Programming Instruction
EMCOTRONIC TM 02
Milling

Edition 91-4

Ref. No. EN7 766

Programming Instr.
EMCOTRONIC M2
91-4 EN7 766

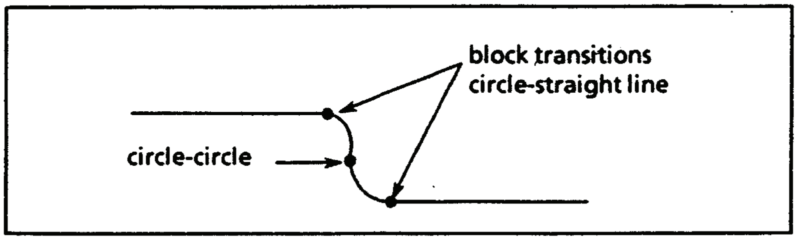
2C [redacted]

Software extension
EMCOTRONIC TM02
DC 5.00

Index

Block transitions without hold

From software 5.00 tangential block transitions to circles can be traversed without tool hold.

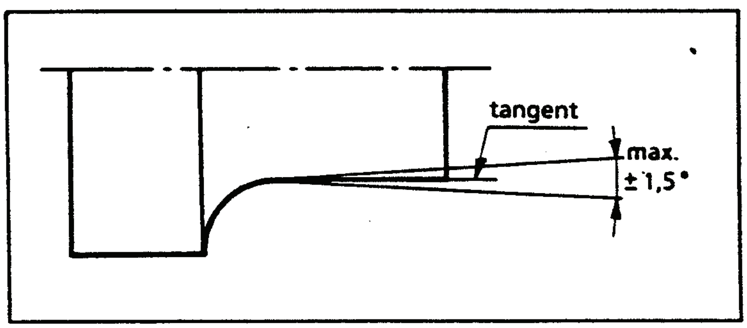


Advantage:

- Time saving
- There is no "free-cutting of the tool" at the block transition.

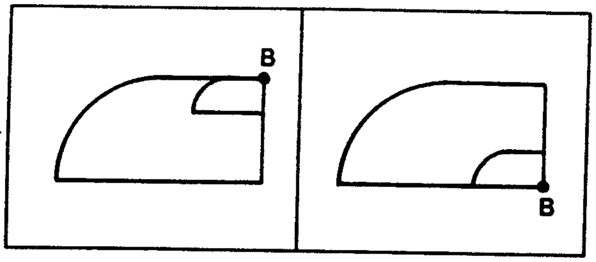
Basic conditions:

- The M39 "precise stop off" has to be active (is deselected with M38 "precise stop on").
- The block transition has to be tangential. A maximum deviation of ± 1,5° is permitted.



- The block after the transition (subsequent block) has to be programmed in the same feed. This limitation will be eliminated in the next software version.
- Both contour elements have to be in the same main plane (X-Y, X-Z or Y-Z plane).
- The feed override switch must not be actuated prior to the block transition.
- The contour elements must not be moved in rapid motion.
- If a contour element is too short or the feed is too large the control has too little time for calculating the following block transition and an exact hold is executed.

Scaling factor



A tool path can be scaled up or scaled down in a linear way from a reference point (B).

Selection of the scaling factor

N 4	G51	X U ±43	Y V ±43	Z W ±43	P_7 ±43
		[mm]	[mm]	[mm]	[ ]

Deselection of the scaling factor

N 4	G50

Data required:

1. Tool path:
   The tool path to be scaled up or scaled down is described in the program between G51 and G50. It can be opened or closed.

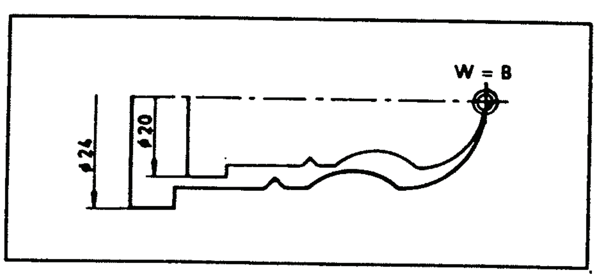
2. Reference point (B):
   The reference point is described with X,Y,Z (absolute) and U,V,W (incremental). It can be situated anywhere on or beside the contour or anywhere in the space.

3. Scaling factor (P_7):
   With P_7 the scale for scaling up or scaling down the tool path is determined.
   It can range from 0 up to ± 9999,999.
   e.g. scale1:2 ..... P_7 = 0,5
   scale1,38:1 .... P_7 = 1,38

Mind!
Thread pitches are scaled up or scaled down correspondingly.

Example 1:

You have written a program for ø 24 and only get blanks with ø 20.

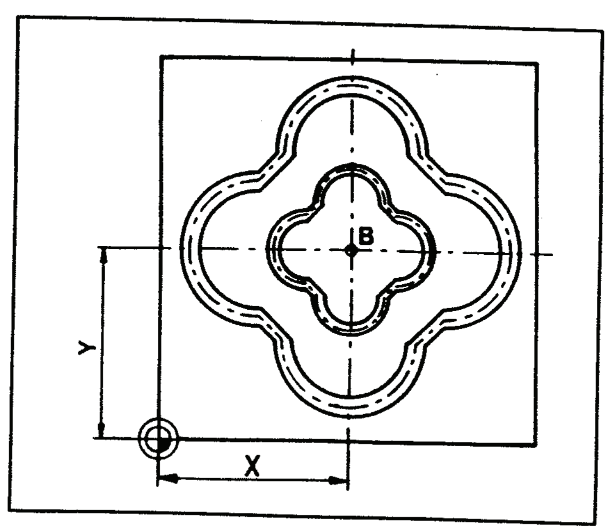


N.... G51, X = 0, Z = 0, $P_7 = \frac{20}{24}$

N..... usual program for Ø 24

N.... G50

Example 2:



Memory extension

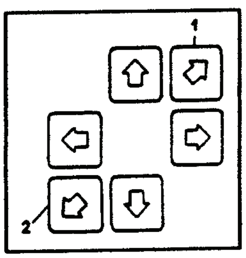
From software version DC 5.00 the Emcotronic control is provided with 64 kB RAM memory (previously 32 kB).

Reason: With the software version 5.00 additional capacity of the RAM memory was occupied (occupied RAM capacity increased from approx. 7 to approx. 12 kB).
Thus, there would be less space available than before for storing the programs.

Additional memory extension (option)
When retrofitting software version 5.00 you are therefore advised to install a memory extension.
The memory extension (ord.no. 276 110) consists of 3 memory modules for extension to 128 kB.

Listing memorized programs (key L)

Due to the memory extension more programs can be in the memory han can be listed on one screen page.



1 .. Display next page
2 .. Display previous page

For this reason the possibility was provided to turn forward and backward in the pages using the jog keys Y + and Y-.
The display can consist of a maximum of 3 pages. 258 programs can be displayed at a maximum. If there are more programs in the memory ALARM 675 "TOO MANY PROGRAMS IN THE MEMORY" is emitted.
Remedy:
Cancel programs that are not required any more.

Loading all stored programs from the machine memory onto cassette

After selecting "OUTPUT ALL" the inserted cassette is registered.

If the first cassette is full the message "INSERT NEXT TAPE" is displayed on the screen.
On the newly inserted cassette that program is started which did not have enough space on the previous cassette. If all programs are stored there is an automatic return into the EDIT mode.

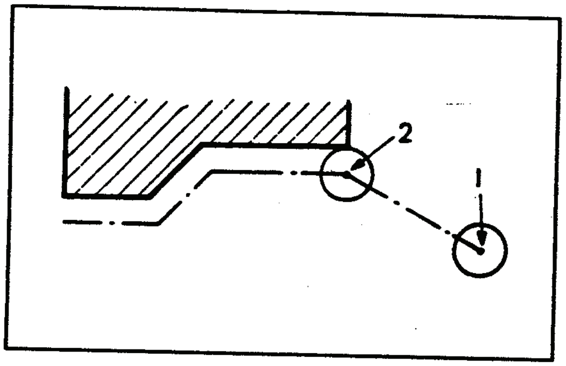
Note:
- Cassettes have to be formatted on the Emcotronic, otherwise there is an alarm and OUTPUT ALL is interrupted.
- The inserted cassette need not be empty; the programs already stored on it will be preserved.
- With the RESET key OUTPUT ALL can be interrupted, all the other keys are not active during OUTPUT ALL.

Changes in the operator monitor

From software DC 5.00 the R-parameters (machine specific data) cannot be changed by anyone in the operator monitor but only by a service engineer.

Reason:
By inexpert changes damage at the machine can be caused.

Note regarding radius compensation (G41/G42)



With G42 a "traversing command to the starting point" should be programmed, e.g the tool position (1) during selection of G42 has to be different from the starting point (2) of the radius compensation.

If this traversing command is not indicated alarm 520 may occur (error during selection/deselection of the compensation).

N.... G00 X_1 Y_1 Z_1
N.... G42 $\underbrace{G00\ X_2\ Y_2\ Z_2}$

traversing command
to the starting point

The dividing attachment (M27)

From software DC 5.00 the dividing attachment TARM HW 125 (Messrs. Walter) can be controlled.

Function:

- Adjust divisions to be carried out (min. 15°) at the control of the dividing attachment.
- The starting command for the division is carried out by M27 in the NC program.
- Subsequently there is a response from the dividing attachment to the machine control and the NC program continues to be executed.

An exact description, installation and connection instructions are to be found in the operating instructions (ord.no. F5Z 140 030) enclosed to the dividing attachment.

Bar loading magazine (M65)

From software DC 5.00 the bar loading magazine LM 1000 of Messrs. KUPA can be controlled.
With M65 the execution of the NC program is stopped until the response is emitted by the bar loading magazine.

Selection required in the operator monitor

Activate M65:
L39 Bit 1 = HIGH ...... M65 selected
L39 Bit 1 = LOW ...... M65 deselected

Activate bar loading magazine:
L25 Bit 2 = HIGH ...... bar loading magazine selected
L25 Bit 2 = LOW ...... bar loading magazine deselected

Notes:

- In your NC program you must not use skip blocks (SKIP) since these are used for bar change.

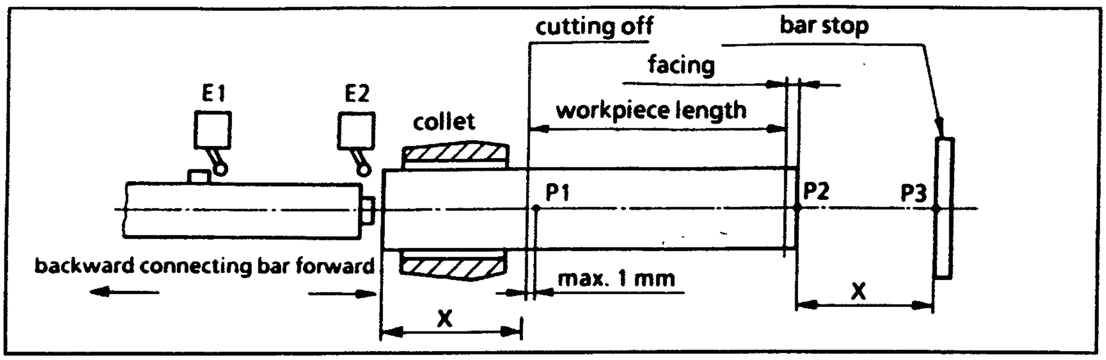
- M65 must not be programmed with active cutter radius compensation.

- The bar loading magazine starts pushing forward the bar if the clamping device is opened when the program is operative (CYCLE START active). You are therefore advised to start NC programs only with closed clamping device since otherwise immediately after pressing CYCLE START the bar is pushed out although it is not provided that the stop is swivelled in or that the tool turret is in the correct position.

- Alarm of the bar loading magazine:
  1. Red led blinks once/sec = material supply off
  2. Red led blinks twice/sec = irregular state

  Remedy: Eliminate state and switch the bar loading magazine on and off.

Functional description



P1 ... Pick-up position of the bar
P2 ... Machining position of the bar
P3 ... Ejecting position of the remaining bar
E1 ... Limit switch 1 for bar end
E2 ... Limit switch 2 for remaining bar ejection

} limit switches have to be adjusted correspondingly

Subprogram for "loading"

T..	Swivel in bar stop Note: The bar stop is clamped in the tool turret. It is absolutely necessary to use a spring-type bar stop.
G00	Move bar stop to P1. Explanation: If the bar stop immediately moves to P2 the bar would be provided with a too high impact speed due to the long approach travel.
M25	Open clamping device, with M25 the connecting rod is activated at the same time and pushes the bar forward.
G04	Dwell time (2 sec) until the bar has reached the bar stop.
G01	Move bar stop to P2 (G94, F3000). The bar is pushed by means of the connecting bar.
M65	Control waits for signal of the bar loading magazine. If the bar is at the bar stop (point P2) a limit switch provides a signal.

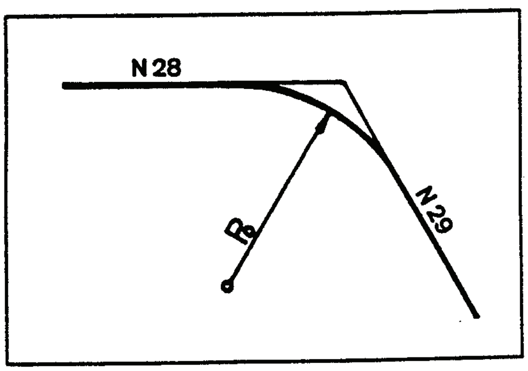
Explanation for the following six blocks:

If the bar is too short for a new workpiece the connecting bar overtravels limit switch 1, a signal is emitted to the Emcotronic thus inactivating "SKIP" until program end M30. (The skip blocks designated with "/" are executed.)

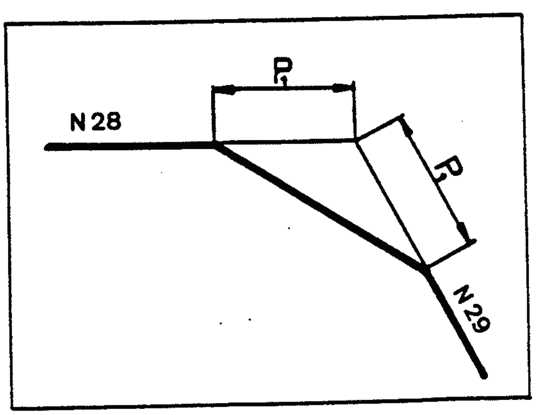
/G00	Move bar stop to P3 The connecting bar ejects the rest of the bar and overpasses limit switch 2. Thus, the connecting bar travels into the rear final position and initiates a bar change.
/M65	Waiting for signal of bar loading magazine: Bar is at the bar stop (point P3).
/G01	Move bar stop to P2 (G94/F3000)
M26	Close clamping device, connecting bar returns back.
M65	Waiting for signal of bar loading magazine: Connecting bar has reached rear final position.
G00	Move to tool change position
M17	Subprogram end

Software Extension
Emcotronic TM 02
DC 5.10

Inserting of chamfers and radii



N28 G01 X.. Z.. F.. P_0..
N29 G01 X.. Z.. F..



N28 G01 X.. Z.. F.. P_1..
N29 G01 X.. Z.. F..

MILLING

N 4	G01	X..	Y..	Z..	P_0 P_1 43

TURNING

N 4	G01	X..	Z..	P_0 P_1 43

- A radius or a chamfer can be inserted between two straight lines (e.g. block N28 and N29)
- A radius is defined with parameter P_0 [mm].
- A chamfer is defined with parameter P_1 [mm]. The chamfer is laid symmetrically into the edge, i.e. length P_1 is identical on both enclosing straight lines.
- P_0 and/or P_1 is attached at the first of the two enclosing blocks (N28).

Conditions

- The length P_1 of an inserted chamfer must not be longer than the shorter of the enclosing straight lines, otherwise it would not result in an intersecting point.

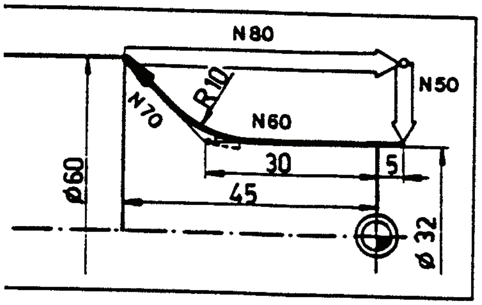
- For the calculation of the chamfer and/or the radius the block in which the chamfer and/or the radius is programmed as well as the subsequent block is necessary.

  In these blocks no PSO change, no tool change and no scaling change must be carried out.

  In the execute mode the subsequent block is not available, therefore chamfers and radii cannot be programmed with P_0 and/or P_1.
- The block in which the chamfer and/or the radius is programmed must contain exactly two position parameters (X+Y, X+Z, Y+Z).

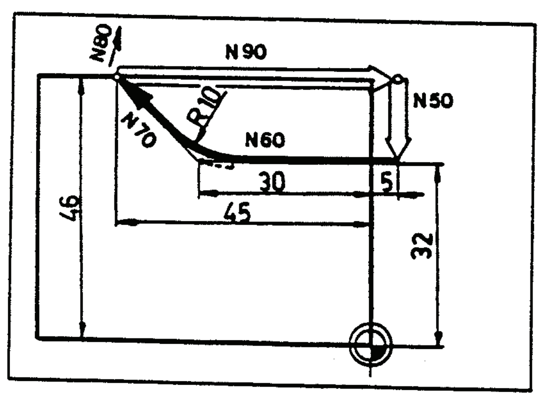
Examples:

NING



)/ G00 / X32.000
)/ G01 / X32.000 / Z-30.000 / F... / P0 = 10.000
)/ G01 / X60.000 / Z-45.000 / F...
)/ G00 / Z5.000

MILLING



.
.
.
N0050 / G00 / Y32.000 / Z-15.000
N0060 / G01 / X30.000 / Y32.000 / F... / P0 = 10.000
N0070 / G01 / X-45.000 / Y46.000
N0080 / G01 / Z1.000
N0090 / G00 / X5.000 / Y46.000

Chapter 1

Technical data, Summary

Chapter 2

General remarks on programming, program structure, syntax etc.

Chapter 3

The reference points of the CNC machine, zero point offsets

Chapter 4

Tool (length) compensation

Terminology - tool compensation
Tool compensation can mean two different things: tool (length) compensation and tool (path) compensation.

Tool (length) compensation:
This means the input and compensation of the tool length.

Correction of tool compensation:
If you see from the milling result that the lengths are wrong (with correct programming), you must correct the (length) compensation value.

Tool (path) compensation:
See G40/G41/G42

Chapter 5

The M-functions

Chapter 6

The G-functions (Path conditions)

* The G-functions are subdivided into groups (see summary 1.4)
* G-functions from the same group cancel each other out.
* If two G-functions of the same group are in one block, the G-funktions last progrmmed (entered) is effective.

Chapter 7

Alarms

Chapter 8

User Monitor EMCOTRONIC TM 02

Chapter 9

Interface RS 232 EMCOTRONIC TM 02

Chapter 1
Technical data, Summary

Technical data
EMCOTRONIC TM 02

Microprocessor 3-axis continuous-path control
Linear and circular interpolation (2 1/2 D)
Program memory for 20 kB (about 110 m of punched tape)

Actual position
Remaining travel path
Spindle speed
Tool compensation
Feed
Other parameters
12" monochrome screen

Input accuracy	0.001 mm (0.0001")
Output accuracy	**≥ = 0.001 mm (0.0001 inch) (set conform to step resolution of the concerning machine, see "Technical Data of the machine")**
Thread pitches	0.01 - 10 mm
Feed override	0 - 120%
Spindle speed override	50 - 120 %
Interpolation range	+/- 9999.999 m
Tool memory	99 tools

Modes
Manual mode (manual traversing of the slides)
Execute (processing of the input memory)
Edit (program input via keyboard, interfaces, tool data and position offset registers, user monitor)
Automatic (execution of the NC programs)

Submodes
Single block, skip block, dry run, reference point, status, change tool

Program format
Structure to DIN 66025
Decimal point input

Permanent program memory for machine data, tool data registers and workpiece programs, position shift offsets.

Data input/output
RS 232 C interface (V24 and 20 mA), 150 - 4800 bd
Cassette recorder (Philips MDCR) 600 characters/sec. (corresponding to 6 kbaud)

Subject to technical modifications and supplements!

The addresses

O Program number (00-6999) (7000-9999 reserved for graphics)

N Block number (0000-9999)

G Path functions (00-99)

G00 = Rapid traverse (positioning behaviour)
G01 = Linear interpolation
G02/03 = Circular interpolation
G04 = Dwell
G25 = Subroutine call
G27 = Unconditional jump
G33 = Thread-cutting in single block
G40 = Cancellation of the cutter path compensation
G41 = Cutter path compensation left
G42 = Cutter path compensation right
G53 = Delete position shift offsets 1 and 2
G54 = Position shift offset 1
G55 = Position shift offset 2
G56 = Delete position shift offsets 3, 4 and 5
G57 = Position shift offset 3
G58 = Position shift offset 4
G59 = Position shift offset 5, also changeable in the program
G70 = Measurements in inches
G71 = Measurements in mm
G72 = Definition of circular boring pattern
G73 = Call-up circular boring pattern
G74 = Definition rectangular boring pattern
G75 = Call-up rectangular boring pattern
G81 = Drilling, centering
G82 = Drilling, spot-facing
G83 = Deep-hole drilling with retraction
G84 = Tapping
G86 = Deep-hole drilling with chip breaking
G87 = Rectangular pocket milling cycle
G88 = Circular pocket milling cycle
G89 = Slot milling cycle
G92 = Set memory
G94 = Specification of the feed speed in mm/min (inch/min)
G95 = Specification of the feed rate in mm/rev. (inch/rev.)
G98 = Withdrawal to starting plane
G99 = Withdrawal to withdrawal plane

X,Y,Z Absolute coordinates

U,V,W Incremental coordinates

I,J,K Interpolation parameters

P0..P7
Auxiliary parameters
D0...D7

F Feed in mm/min, µm/revolution
Thread pitch in µm

S Spindle speed/spindle position for M19

T Tool call-up, selection of tool compensation (four digits)

L Subroutine number/repetitions (four digits)
Jump target

M (00-99) Additional functions
M00 Programmed stop
M03 Spindle clockwise direction
M04 Spindle counterclockwise direction
M05 Spindle stop
M08 Coolant on
M09 Coolant off
M17 Subroutine end
M19 Spindle precise stop
M30 Program end with return to program start
M38 Precise stop on
M39 Precise stop off
M50 Cancellation of the direction logic of the tool turret
M51 Selection of the direction logic of the tool turret
M90 Cancellation of the mirroring function
M91 Mirroring on the X-axis
M92 Mirroring on the Y-axis
M93 Mirroring on the X and Y-axes

Permanent program memory for machine data, tool data registers and workpiece programs, position shift offsets and automatic approaching of the reference point.

Data input/output
RS 232 C interface (V24 and 20 mA(), 150 - 2400 bd
Cassette recorder (Philips MDCR) 600 characters/sec.
(corresponds to 6 kbaud)

Subject to technical modifications and supplements!

Group structure and initial status of the G-functions

Group 0		G00:	Rapid traverse
		G01:	Linear interpolation
		G02:	Circular interpolation clockwise
		G03:	Circular interpolation counterclockwise
	*	G04:	Dwell
		G72:	Definition circular boring pattern
		G74:	Definition rectangular boring pattern
		G81:	Drilling, centering
		G82:	Drilling, spot-facing
		G83:	Deep-hole drilling with retraction
		G84:	Tapping
		G86:	Deep-hole drilling with chip breaking
		G87:	Rectangular pocket milling cycle
		G88:	Circular pocket milling cycle
		G89:	Slot milling cycle
Group 2		G94:	Feed in mm/min or 1/100 inch/min
	**	G95:	Feed in µm/rev. or 1/10000 inch/rev.
Group 3	**	G53:	Cancellation of offsets 1 and 2
		G54:	Call-up of offset 1
		G55:	Call-up of offset 2
Group 4	*	G92:	Set offset 5
Group 5	**	G56:	Cancellation of offsets 3,4,5
		G57:	Call-up of offset 3
		G58:	Call-up of offset 4
		G59:	Call-up of offset 5
Group 6		G25:	Subroutine call-up
		G27:	Unconditional jump
Group 7	□	G70:	Measurements in inches
	□	G71:	Measurements in mm
Group 8	**	G40:	Cancellation of the tool path compensation
		G41:	Cutter path compensation left
		G42:	Cutter path compensation right
Group 9	□ **	G17:	1st axis switching
	□	G18:	2nd axis switching
	□	G19:	3rd axis switching
	□	G20:	4th axis switching
	□	G21:	5th axis switching
	□	G22:	6th axis switching
Group 11	**	G98:	Withdrawal to starting plane
		G99:	Withdrawal to withdrawal plane
Group 12		G73:	Call-up circular boring pattern
		G75:	Call-up rectangular boring pattern

* effective blockwise
** initial status
□ Initial status can be established in the user monitor (MON) mode.

Group structure and initial status of the M-functions

Group 0	 * 	M03: Spindle ON in clockwise direction M04: Spindle ON in counterclockwise direction M05: Spindle stop M19: Spindle precise stop
Group 1	 **	M38: Precise stop ON M39: Precise stop OFF
Group 2	* * *	M00: Programmed STOP M17: Subroutine end M30: Program end with return to program start
Group 3	 **	M08: Coolant ON M09: Coolant OFF
Group 8	□ □ 	M50: Cancellation of the direction logic with bidirectional tool turret M51: Selection of the direction logic with bidirectional tool turret
Group 10		M90: Cancellation of the mirroring function M91: Mirroring on the X-axis M92: Mirroring on the Y-axis M93: Mirroring on the X and Y-axes

* effective blockwise
** initial status
□ Initial status can be established in the user monitor (MON) mode.

<u>Note:</u> The implementation of the individual M-functions depends on the hardware of the machine in question.

Addresses and their input dimensions

Addresses	metric	inch
Path addresses absolute X,Y,Z	+/- (mm)	+/- (inch)
Path addresses incremental U,V,W	+/- (mm)	+/- (inch)
Circular interpolation parameters I,J,K	+/- (mm)	+/-(inch)
1. F thread pitch (G84)	(µm)	(1/10000 inch)
2. F Feed per minute (G94)	(mm/min)	(1/100 inch/min)
3. F Feed per revolution (G95)	(µm/rev.)	(1/10000 inch/rev.)
1. S speed programming	(rpm)	(rpm)
2. S angle position (M19)	(°)	(°)

The P-parameters in the program

Possible input: 0 - +/- 10000,000

Parameter		Default option
P_0	G72: Circle diameter (mm)	
	G74: Horizontal distance (mm)	
	G87: Pocket length in X (mm)	
	G89: Length of the slot (mm)	
P_1	G74: Vertical distance (mm)	
	G87: Pocket length in Y (mm)	
	G88: Pocket diameter (mm)	
	G89: Width of the slot (mm)	
P_2	NOT USED	
P_3	G81, G82, G83, G84, G85, G86,G87 G88, G89 Definition of the withdrawal plane (mm) Absolute from the workpiece zero point	
P_4	G81, G82, G83, G84, G85, G86, G87, G88, G89 Definition of the withdrawal plane (mm) Incremental from the starting plane	
P_5, P_6, P_7	NOT USED	

The D-parameters in the program

Possible input: 0 - 32,767

Parameter		Default option
D_0	G72: Number of boring pattern elements ( )	
	G74: Horizontal number of boring pattern elements	
D_1	G74: Vertical number of boring pattern elements	
D_2	G72: Starting angle (° x 10)	0
	G88: Horizontal infeed ( )	D_2 = 1.7 x cutter radius
	G89: Angle of slot related to X-axis (° x 10)	0
D_3	G72: Total angle (° x 10)	3600
	G83: Infeed per cut (µm)	no cut segmentation
	G86: Infeed per cut (µm)	no cut segmentation
	G87: Infeed per cut (µm)	no cut segmentation
	G88: Vertical infeed ( )	machining peformed with one infeed
	G89: Vertical infeed ( )	machining performed with one infeed
D_4	G04: Dwell (1/10 sec.)	no dwell
	G82: Dwell (1/10 sec.)	no dwell
	G85: Dwell (1/10 sec.)	no dwell
	G86: Dwell (1/10 sec.)	no dwell
	G88: Finishing parameter ( )	D_4 = 1
	G89: Finishing parameter ( )	D_4 = 1
D_5	G83: Percentage reduction (%)	0
	G86: Percentage reduction (%)	0
	G87: Climb/conventional mill ( )	D_5 = 3
	G88: Climb/conventional mill ( )	D_5 = 3
	G89: Climb/conventional mill ( )	D_5 = 3
D_6	NOT USED	
D_7	G72: Take-over of parameters ( )	0
	G74: Take-over of parameters ( )	0
	G87: Type of vertical infeed ( )	1
	G88: Type of vertical infeed ( )	1
	G89: Type of vertical infeed ( )	1

Observe the D-parameters in the user monitor (MON).

Chapter 2

General remarks on programming, program structure, syntax etc.

The CNC-Program
The Program Set-up

A CNC program contains all instructions and informations necessary for the production of a workpiece.

It consists of:
- program number
- NC-blocks and
- program end information

The Program numbers:

O 28
N 00 00
N 00 10
N 00 20

Each program has to start with a program number.

Address: Letter O

Possible program numbers: O 00 to O 99

The Program Blocks/NC-Blocks

Address: N

Block numbers: N 0000 to N 9999

It is advisable to number the blocks in increments of 10. In this way blocks can be added later.
The control automatically suggests a 10-step system of blocks.

The Words:

N 0010/X 20./Y 10./Z 5./F...

The block consists usually of various words.

The Word

Address — Combination of figures

$\boxed{\text{G 01}}$ — Word

The word consists of a letter (the address) and a combination of figures. Each address (letter) has a specific meaning.

Block Rules:

If two or more G- or M-codes of the same group are in one block (no sense), the code programmed last is valid.

Decimalpoint Programming

X,Y,Z,U,V,W,P_0,P_1,P_3,P_4,I,J,K values must be programmed with the decimalpoint.

The values would be calculated as µm (with G71) resp. as 1/10 000 inch (with G70). Leading and following zeros need not be programmed.

Modal Codes and Word Contents

Compare the group structure of G- and M-Codes.

The programming work should be as simple as possible; thus self-holding codes.

1) Take-over of G-, M-Codes

General:

Self-holding codes and instructions remain valid in the program until they are called off.

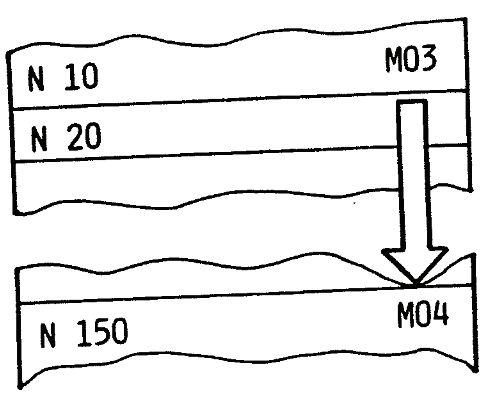
Call off:

1. By instruction of the same group, e.g. G00 is called off with G01.
2. Call off instruction:
   For some codes there are special call off instructions, e.g. G41 is called off with G40.

N 100	G00	X-20.	Y 5.	Z 40.
N 110	G00	X 10.	Y 5.	Z 5.
N 120	G01	X 10.	Y 5.	Z- 5.

Example:

In block N 110 no programming of G00 necessary.



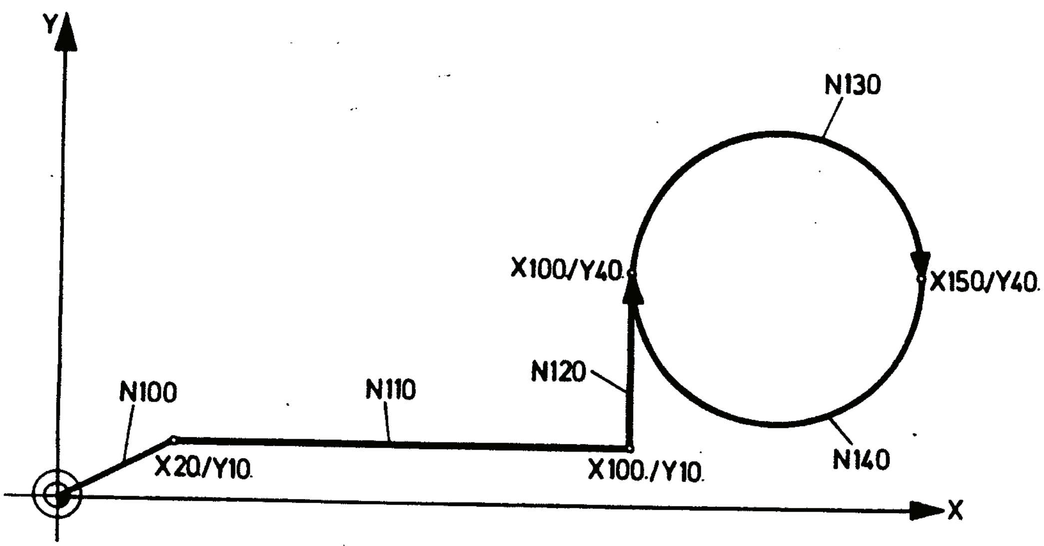
Example:

M03 is called off by M04. In blocks from N20 to N 140 no programming of M03.

2) Take-over of Modal Word Contents

Purpose: Simplify programming

Constant word contents of X,Y,Z,F,S,T are taken-over in the subsequent blocks.



N 100	G00	X 20.	Y 10.	Z-5.				S 2000	T0101
N 110	G01	X 100.	Y 10.	Z-5.			F 100	[redacted]	[redacted]
N 120		X 100.	Y 40.	Z-5.			F 100	[redacted]	[redacted]
N 130	G02	X 150.	Y 40.	Z-5.	(I 25.)	J 0.	F 100	[redacted]	[redacted]
N 140		X 100.	Y 40.		(I -25.)	J 0.	F 100	[redacted]	[redacted]
N 150									
N 160									

3) Take-over of Modal Parameters with Drilling and Milling Cycles

Purpose: Simplify programming

		X	Y	Z	D_3	D_5	D_6
N 110	G83	....	....	....	....	....	....
N 120	G83	....	....	....			
N 130	G83	....	....	....			
N 140	G86	....	....	....	D_3	D_5	D_6

As long as the same G-codes are valid, the parameters and their values are taken-over into the next blocks.

As soon as another G-code (e.g. G00/G01/G86) of group zero is called up, the parameters are erased.

The Initial Status of the EMCOTRONIC TM 02

The initial status is determined by the manufacturer of the control.
This is done for reason of practical operation and safety.

Example:

M05: When switching on the control the main spindle must not run up.

☐ G71: Since everywhere (exception in the USA) they program in mm, the initial status is fixed as in mm.

Initial Status EMCOTRONIC TM 02

The following codes are valid when switching on and need not be programmed anymore.
They are also indicated in operation mode STATUS.

G-Codes:

G40 Neutralization of the cutter tool correction
☐ G71 Measurements in mm
G53, G56 Position shift offset erased
G94 Data of feed speed in mm/min - inch/min

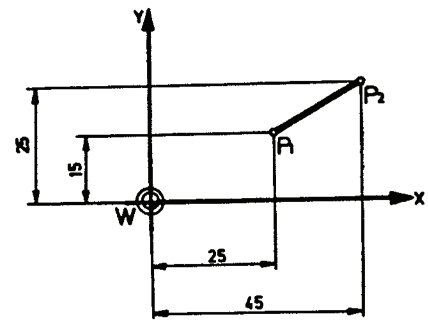
G98 Withdrawal to starting plane

M-Codes:

M05 Spindle stop
M09 Coolant off
M39 Precise stop off

☐ Can be changed by the customer in mode MON, G71 to G70.

Absolute and Incremental Value Programming



Absolute Value Programming

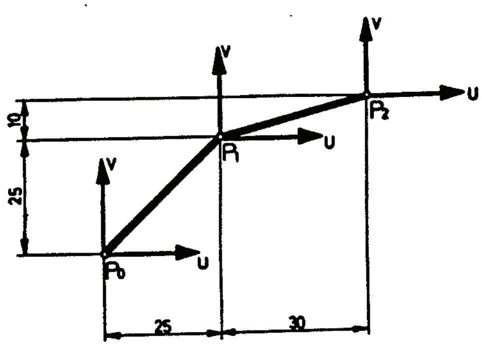
The description runs under the addresses

X, Y, Z

The X,Y,Z data always relate to the actual origin of the coordinates system.

P 1 : X = 25/Y = 15

P 2 : X = 45/Y = 25



Incremental Value Programming

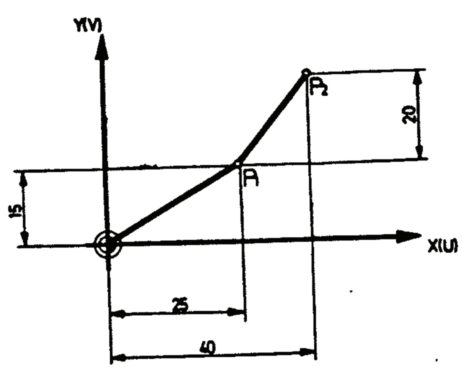
The description runs under the addresses

U, V, W

The U,V,W addresses refer to the starting point of each block.

P 1 : U = 25/V = 25

P 2 : U = 30/V = 10



Mixed Programming

The programming can also be mixed.

P 1 : X = 25/Y = 15

P 2 : X = 40/V = 20

Note

The programming of
G90: X,Y,Z data absolute
G91: X,Y,Z data incremental
is not necessary with the EMCOTRONIC TM 02 control. X,Y,Z are automatically absolute; U,V,W incremental.

G-Codes, their Formats and Description of Formats

Specific addresses are asigned to most G-Codes.

Example:

G00/X±....,.../Y±....,.../Z±....,...
or
G01/X±....,...Y±....,.../Z±....,.../F....

For a short and easy to understand description of pertaining addresses (format description) the data are encoded.

Code:

1) Instead of giving the possible inputs, the number of decades is given.

   Example:

   Instead: N from 0 to 4000
   or N .... we write N4.

   $N\underbrace{....}_{4} \Rightarrow \underline{N4}$

2) The specification of the possible decades before or after a decimal point is coded with two figures.

   $X\underbrace{....}_{4},\underbrace{...}_{3} \Rightarrow X43$

   The first figure: Decade before decimal point
   The second figure: Decade after decimal point

3) If the values could be negative or positive a + sign is written between address and number.

   X + 43

   Remark:
   For better determination quite often a ± sign is written (X±43).

Example:

N4	G01	X±43 / U±43	Y±43 / V±43	Z±43 / W±43	F4

N4: Four digits without decimalpoint and sign.

X Y Z ± . . . . , . . .
U V W . . . . , . . .

± sign possible	Four digits before decimal point	Three digits after decimal point

F4: Four digits without decimal point and sign.

Example:

N4	G02 / G03	X±43 / U±43	Y±43 / V±43	Z±43 / W±43	J±43	K±43	F4

Example:

N4 / G04 / D_4 5

D_4 5 Five digits without decimal point and sign.

For maximum values and input dimensions (mm or µm etc.) compare chart of specific machine (1.7, 1.8)

Programming Hints

The Take-over of Codes and Block Contents into the next Program

The self-holding codes, not called off by M30, and the tool called up at last are taken over into the next program, if they are not called off (or RESET key, Emergency button).

Example: Take-over of tool

```
O 10
N  0000
N  100 ........................T0303
N  230 ........................M30
-----------------------------------
O 20
N  0000 / G00 /
```

The tool T0303 would be active in program O 20.

Example: Zeropoint offset

```
O 10
N  000
N  20  ........................G54
N  100 ........................G57
N  200 ........................M30
-----------------------------------
O 20
N  0000
N / O / G00
-----------------------------------
```

G54 and G57 would be active.

Example: G-, M-Codes

O 10
N 0000
.
.
.
N 200 / G00 / X_1 / Y_1 / Z_1 /
N 210 / M30

O 20
N 000 / X_2 / Y_2 /

G00 would be active

Thus you should observe specific rules for program start and program end.

Program start - Program end
Programming Rules:

There are no general rules for a specific type of programming.
Each programmer will set up the program what he believes is simple and distinct.
However, there are some guide lines to observe as concerns program start and program end.
Bear in mind that other persons beside you could put in the program and that there are, maybe, some self-holding codes still valid.

Program Start:

1) Programming of G94 or G95 and feed F
2) Zeropoint offset(s) and tool correction call up
3) Direction of rotation
4) R.p.m.
5) Precise stop on/off (M38/M39)
6) Traverse instruction G00 (G00 is obligatory after T-call-up and zeropoint offset)

The sequence is optional. By a new input you have over-written still valid instructions of an old program.

Program end:

* Calling off zeropoint offsets
* Calling off the active tool

If a zeropoint offset e.g. would be active, the consequence could be a collision.

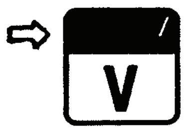
Skip-Blocks

For some cases (trial cut, serial production) it is quite useful that blocks can be skipped.

Skipped blocks are marked with a diagonal stroke (slash). It has to be put in after the block number.

```
N  90   G00 X20. Y25. Z30.
N 100   / M00 ———————— skip block
```

Key



Sequence in Program

SKIP Key pressed:
Skip blocks will be not executed.

SKIP Key not pressed:
The skip blocks will be executed.

Chapter 3

The reference points of the CNC machine, zero point offsets

The Reference Points of the CNC-Machine

Calculation/Compensation:

1. Reference Point R
2. Machine Zeropoint M
3. Tool Holding Reference Point N
4. Workpiece Zeropoint W

1. Reference Point R

The reference point serves for the synchronization of the measuring system. After switching on the machine the reference point has to be approached.

The position of the reference point differs from machine to machine. The position is fixed by the manufacturer.

The Criterion to Fix the Position:

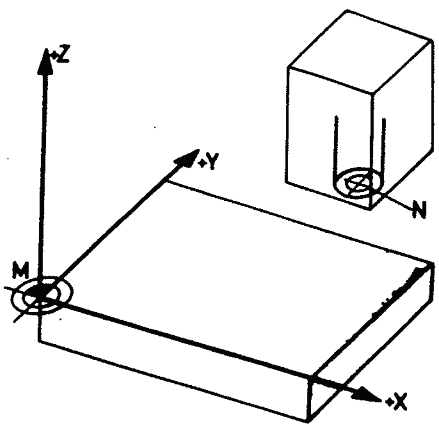
The reference point is usually situated out of the working area. Thus it is possible to approach it even with workpiece or tool mounted.

Approaching the reference point - compare Instruction of specific machine.

The Machine Zeropoint M ⊕
The Tool Holding Reference Point N ⊕

Their position of the machine zeropoint and of the tool holding reference point is determined by the machine manufacturer.

2. The Machine Zeropoint M ⊕

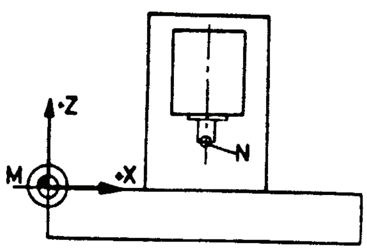


The machine zeropoint is the origin of the coordinate system. The origin of the coordinate system can be offset with codes G54, G55, G57, G58, G59.

Position of M on VMC-100 and VMC-200
Left-hand front edge of the table surface.

3. The Tool Holding Reference Point N ⊕

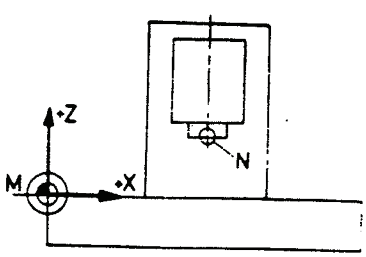
The tool lenghts are described from this point.



3.1 The position of the tool holding reference point N on VMC-100

N lies in the spindle axis at the front of the reference tool.

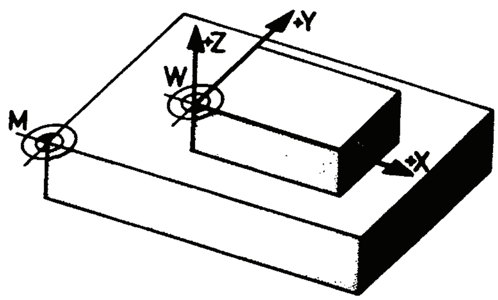
Note:
If the main spindle is freely accessible, the point N in milling machines is established on the front of the milling spindle. On the VMC-100 the front is not easily accessible and therefore N is established on a reference tool.



3.2 The position of the tool holding reference point N on VMC-200

N lies on the spindle axis at the front of the main spindle.

4. Workpiece Zeropoint W



The workpiece zeropoint is determined by the programmer.

The programming is done with G-Codes G54/ G55/G57/G58/G59.

5. ad) Machine Zeropoint, Tool Holding Reference Point

REMARKS FOR BETTER UNDERSTANDING

1) The manufacturer determines the position of M and N.

   Criterion for the Determination of M

   It should be easy to get the measurements up to the workpiece zeropoint.

2) The manufacturer measures the distances between M and N and puts them into the control (measurement is done with reference point approached).
   The control knows the distances M --> N.

Survey

Zeropoint Offsets

1. G53 - G59 Zero-Point Offsets with Position Shift Offset

[redacted]

The coordinates system can be offset (dis-placed) from the machine zero-point or from a selected zero-point using G54, G55, G57, G58, G59 .

With G53 the offset of G54, G55 is erased.

With G56 the offset of G57, G58, G59 is erased.

Structure

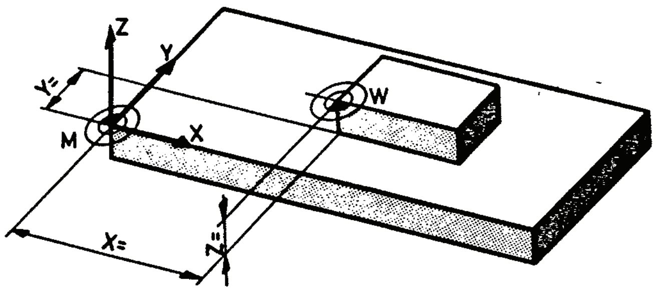
Group 3	G53	Erase G54,G55
	G54 ≙ 1 G55 ≙ 2	Call up position shift offsets 1,2.
Group 5	G56	Erase G57, G58, G59
	G57 ≙ 3 G58 ≙ 4 G59 ≙ 5	Call up position shift offsets 3,4,5.

2. G92 - Set Register

[redacted]

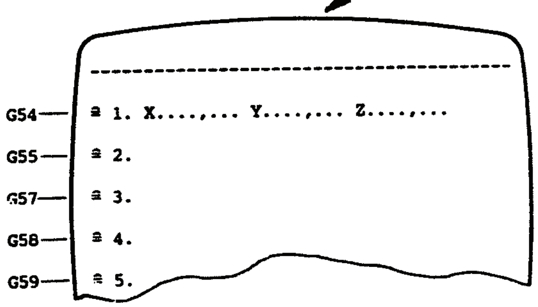
Schema of Zeropoint Offset

Data for offset values X,Y,Z (U,V,W)



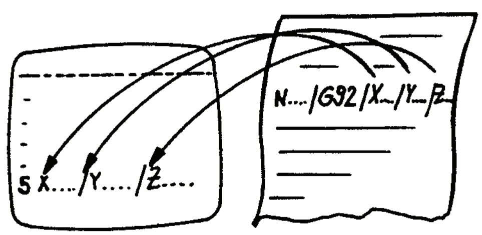
Data Input in Position Shift Offset Register

Two possiblities



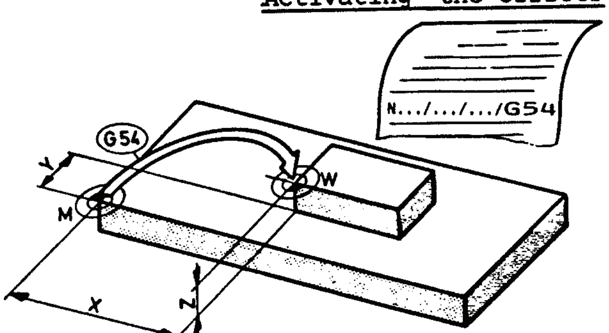
Manual input in position shift offset register 1 to 5.

Set register G92



To set register means:
The offset values are written in the parts program and stored in offset register number 5.

Activating the Offsets



The offsets are activated when G54,G55 G57,G58,G59 is activated in the parts program.
The zeropoint is offset by the values in the register.

Set Back with G53 resp. G56

The zeropoint is set back - compare detailed explanations on the following pages.

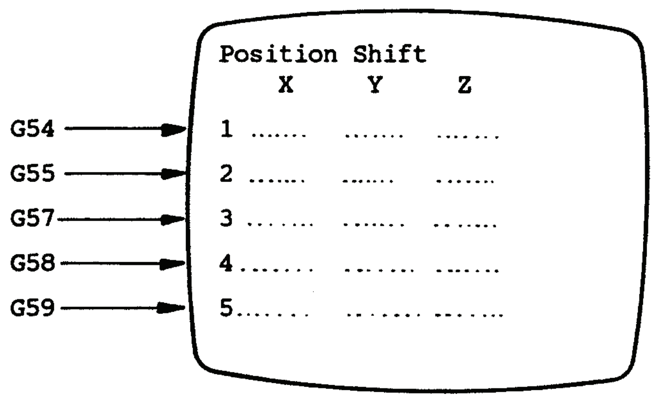
1. G53 - G59 Zeropoint Offset with Position Shift Offset

1.1 Notes and Rules for G53 - G59

Input of Data:

The measurements for the offset are put into the position shift offset with the correct sign.

G-Commands Position Shift offset 1 - 5



Structure and Offset/ Erasion of an Offset:

Various Instructions ot the same Group in one Program:
The instruction first programmed is always valid.
The previous instruction is disactivated by the following one (compare example 2).

Two Instructions out of Various Groups:
Instructions out of various groups are added vectorially.(There is no disactivation effect!) - Compare example 3

Set back Instructions

G53 sets back G54 and G55
G56 sets back G57/G58/G59.

Syntax Regulations:

The offset instructions have to programmed in connection with G00 instructions.

Possibility 1

In the same block as G00

N 100/G00/X.../Y.../Z.../G54

Possibility 2

The following traverse instruction is a G00 block.

N 100/G54
N 110/G94/F 120
N 120/G00/X.../Y.../Z...

1.2 Examples G53 - G59

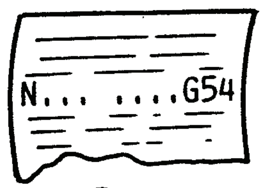
Example 1: A zeropoint offset - erase

Example 2: Two or more G-instructions of the same group - erase

Example 3: Group 3 and group 5 instructions - erase

Example 1: (G53 - G59)

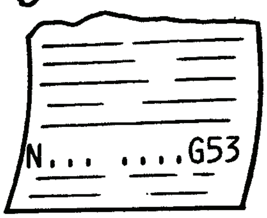
Zeropoint Offset - Erase



- Activation

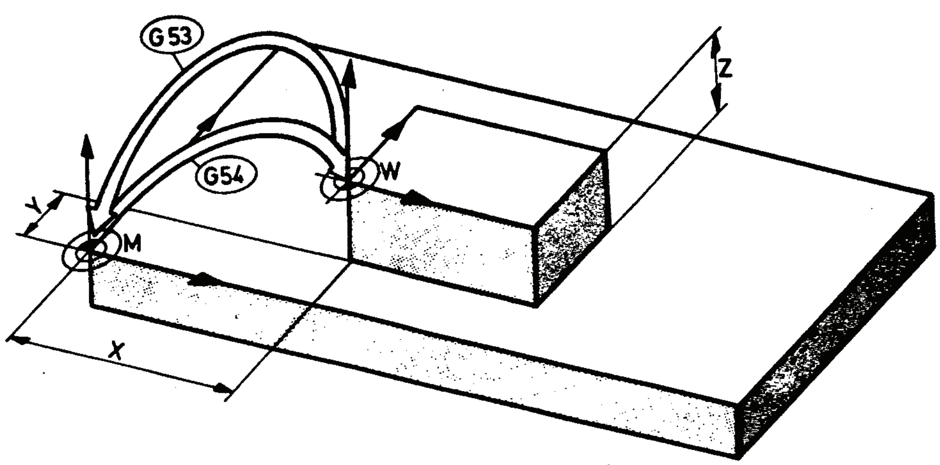
G54: The zeropoint is offset with G54 by the values X,Y,Z to the point W.

The X,Y,Z values are in the position shift offset.



- Disactivation

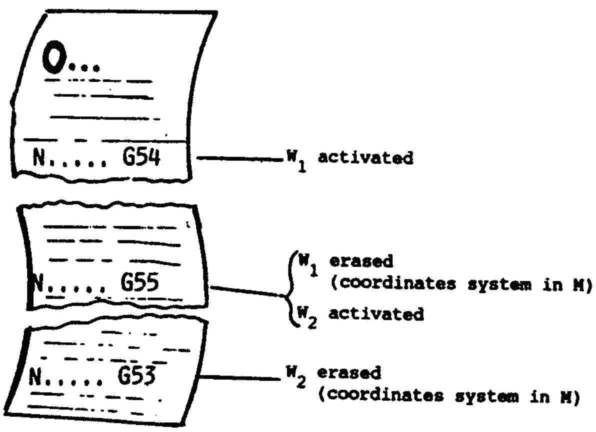
G53: The G54 offset is erased with G53. The zeropoint of the coordinates system is relocated to original point by the values -X,-Y,-Z. In this example to point M.



Example 2: G53 - G59

Two or more Instructions of the same Group - Erase

If two of more zeropoint offsets of the same group are called up in one program, the previous one is erased.



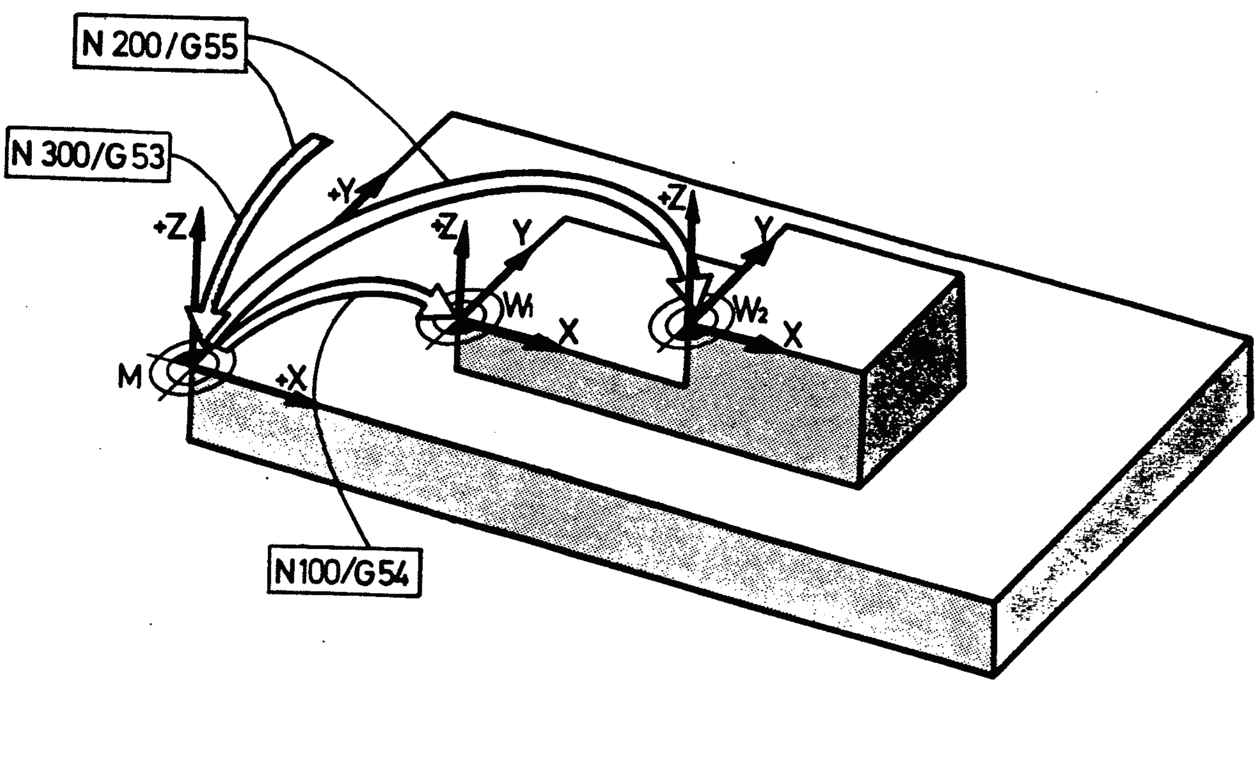
Example:

For the operation on a workpiece it is useful to program for

plane 1 W_1 and for

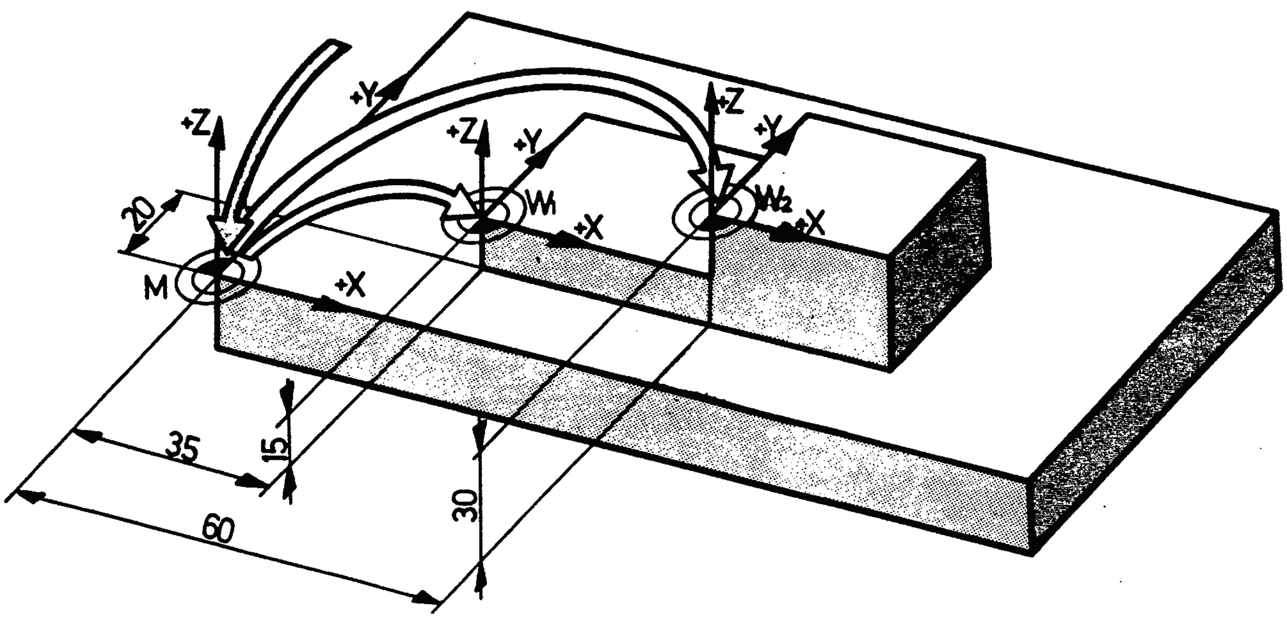
plane 2 W_2 .

In the position shift offset the values for W_1 and W_2 are put in.



ad) Example 2: Offset Values

	X	Y	Z
G54	≙ 35 000	20 000	15 000
G55	≙ 60 000	20 000	30 000



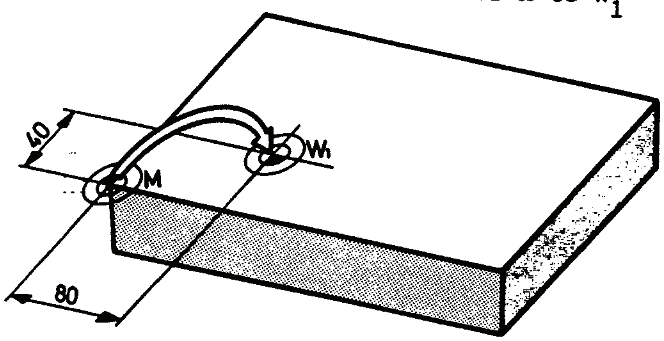
Example 3: G53 - G59

Group 3 and Group 5 Instructions in one Program

Various G-instructions of group 3 and group 5 in one program are not dis-activating one another, but will be added to one another.

Call up of G54

Offset of M to W_1

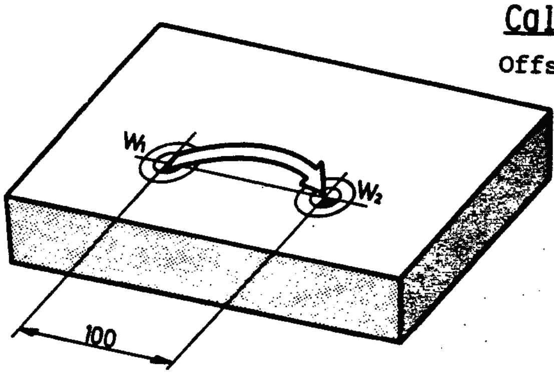


PSO1 - values

G54 ≙1.X = 80/Y = 40/Z = 0

Call up of G57

Offset of W_1 to W_2



PSO3- values

G57 ≙ 3.X = 100/Y = 0/Z = 0

Total Offset = G54 + G57 values

G54:	X = 80	Y = 40	Z = 0
G57:	X = 100	Y = 0	Z = 0
Sum:	X = 180	Y = 40	Z = 0

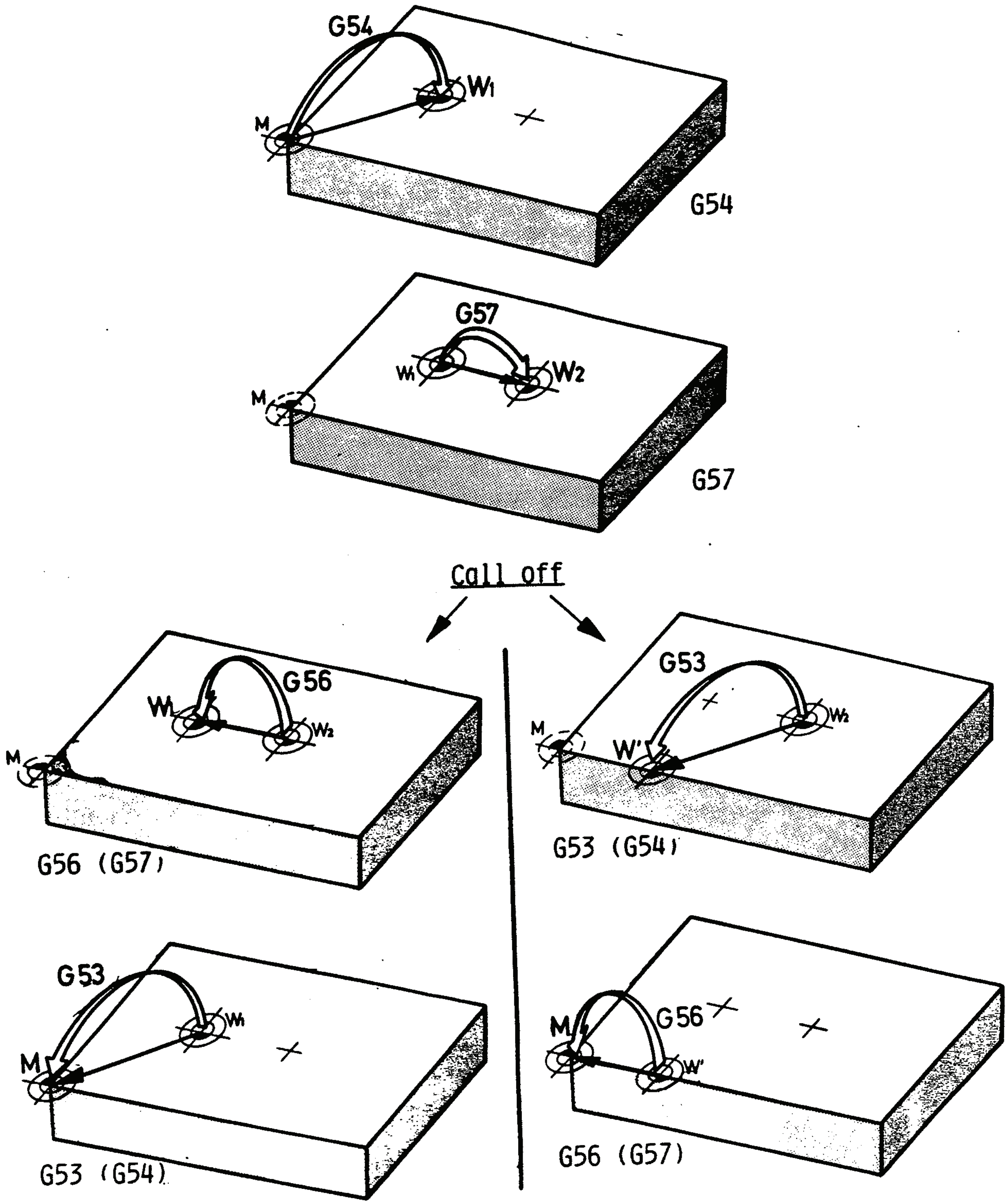
Note:

Pay attention to the signs!

G54:	X 120	Y 45	Z 38
G57:	X -25	Y 10	Z -15
Sum:	X 95	Y 55	Z 23

ad) Example 3: G53 - G59
Erasion of Offsets with two Groups

Pay attention : Erasion instructions G53/-
G56 are different groups.



2. G92 Set Register

2.1 Rules

Programming of Offset Values:

The measures for the offset are written into the parts program under G92.

Example:

N..../G92/X -14.2/Y +13./Z +14./

Activating of Offset:

1. When G92 appears in the program, the offset values are written in the position shift offset in register number 5.
2. Register number 5 is activated by the G59 command.

   Example:

   N..../G92/X -14.2/Y +13./Z +14.
   N..../G59

Notes:

* G59 cannot be programmed in the same block with G92, but G59 must be programmed in the following blocks.
* If G59 is programmed in a block before G92, an alarm sign will appear.
* If G92 + G59 follow a G-instruction of group 3, then the two instructions will be added to one another.

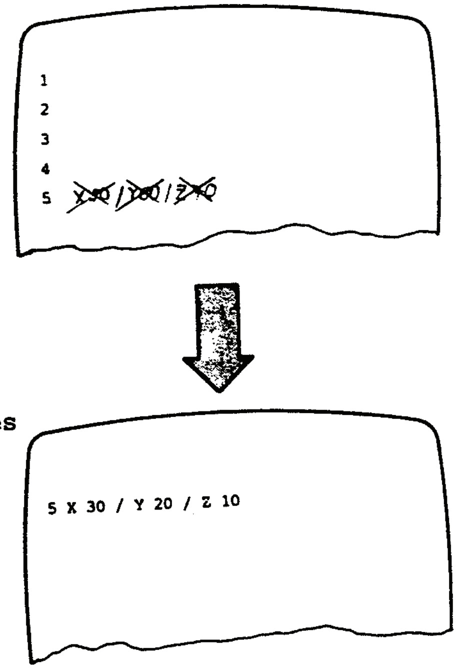
Erasion:

The erasion in the program is done with G56.

Types of Measurements in G92

Old values

Active values



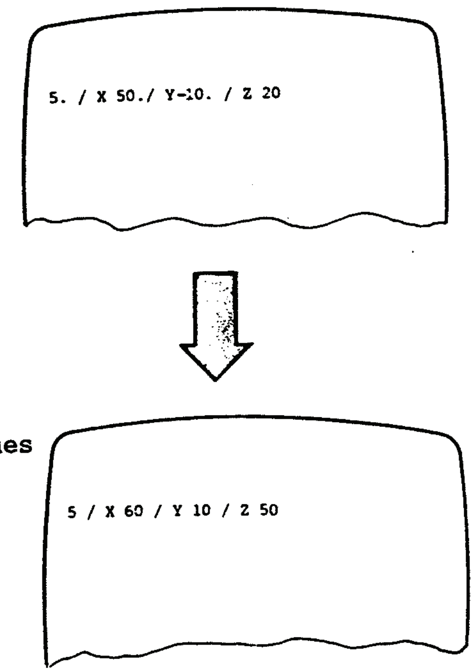
1. Absolute Values:
If under G92 the offset values are described with X,Y,Z the old values of G59 in the position shift offset are erased and the G92 values are active.

N 100/G92/X30./Y20./Z10.

N..../G59

Old values

Active calues



2. Incremental Values:
If under G92 the offset values are described with U,V,W then the U,V,W measures will be added to the measures of the position shift offset.

N 100/G92/U10./V20./W30.

N..../G59

d values

5 / ~~X 20.~~ / Y 30./ Z 40.

ive values

5 / X 15. / Y 35./ Z 52.

3. Mixed Values

If the measurements are indicated under G92 in a mixed up sequence, absolute with X,Y,Z and incremental with U,V,W then

* the absolute G92 measurements are taken over into the register.
* The incremental G92 measurements are added to values of the position shift offset.

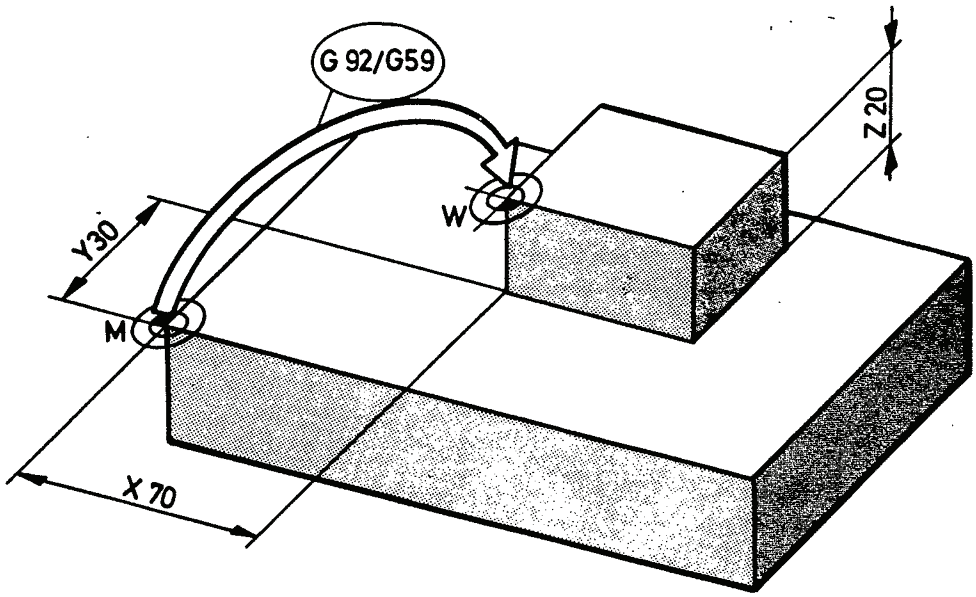
N..../G92/X15./V +5./W +12./

2.3 Examples G92:

Example 1:

1. G92 + G59 Instruction:

Offset from the machine zeropoint.



Program:

N 100/G92/X70./Y30./Z20./

..

..

..

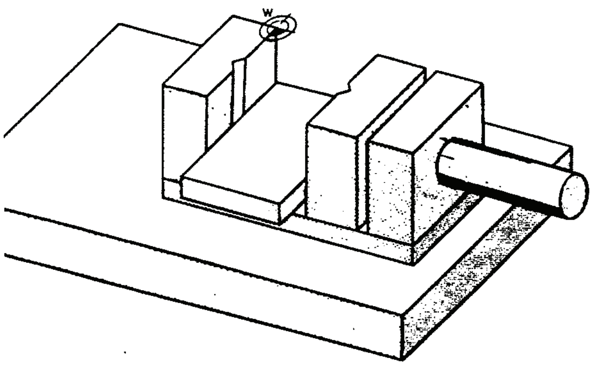
N 150/G00 /G59

The offset is executed in block N 150.

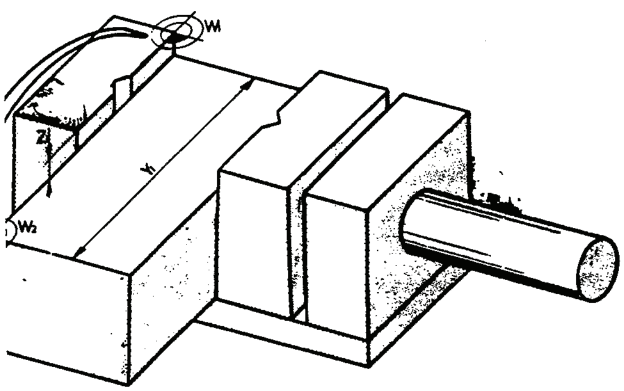
EXAMPLE 2:

2. Offset with Group 3 Instructions and Offset in Workpiece Program with G92 , G59

This proves to be quite a practical method:



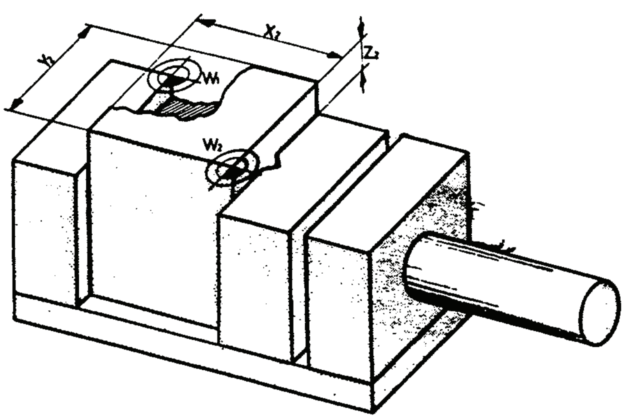
* The zeropoint is offset from M to W_1 using group 2 instruction.
* From this point the workpiece zeropoint of the respective workpiece W_2 is programmed using G92, G59.



Example 1:

N..../G54

N..../G92/X_1 /Y_1 /Z_1 /G59



Example 2:

N..../G54

N..../G92/X_2 /Y_2 /Z_2 /G59

Chapter 4

Tool (length) compensation

Terminology - tool compensation

Tool compensation can mean two different things: tool (length) compensation and tool (path) compensation.

Tool (length) compensation:

This means the input and compensation of the tool length.

Correction of tool compensation:

If you see from the milling result that the lengths are wrong (with correct programming), you must correct the (length) compensation value.

Tool (path) compensation:

See G40/G41/G42

Tool Programming and Compensation

1. T-Address:

Tools are programmed under the T-address using a 4-digits number.

Code

T .. ..

01 - 10 (12)

01 - 99

Tool Number

The first two digits are the tool number (number of tool position on tool pallet with automatic tool change).

For VMC-100: 10 tool numbers admissible

For VMC-200: 12 tool numbers admissible

Tool Compensation Number

Number code for tool data (length, radius).
The tool compensation number is listed in the tool data memory.

2. Call-up

Every new T-address has to be called-up with a G00 block (otherwise Alarm sign).

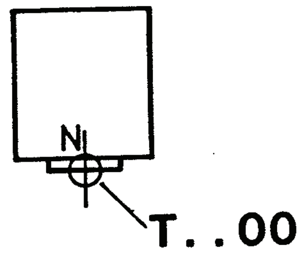
Example: Call-up in same block with G00

N 90 / M00
N 100 / G00 / X.../Y.../Z.../ T02 02

Example: After the T-call-up a G00 traverse instruction follows.

N 100 / T02 02
N 110 / G94 / F 130
N 120 / G54
N 130 / G00 / X.../Y.../Z.../

3. Calling-off the Tool(length) Compensation



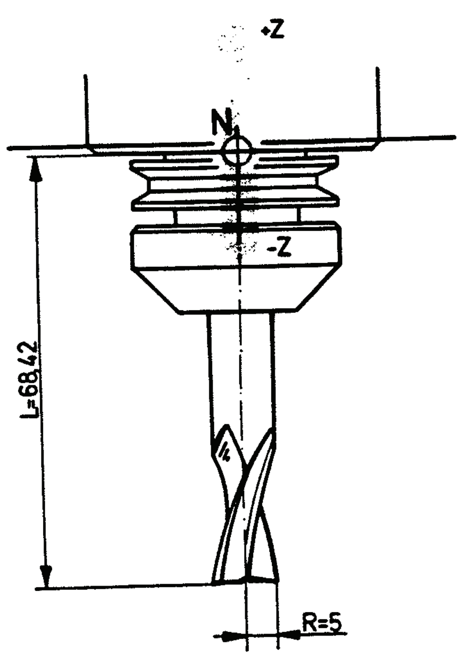
T.. 00

If the reference number T.. 00 is programmed, the coordinates (measurements) system refers to the reference point for the tool mounting.

4. Alarms:

1. T-call-up not in connection with G00 traverse instruction
2. Unacceptable call-up T00 04:
   Tool number ≠ 1 - 99
   But compensation number ≠ 0
   T 02 00 is acceptable, but does not make sense.

5. The Tool Compensation Values:
Tool Length, Tool Radius



1. Tool Length in mm (Inch with G70 being active)with sign:

   Imagine the coordinate system in point N. The tool lengths are taken from point N on.

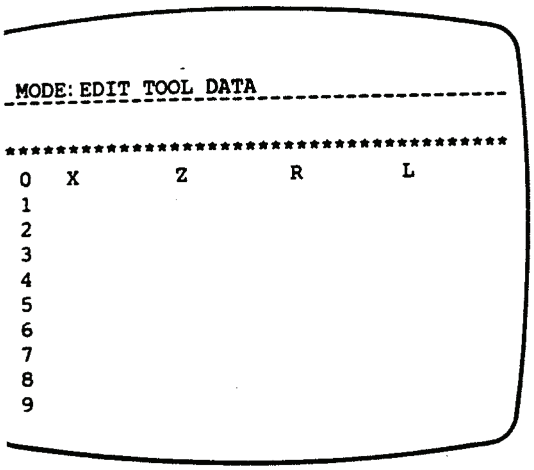
2. Cutter Radius in mm (Inch with G70 being active)

   The information on the radius is necessary with G41/G42.

   Input:

   Z - 68.42/R5

6. Input - Compensation:



The tool data are entered into the tool memory in mode EDIT.

1 corresponds to correction T.. 01
20 corresponds to correction T.. 20

Tool length: under Z-address
Radius: under R

Compensation:

If a tool is called-up in the program the computer fetches the data Z(length) and R (radius) which were put in under the code number.

7. Programming Hint:

Number of compensation and of tool need not be the same; e.g. T 05 01. For a better overlook it is useful that compensation number and tool number are identical.

For VMC-100 the numbers 01 - 10 can be entered as tool numbers. If a tool number greater than 10 is programmed, an ALARM is given.

For VMC-200 the numbers 01 - 12 can be entered as tool numbers. If a tool number greater than 12 is programmed, an ALARM is given.

Correction of Tool (length) Compensation

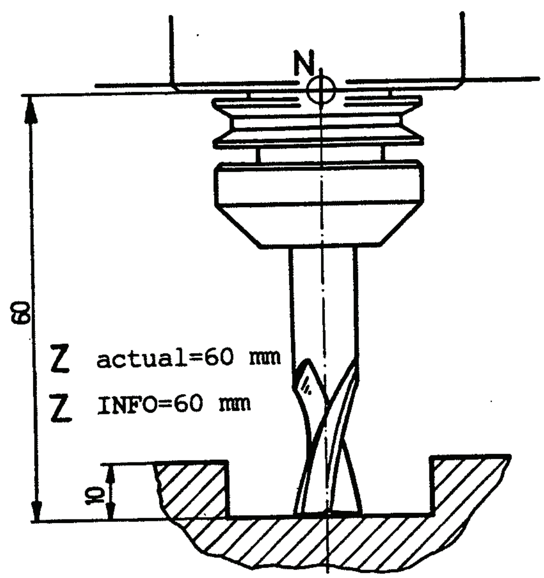
Correction of length measurements

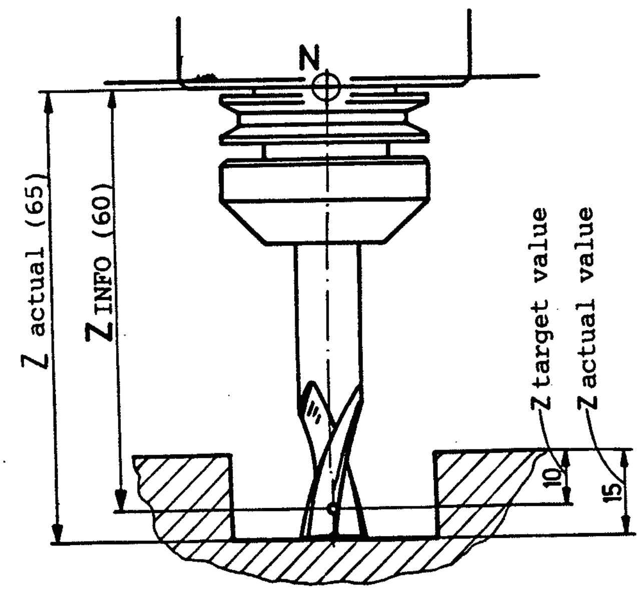
By measuring the workpiece you find out possible faults, which are caused by non-accurate tool data.

Correction:

* Erase wrong data
* Put in correct data

Example:





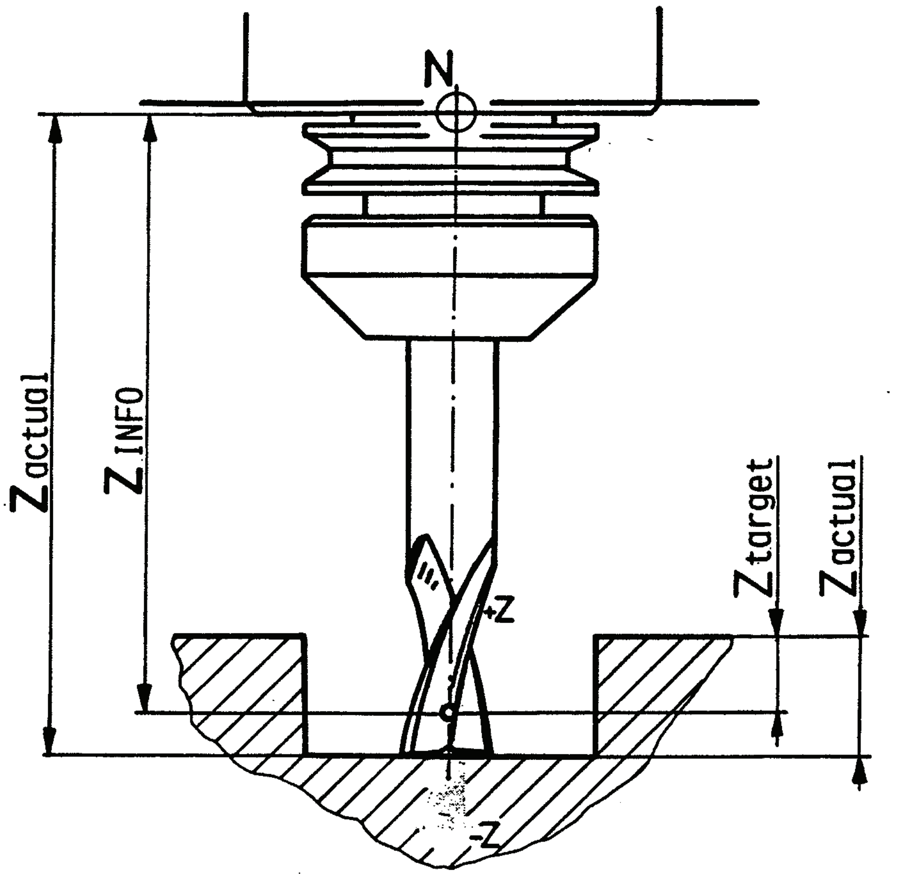
Z actual is the same as
Z INFO in the tool data memory

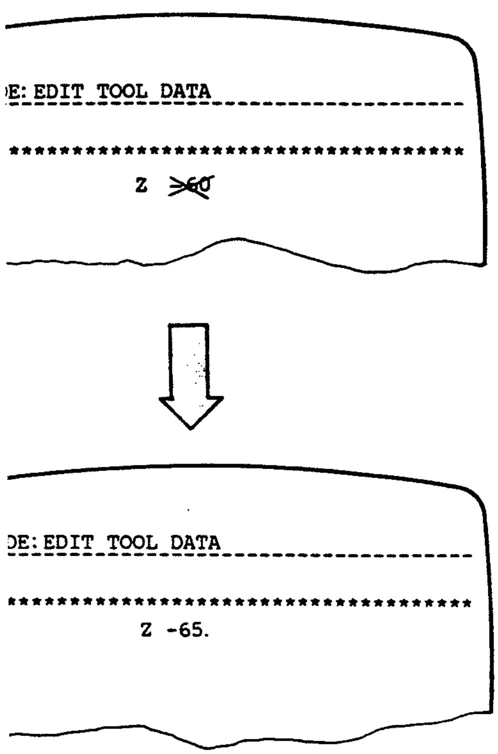
Measurement in drawing and on work-piece are correct.

Z actual is not equal to Z INFO:

Consequence: Wrong measurements on workpiece (15 mm instead 10 mm)

Correction of the Length Data





* Imagine the coordinates system in the target value.

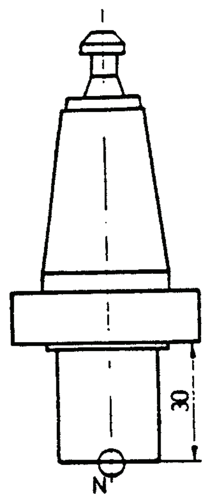
* Measure difference between target value and actual value: Z = -5mm

* Add this value with the correct sign to the value in the tool data memory.

  Z - 60 mm + (-5mm) = -65 mm

* Write this value into the tool data memory.

The reference tool - VMC-100



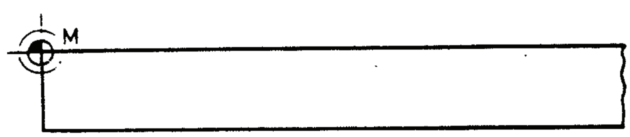
* A reference tool is supplied with the VMC-100.
  The point N is located on the rotating axis on the front of the reference tool.

* If no zero point offset is active and no tool compensation selected, the screen displays the measurements M N.

* The tool lengths are described from the point N.

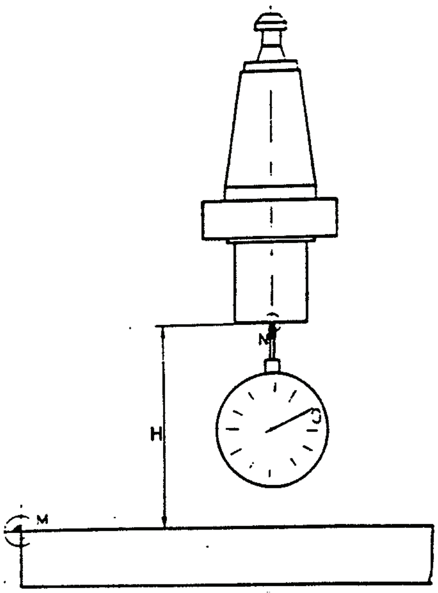
* This reference tool is required for the direct take-over of tool lengths when touching with a dial gauge.

* The point N lies 30 mm below the ball bearing shoulder.



Direct take-over of the tool lengths - VMC-100

MAN
Mode



With EMCOTRONIC TM 02 it is possible to take over the tool lengths directly. This is a very accurate and comfortable method.

Option 1: Touching with dial gauge

* Select manual mode.

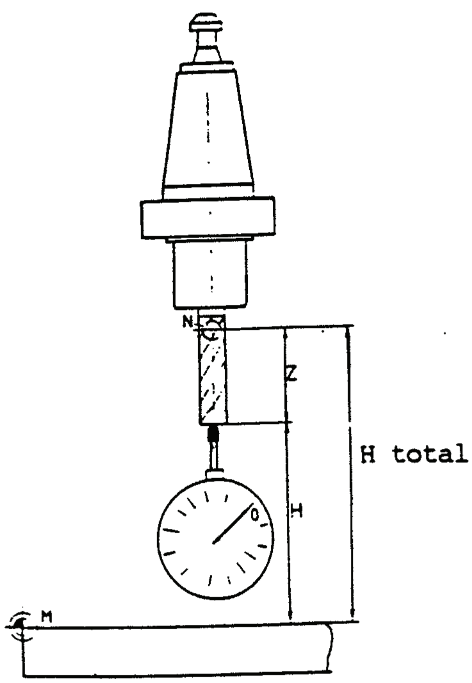
* Mount reference tool and swivel in.

* Touch dial gauge with reference tool. Set dial gauge to zero.

TO 0 0

ENTER

The height H is stored in the control by TO 00 ENTER.



* Swivel in the tool to be measured and touch dial gauge (gauge must show 0).

TO 0 1 Tool correction number wanted

ENTER

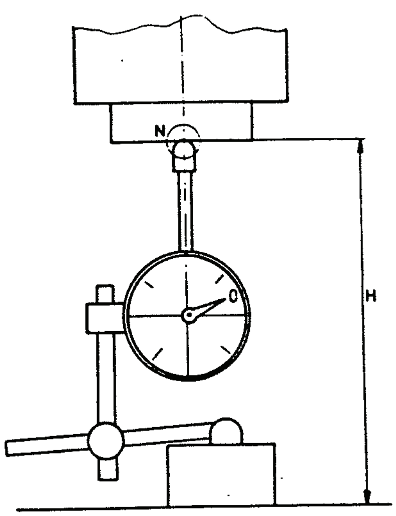
The tool length Z is stored under tool correction number T..01.

Direct take-over of the tool lengths - VMC-200

Direct take-over of the tool lengths is very convenient and applicable in most cases.

Touching the dial gauge

MAN
Mode



1. Touch dial gauge with point N. Set dial gauge to zero. When dial gauge indicates 0, a specific height H is reached.

   TO 0 0

   ENTER

   The height H is stored. The computer can calculate the tool value Z when touching with a tool.

2. Touch dial gauge with tool (gauge must indicate 0, then the height Z is reached).

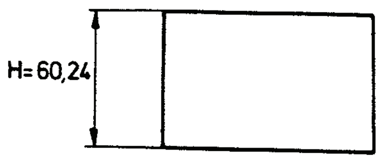
   TO 0 1

   ENTER   **Tool correction number wanted**

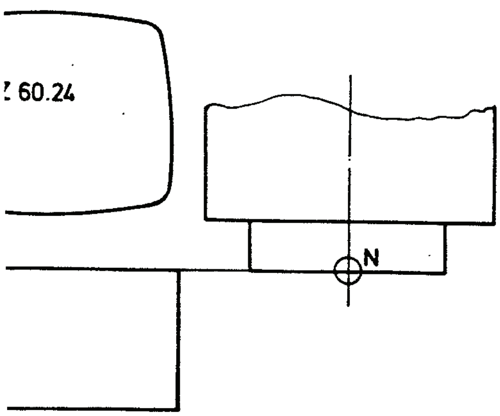
   The measurement Z of the tool T01 is stored under the compensation number T..01.

Scratching a trial workpiece

MAN
Mode



1. Measure the height (H) of the trial workpiece to be scratched.

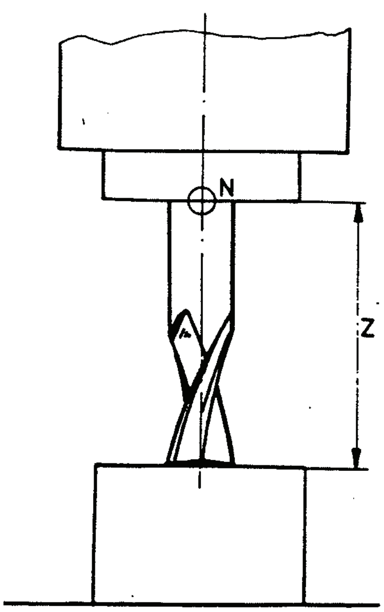


2. Move the point N to the Z-value H (screen display)

   TO 0 0

   ENTER

   The reference measurement H is stored.



3. Scratch the workpiece surface.

   TO 0 1

   ENTER current tool number

   The measurement Z is stored under the compensation number T..01.

Chapter 5
The M-functions

The M-functions

The M-Codes

As the G-Codes, tne M-Codes are divided into different groups. An instruction of one group erases the other instruction of the same group.

That means the last programmed M-instruction cancels out the previous instruction from this group.

Programming
M-functions are switching and additional functions. The M-commands can stand alone in a program block or together with other instructions.

Note:
The following pages contain a list of the M-functions which are standard features of the EMCOTRONIC TM 02. Whether these M-functions are active on the machine in question depends on the machine version.

Group structure and initial status of the M-functions

Group 0	 * 	M03: Spindle ON in clockwise direction M04: Spindle ON in counterclockwise direction M05: Spindle stop M19: Spindle precise stop
Group 1	 **	M38: Precise stop ON M39: Precise stop OFF
Group 2	* * *	M00: Programmed STOP M17: Subroutine end M30: Program end with return to program start
Group 3	 **	M08: Coolant ON M09: Coolant OFF
Group 8	□ □	M50: Cancellation of the direction logic with bidirectional tool turret M51: Selection of the direction logic with bidirectional tool turret
Group 10		M90: Cancellation of the mirroring function M91: Mirroring on the X-axis M92: Mirroring on the Y-axis M93: Mirroring on the X and Y-axes

* effective blockwise
** initial status
Initial status can be established in the user monitor (MON) mode.

Note: The implementation of the individual M-functions depends on the hardware of the machine in question.

M00 - Programmable intermediate stop

N4/M00

The slides are stopped and the main spindle and coolant switched off.
Effect: end of block
Application: If measurements are necessary during a machining operation or other checks are to be performed.
Tool change, remounting of the workpiece etc.

M03 - Main spindle ON clockwise rotation

N4/M03

M04 - Main spindle ON counterclockwise rotation

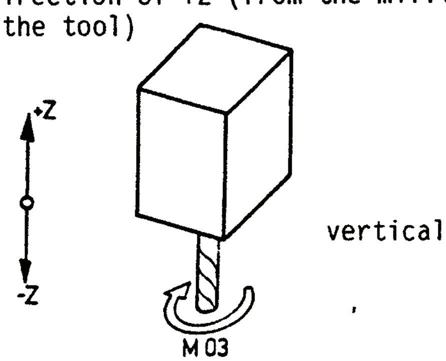
N4/M04

M05 - Main spindle STOP

N4/M05

M05 is activated by M30 at the program end.
Effect: start of block

Establishing clockwise rotation, counterclockwise rotation:
Always view the direction of rotation from the -Z side in the direction of +Z (from the milling spindle towards the tool)



M08 - Coolant ON

N4/M08

M09 - Coolant OFF

N4/M09

M09 is activated by M30 at the program end.
Effect: start of block

M17 - Subroutine end

N4/M17

The subroutine is concluded with M17. M17 causes a jump back to the next higher plane of the part program.
For details, see subroutine technique G25/M17.
Effect: end of block

M19 - Main spindle precise stop

N4/M19/S4

[°]

The main spindle can be positioned by programming M19.
The position of the main spindle can be entered (in degrees °) under parameter S.
Input range: 0-360°

M30 - Program end with return to program start

N4/M30

Effect: Block end/program end, return to program start

M30 also produces:
- coolant OFF
- main spindle OFF
- G40

M38 Precise Stop ON

N4/M38

M39 Precise Stop OFF

N4/M39

Explanations on M38/M39
Effect: block start

If you want a sharp transition you have to program M38.
The axis movement in the programmed target point stops completely and only then the next block is traversed.

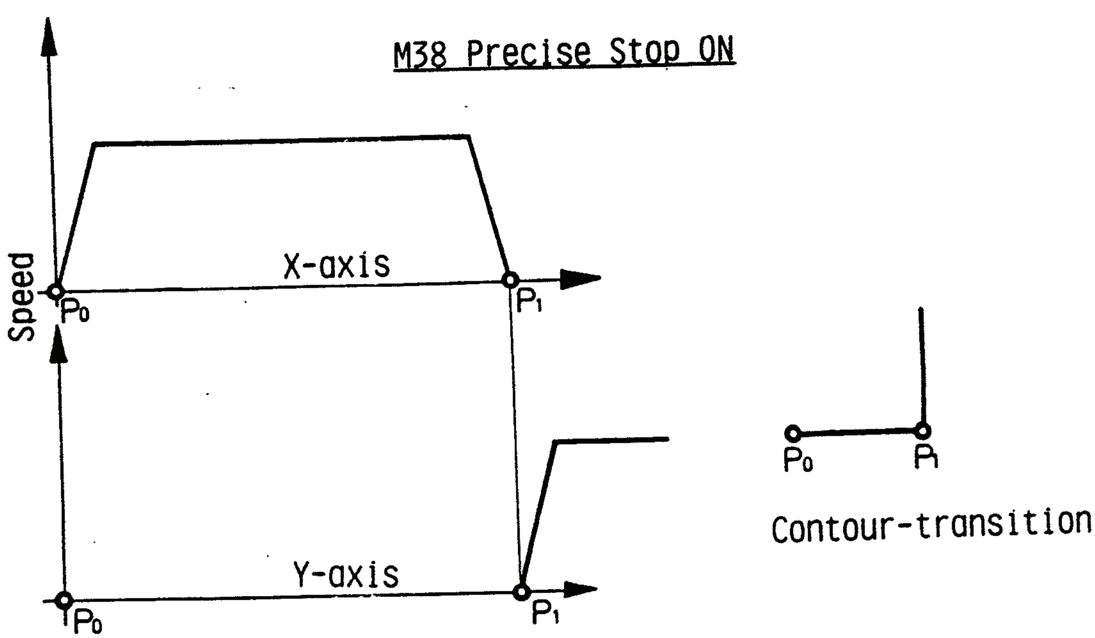
Note:

What has been said for XY-axis is of course also valid for the other three axes.

Remarks: * Note down the time difference when manufacturing a workpiece with and without precise stop.

* The control knows the contents of the following traverse instruction

M38 Precise Stop ON

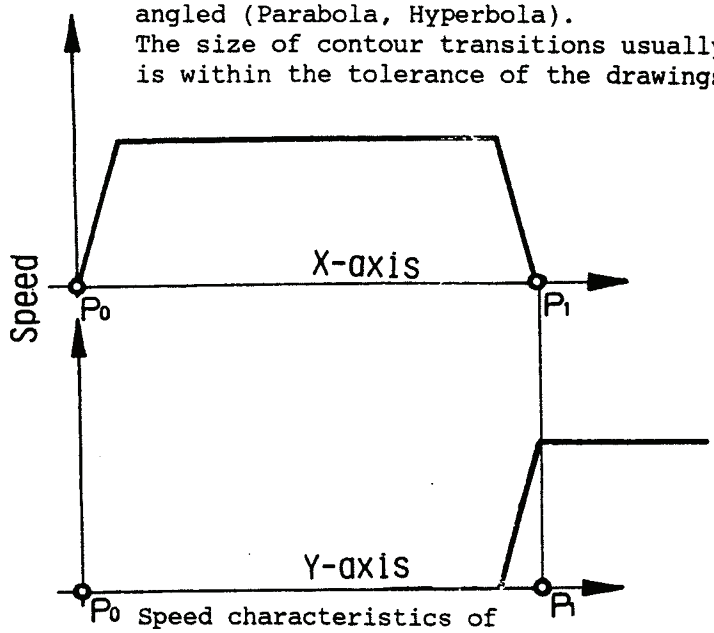


Dwell: 20 m/sec. at point P_1

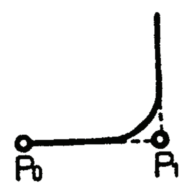
M38/M39 Continuation

M39 Precise Stop OFF

The EMCOTRONIC TM 02 control is laid out to accelerate in Y-axis already before reaching the target point in X-axis.
Thus a continuous movement with contour transitions is achieved.
The contour transition is not acute-angled (Parabola, Hyperbola).
The size of contour transitions usually is within the tolerance of the drawings.



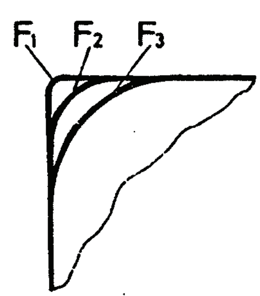
Speed characteristics of slides with M39



Contour-transition

Size of Transition Arcs in Relation to the Speed of Feed:

The larger the feed, the larger the arc of circle.



M50 - Cancel direction logic

N4/M50

The tool magazine is activated by the control in such a way that it only indexes in one direction.

M51 - Select direction logic

N4/M51

The direction of rotation of the tool magazine is selected by the control so that the programmed tool magazine position is reached in the shortest way.

Establishing the initial status

The initial status (M50 or M51) can be established in the user monitor (MON) under the parameter O_{11}.

M50: O_{11} bit 3 = 0 (low) value 0

M51: O_{11} bit 3 = 1 (high) value 4

(See also description of user monitor)

Mirror functions

Mirroring of contour elements

With the selection of a mirror function all subsequent positions are mirrored about the workpiece zero point axis.

M90 - Cancellation of the mirror function

M91 - Mirroring on the X-axis

M92 - Mirroring on the Y-axis

M93 - Mirroring on the X and Y-axes

Conditions for the selection and cancellation of mirror functions

When working with mirror functions the following conditions must be observed:

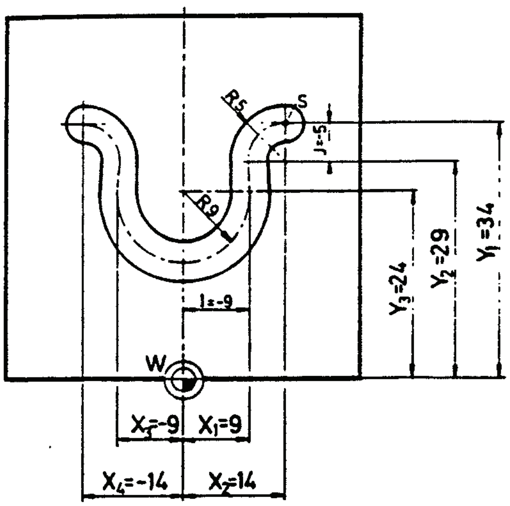
- o Neither a circular interpolation (G02, G03) nor a call-up of the tool path offset (G41, G42) may stand in NC blocks in which there is a selection or cancellation of a mirror function.
- o Selection or cancellation of a mirror function may only take place when the tool path offset (G40) is cancelled.

Notes:

- o Mirroring of G02, G03, G41 and G42:
  Depending on the type of mirror function, it is possible that the direction of rotation of a circular interpolation or the type of tool path offset changes. You can see the circular interpolation or tool path offset actually covered by switching over to the SPECIAL page.
- o Mirroring of boring patterns:
  * Definition parts of a boring pattern cannot be mirrored. M90 must be active.
  * The execution section of a boring pattern can be mirrored.
    Limitation - circular boring patterns:
    The circular boring pattern must be closed (D_3 = 3600).

Example:

mill diameter: 10 mm



milling depth: 5 mm

Absolute programming:

Main program: 0 0025

```
N..../....
N..../M90
N..../G00/X14,000/Y34,000
N..../G01/Z-5,000/F....
N..../G25/L7001
N..../G00/Z5,000
N..../G00/X-14,000/Y34,000
N..../M92
N.... /G25/L7001
N..../M90
N.../.....
```

Subroutine: 0 0070

```
N..../G03/X9,000/Y29,000/J = -5,000/F....
N..../G01/X9,000/Y24,000/F....
N..../G02/X00,000/Y15,000/I = -9,000/
   F....
N..../M17
```

Chapter 6

The G-functions
(Path conditions)

* The G-functions are subdivided into groups
  (see summary 1.4)

* G-functions from the same group cancel each other out.

* If two G-functions of the same group are in one block, the
  G-funktions last progrmmed (entered) is effective.

The G-functions

Group structure and initial status of the G-functions

Group 0	 *	G00: Rapid traverse G01: Linear interpolation G02: Circular interpolation clockwise G03: Circular interpolation counterclockwise G04: Dwell G72: Definition circular boring pattern G74: Definition rectangular boring pattern G81: Drilling, centering G82: Drilling, spot-facing G83: Deep-hole drilling with retraction G84: Tapping G86: Deep-hole drilling with chip breaking G87: Rectangular pocket milling cycle G88: Circular pocket milling cycle G89: Slot milling cycle
Group 2	 **	G94: Feed in mm/min or 1/100 inch/min G95: Feed in µm/rev. or 1/10000 inch/rev.
Group 3	**	G53: Cancellation of offsets 1 and 2 G54: Call-up of offset 1 G55: Call-up of offset 2
Group 4	*	G92: Set offset 5
Group 5	**	G56: Cancellation of offsets 3,4,5 G57: Call-up of offset 3 G58: Call-up of offset 4 G59: Call-up of offset 5
Group 6		G25: Subroutine call-up G27: Unconditional jump
Group 7	□ □	G70: Measurements in inches G71: Measurements in mm
Group 8	**	G40: Cancellation of the tool path compensation G41: Cutter path compensation left G42: Cutter path compensation right
Group 9	$\boxed{**}$ □ □ □ □ □	G17: 1st axis switching G18: 2nd axis switching G19: 3rd axis switching G20: 4th axis switching G21: 5th axis switching G22: 6th axis switching
Group 11	**	G98: Withdrawal to starting plane G99: Withdrawal to withdrawal plane
Group 12		G73: Call-up circular boring pattern G75: Call-up rectangular boring pattern

* effective blockwise
** initial status
□ Initial status can be established in the user monitor (MON) mode.

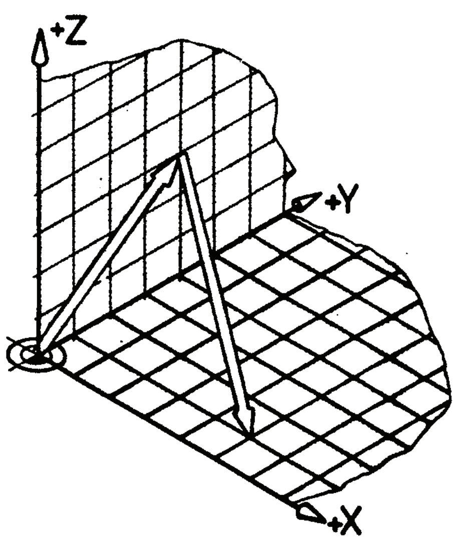
G00 Rapid Traverse

N4	G00	X U ±43	Y V ±43	Z W ±43

The movement is executed with rapid traverse in all three axes simultaneously.

Absolute Mode

The target point is described from the previously defined zeropoint of the coordinate system.



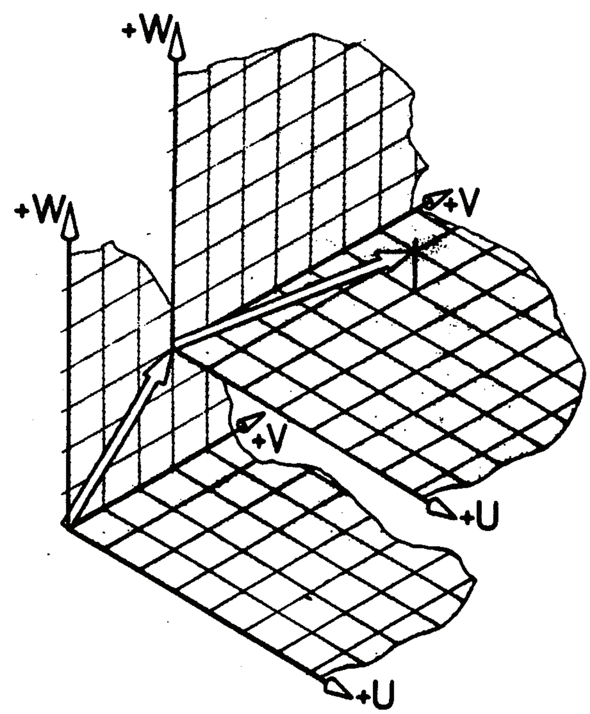
Example:

N 100/G00/(X 0)/Y 4./Z 3./
N 110/G00/X 5./Y 1./Z 0./

N 100 X = 0 need not to be programmed.

Incremental Mode

The target point is described from the starting point of the block (difference in distance).



Example:

N 100/G00/ (U 0)/V 3./W 3./
N 110/G00/U 2./V 5./W 1./

U = 0 need not to be programmed since there is no change in U(X) direction.

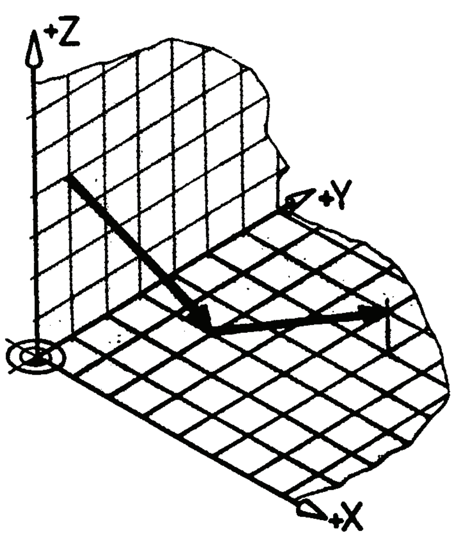
G01 Linear Interpolation

N4	G01	X / U ±43	Y / V ±43	Z / W ±43	F4

The tool can traverse with the programmed feed speed (or feed per revolution) in all three axes simultaneously.

Absolute Mode

The target point is described from the previously defined zeropoint of the coordinates system.

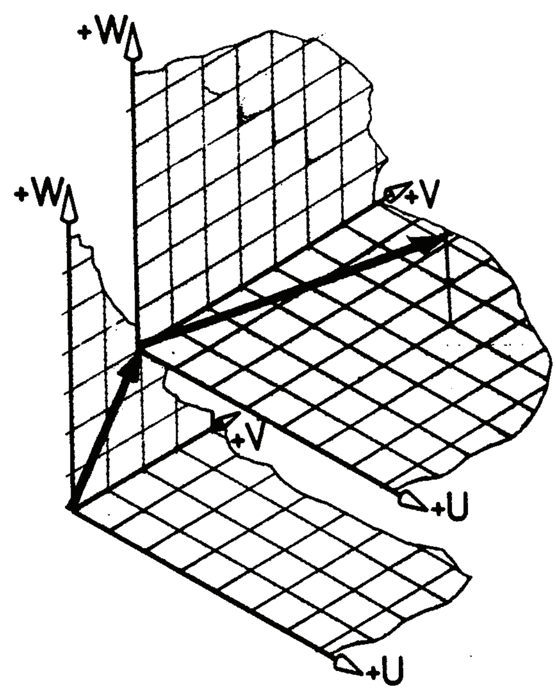


Example:

N 100/G01/X 2./Y 3./Z 0./
N 110/G01/X 5./Y 5./Z 1./

Incremental Mode

The target point is described from the starting point of the block (difference in distance).



Example:

N 100/G01/(U 0)/V 2./W 3./F..
N 110/G01/U 4./V 5./W 2./F...

F-Data

G94 : mm/min (0 - 2200 mm/min)
G95 : µm/U = (0 - 2000 µm/rev.)
Initial status G94

Circular Interpolation
G02 Clockwise
G03 Counterclockwise

N4	G02 G03	X U ±43	Y V ±43	Z W ±43	I±43	J±43	K±43	F4

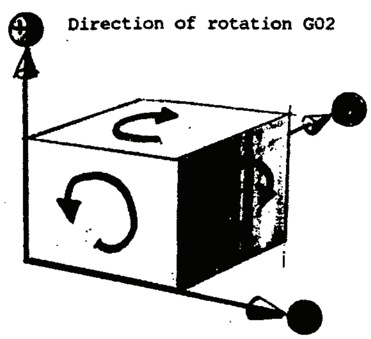
General Remarks

* With the EMCOTRONIC control you can program circles and arcs of a circle in all three planes.

* For the maximum radii please compare the technical data of the machine.

* The programming is based on center point coordinates.

The Direction of Rotation G02/G03



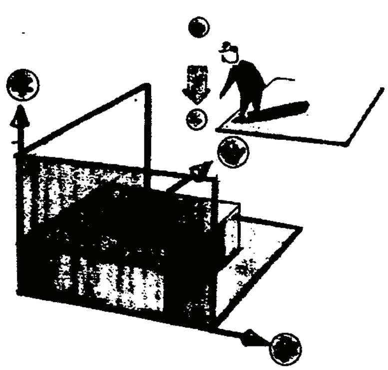
In order to determine right-hand rotation and left-hand rotation it is necessary to fix the direction of view.

Determination

View the direction of rotation in a plane always from the positive direction of the third axis.

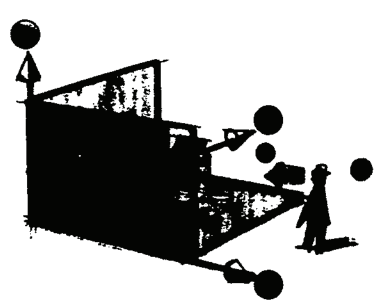
Example:

Direction of rotation G02 clockwise



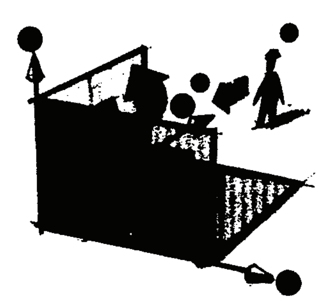
XY-plane:

View from +Z direction to -Z direction



YZ-plane:

View from +X to -X



XZ-plane:

From +Y to -Y

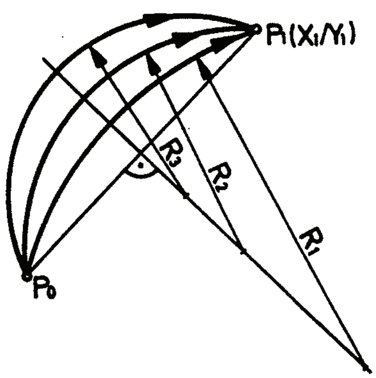
In this drawing the direction in the XZ-plane thus seems reversed.

Programming:

1. Direction of rotation
2. Description of position and size of the arc.
3. Feed

Programming an Arc (of a Circle)

(Center point programming)

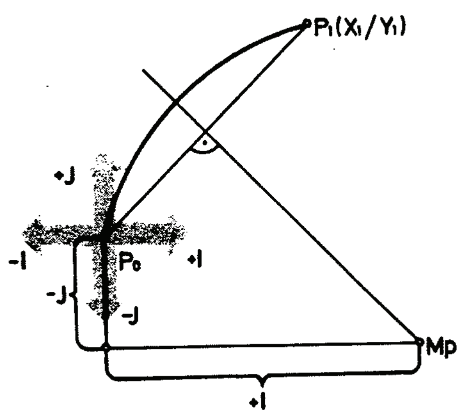


1. <u>Programming the target coordinate (P_1) absolute or incremental</u>

   (The starting coordinate is known to the computer.)

   N..../G02 /X_1..../Y_1....

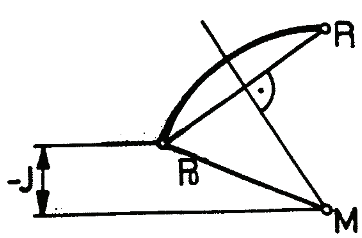
   With this description the position is defined but not the radius.



2. <u>Incremental description of the center point coordinate Mp from the starting point.</u>

   The center point is on the axis of symmetry of points P_0 and P_1.
   If the center point coordinates are described then the size of the arc is fixed.
   The center point is described from the circle starting point with addresses I,J,K
   Imagine the I,J,K coordinate system puts in the circle starting point.

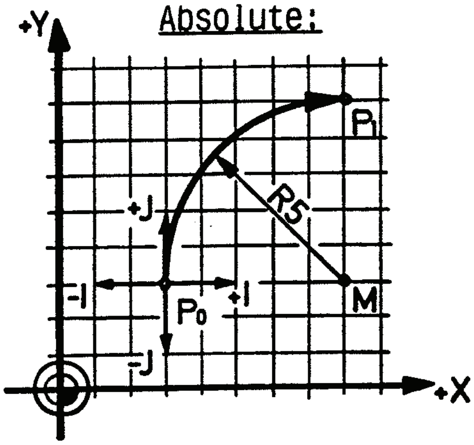
   N..../G02/X_1/Y_1/I..../J..../F....



> Note:
>
> The arc is defined with the description of a centre point value. The second coordinate value must, however, be programmed.

Programming
Example Arc (of a Circle) 90°

Absolute:



1. Direction of rotation

N..../G02

2. Final coordinates of arc P_1 absolute

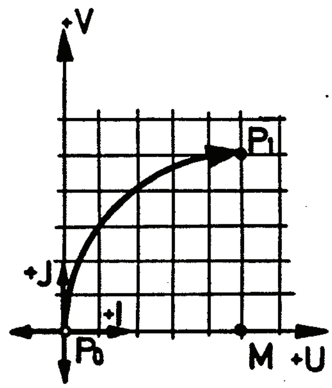
N..../G02/X 8./Y 8./Z..

3) Incremental description of center point from arc starting point.

The center point oft the circle is described with addresses I,J,K. Imagine the incremental axis system in the arc starting point and describe the center point.

N..../G02/X 8./Y 8./(Z)/I 5/J 0

Incremental:



1. Direction of rotation

N..../G02

2) Final coordinates (P_1) of arc from starting point.

N..../G02/U 5/V 5/

3) Center point coordinates described from starting point (P_0).

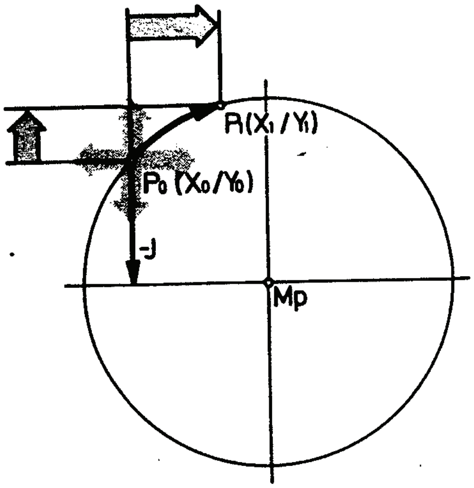
N..../G02/U 5/V 5/I 5/J 0

The Description of the Center Point Coordinates

(Example in XY-plane)

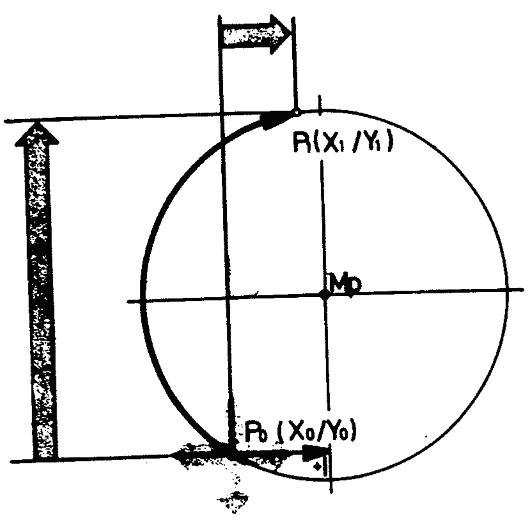
An arc (of a circle) is determined by the description of points P_0 and P_1 and by the value of a center point coordinate (I or J).

Therefore, following rules for centre point coordinates on the EMCOTRONIC TM 02 control
Both centre point coordinates must always be programmed.



Example 1:

N..../G02/X_1.../Y_1.../I.../J.../F...

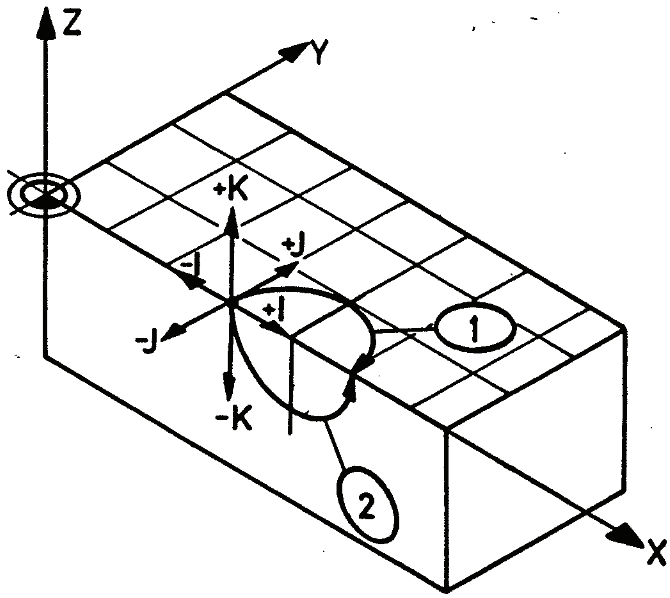


Example 2:

N..../G02/X_1.../Y_1.../I.../J.../F...

Special Case:

Arcs 1 and 2 have the same sense of rotation. The target coordinates and the center point coordinates are the same for both arcs.



Programming:

N..../G02/X 5./Y 0/I 1./J 0/K 0

This type of programming could mean arc 1 and arc 2.

Regulation:

The control reacts to alphabetical sequence of description of the planes.

Arc 1:

N..../G02/X 5./Y 0/(I 1.)/J 0

Arc 2:

N..../G02/X 2/(Y 0)/Z 0/(I 1.)/K 0

J = 0 must not be programmed, otherwise the control will give priority to address J and executes arc 1.

Arcs will only be executed in the planes or parallel planes. Thus it is not necessary to determine the third coordinate of the point.

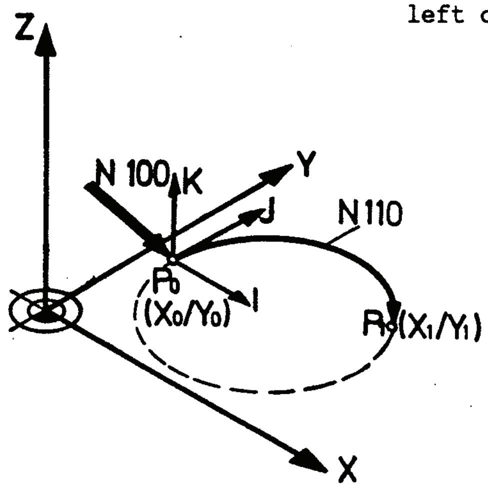
Example: Arc is XY-plane

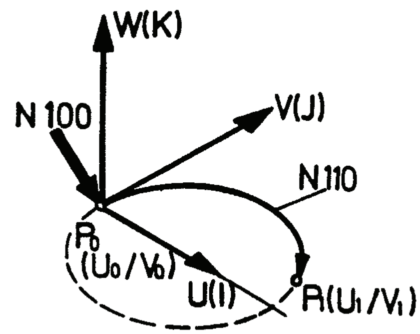
Absolute Value Programming:

Incremental Value Programming

N 100: P_0 is approached;
The control knows the position of the circle plane (distance Z).

N 110: The programming of the Z(W) coordinate and the K coordinate can be left out.





N $100/G01/X_0.../Y_0.../Z_0.../F...$
N $110/G02/X_1.../Y_1/(I...)/J.../F...$

N $100/G01/U_0.../V_0..../F...$
N $110/G02/U_1.../V_1..../(I...)/J.../F...$

Allocation of the I,J,K Addresses

X(U)	Y(V)	Z(W)
↓	↓	↓
I	J	K

A Rule of Memorisation:

X,Y,Z and I,J,K have the same alphabetical sequence: Thus I to X(U), J, to Y(V),K to Z (W) .

G04 - Dwell

N4	G04	$D_4$5

The time of dwell is programmed in 1/10 seconds under the parameter D_4.

G04 is a Modal Function

Possible Inputs

1 - 10 000 (1/10 - 1000 sec.)

Example:

Dwell 2 seconds.

N..../G04/D_4 20

Effect of G04

G04 is active at the end of the block,also then when G04 is not written at the end of the block.

Example:

N 10/G04/$D_4$20/M03
N 20/G00/X 50./Y 10.

Block 10: Main spindle is switched on, then 2 seconds dwell to block N 20.

G17 Switching of axis

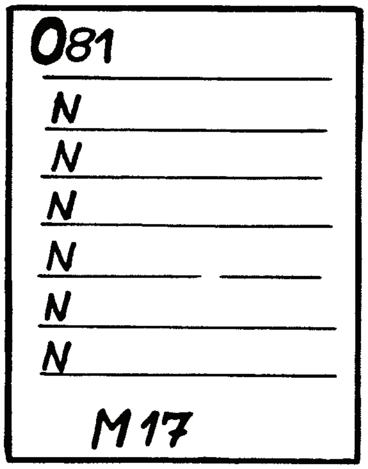
N4	G17

With G17 the axis system is established for vertical milling machines VMC-100, VMC-200.
With these machines there are no possible applications for other G-functions of the group 9 (G18-G22).

G25 Subroutine call
M17 Return command

Subroutine numbers: O 0080 - O 0255

Nesting limit: 10



A subroutine is called by the main program or a subroutine. In principle, the subroutine, as such, has the same structure as a main program.

It consists of:

+ Program number:

  Possible program numbers
  0 80 - 0 99 (see also remark)

+ Blocks

+ M17:
  Program end with return command.

Subroutine call G25

A subroutine is called by the main program or a subroutine.

Format:
N4/.../G25/L4

Subroutine call:

+ G25 Subroutine call

+ L.... Address for subroutine number and number of runs.

L .. ..

Subroutine number 80 - 99

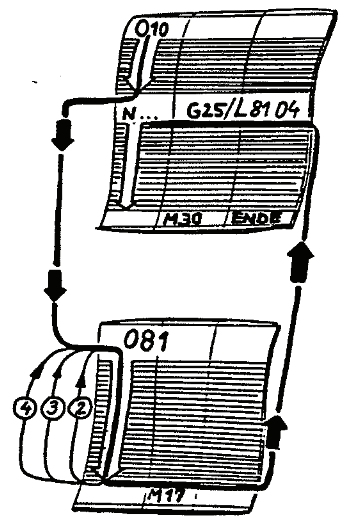
Number of runs (1 - 99).

Example 0 81:

Subroutine with 4 runs

MAIN PROGRAM
0 1Ø

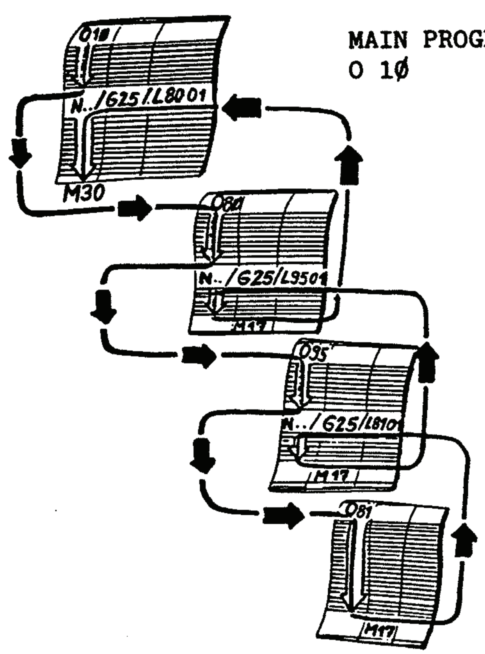
Subroutine 0 81



Example:

Nesting of subroutines

From subroutines, additional subroutines can be called. (Nesting of subroutines)
The EMCOTRONIC permits a ten-fold nesting.



MAIN PROGRAM
0 1Ø

Sub 1 0 8Ø
Nesting limit 1

Sub 2 0 95
Nesting limit 2

Sub 3 0 81
Nesting limit 3

Remark:

Program numbers for subroutines

For easier identification, main programs and subroutines should be numbered so as to keep them apart.

For this reason, the following is specified by the manufacturer:

Possible main program numbers O 0000 - O 6999
Possible subroutine numbers O 80 - O 255

The numbers O 0000 - O 6999 can be used for the main program (sensibly, the numbers O 0080 - O 0255 are not used for main programs, where subroutines are also used by you):

Only the numbers O 0080 - O 0255 can be used as subroutine numbers, otherwise ALARM 630 is actuated.

Remark:

The numerical range for subroutines can be changed by you in the MONITOR operating mode.

Example:

You wish to put in subroutines from program number O 60:

O22 80: Erase number 80, and put in

O22 60: the number 60 under O22.

G27 Unconditional Jump

N4	G27	L4

The G27 instruction causes a jump within the program sequence.
The block number to be jumped on is programmed under the L address.

Example:

N 100/G27/L 320.

The program jumps from block N 100 to block N 320.

G40 Neutralization of the Cutter Tool Correction
G41 Cutter Path Correction left-hand
G42 Cutter Path Correction right-hand

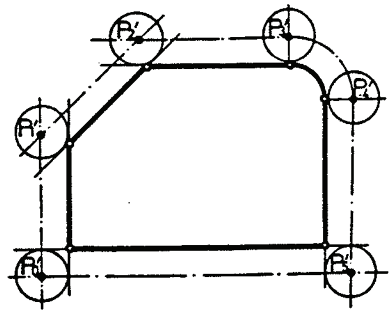
Index G40/G41/G42

Terminology:

G40 Neutralization of the Cutter Tool Correction

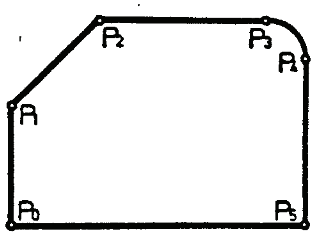
Cutter tool correction is the official DIN term. One means the cutter tool path correction and not the cutter tool length correction.

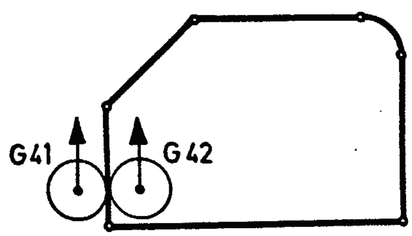
1. Purpose of Radius Compensation

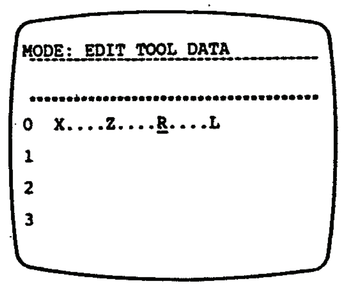


- In technical drawings the contour of the workpiece is dimensioned.
- To program without G41, G42 it is necessary to program an equidistant path. The auxiliary points P_0', P_1', P_2', etc.have to be calculated. These calculations are done by the computer, if radius compensation is programmed.

Schematic Sequence with Radius Compensation



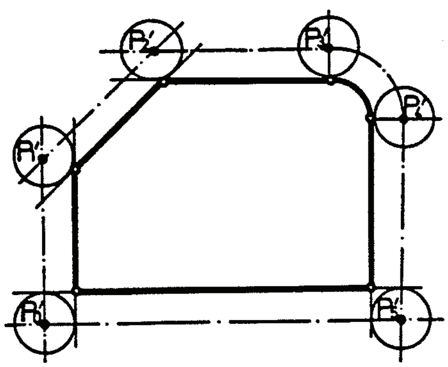




1) The contour points P_0, P_1,P_2 are programmed.

2) The information whether the contour has to be left or right hand of the cutter is given with G41 or G42.

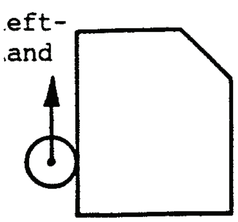
3) The information on the radius is called by the computer from the tool register.



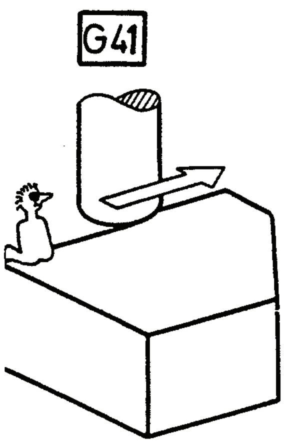
Based on these data the computer calculates the center path of the cutter P_0', P_1', P_2', etc.

2. Definition G41,G42

Cutter Path Correction
t-hand



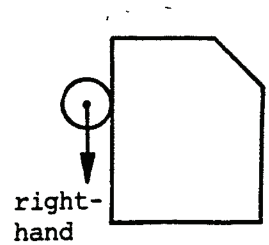
tool is on the left-
l side of the work-
:e, looked at in direc-
ı of the relative tool
ement.



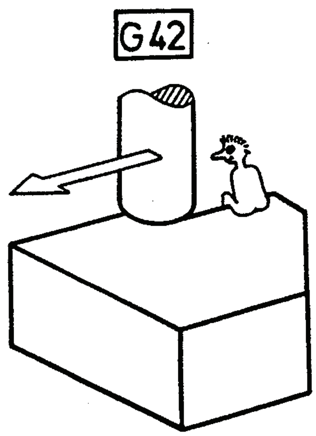
: of Memorisation:

on the workpiece and
: in direction of the
:.
he tool is left-hand
1.

G42 Cutter Path Correction right-hand



The tool is on the right-hand side of the workpiece. looked at in direction of relative tool traverse.



Rule of Memorisation:

Look after the cutter, the cutter is on the right-hand side of the workpiece - G42.

G40 Neutralization of the Cutter Correction

The programmed path is again the center path of the cutter.
M30 neutralizes also the cutter correction (G40 status).

3. General Remarks for a Better Understanding of the Radius Compensation

3.1 Plane of Compensation

* The compensation is only active in the selected main plane. This is always the XY-plane. This plane we call plane of compensation.

3.2 G41,G42,G40 Modal Codes

* G40,G41,G42 are modal selfmaintaining commands;
  Initial status is G40

Disactivation of G41,G42

1) G40
2) M30
3) RESET-key
4) Emergency-off key or main switch

3.3 Type of Interpolation

* The mentioned examples and rules are described in the absolute system. Of course, similar issues are valid for the incremental programming.

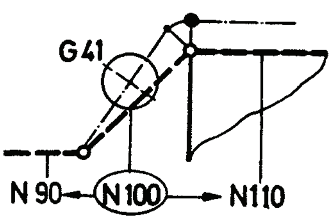
3.4 Activation/Disactivation Rules

Activation and Disactivation has always to be done in conjunction with G00,G01. When activating or disactivating, the G00,G01 instructions have to contain X or Y or XY changes as to the previous traverse instruction. If only the Z-value is changed --> ALARM 520.

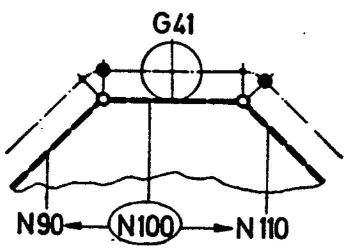
3.5 What Happens in the Computer

When a radius compensation is on, the computer knows the previous and the following traverse instruction in the plane of compensation.

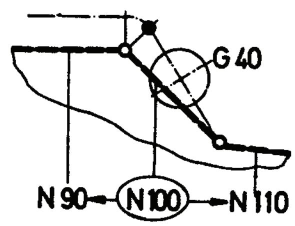
ıctivation Block



Whilst G41, G42 are active



When Disactivating



Pay Attention:

You must not program more than five "empty blocks" between the call and the first change of XY-value, or while G41, G42 is active.
"Empty blocks" are without a change in the XY-value. Pure "Z-blocks"are empty blocks, too.

ɛample: Activation

N ... G40
N 90/ G00 / X_0 / Y_0
N 100 G41
) N 110 M03 / S 1000
) N 120 M39
) N 130 G94 / F 120
) N 140 G00 / X_0 / Y_0 / Z_1
) N 150 M08
) N 160 F 180
N 170 G00 / X_1 Y_1 Z_1

ıe computer indicates alarm 500

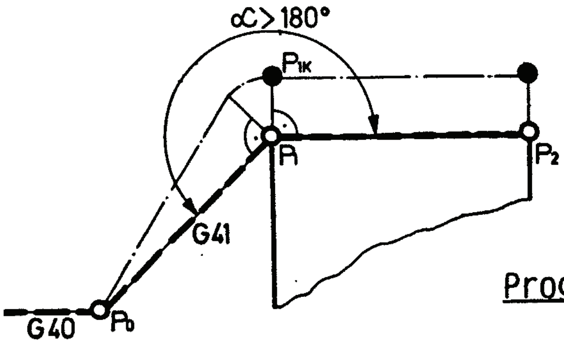
Example: 5 "Empty Blocks" with G41,G42 activ

N .../ G41
N 90 / G00 / X_0 Y_0 Z_0
N 100 / G00 / X_1 Y_1 Z_1
N 110 / M04
N 120 / G94 ....
N 130 ....
N 140 .... Alarm 510
N 150 ....Z_2
N 160 ....
N 170 / G01 / X_2 / Y_2 / Z_2

4. Activation and Disactivation, Cutter Paths, Position Read-out

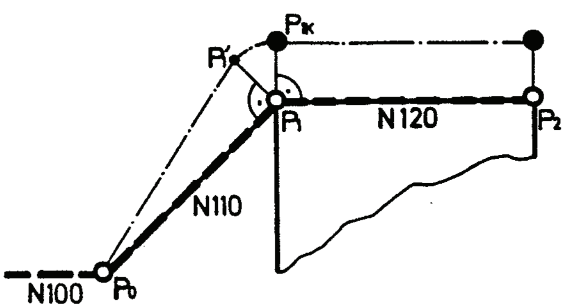
4.1 Activation of Cutter Radius Compensation G41/G42

4.1.1 External Corner $\alpha > 180°$
Definition External Corner



The angle α between programmed approach path ($\overline{P_0P_1}$) and contour ($\overline{P_1P_2}$) is larger 180°.

Programming:



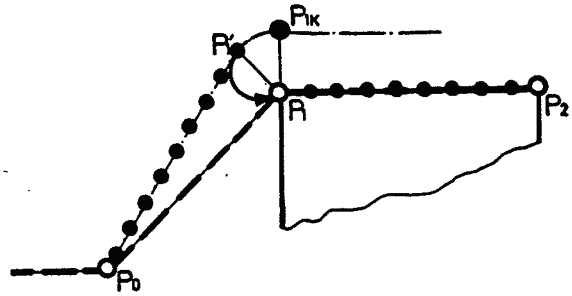
1) Approach of point P_1'.
   $\overline{P_1P_1'}$ are perpendicular to the programmed path $\overline{P_0P_1}$ at the distance of the cutter radius.

2) Cutter moves around point P_1 until cutter radius stands at right angle to distance $\overline{P_1P_2}$.
   If P_{1K} would be approached directly then a damage to the contour would occur.

Programmed path P_0P_1

P_0	N 100/G../P_0/(G40)
$\overline{P_0P_1}$	N 110/G../P_1/G41
$\overline{P_1P_2}$	N 120/G../P_2/P_2/

Position Read-out

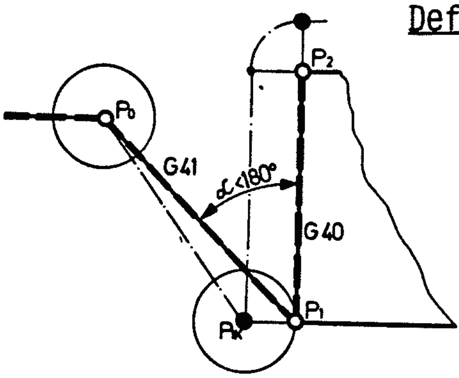


Center path of cutter until begin of transition circle; then the digital read-out jumps to P_1. During the transition circle P_1 is indicated.

- Programmed path
- Center path of cutter
- Digital position read-out
- Programmed end of block P_X
- End of block center point of cutter P_{XK}

4.1.2 Interior Corner α < 180°

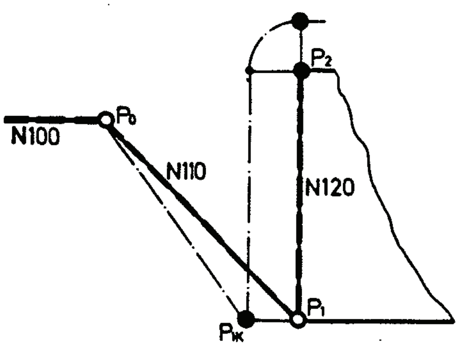
Definition:



The angle α between programmed approach path and P_1P_2 is smaller 180°

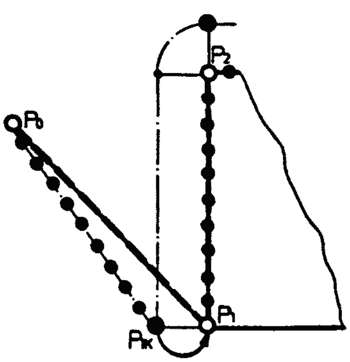
Programming:

N 100/G../XYZP_0/(G40)
N 110/G../XYZP_1/G41
N 120/G../XYZP_2/....



Programmed path: $\overline{P_0P_1}$
The point P_{1K} is directly approached, $\overline{P_1P_{1K}}$ is perpendicular to the programmed path $\overline{P_1P_2}$.

Position Read-out



Center path of cutter $\overline{P_0\ P_{1K}}$
When P_{1K} is reached the read-out jumps to P_1.

- Programmed path
- Center path of cutter
- Digital position read-out
- Programmed end of block P_X
- End of block center point of cutter P_{XK}