[FWD]	Forward in program block by block
[REV]	Backward in program block by block
[→]	Forward in block word by word
[M]	Input of M-functions

Program hold:

[INP] + [FWD]

Program interruption

[INP] + [REV]

Delete program

[DEL] + [INP]

First [DEL] then [INP]

[DEL] remains pressed.

Delete alarm

[INP] + [REV]

Insert block

[~] + [INP]

Delete block

[~] + [DEL]

Single block mode

[1][2][3] etc. + [START]

Testrun:

[M]

Operation – Magnetic tape

Storing of program on tape

G65 [INP] → [FWD] → Put in program number → [INP]

Transmit program from tape to memory

G65 [INP] → [INP] → Select program number → [INP]

Delete tape contents

G65 [INP]

↓

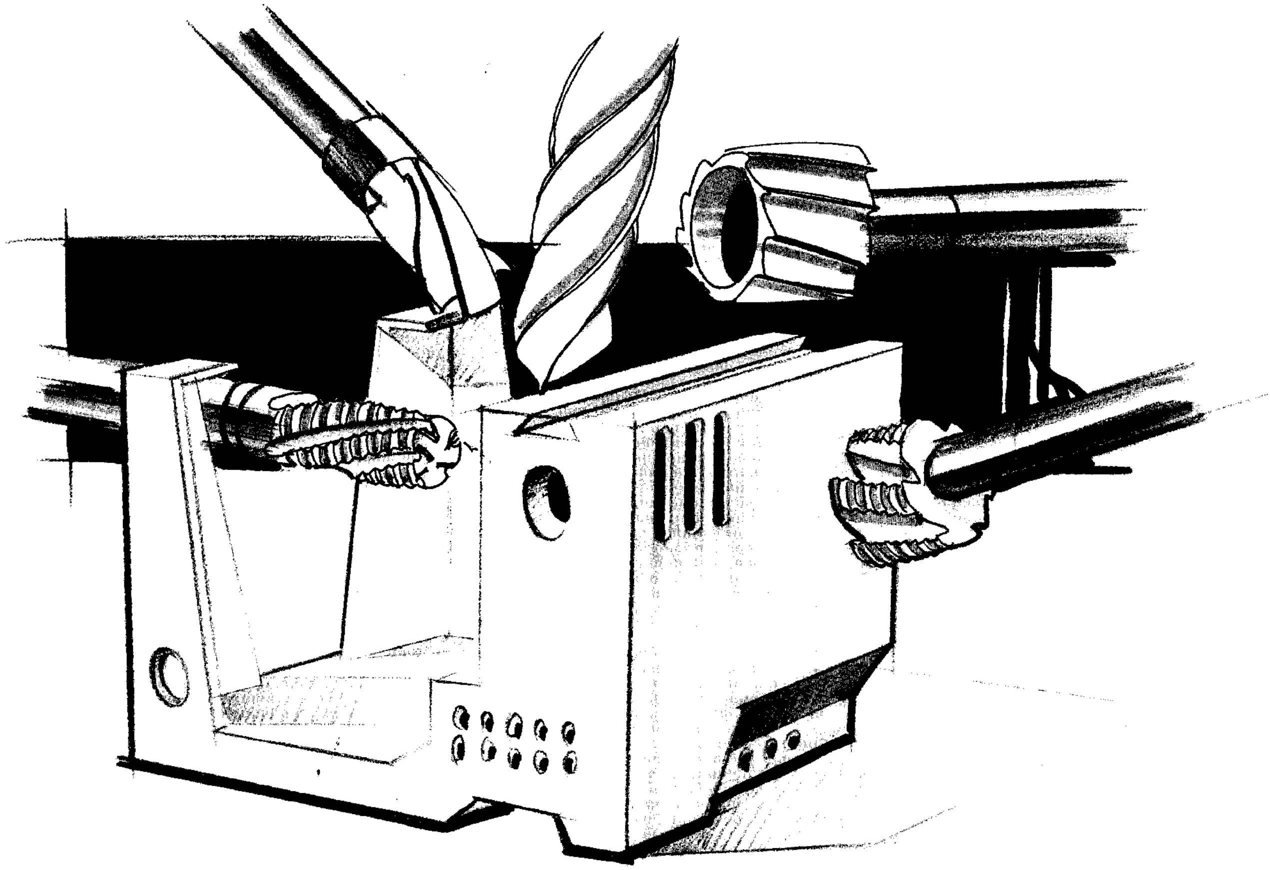
[→] + [DEL]

pages 3.8 & 3.10 are blank

Chapter 6

Tools, tool lengths compensation, radius compensation of milling cutter

The Programming of the Tools



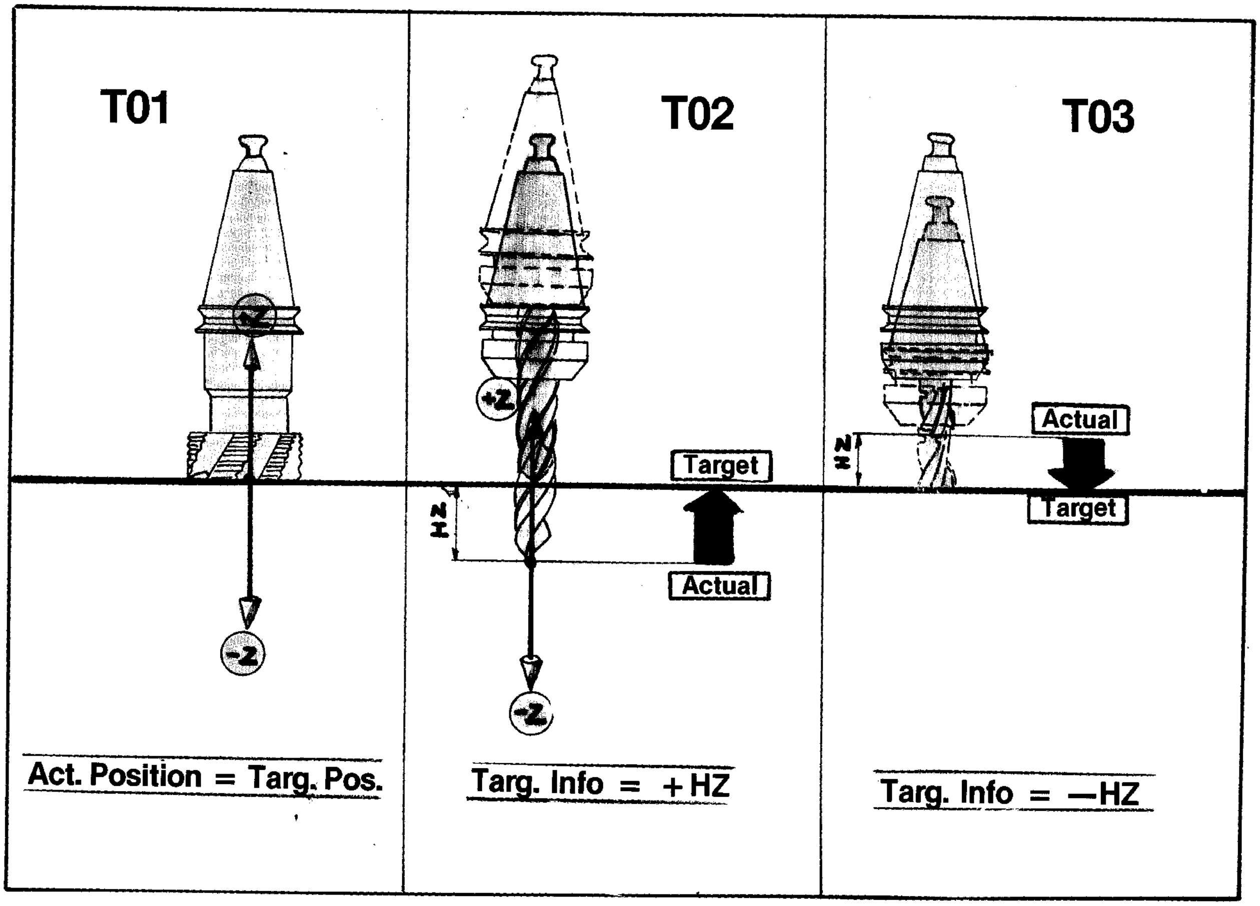
Tool magazines of industrial NC-machines are equipped with up to 50 or more tools.

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.

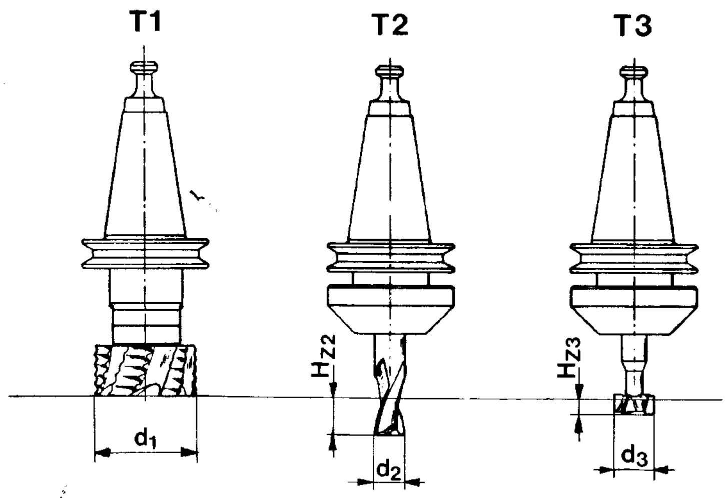
Imagine the coordinate system transferred into the reference plane of the tool.

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

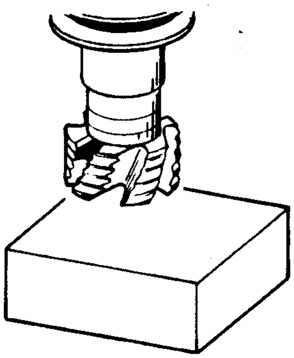
1. The milling cutters are of different diameters. These are known to you.

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).

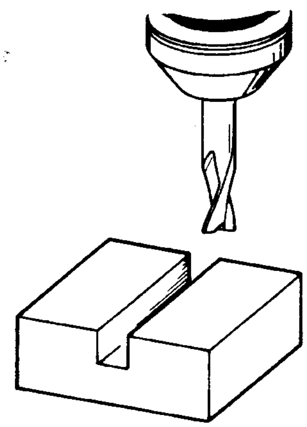
page 6.6 is blank

Procedure

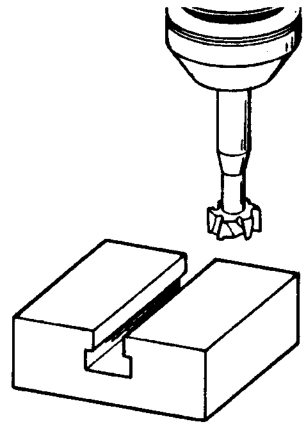
1. Determining the tool sequence



Facing with T1

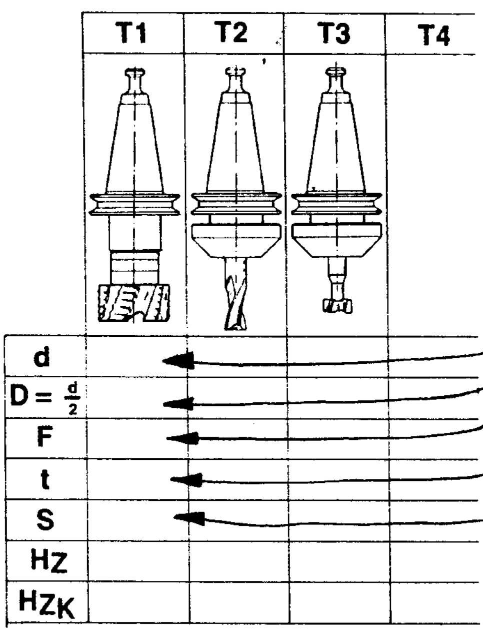


Milling a slot with T2



Milling a T-slot with T3

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



Entering the data

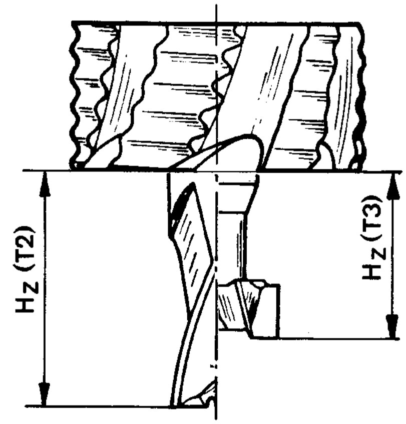
1. Stick the tools into the correspon-ding column.

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.

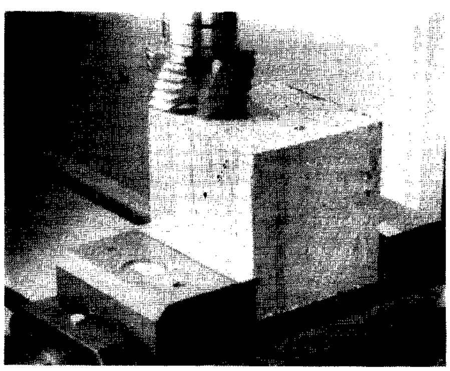
The difference is called H_Z.

Procedure

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

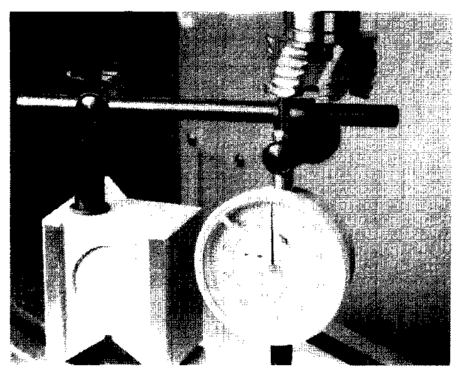
Detection of data by scratching

Scratching only when cutter is turning

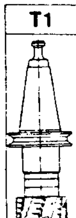


Detection of data with dial gauge.

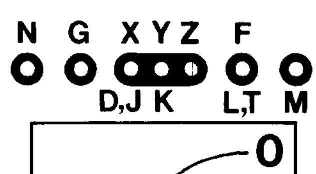
Set dial gauge when machine is at stand-still.



Set dial gauge to 0.

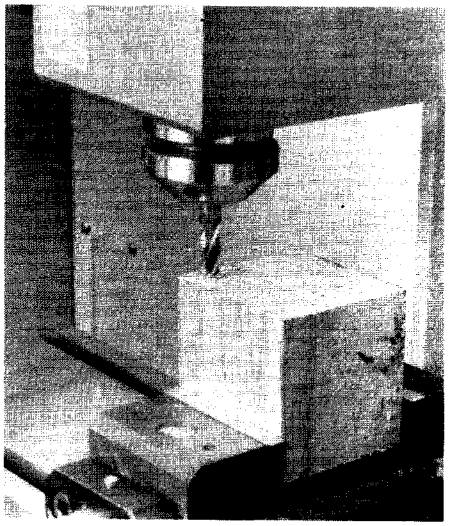
	T1	T2	T3	T4
				
d				
$D = \frac{d}{2}$				
F				
t				
S				
Hz	0			
Hz_K				

Press key DEL, the Z-value display is set to 0.

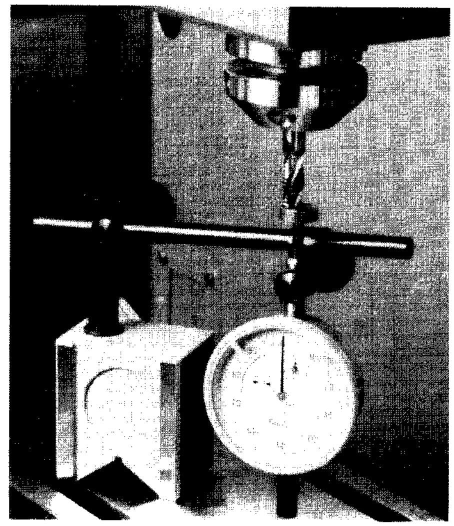


page 6.10 is blank

Mount T2



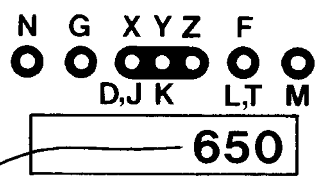
Scratch surface



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.



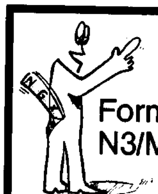
Enter value into tool data sheet. In this way you determine all tool lengths.

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)

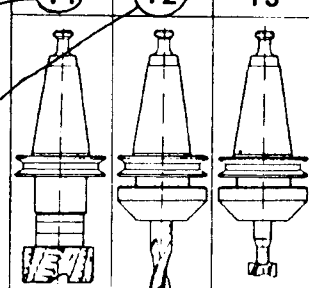


Format M6
N3/M06/D(X)4/S4/Z(Hz) ± 4/T(F)3

The data are entered into the programming sheet.

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

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	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.

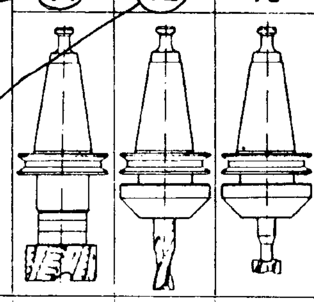
Calculation of tool length M06 (Tool lengths compensation) (Programming)

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

The data are entered into the programming sheet.

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	2000	1100	0	1
N...	M06	500	2000	650	2



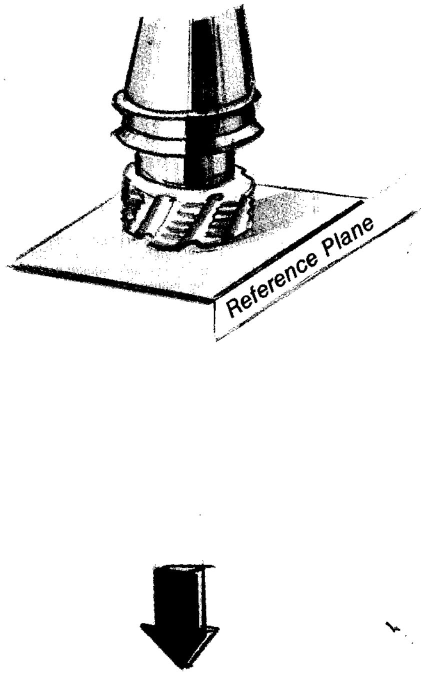
	T1	T2	T3	T4
d	40	10	16	
$D = \frac{d}{2}$	20	5	8	
F	80	160	40	
t	0,7	5	8	
S	1100	2000	2000	
HZ	0	650	-320	
HZK				

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

⇩

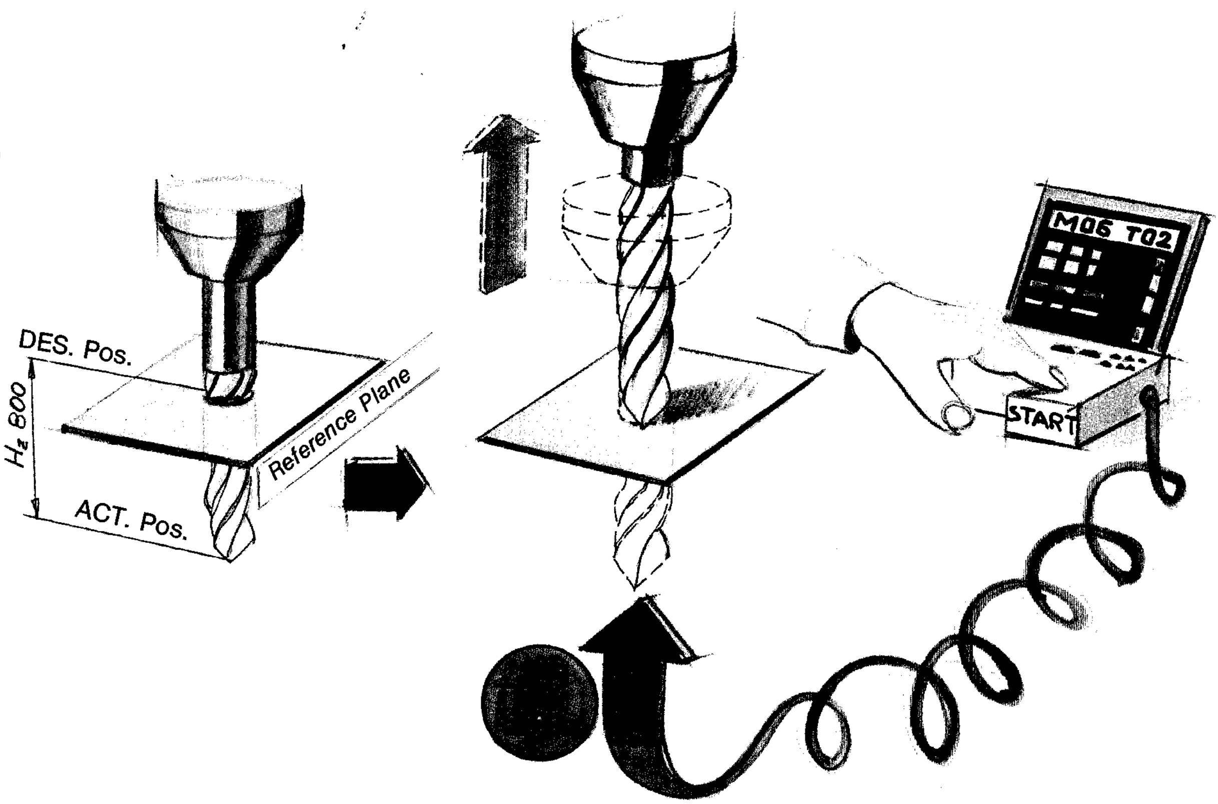
N.../M06/D500/S2000/Z(H_Z) = 800/T02

⇩

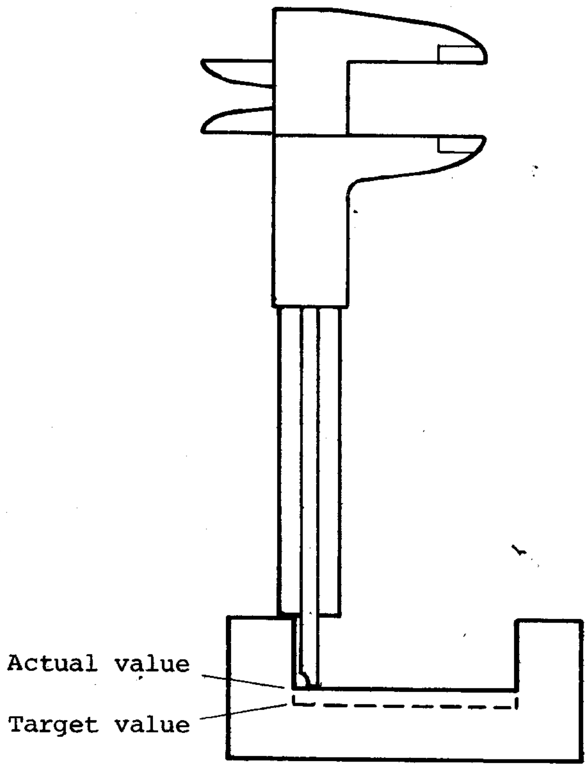
Start

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

Then the manufacture itself starts.



Tool Lengths Corrections



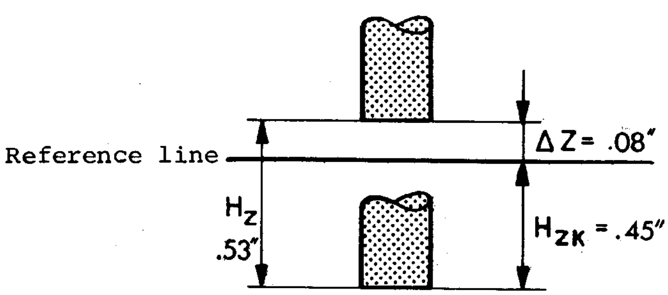
You have finished the manufacture of a workpiece and find out that the Z-measurement is not correct.

- 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

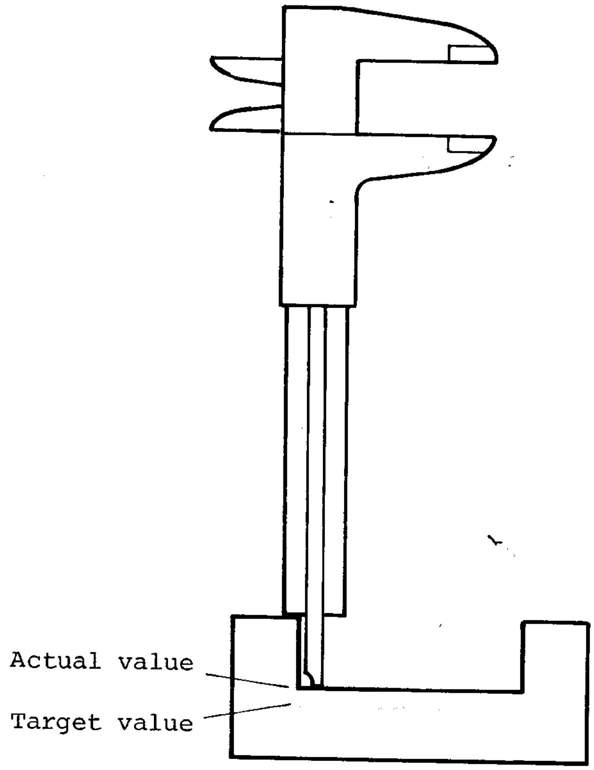
The target information Hz has to be corrected.

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

Hzk = Hz + ($\pm$ correction value $\triangle$ Z)

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

Tool Lengths Corrections



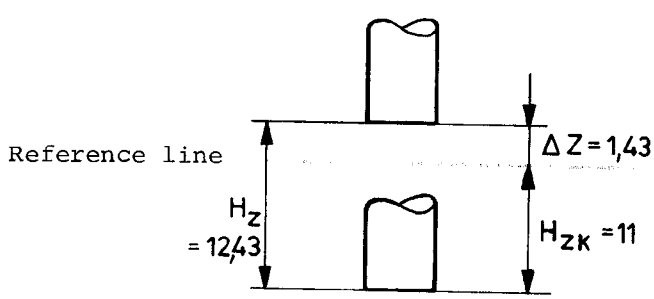
You have finished the manufacture of a workpiece and find out that the Z-measurement is not correct.

- 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

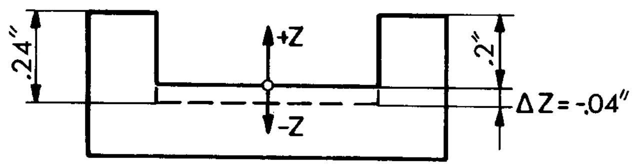
The target information Hz has to be corrected.

Hzk = Corrected target information

Hzk = Hz + ($\pm$ correction value $\triangle$ Z)

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

Example of a Correction of the Hz-value

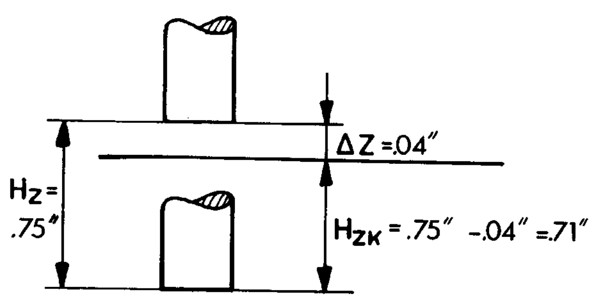


You may

1. Measure tool once again

2. Detect the correction value by measuring the workpiece.

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



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

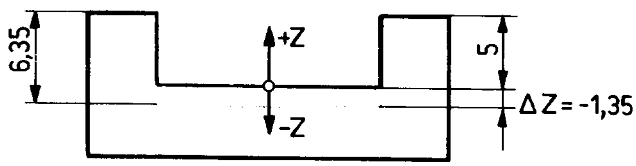
- Add the correction value △ Z to the target information Hz of the tool bit.

Pay attention: △ Z may have ± 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

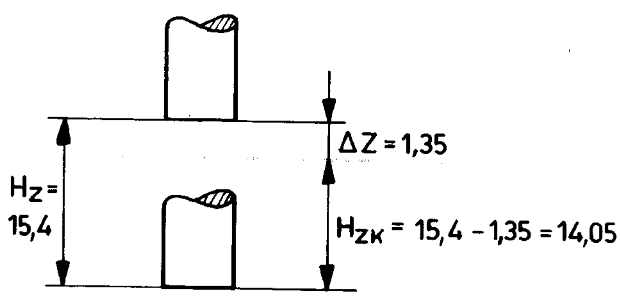


You may

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.

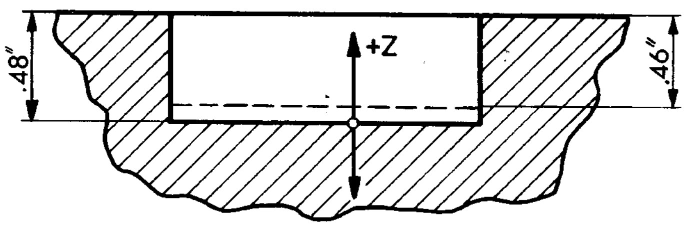
$$
\begin{aligned}
Hzk &= Hz + (\pm \ \Delta Z) \\
&= 15.4 + (- \ \Delta Z) \\
&= 15.4 - 1.35 \\
&= 14.05
\end{aligned}
$$

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)

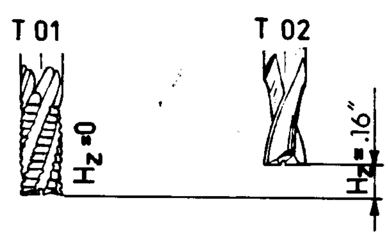
Pay attention to the sign of Δ Z.

Hzk = ...

Example

Hz of TO1 = 0

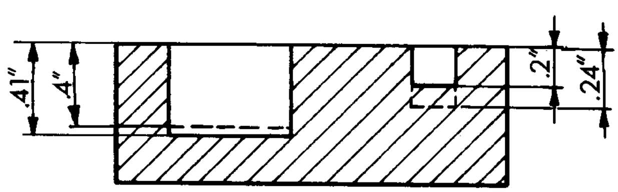
Hz of TO2 = .16″



<u>Workpiece:</u>

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

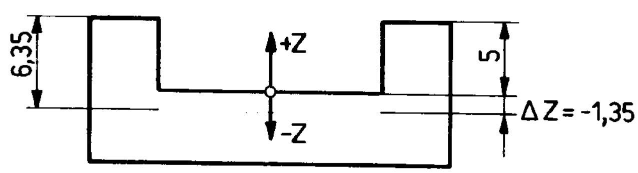
Target value TO1 = .4″
Target value TO2 = .24″



Correct the Hz-values of TO1 and TO2.

	TO1	TO2
Hzk		

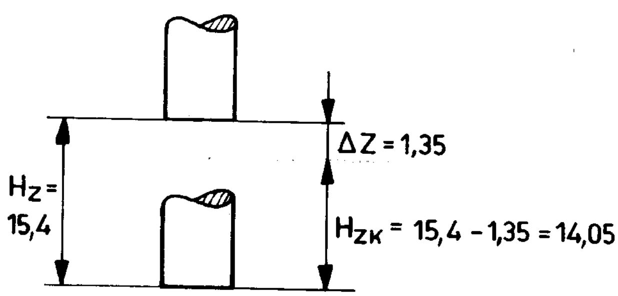
Example of a Correction of the Hz-value



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 △ Z value.

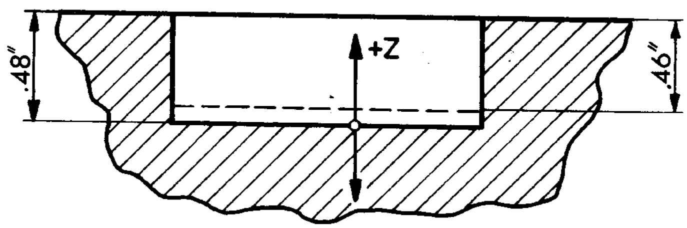


- Imagine the coordinate system transferred to the Z-actual position of the workpiece.
- Add the correction value △ Z to the target information Hz of the tool bit.

Pay attention: △ Z may have ± 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″

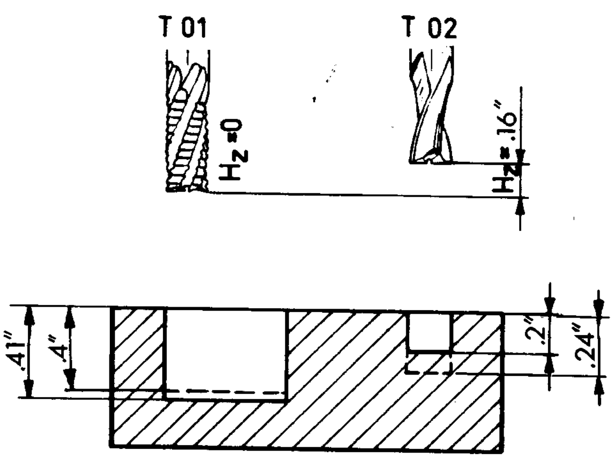
Workpiece measurements: Actual and target, compare drawing.

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″

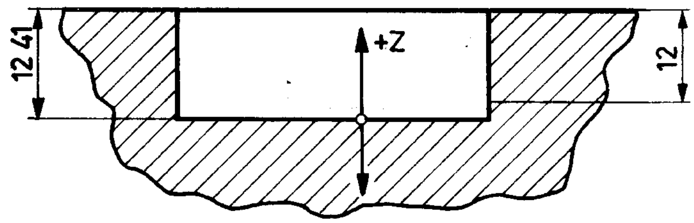
Correct the Hz-values of TO1 and TO2.

	TO1	TO2
Hzk		

Example

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

Workpiece measurements: Actual and target, compare drawing.



Correct the Hz-value

Hzk = Hz + (± △ Z)

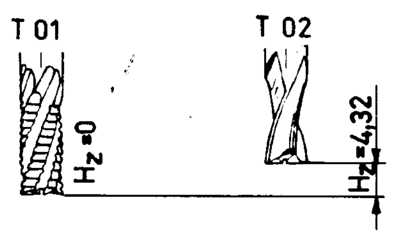
Pay attention to the sign of △ Z.

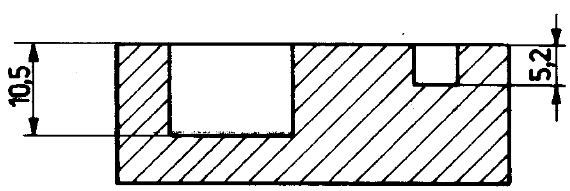
Hzk = ...

Example

Hz of TO1 = O

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.

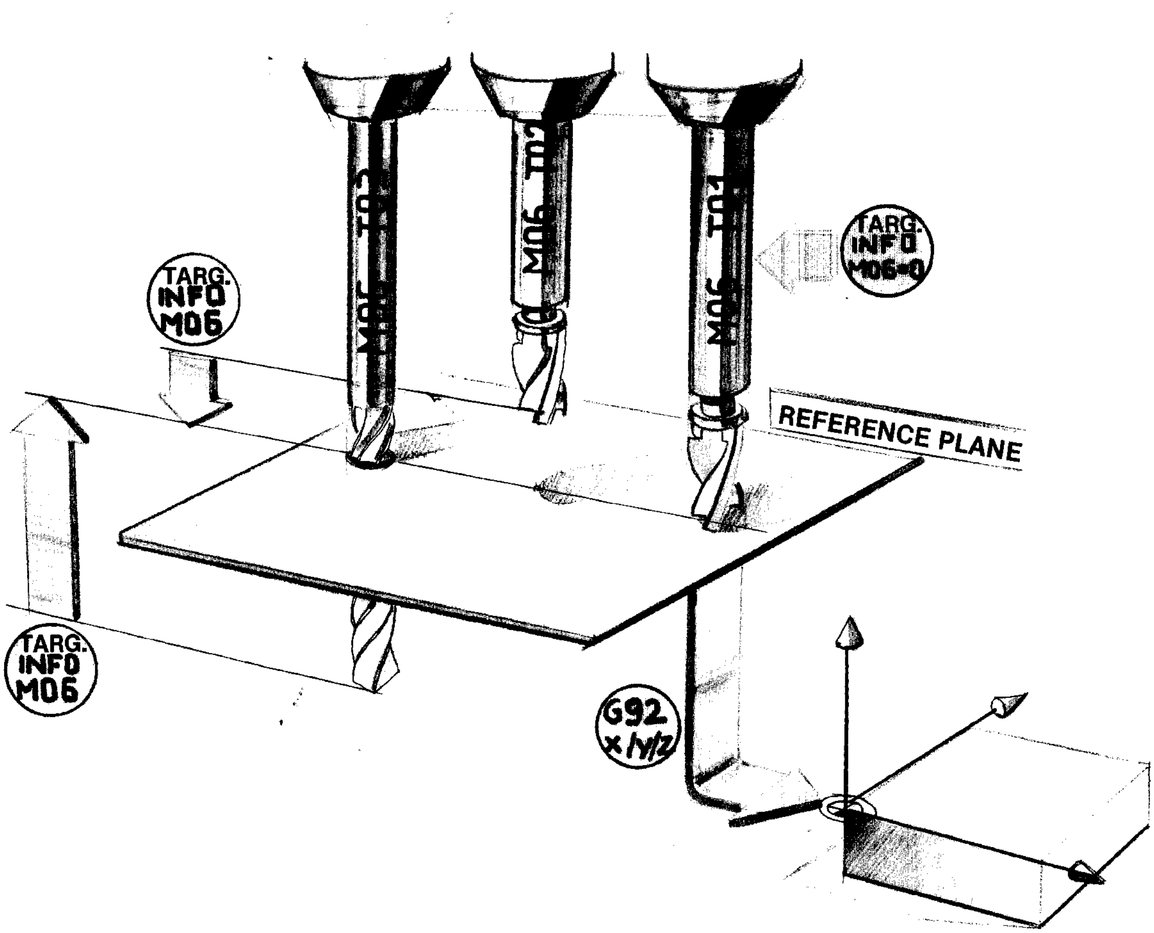
<u>Clearing offsets at end of program</u>

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.

page 6.24 is blank

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.

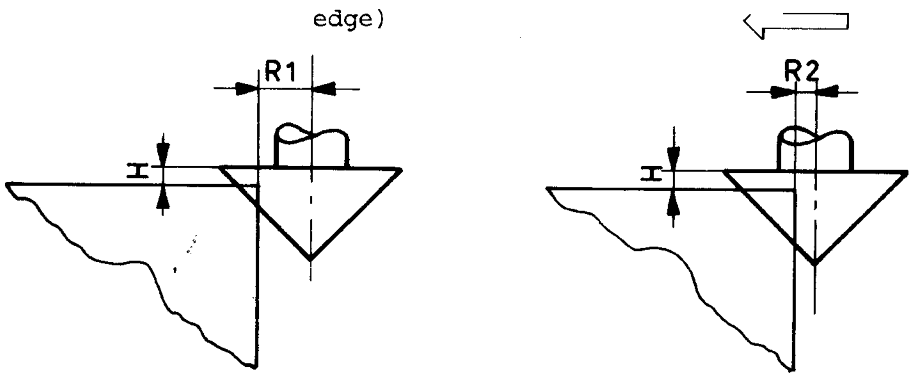
page 6.26 is blank

Milling of Chamfers

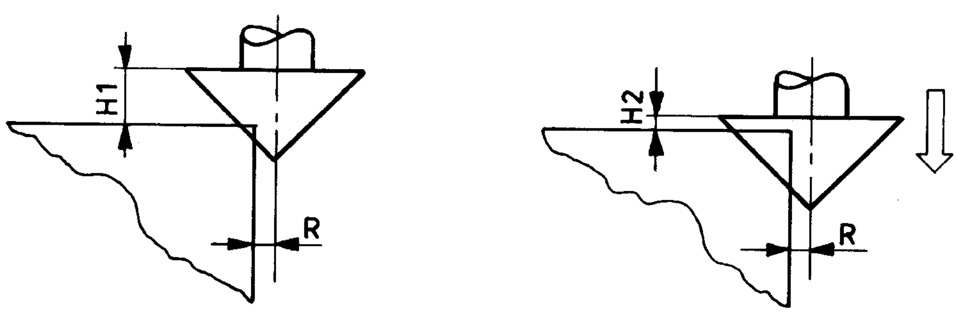
Chamfers are usually milled at an angle of 45°.

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

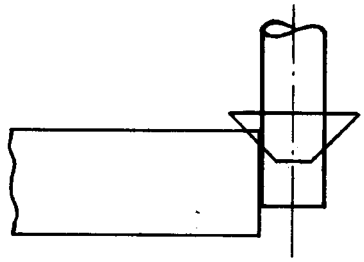
1. Chamfer size determined by different cutter paths (different distances between cutter axis and workpiece edge)



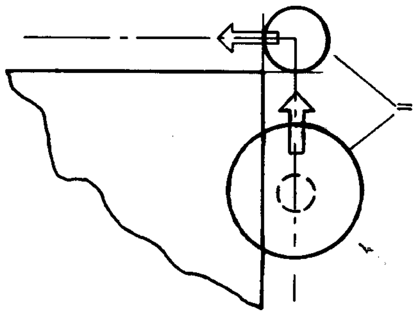
2. Chamfer size determined by different infeed and Z-direction. The cutter path remains unchanged.



Programming a Chamfer with Cutter Path unchanged



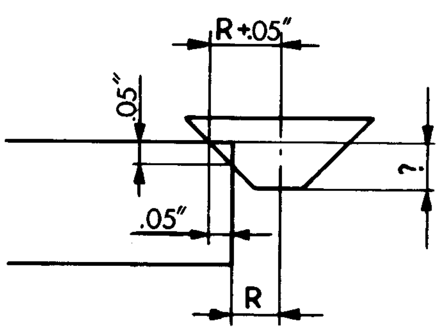
The contour is milled with a cutter of .500″ dia.
To avoid the necessity to program a new cutter path for chamferring, the angle cutter shall be programmed in Z-direction such that a chamfer .05″x.05″ is reached.



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

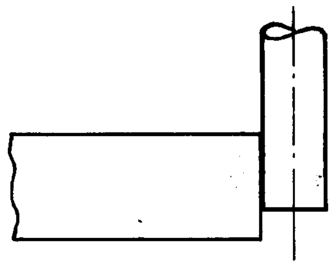
+

Width of chamfer

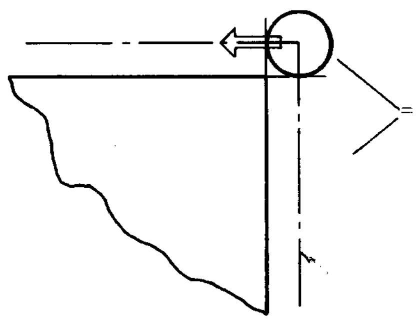
With a mill path using a .5″ shank, dia. .6″ , the radius of the angle cutter produces the chamfer .05″×45°.

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

Programming a Chamfer with Cutter Path unchanged



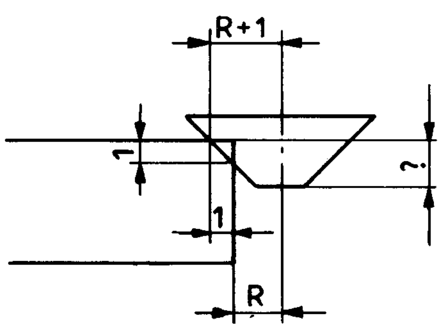
The contour is milled with a cutter of 10 mm dia.
To avoid the necessity to program a new cutter path for chamferring, the angle cutter shall be programmed in Z-direction such that a chamfer 1x1 mm is reached.



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

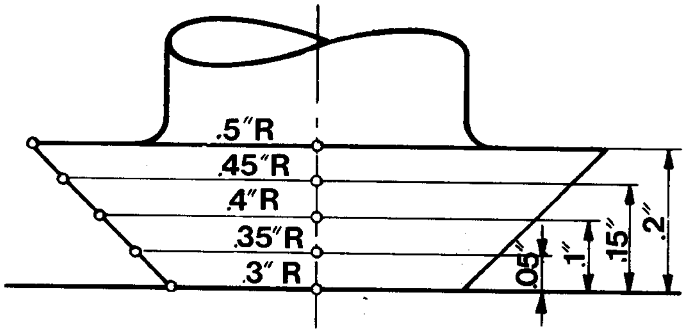
+

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

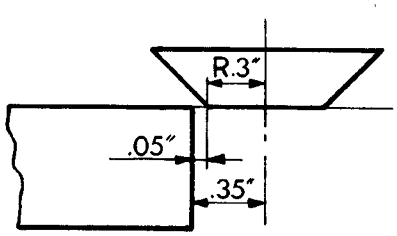
Angle cutter, dia. 1″ × .2″



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″

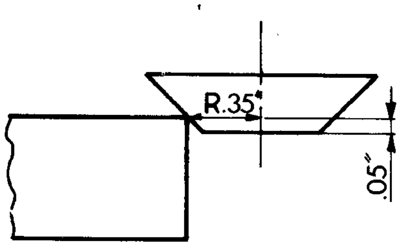


1. Cutter at height 0

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.

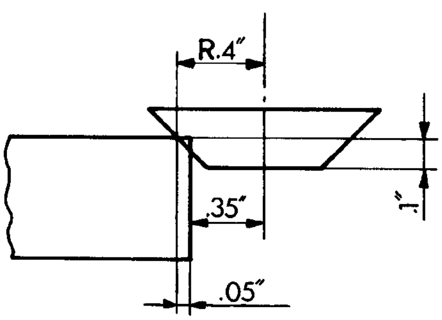
Measure of total depth:

Measure until radius mill path (.05″)

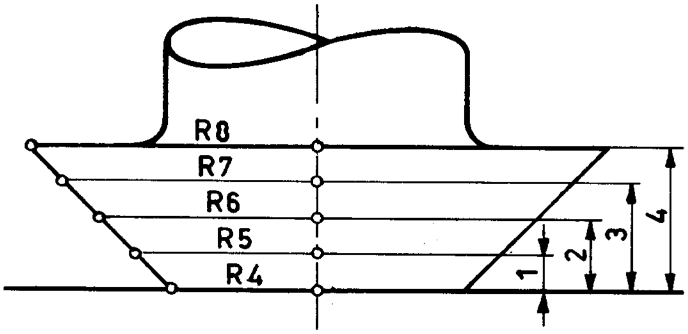
+

Width of chamfer (.05″)

= .1″



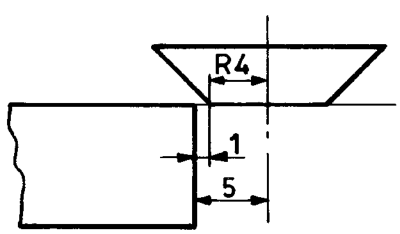
Angle cutter, dia. 16 × 4 mm



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

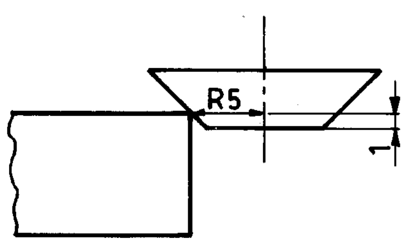
Example

Radius of mill path 5 mm



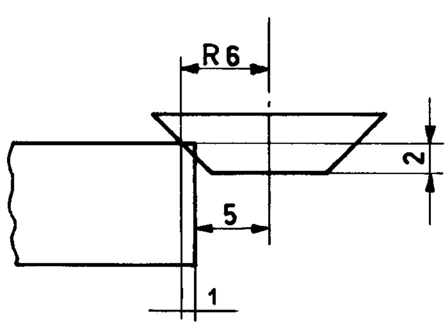
1. Cutter at height 0

Distance to workpiece = 1 mm



2. Cutter fed in by 1 mm

Radius 5 mm touches edge.



3. Cutter fed in by 2 mm

Chamfer 1x45° is produced.

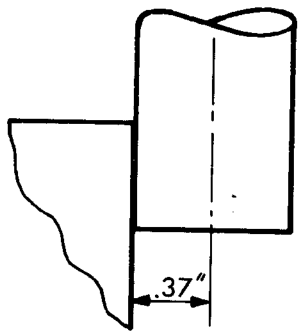
Measure of total depth:

Measure until radius mill path (1 mm)

+

Width of chamfer (1 mm)

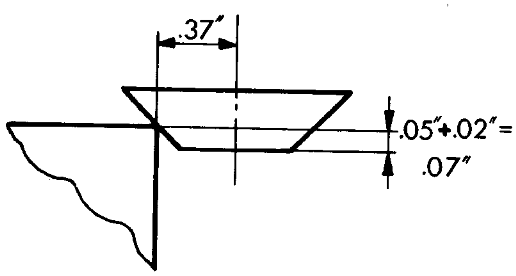
= 2 mm



Example

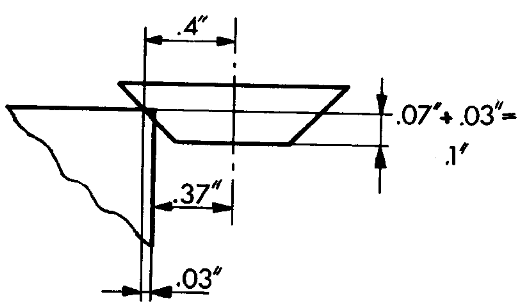
Unchanged mill path

- 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.

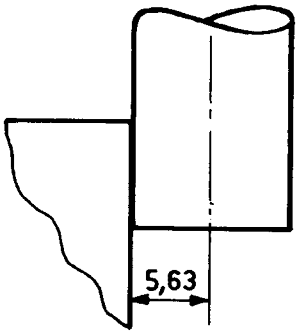
	.37″	radius cutter path
+	.03″	width of chamfer
	.4″	

Cutter infeed

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

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

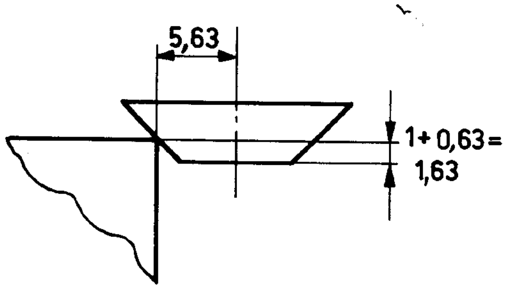
Example



Unchanged mill path

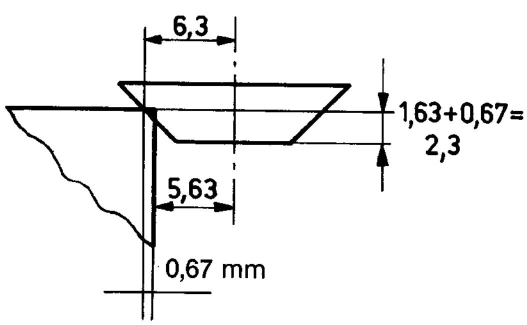
- Radius end mill: 5,63 mm

- 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



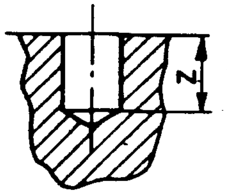
Radius 6,3 mm produces the chamfer contour.

	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



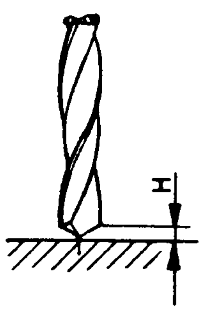
Blind holes are dimensioned down to the flat ground of the bore.

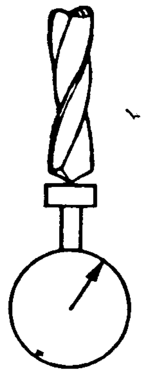
If you want to calculate the tool length you either scratch the surface with the point of the drill bit or you take measurement of the length of the tool.

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

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

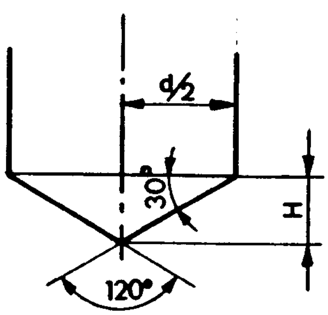
$$H = (tg30^{o}) \times \left(\frac{d}{2}\right)$$



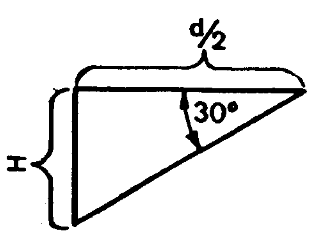


Chart

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

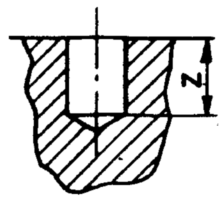


Drill Data for the Tool Sheet



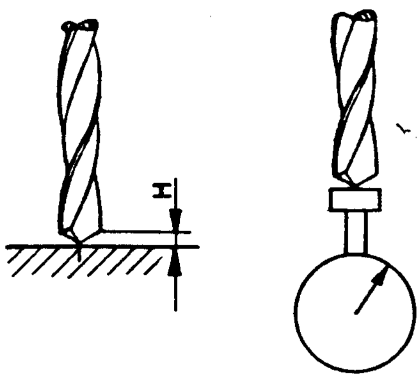
Always deduct value H from the measured data when you enter it. You need not to calculate anymore and can program the dimensions of the drawing directly.

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.



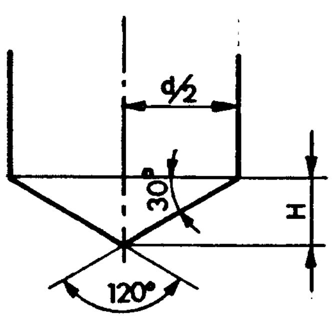
In order to program the indicated depth of bore you have to add the length of the tool point.

$$\mathrm{tg}30^{\circ} = \frac{H}{\frac{d}{2}}$$

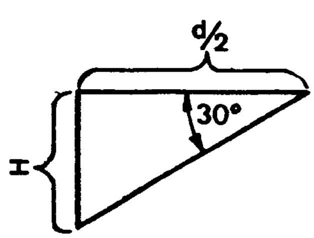
$$H = \mathrm{tg}30^{\circ} \times \frac{d}{2}$$

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



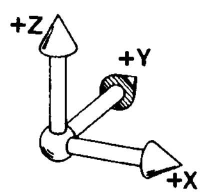
Always deduct value H from the measured data when you enter it.
You need not to calculate anymore and can program the dimensions of the drawing directly.

Tool Data Sheet (inch)

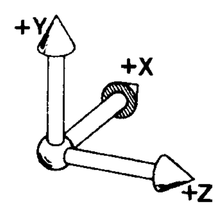
	T1	T2	T3	T4	T5	T6	T7	T8
d								
$D = \frac{d}{2}$								
F								
t								
S								
H_Z								
H_{Z_K}								

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 ____________

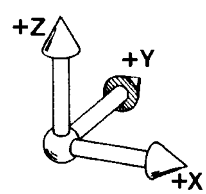
Drawing no.:
Denomination:
Workpiece material:
Program no.
Name:
Date:

Tool Data Sheet

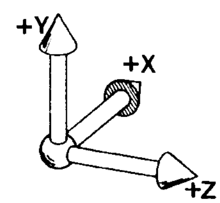
	T1	T2	T3	T4	T5	T6	T7	T8
d								
$D = \frac{d}{2}$								
F								
t								
S								
H_Z								
H_{Z_K}								

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: