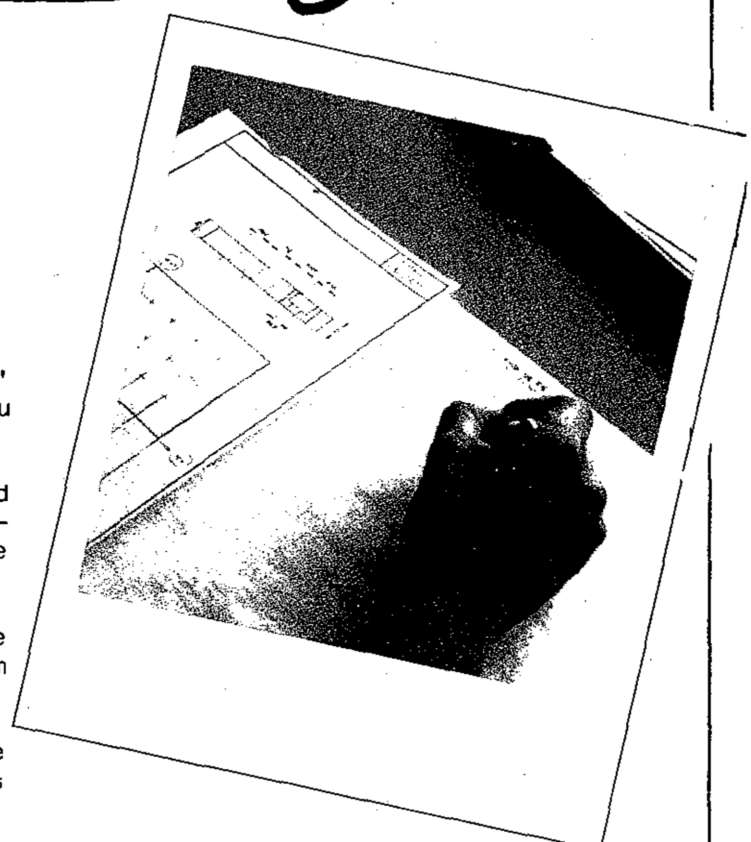
6. Programming



In this chapter you will find:

- first of all, what 'NC programming' means (page 6-1) and what you should know about it (page 6-6);
- what motions your tool (with and without tool compensation) can perform and how these tool motions are obtainable (as from page 6-10);
- what programming aids you can use to facilitate your job (as from page 6-10);
- what preprogrammed cycles are available for routine operations (as from page 6-84);
- how you can use the 4th axis (C axis) (page 6-144).

Programm

On conventional milling machines, all operations used to be performed consecutively by manual control.
Modern milling machines are equipped with a numerical control (NC) unit. The entire sequence of operations required for machining a workpiece is described in a program.
With the aid of such a program, the control can machine the workpiece automatically.

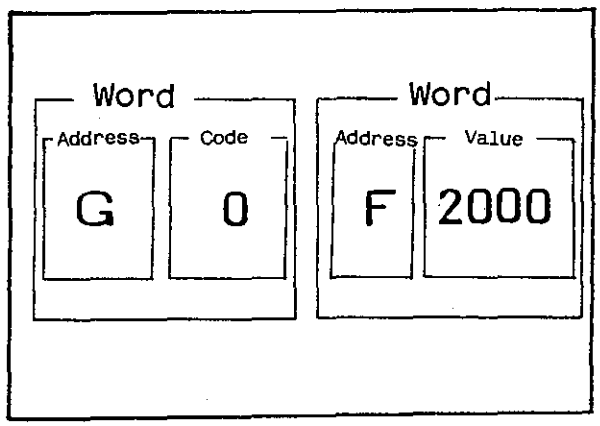
Blocks

The program for machining a workpiece is written in a language understood by the control system.
Programs are subdivided into blocks containing all necessary data such as feed rate, spindle speed , cutter path etc.

Words

A word is composed of an address and a number. The number may be

- either a code, for example G0, which means 'Moving at rapid traverse rate',
- or a value, for example F2000, which means 'Feed rate 2000 mm/min'.

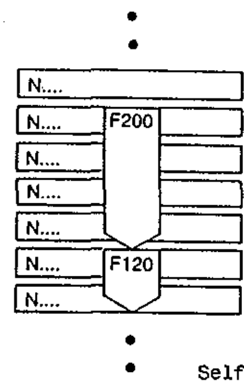


Leading zeros in words or block numbers may be omitted in writing a program, so you may write N1 instead of N0001, or G0 instead of G00.

Commands - self-retaining/ effective in only one block

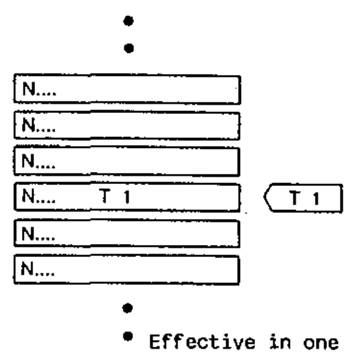
Each word in a block is a command for the control system of the milling machine.

Self-retaining commands, sometimes also referred to as modal commands, are effective until they are either cancelled or replaced by a different command with the same address letter.



Self-retaining

Commands effective in only one block are only active in the block in which they stand.



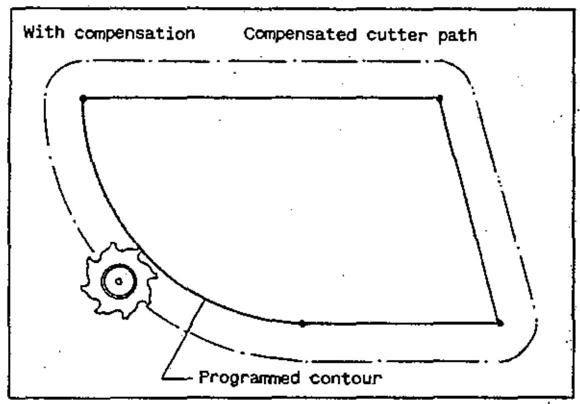
Effective in one block only

Tool compensation

To obtain the one and same contour on a workpiece with different cutting tools, these tools have to describe different paths. However, to facilitate programming, it will be sufficient if you simply program the workpiece contour and let the control do the rest: it can automatically calculate the cutter path required for machining such contour.
The different tool radii and tool lengths can be stored either in the tool store T or in the compensation store D of the control. In tool store T, the data are assigned to the tool number, in tool compensation store D to a compensation number. The tool compensation values can then be called in your program by entering either the corresponding tool number T ... or the corresponding compensation number D ...

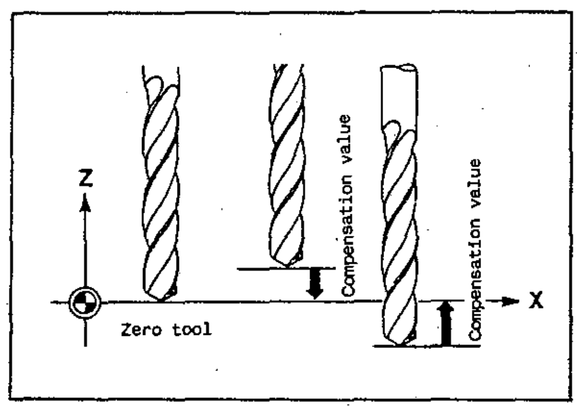
Compensation on contour

When compensation on the contour is called, the control compensates for the cutter radius.



Tool length compensation

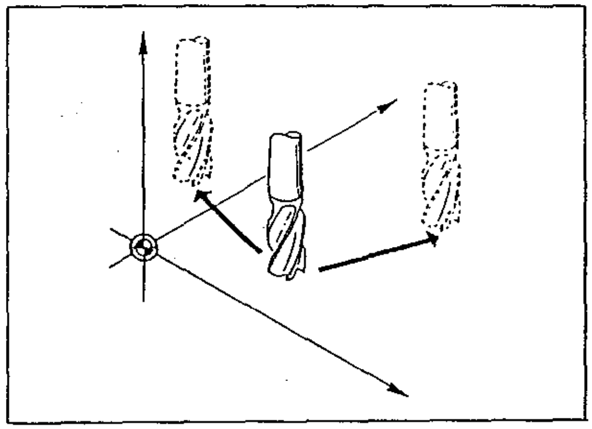
When tool length compensation is called, the control compensates for a deviating tool length.



If you have determined the zero point by contacting the workpiece surface with a tool (page 4-12), the reference length is the length of the tool used in setting up the machine. Any tool lengths deviating from this 'zero tool' are stored by the control and applied as required.
If a preset tool has been used for determining the zero point (page 4-10), the control stores the absolute lengths of all tools and applies them as required.

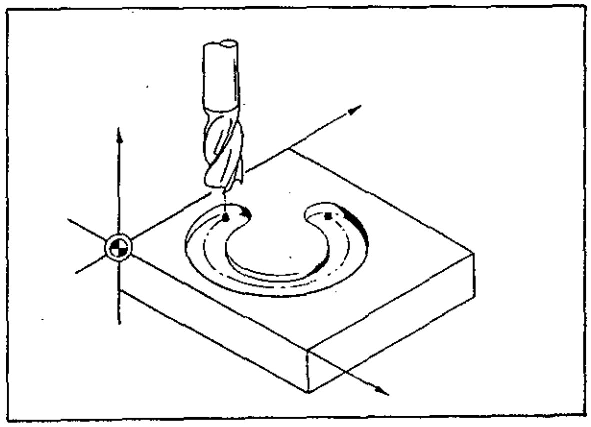
Tool motions

Straight motions



The tool can move along a straight path to any point within the working area of the machine.
On the basis of the coordinates of the start-off and target points, the control calculates the cutter path and the way in which the feed motions have to be coupled to move the tool along a desired path.
This process is called 'linear interpolation' (for programming examples see page 6-12).

Circular motions



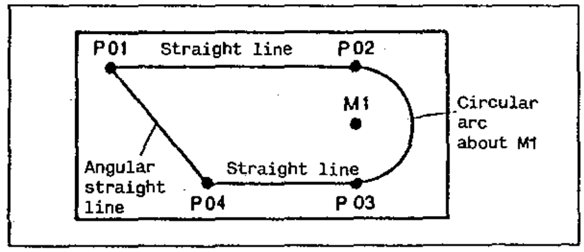
The tool can also move along a circular path, provided such circular motion is in one of the main planes of the coordinate system.
The control calculates the cutter path and the necessary linkage of the feed motions if the starting point, the end point and the centre of the full circle or circular arc are known.
This process is called 'circular interpolation' (for programming examples see page 6-18).

Rapid traverse motions

Rapid traverse means that each machine slide moves at maximum speed until it has reached its target position.

Program structure

A workpiece contour is subdivided into straight lines and circular arcs.
In each machining step (program block), the control can move the cutting tool along one of these geometrically simple contour elements.
The blocks are programmed one by one.



Dividing up the contour: subdivision of machining operation into individual steps.

Preparation of program.

Input of program into control

Start of program

The control system controls the machining operation.

Program structure

1st block

- Tool motion to point P01 (shortest way)
- Rapid traverse rate
- Spindle on
- Coordinates of point P01

2nd block

- Tool motion to point P02 (straight cut)
- Desired feed rate
- Coordinates of point P02

3rd block

- Tool motion to point P03 (circular cut about M1)
- Desired feed rate
- Coordinates of point P03
- Coordinates of point M1

4th block

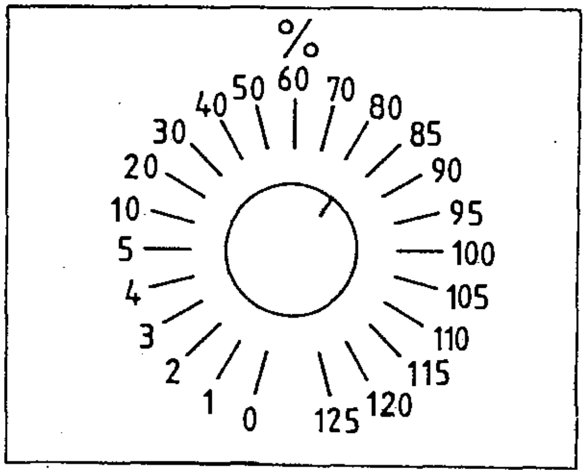
- etc

Block number - N

The block number is an identification number and can be freely selected between N0000 and N9999.
It has no effect on the sequence of the blocks in a program.
The sequence of the machining steps is determined by the sequence of input.
Leading zeros may be omitted, so you may write N1 instead of N0001.

Each block number should be used only once in a program.

Feed rate - F



Feed rates are programmed under an F address.
The lowest feed rate obtainable is F2.
Within the feed range of your machine (see 'Technical data') you may program any feed value (whole numbers only).
The maximum feed value available on your machine can be used for fast positioning.

Overriding control (6) permits the variation of the programmed feed rate: the feed rate can be reduced to 0 % or increased to 125 %.

Feed rates are self-retaining (page 6-2).

Feed motions are only obtainable while the spindle is running.

For calculating the feed rate F you may use the Cutting Data Calculator of the DIALOG 4 control (see Chapter 7).

Rapid traverse rate - G0

Command G00 (or G0) is used for programming tool motions at rapid traverse rate.

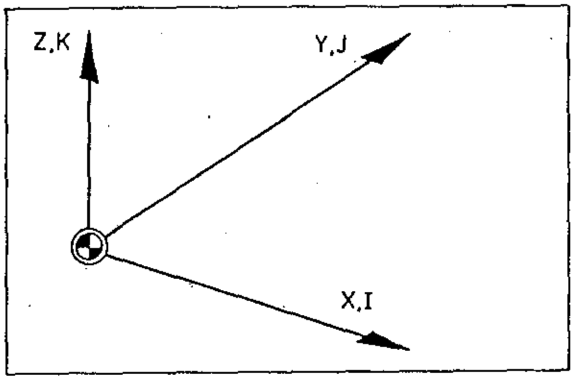
Each slide moves at its maximum rate until it has reached its target position. This means that the tool does NOT move to the target point along a straight line.

Rapid traverse motions are also obtainable while the spindle is not running.

The target point for rapid traverse motions can be programmed
- in absolute or chain dimensions,
- in Cartesian or polar coordinates.

Coordinates – X/Y/Z

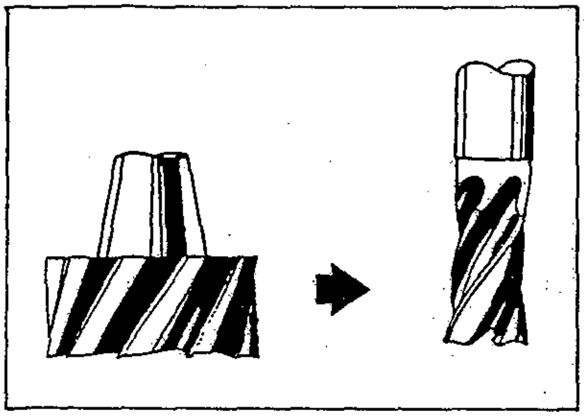
Coordinate values are programmed under the addresses X, Y and Z, either in absolute dimensions or in incremental (chain) dimensions.



Coordinates – I/J/K

The coordinates of a pole or centre of circle are programmed under the addresses I, J and K.

Tool change – T



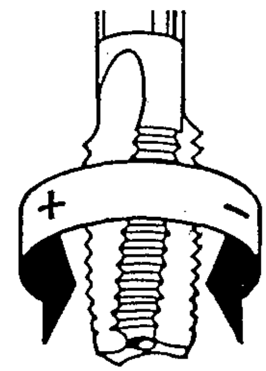
A tool change is programmed by a T address and a four digit numeral (T1 to T9999) identifying the tool to be used for the next machining operation. Together with the T address and the tool number you call the associated tool compensation values from the tool store. When a block contains a T address, the program is interrupted upon completion of that block: 'Tool change' button (21) as well as the indicating lamp in the remote-control handbox start flashing. The operator has to mount the tool called.
Upon acknowledgement of the tool change by means of button (21), the program can be continued by pressing 'Cycle start' button (20).

A tool change T cancels the previously programmed feed rate F and the programmed spindle speed S. For this reason, the first block after a tool change must always be a main block.
Command T0 cancels the call of tool compensation values: compensation value = 0.
Tool compensation values called by a T address are self-retaining as from the end of the block in which they are called.

Coolant supply – M8, M9

Command M8 is used for starting, M9 for stopping, the coolant supply.
M8 and M9 are self-retaining.

Spindle speed - S



Spindle speeds are programmed under an S address directly in rpm. The following spindle speeds are obtainable on FP2/3/4 NC and FP42 NC machines (equipped with a 3-phase AC drive motor):

0, 31.5, 40, 50, 63, 80, 100, 125, 160, 200, 250, 315, 400, 500, 630, 800, 1000, 1250, 1600, 2000, 2500, 3150 rpm.

If intermediate values are programmed, the control will automatically use the next lower speed; for values below 31.5, the control will use 31.5 rpm.

For the vertical spindle, the range can be extended to 6300 rpm by means of a mechanical selector. The above table of speeds will then start at 63 rpm and continued with 4000, 5000 and 6300 rpm.
The control has to be informed about any such change by selecting mode 16 and entering command S1 or S2 (see Chapter 7).

S+0 serves to disengage the spindle from the transmission in program-controlled operation (shifting to neutral).

On FP5/6/7 NC machines (equipped with a DC drive motor), the spindle speed is steplessly programmable.

Speed ranges:	FP5 NC:	18-6300 rpm
	FP6/7 NC:	35-1800 rpm
		or 35-2500 rpm

Important: When the grinding head is used (on FP2/3/4 NC and FP42 NC), the maximum permissible spindle speed is 1000 rpm. Failure to observe this instruction may lead to accidents.

Direction of spindle rotation:
+ means clockwise rotation;
- means counterclockwise rotation.

To save time, always program the spindle speed one block before the tool contacts the workpiece.
Input for spindle stops: S0 without sign.
Spindle speeds are self-retaining.

For calculating the spindle speed S you may use the Cutting Data Calculator of the DIALOG 4 control (see Chapter 7).

Lube pulse - M7

Lube pulses can be programmed by command M7. After a block with M7, a lube pulse is released, either the next time all machine slides are standing still or during the next rapid traverse motion.

These lube pulses are indispensable in operations involving numerous short slide motions (less than 15 mm), otherwise the oil film might be disrupted.

To ensure adequate lubrication, a lube pulse (M7) and a long slide stroke (min 100 mm) at rapid rate should be programmed after approx 50 short strokes.

End of program - M2, M30

The end of a program is identified by the word M2 or M30: spindle and feed motions will be stopped.

Upon command M2, a program section can be repeated by calling the block where such repetition is to start, or the program can be continued.
If M30 is used, a new machining operation is initiated, starting with the first program block.

Program stop - M0

Command M0 is used for stopping a program: the machining operation is interrupted upon the completion of the block in which the command is programmed. To resume the program, press 'Cycle start' button (20).

The first block after a program interruption by M0 must be a main block.

Dwell - G4 F

Command G4 F interrupts the feed motion for a certain time.

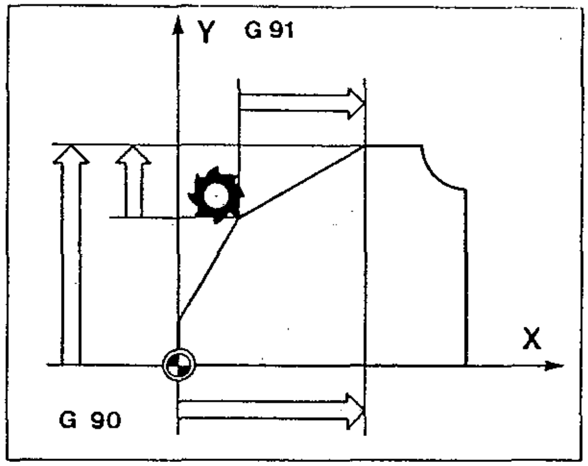
The duration of such a dwell can be programmed in seconds between 0.1 and 99.9 sec. G4 F means a dwell of 10 seconds.

The passing of the dwell time is shown in seconds in the upper right corner of the display screen.

Dwells are NOT permissible within compensations on the contour (page 6-33).

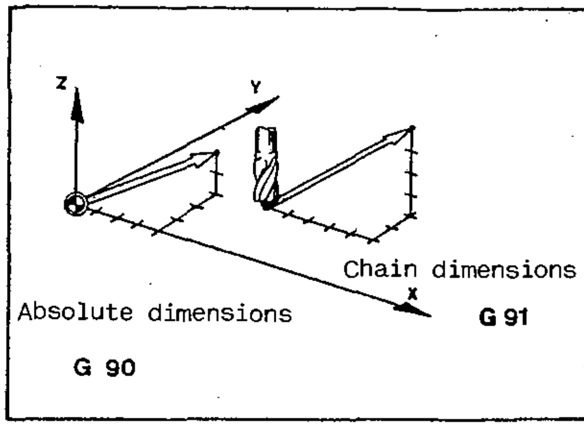
Linear interpolation – G1
Input in Cartesian coordinates

This method is recommended if the target point of an angular straight line is dimensioned in Cartesian coordinates in the part drawing.
Command G1 need not specially entered because it is activated automatically when the control is switched on.
Use addresses X, Y and Z for entering the coordinate values.



If absolute input (G90) is active, the coordinates define the point to which the tool is to move.

If incremental input (G91) is active, the coordinates define the distance by which the tool is to move.



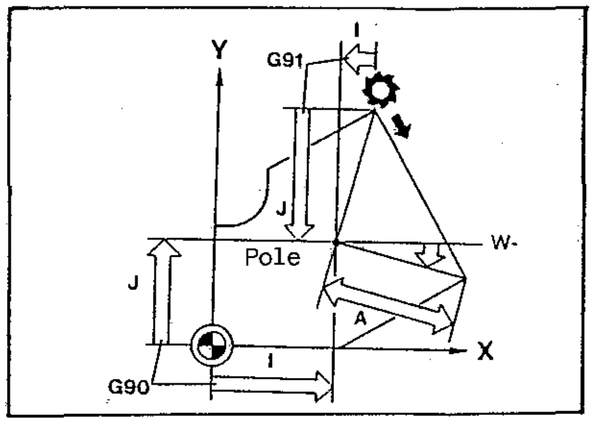
Linear interpolation – G1
Input in polar coordinates

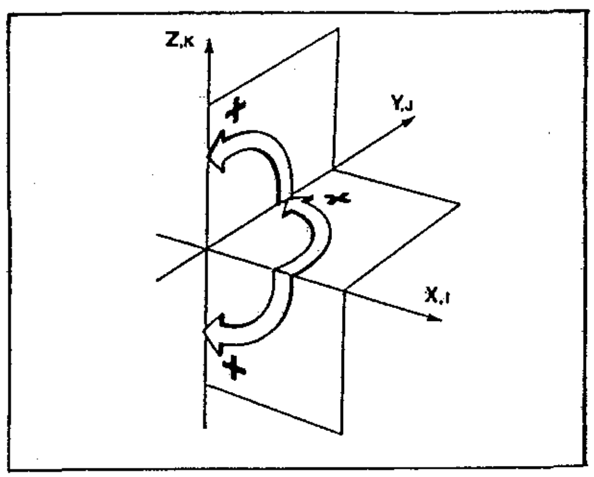
INPUT DIALOG

Upon entering and transferring command G9, the control requests the addresses listed on the right.
These addresses should be completed and transferred to the control by operating the transfer key.

The maximum angle that can be programmed is ±360°.

G9	Linear interpolation with input in polar coordinates
	The control requests:
G	Preparatory function G1
A	Distance pole/target point
W	Angle between the line from pole to target point and the angle reference axis
G9	Absolute/incremental input (G90/G91) of pole coordinates
I(J)(I)	1st coordinate of pole
J(K)(K)	2nd coordinate of pole

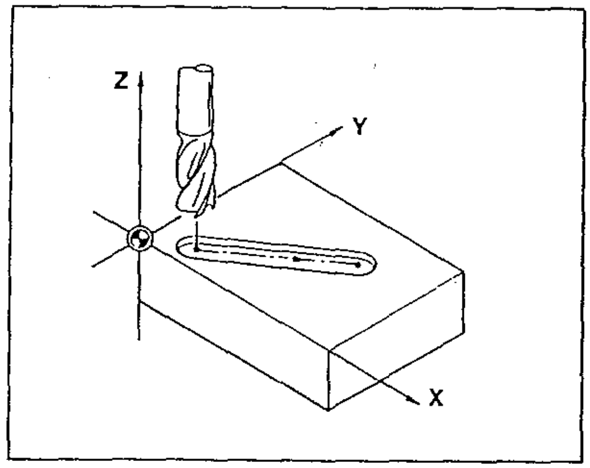




Machining plane parallel to ... plane:		Polar coordinates with addresses:		Angular reference axis parallel to ... axis	
	X,Y		I,J		X+
	Y,Z		J,K		Y+
	Z,X		I,K		X+

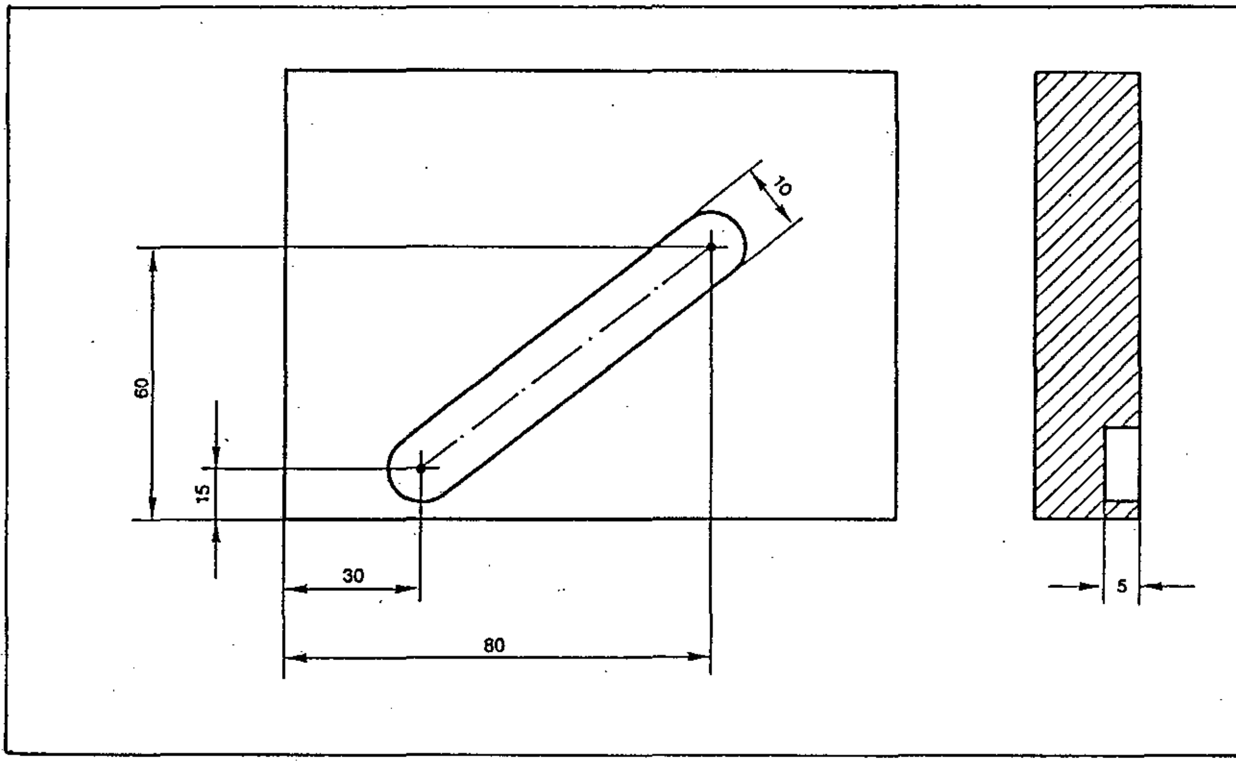
Linear interpolation – G1
Input in Cartesian coordinates

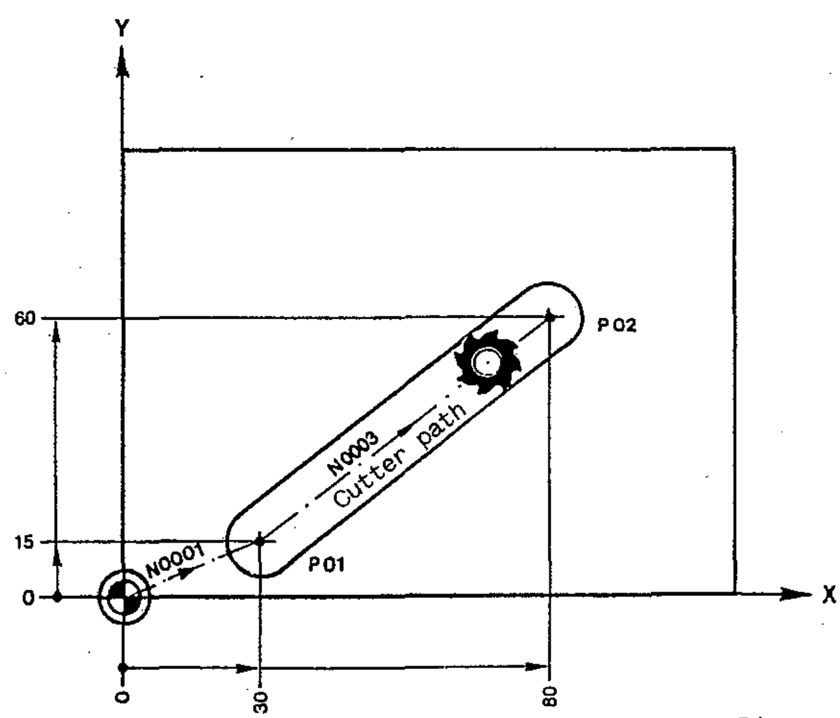
EXAMPLE
Diagonal cut



%15	Program number
N1 G0 X30 Y15 Z1 S+500	Tool moves at rapid rate to point P01 identified by coordinates X30, Y15, Z1; spindle speed S+500.
N2 Z-5 F100	Downfeed to Z-5, feed 100 mm/min.
N3 X80 Y60 F200	Tool moves along angular path from P01 to P02; (feed rate 200 mm/min).
N4 G0 Z100	Tool retracted in Z axis (rapid rate).
N5 M30	End of program.

Linear interpolation – G1
Input in Cartesian coordinates

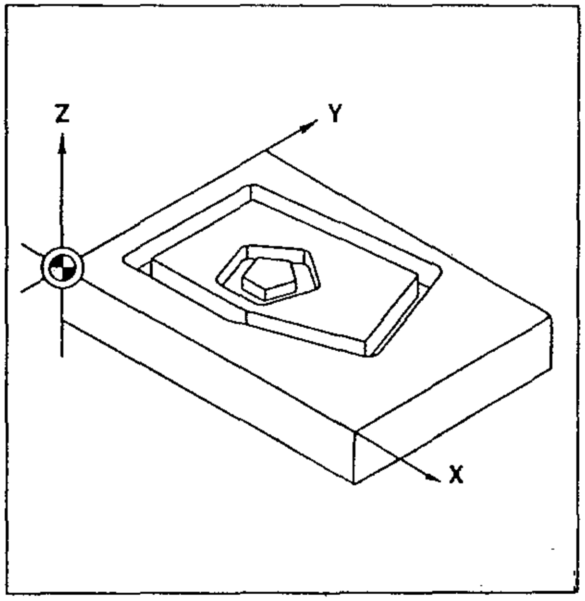




Dimensions in mm

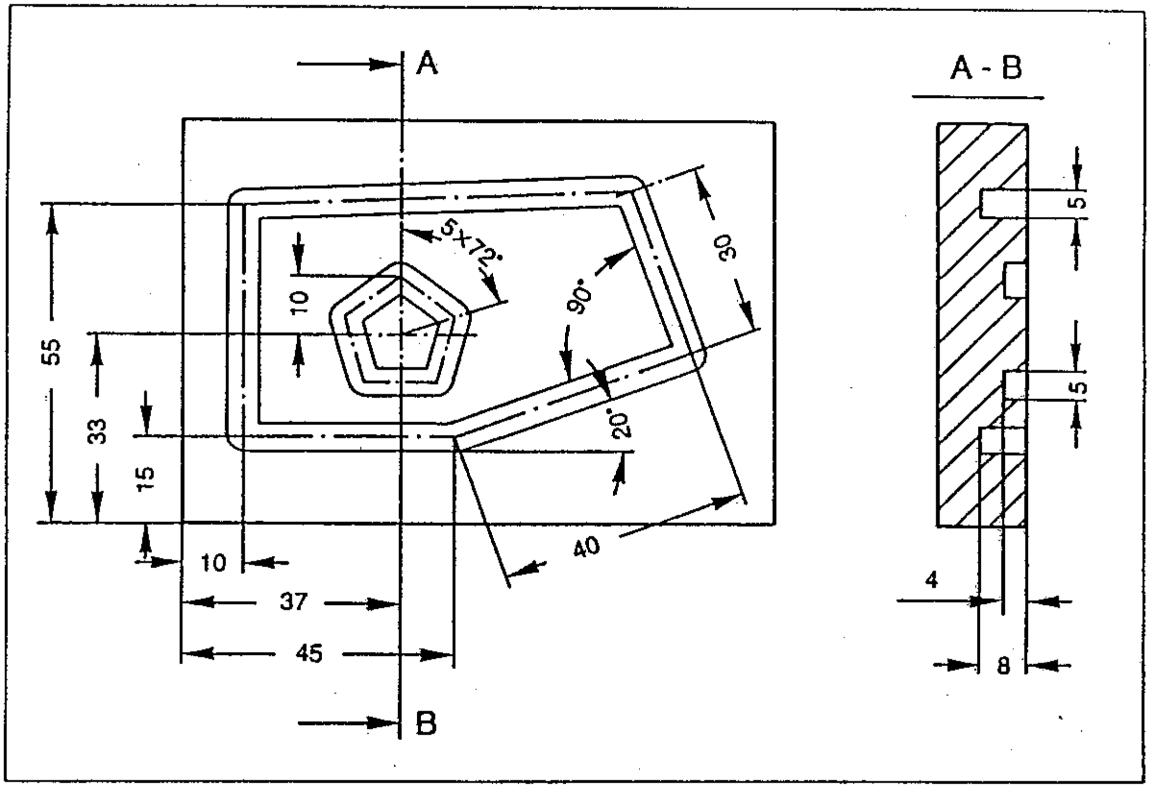
Linear interpolation – G1
Input in polar coordinates

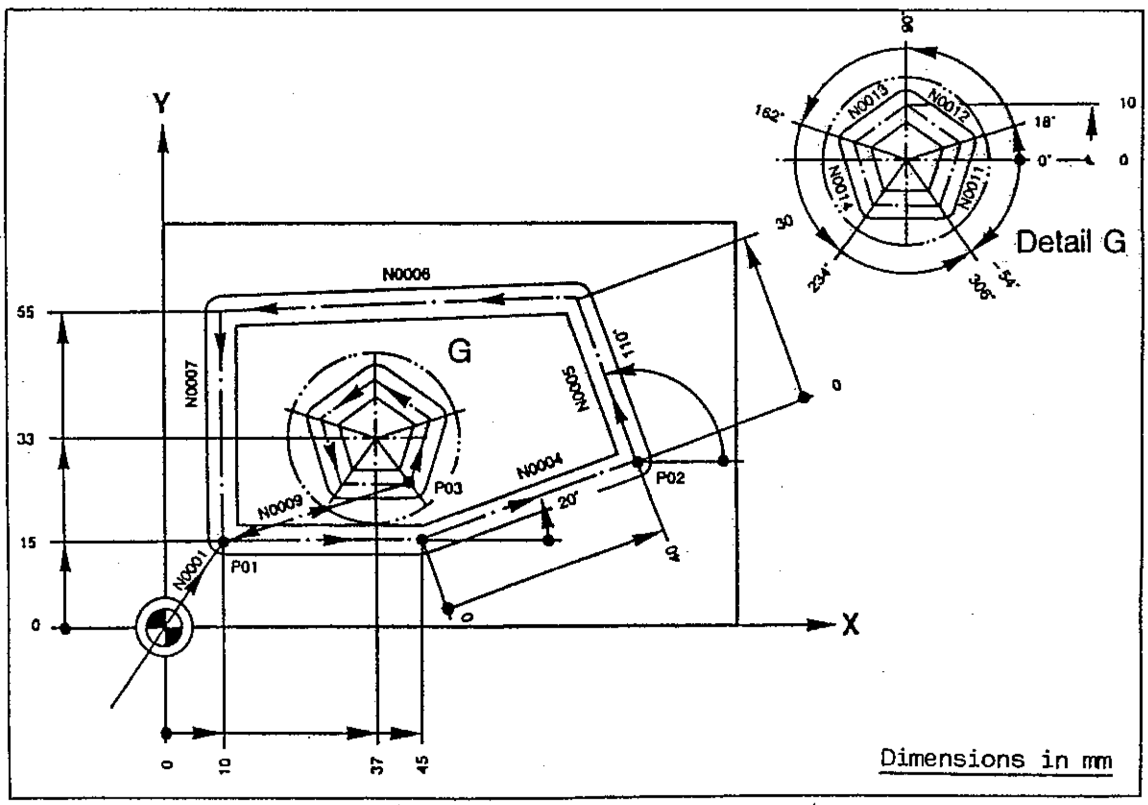
EXAMPLE
Slots



%20	Program number
N1 G0 X10 Y15 Z2 S+500 N2 Z-8 F150	Tool moves to P01 at rapid rate, spindle speed S+500. Infeed to Z-8 at feed F150.
N3 X45 F250	Tool cuts first slot, feed rate F250.
N4 G9 G1 A40 W20 G90 I45 J15	The next target point is dimensioned in polar coordinates (G9): 40 mm away from pole, angle 20° from angle reference axis (X axis). Pole dimensioned in absolute values (G90) from program zero, having coordinates I45 and J15.
N5 G9 G1 A30 W110 G91 I0 J0	Target point dimensioned in polar coordinates. Since coordinates of pole are not known and current tool position coincides with pole, current position (P02) is used as pole. Incremental input (G91) of pole coordinates from tool position.
N6 X10 Y55 N7 Y15	Finish milling of outer groove.
N8 G0 Z2	Tool retracted in Z.
N9 G9 G0 A10 W-54 G90 I37 J33	Tool positioned above P03 at rapid rate. P03 dimensioned in polar coordinates from centre of pentagonal slot.
N10 Z-4 F150	Downfeed in Z.
N11 G9 G1 A10 W18 G90 I37 J33 N12 G9 G1 A10 W90 G90 I37 J33 N13 G9 G1 A10 W162 G90 I37 J33 N14 G9 G1 A10 W234 G90 I37 J33 N15 G9 G1 A10 W306 G90 I37 J33	Milling of internal slot: all corners of pentagonal slot defined by polar coordinates from centre.
N16 G0 Z100 M30	Tool retracted in Z and end of program.

Linear interpolation – G1
Input in polar coordinates





Dimensions in mm

Circular interpolation - G2, G3

Commands G2 and G3 can be used to program circular cutter paths. The circular contour must be in one of the main planes of the coordinate system.

The contour is defined by

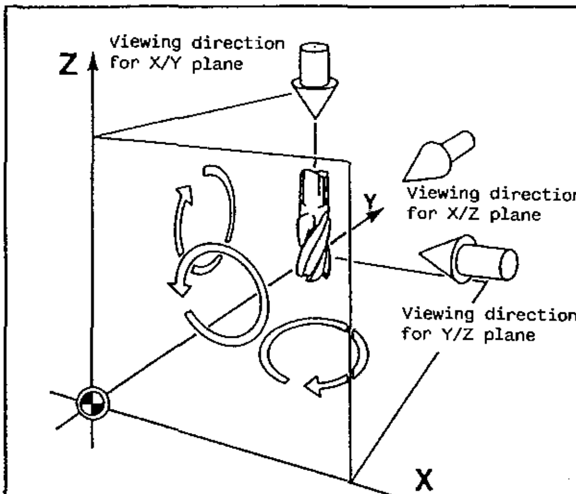
- the radius (from 0.2 to 16 000.00 mm),
- the starting point (tool position from preceding block),

and, in the case of circular arcs,

- by the target point.

G2 means clockwise travel,
G3 counterclockwise travel,

in the main plane concerned.



Direction of travel (G2) in main planes

The viewing direction is always the direction opposite the third coordinate axis (axis NOT used in defining the plane).

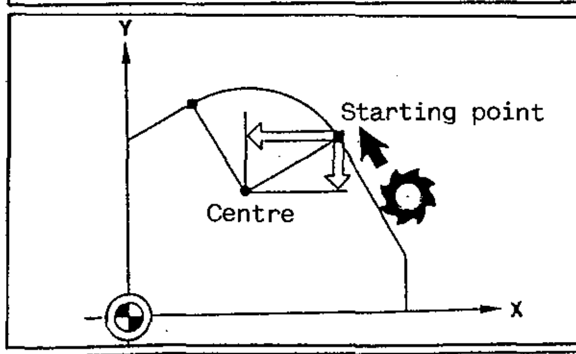
Circular interpolation - G2, G3 Input in Cartesian coordinates

INPUT DIALOG

Upon entering and transferring command G2 or G3, the control requests the addresses listed on the right.
These addresses should be completed and transferred to the control by operating the transfer key.

G2 G3	Clockwise rotation or counterclockwise rotation
	The control requests:
F	Feed rate (if not required: transfer key
X(Y)X	1st coordinate of target point (= end of circular path) (full circle: transfer key)
Y(Z)(Z)	2nd coordinate of target point
I(J)(I)	1st coordinate of centre
J(K)(K)	2nd coordinate of centre

The coordinates of the circle centre should always be programmed from the circle starting point.



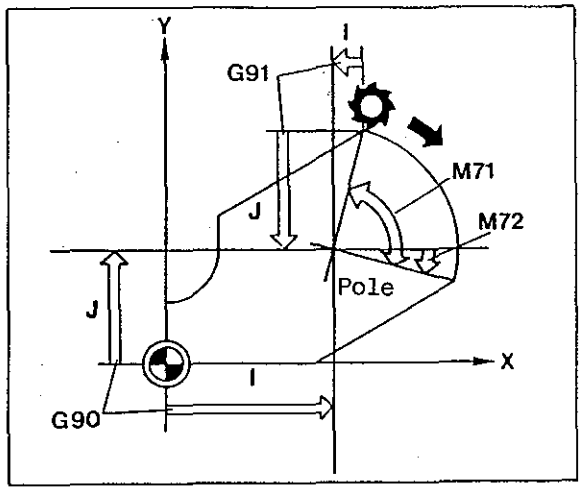
Circular interpolation - G2, G3
Input in polar coordinates

This method is recommended if either the starting point and the target point, or only the target point, of a circular path is dimensioned in polar coordinates (distance and angle) from the centre point.

The angle can be defined in two ways:

(a) angle of sector between starting and target points of circular arc (M71);

(b) angle between between angle reference axis and target point of circular arc (M72).



INPUT DIALOG

Upon entering and transferring command G9, the control requests the addresses listed on the right.
These addresses should be completed and transferred to the control by operating the transfer key.

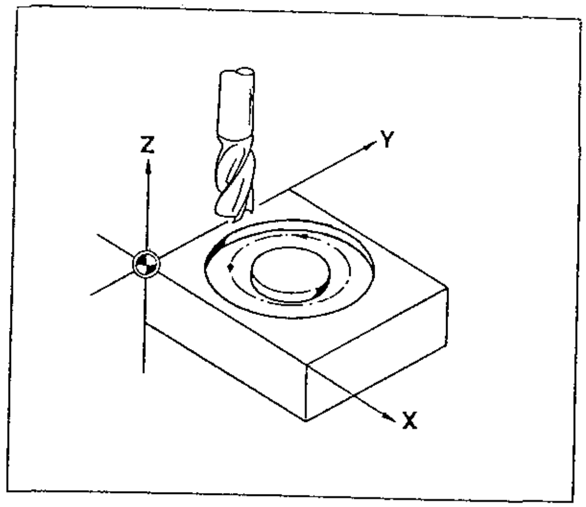
G9	Circular interpolation - G02/G03 with input in polar coordinates The control requests:
G	G2 clockwise or G3 counterclockwise cut
M7	Kind of angle definition used M71: angle of sector M72: angle from reference axis
W	Angle counterclockwise (G3): positive, clockwise (G2): negative
G9	Absolute/incremental input (G90/G91) for circle centre coordinates
I(J)(I)	1st coordinate of circle centre
J(K)(K)	2nd coordinate of circle centre

Machining plane parallel to ... plane:	Polar coordinates with addresses:	Angular reference axis parallel to ... axis
X/Y	I,J	X+
Y/Z	J,K	Y+
Z/X	I,K	X+

Circular interpolation – G3

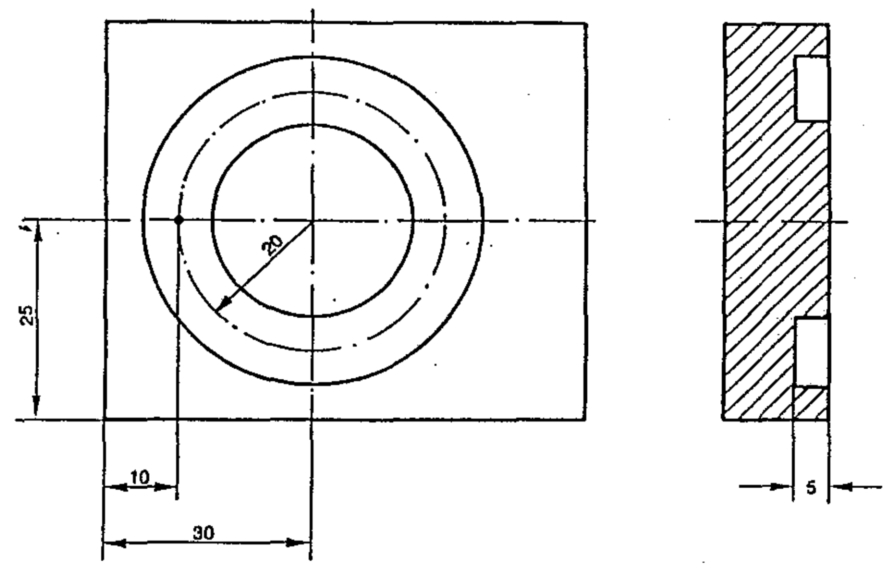
EXAMPLE
Full circle
Cartesian coordinates

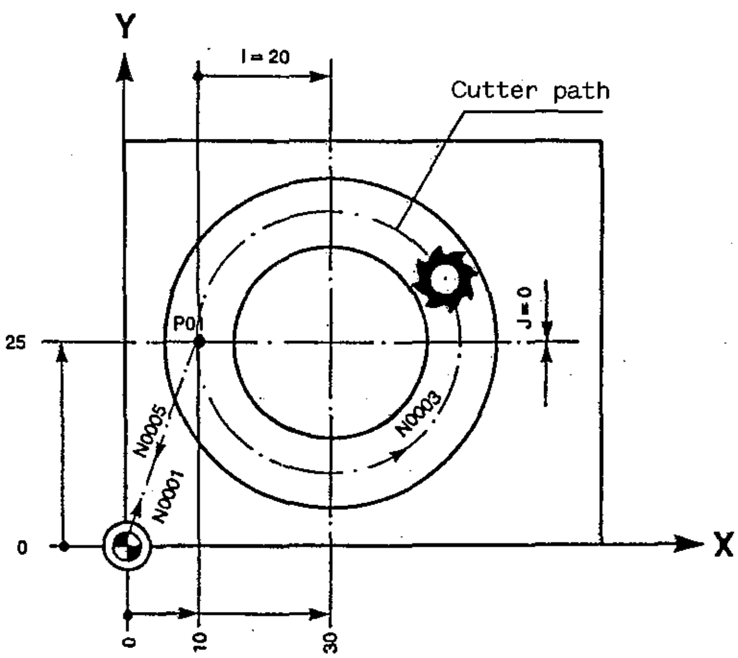
This example shows how to program a full circle. Input in Cartesian coordinates is used because both the centre and the target point of the circle are dimensioned in Cartesian values in the drawing. The coordinates of the target point need not be programmed for full circles.



%25	Program number
N1 G0 X10 Y25 Z1 S+1250	Tool moves to point P01 at rapid rate.
N2 Z-5 F100	Downfeed to Z-5.
N3 G3 F125 I20 J0	Tool moves along full circle in a counterclockwise direction (G3). Sign of I is plus because centre of circle is located 20 mm in direction of X axis from starting point of circle. J = 0 because centre of circle is at a distance of 0.0 mm in Y axis from P01.
N4 G0 Z100	Tool retracted in Z axis.
N5 G0 X0 Y0	Tool moves from P01 to program zero at rapid rate.
N6 M30	End of program.

Circular interpolation - G3

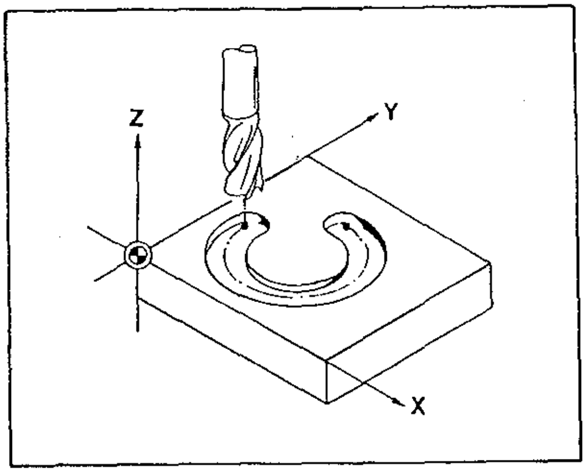




Dimensions in mm

Circular interpolation – G2

EXAMPLE
Circular arc
Cartesian coordinates

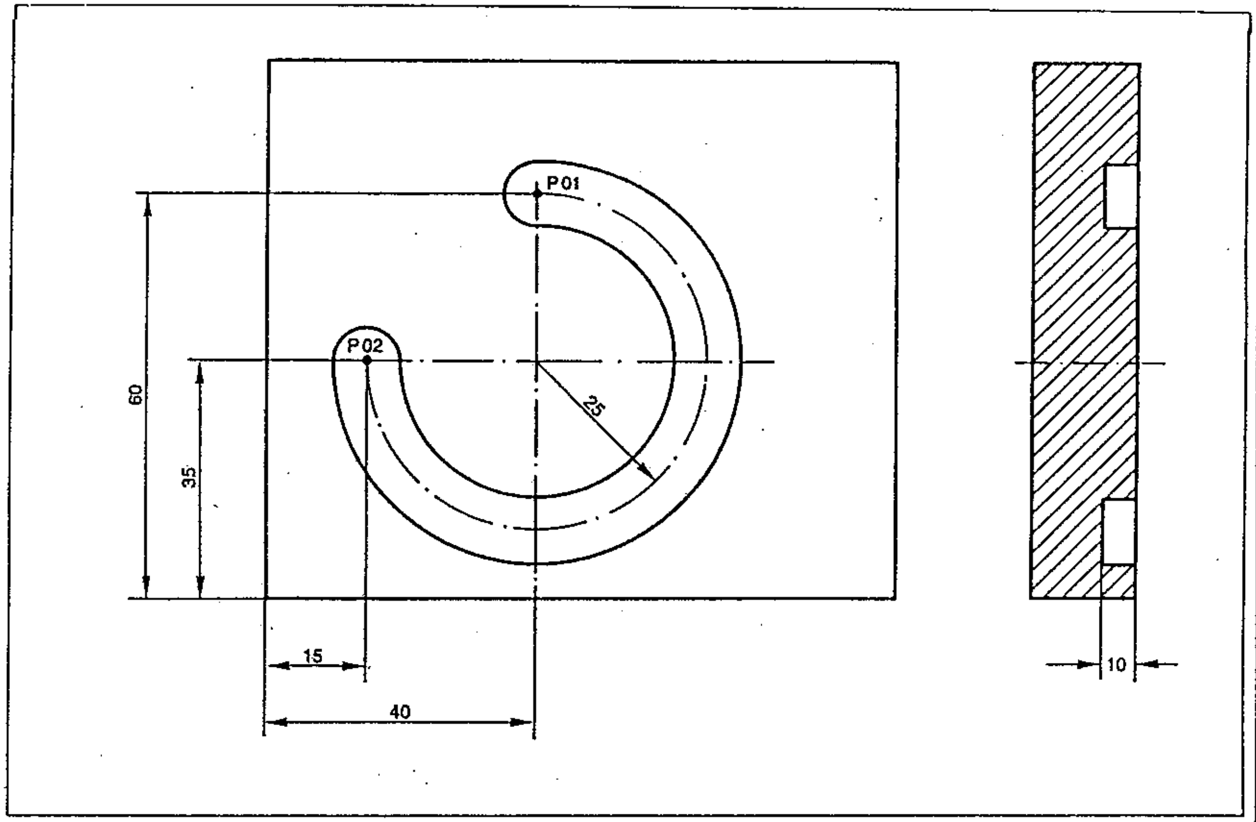


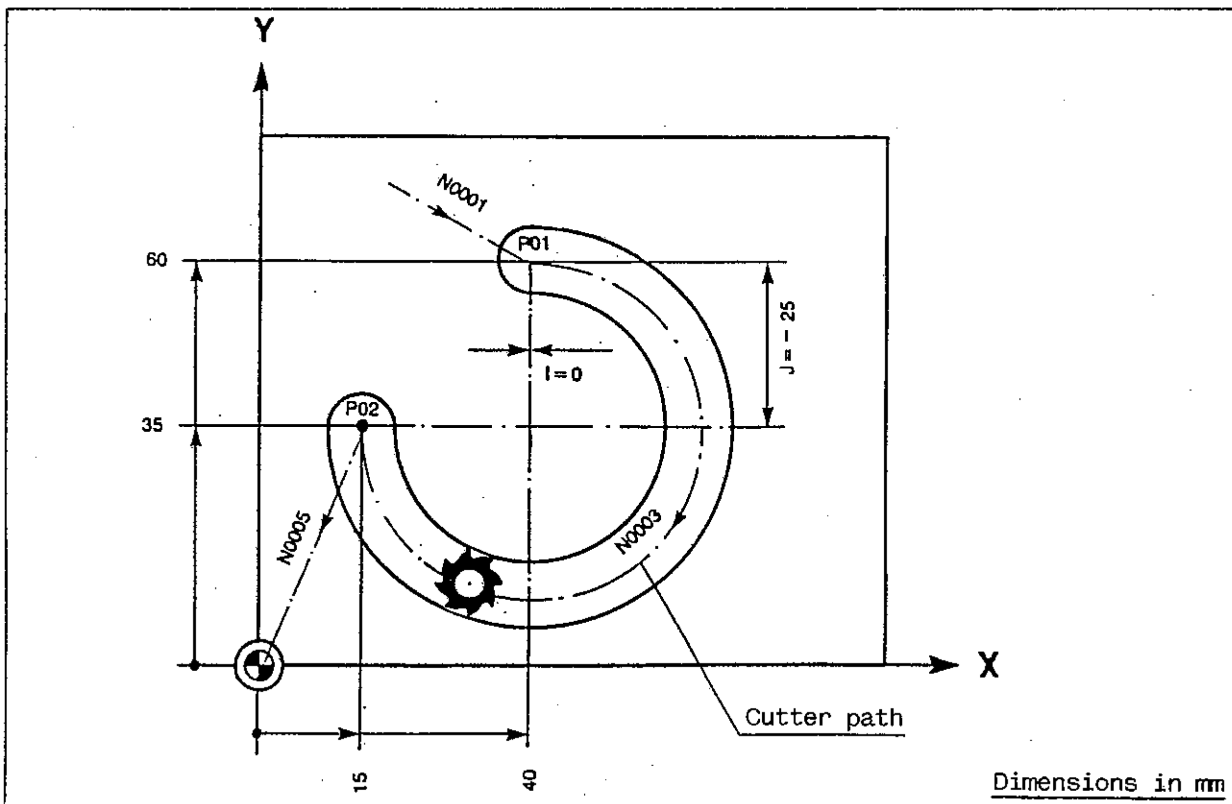
In this example, the target point (X/Y coordinates) of the tool has to be programmed because the cutter path does not constitute a full circle.

Input in Cartesian coordinates is used because both the centre and the target point of the circular arc are dimensioned in Cartesian coordinates in the part drawing.

%30	Program number
N1 G0 X40 Y60 Z1 S+1250	Tool moves to point P01 at rapid rate.
N2 Z-10 F100	Downfeed to Z-10.
N3 G2 F200 X15 Y35 I0 J-25	Tool moves along circular path from P01 to P02 in a clockwise direction (G2). I = 0 because centre of circle is at the same level in the X (I) axis as the starting point. Sign of J is minus because centre of circle is located 25 mm in direction of the negative Y (J) axis from starting point P01.
N4 G0 Z100	Tool retracted in Z axis.
N5 G0 X0 Y0	Tool moves from P02 to program zero at rapid rate.
N6 M30	End of program.

Circular interpolation - G2

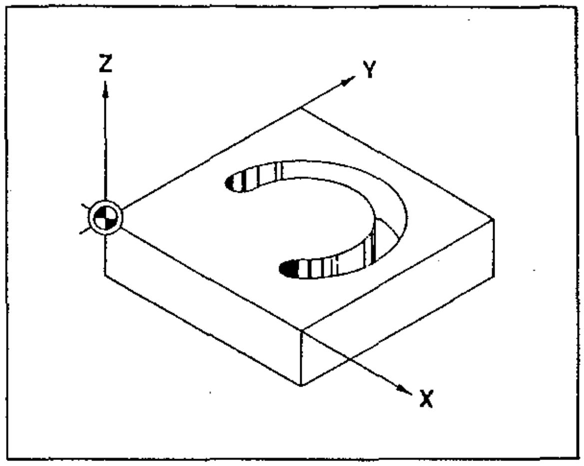




Circular interpolation – G2

EXAMPLE
Circular arc
Polar coordinates
Absolute input

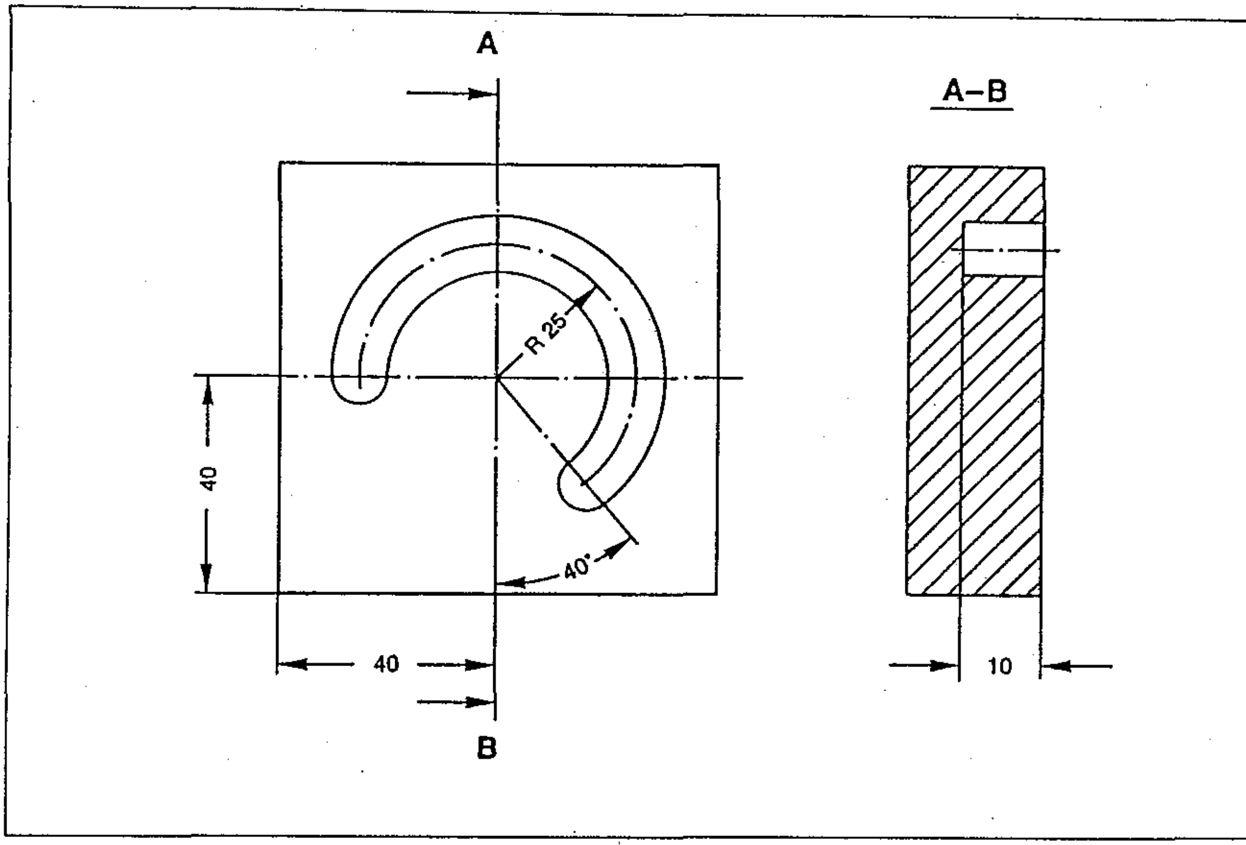
In this example, the target point of the tool is defined by a radius and an angle referred to the circle centre. It is thus advisable to use input in polar coordinates.
The angle is specified relative to a straight line parallel to the Y axis, that is why the angle is defined in relation to the angle reference axis (M72).

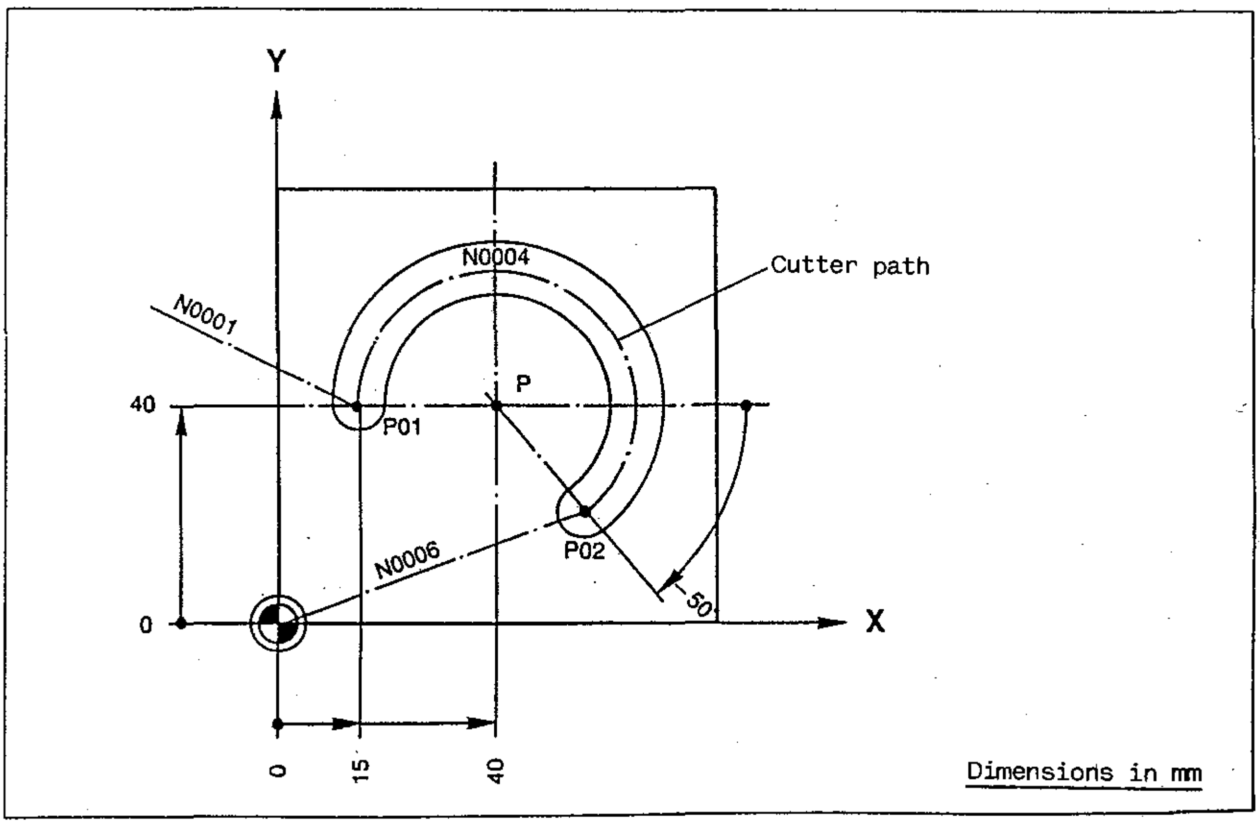


%35 — Program number

N1 G0 X15 Y40 Z1 S+800	Tool moves to point P01 at rapid rate.
N2 Z-10 F100	Downfeed to Z-10.
N3 F200	Change of feed rate.
N4 G9 G2 M72 W-50 G90 I40 J40	Circular interpolation clockwise with input in polar coordinates. Angle defined by absolute values from X axis (M72; I, J). Polar coordinates defined in absolute values. Tool moves from P01 to P02 along circular path about P.
N5 G0 Z100 N6 G0 X0 Y0	Tool retracted in Z axis and moved above program zero.
N7 M30	End of program.

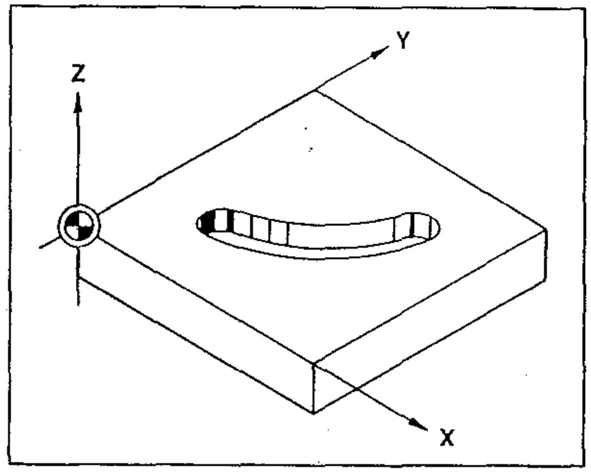
Circular interpolation - G2





Circular interpolation - G2

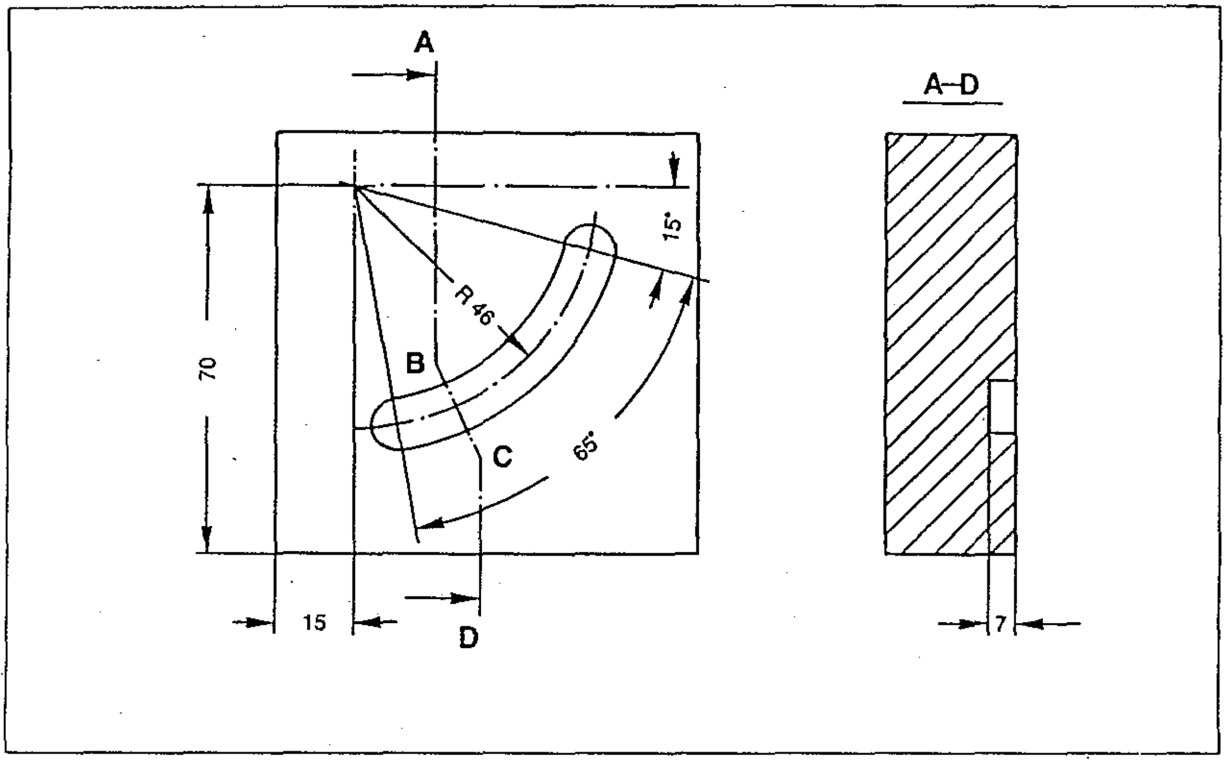
EXAMPLE
Circular arc
Polar coordinates
Incremental input

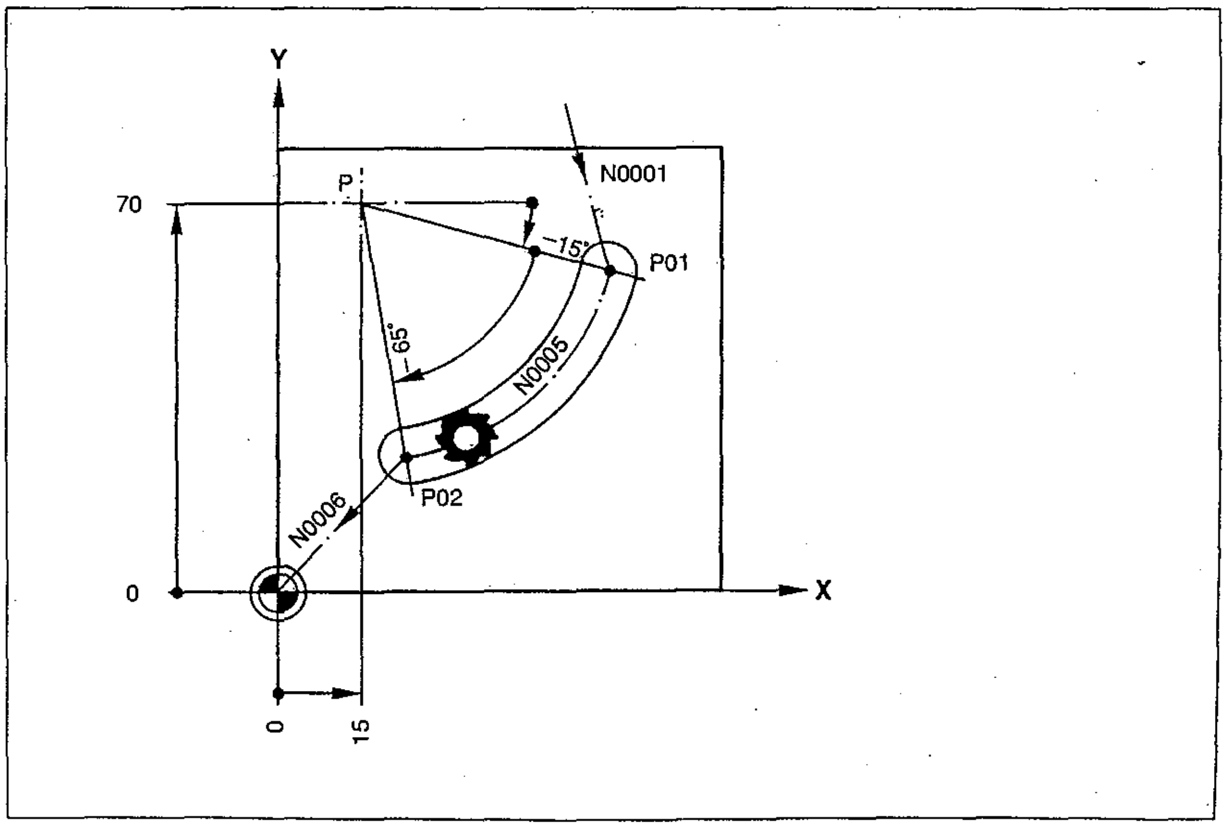


In this example, both the starting point and the target point of the tool are defined by a radius and an angle referred to the centre point: input in polar coordinates.
The angle of the circular arc between the starting point is specified relative to a straight line parallel to the Y axis, that is why the angle is defined in relation to the angle reference axis (M71).

%40	Program number
N1 G9 G0 A46 W-15 G90 I15 J70 N2 G0 Z1 S+1600	Tool positioned above P01 at rapid rate.
N3 Z-7 F100	Infeed in Z.
N4 F200	Change of feed rate.
N5 G9 G2 M71 W-65 G90 I15 J70	Circular interpolation clockwise with input in polar coordinates; angle in incremental values (M71). Polar coordinates of angle in absolute values (G90). Tool moves from P01 to P02 along circular path about point P.
N6 G0 Z100 N7 G0 X0 Y0	Tool retracted in Z and moved above program zero.
N8 M30	End of program.

Circular interpolation – G2





Tool compensation values

If your machine is equipped with a DIALOG 4 control, you can store your tool compensation values

o either in tool store T

o or in tool compensation store D.

When using DIALOG 3 programs, always use compensation store D for entering your tool compensation values.

Make it a rule to decide before program input which of the two stores you are going to use, either T or D. A mixture is not advisable, since each call of tool compensation values from store T has to be deleted before a tool compensation value can be called from store D, otherwise the control would add the two compensation values.

Tool store T

The following points should be observed if tool compensation values are called from tool store T:

- the tool compensation values are self-retaining;
- tool length and tool radius compensation values should be programmed together with a tool change;
- in the input dialog for compensation on the contour (page 6-33), the request for D (tool compensation number) should be answered by simply pressing the transfer key;
- command T0 is used for deleting a tool compensation value;
- tool compensation values called by address T or deleted by T0 will only be activated as from the end of the block in which they are programmed.

Tool store T contains a tool table. For each tool number in this table you enter the radius, radius allowance, length and length allowance to be applied to the tool concerned.
If you wish to use a tool for several operations (for example, roughing and finishing), proceed as follows:
under T/tool number you enter the basic compensation values (e.g. for roughing);
under T/tool number/*/another number you may enter additional compensation numbers (e.g. for finishing).

Tool table

T1	R	RA	L 1,0	A 0,1
T1*1	R	RA	L 2,0	A 0,5
T1*2	R	RA	L	A 0,1

If no values are entered under T...*... for radius, length and/or allowance, the preceding compensation values will be applied.
Important: tool calls (T...*...) omitted by a jump command (L0 N..) are disregarded by the control.

Program				
N1	G0	Z0	Z value=0	Jump
N2	G17	T1		command
N3	G0	Z0	Z value=1.1	in block 6:
N4	G17	T1*1		
N5	G0	Z0	Z value=2.5	
N6	G17	T1*2		
N7	G0	Z0	Z value=2.1	Z value=1.1

Tool compensation store D

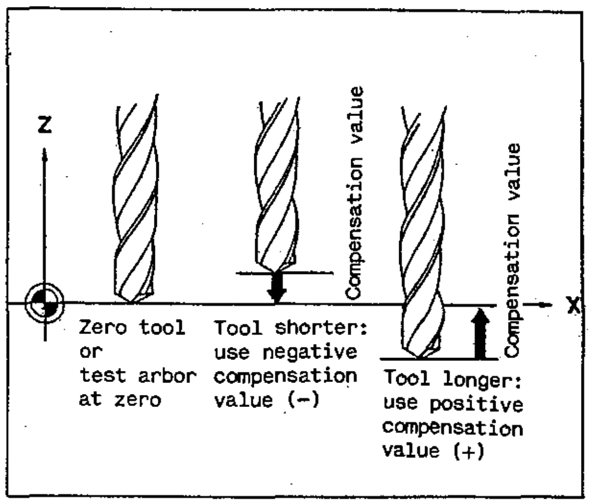
The following points should be observed if tool compensation values are called from tool store D:

- the tool compensation values are effective in only one block;
- tool length and tool radius compensation values should be programmed together with the feed/infeed motion.
- Important: if tool compensation values have already been called from tool store T without having been deleted, the control will add the length compensation values to the compensation values from store D.

Tool length compensation

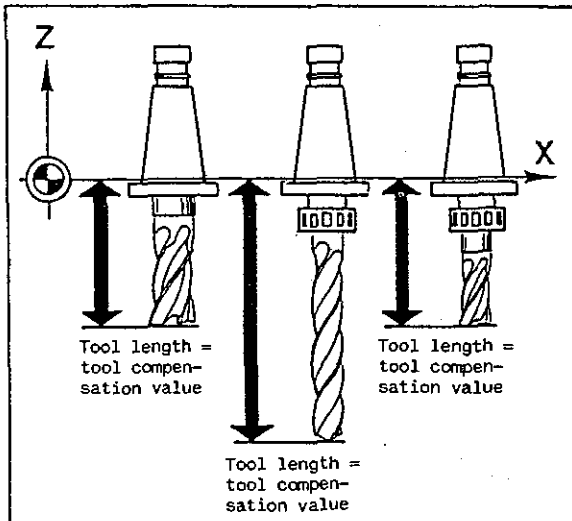
When entering tool length compensation values, you should differentiate between two alternatives, depending on whether the zero point has been determined by contacting the workpiece surface with a zero tool (page 4-12) or by using a preset tool (page 4-10).

Contacting with zero tool:
the difference in length between the zero tool (or a test arbor) and the tool to be used for cutting should be entered as tool length compensation value.
The compensation value carries
- a negative sign if the tool is shorter, than the zero tool;
- a positive sign if the tool is longer, than the zero tool.



6. Programming – Tool motions

Preset tools:
the absolute tool length is the tool length compensation value.



In tool store T, the tool length compensation value with address T and a tool number are entered.
(For input of tool compensation into, and output from, tool store T see Chapter 7).
You program the tool compensation values together with a tool change, using address T and the tool number. In addition, you have to assign a machine axis to the tool length compensation value:

G17	Z axis
G18	Y axis
G19*1	+X axis
G19*2	–X axis

With each tool compensation call you should also program the axis (G17, G18, G19*1, G19*2).
This ensures that the length compensation is always applied to the correct axis.
Commands G19*1 and G19*2 are programmed when the machine is used with the vertical spindle head tilted through an angle of 90° to the right or left.

T and tool number:
call tool change and tool compensation values.

T* and tool number:
call tool compensation value.

T and tool number and * and tool number:
call tool change and tool compensation value
OR: only call tool compensation value (if 1st tool number is that of tool already called.

T0: delete tool compensation value.

T*: delete command T0; tool compensation values are reactivated.

G17: Z value = Z coordinate + length compensation value + length allowance

G18: Y value = Y coordinate + length compensation value + length allowance

G19*1: X value = X coordinate + length compensation value + length allowance

G19*2: X value = X coordinate – length compensation value – length allowance

The tool compensation values called from tool store T are self-retaining.

The tool compensation values called from tool store T are self-retaining.

Command T0 serves to delete tool compensation values (compensation value = 0).

N ....

N3 G17 T1	Call of tool length and tool radius compensation values stored under tool No. 1, and tool change. Length compensation applied to Z axis.

N ....

In tool compensation store D, the tool length compensation value with address D and a compensation number are entered. (For input of tool length compensation into, and output from, tool compensation store D see Chapter 7).
You program the desired compensation number (under address D) at that point of the program where the tool length compensation is to be applied.
Tool compensation values called from tool compensation store D are effective in only one block.

N ....

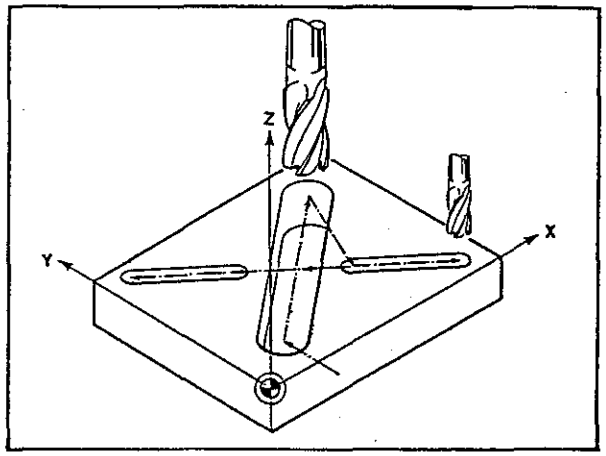
N100 Z-10 D3 F100	Control adds Z-10 to compensation value stored under compensation number D3. Tip of tool is advanced to Z-10.

N ....

Tool length compensation

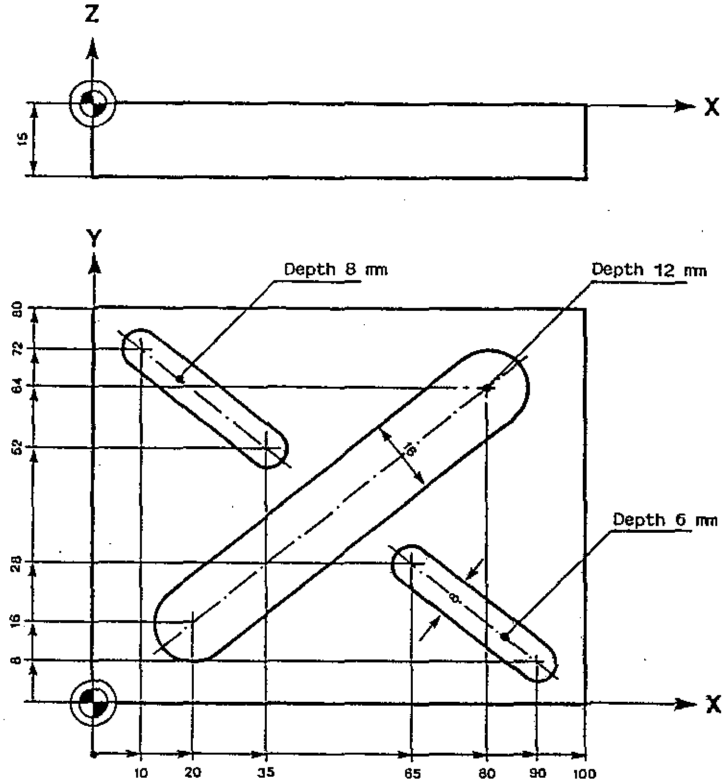
Example

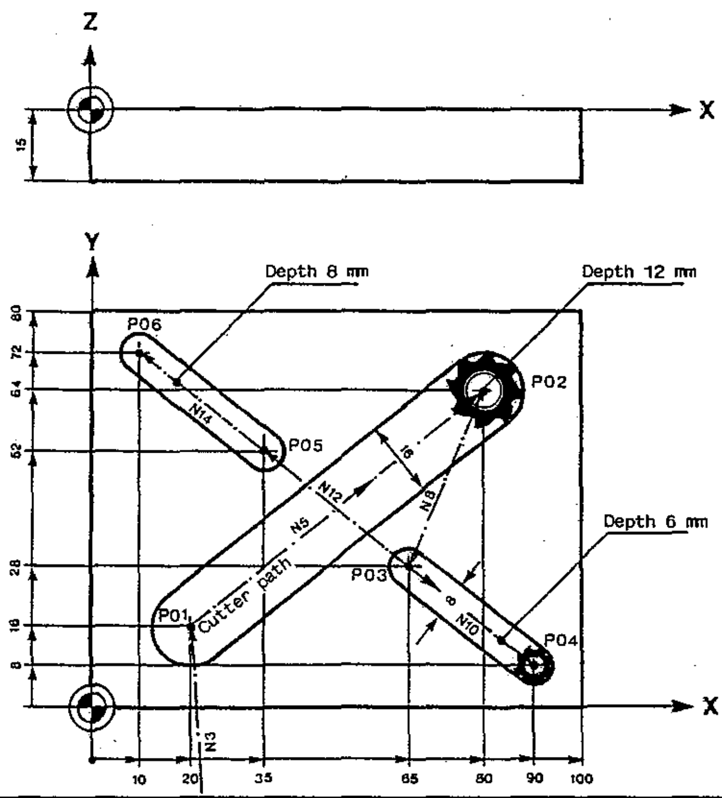
Enter tool length compensation values in tool table (in tool store T) before you start programming. Remember that a machine axis (Z axis = G17, Y axis = G18, +X axis = G19*1, -X axis = G19*2) has to be assigned to each tool length compensation before such compensation is called. The compensation is then called by programming the address T and the desired tool number.



%46	Program number
N1 G0 Z100	Tool retracted at rapid rate.
N2 G17 T1	Tool change, assignment of Z axis to length compensation, call of tool length compensation.
N3 G0 X20 Y16 Z2 S+800	Tool moves to P01 at rapid rate. Call of tool length compensation. Tip of tool moves to Z2.
N4 Z-12 F80	Downfeed to Z-12.
N5 X80 Y64 F125	Tool moves to P02 at feed rate.
N6 T0	Tool length compensation deleted.
N7 G0 Z100 S0	Tool retracted, spindle stop.
N8 G17 T2	Tool change, call of new tool compensation value.
N9 G0 X65 Y28 Z2 S+1600 N10 Z-6 F60 N11 X90 Y8 F100 N12 G0 Z2	Tool cuts slot from P03 to P04, then tool retracted.
N13 G0 X35 Y52	Tool moves to P05 at rapid rate.
N14 Z-8 F60 N15 X10 Y72 F100 N16 T0 N17 G0 Z100 S0 M30	Tool cuts slot from P05 to P06, then tool length compensation is deleted. Tool retracted at rapid rate without length compensation. End of program.

Tool length compensation





Dimensions in mm

Tool radius compensation

Similar to tool length compensations, tool radius compensations can also be stored either in tool store T or tool compensation store D.

In tool store T, the tool radius compensation value with address T and a tool number are entered.
(For input of tool compensation into, and output from, tool store T see Chapter 7).
You program the tool compensation values together with a tool change, using address T and the tool number.

T and tool number:
call tool change and tool compensation values.

T and tool number and * and tool number:
call tool change and tool compensation value
OR: only call tool compensation value (if 1st tool number is that of tool already called.

T* and tool number:
call tool compensation value.

T0: delete tool compensation value.

T*: delete command T0; tool compensation values are reactivated.

The tool compensation values called from tool store T are self-retaining.
Command T0 serves to delete tool compensation values (compensation value = 0):.

N ....

N2 G18 T5	Call of tool length and tool radius compensation values stored under tool No. 5, and tool change. Length compensation applied to Y axis.

N ....

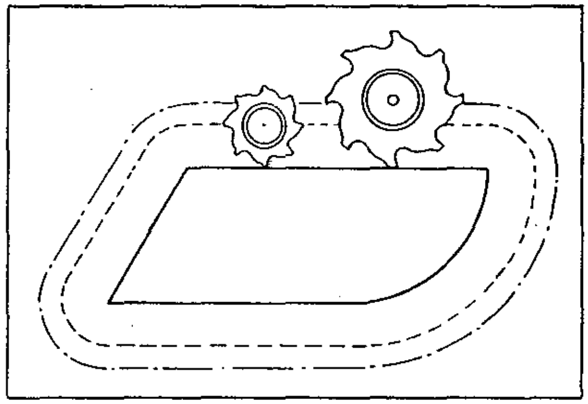
In tool compensation store D, the tool radius compensation value with address D and a compensation number are entered.
(For input of tool length compensation into, and output from, tool compensation store D see Chapter 7).

You program the desired tool radius compensation number in reply to the input dialog request for radius compensation on the contour.

The control applies the tool radius compensation until the command 'tool radius compensation on contour' is deleted again in the course of the program.

Compensation on contour

For compensation on the contour you simply program the coordinates of the contour, enter the cutter radius as a compensation value and specify on which side of the programmed contour the tool is to move.
The control will then calculate the cutter path automatically.



Call of compensation on contour

INPUT DIALOG

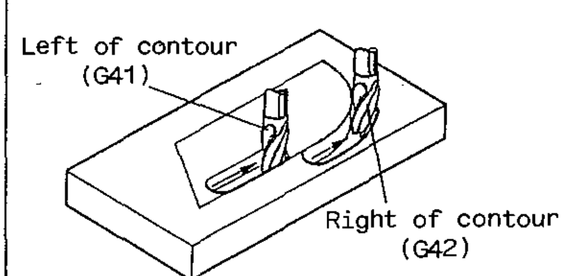
Upon entering and transferring command G41 or G42, the control requests the addresses listed on the right. These addresses should be completed and transferred to the control by pressing the transfer key.
As long as compensation on the contour is active (between call and deletion), only coordinates in a single plane can be programmed.
Infeed motions are not permissible, nor are feed motions: they have to be entered (in a block identified by M70) before calling compensation on the contour.
Blocks with M70 are first disregarded, but activated immediately upon the call of compensation on the contour.

Difference G41/G42:
G41 – tool moves left of the contour, as viewed in cutting direction;
G42 – tool moves right of the contour, as viewed in cutting direction;

D = tool compensation number:
if tool radius compensation value has already been called together with tool change T, press transfer key.

G41/G42 Compensation on contour

	The control requests:
D	Tool compensation number (if not required: transfer key)
G4	Approach command G45, G46, G47
A	Approach distance
X	1st coordinate of 1st point on contour
Y	2nd coordinate of 1st point on contour
G0	Preparatory function G0, G1 for approach
G6	Contouring command G60, G61, G64
M6	Feed optimization M60, M61, M62



Call of compensation on contour

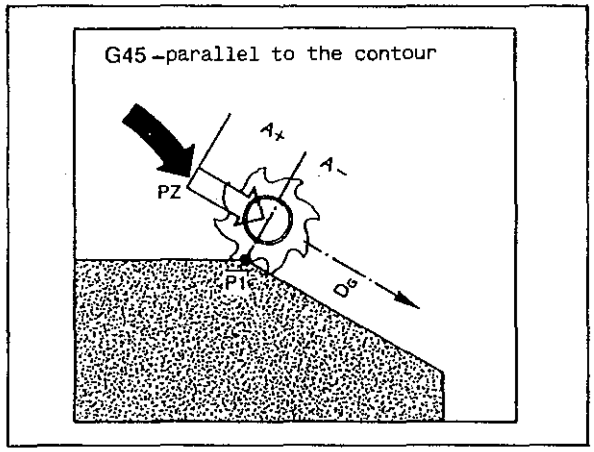
INPUT DIALOG

An approach command determines along what path the tool is to approach the contour after the compensation call (if not required: press transfer key).

G45 Approach parallel to the contour

G46 Approach in semi-circle

G47 Approach in quarter circle

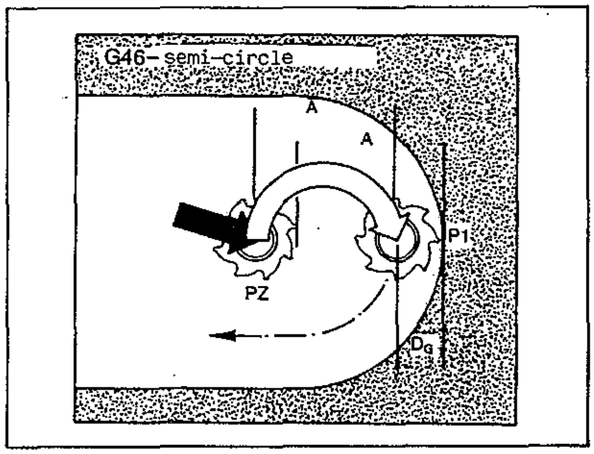


A: Approach distance

When G45 is used,
A = distance between first point on contour (P1) and start-off point (PZ) (sign change and A 0 permissible).

When G46 is used,
A = diameter of approach circle (at least 0.02 mm).

When G47 is used,
A = radius of approach circle (at least 0.02 mm).



G0/G1: rate of motion to start-off point.

G0: rapid traverse rate

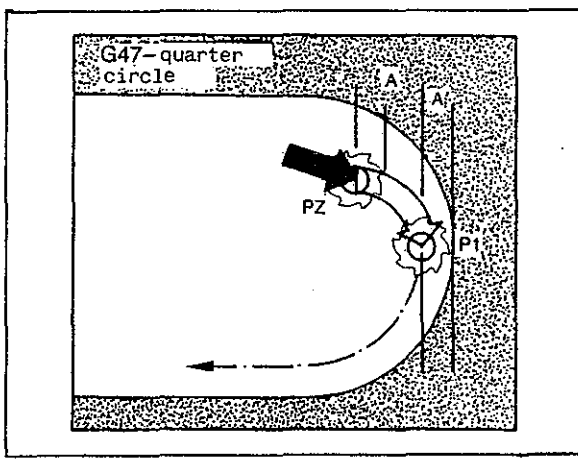
G1: feed rate along straight line

(zu Abb. rechts):

G45 - parallel to contour

G46 - semi-circle

G47 - quarter circle



Rapid rate (feed rate also possible)

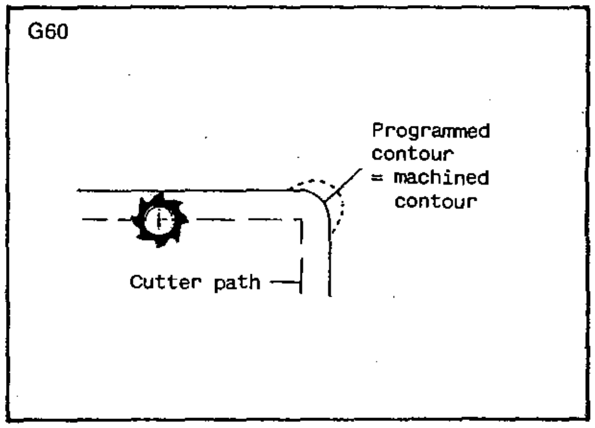
Feed rate

Call of compensation on contour

INPUT DIALOG

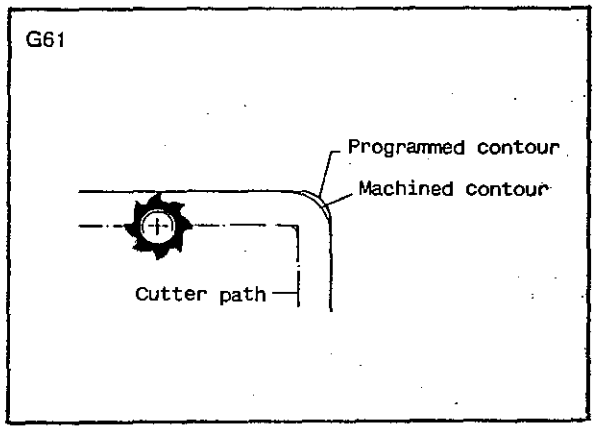
G60/G61/G64

These are contouring commands determining the cutter path at contour transitions (especially internal corners).



G60

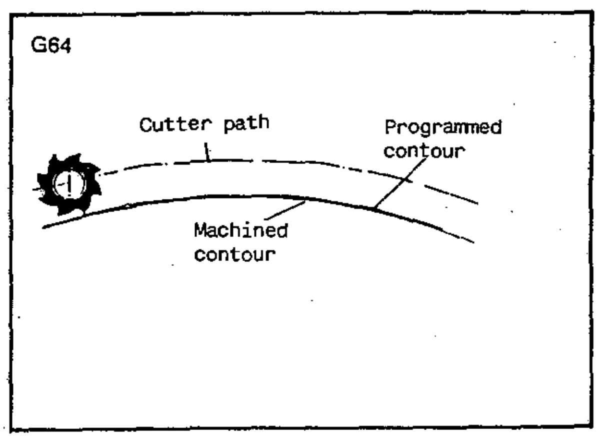
At internal corners, the tool moves exactly to the corner point, stops for a moment and then continues.
G60 is used for finishing internal corners to size.



G61

At internal corners, the tool moves along a transition radius to prevent the tool from being 'pulled' into the corner. In G61 the control interposes a blending arc at a radius which is
- 10% larger than the cutter radius for cutters of more than 4 mm radius;
- 0.4 mm larger than the cutter radius for cutters up to 4 mm radius.

G61 should be programmed for rough machining of internal corners to prevent damage to the contour.



G64

The contour sections are 'blended', dependent on the feed rate (machine slides are not brought to a stop between the individual stops).

Call of compensation on contour

INPUT DIALOG

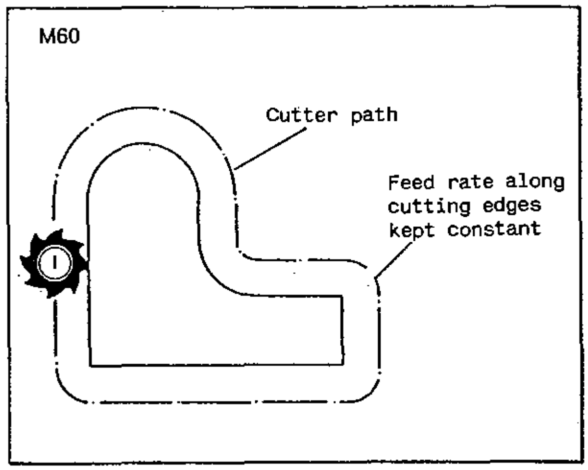
M60/M61/M62

Feed rate optimization

M60

To obtain a constant rate at the cutting edges, the feed rate is reduced along internal radii and increased along external radii.

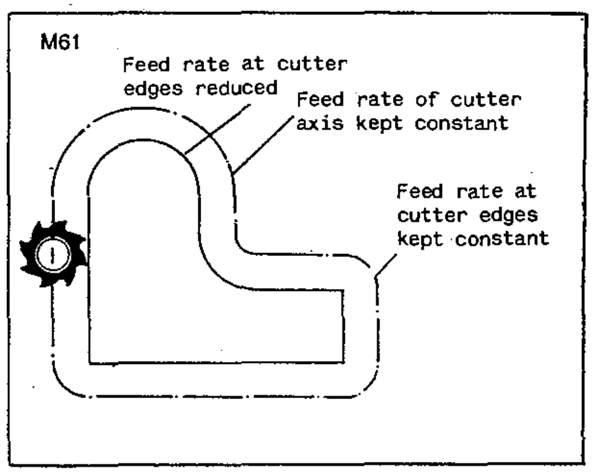
This mode is used for finishing contours when the end face of the cutter is not in contact with the bottom of the cavity or groove.



M61

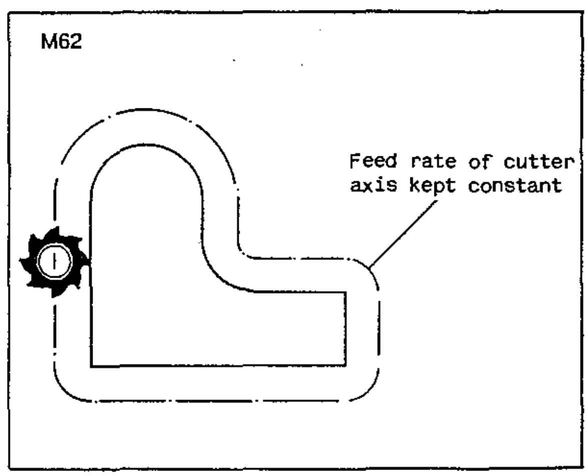
The rate of the cutting edges along the contour is kept constant in milling internal radii, as in M60.

Along external radii, however, the feed rate of the cutter spindle is kept constant, which means that the feed along the contour (cutting edges) is reduced. This mode is used for finishing a contour when the end face of the cutter is in contact with the bottom of the cavity or groove. Without such reduction, the feed rate along the bottom surface on the side away from the workpiece centre might be getting too high.



M62

The rate of feed of the cutter axis is kept at a constant level.



Deletion of compensation on contour

INPUT DIALOG

Upon entering the word G40 for deleting a compensation on the contour, the control requests the words shown on the right. These addresses should be completed and transferred to the control by pressing the transfer key.

G 40	Deletion of compensation on contour
	The control requests:
G4	Departure command G45, G46, G47
A	Distance

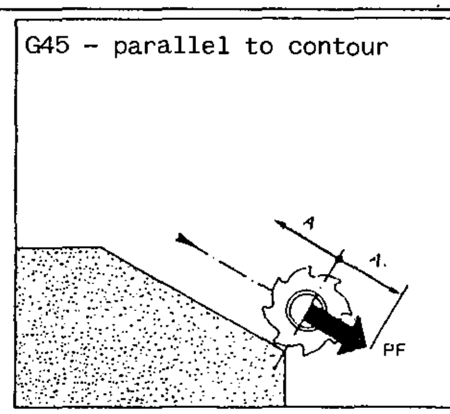
G45/G46/G47

A departure command defines the path along which the tool is to leave the contour (if not required: transfer key).

G45 Departure parallel to the contour

G46 Departure in semi-circle

G47 Departure in quarter circle

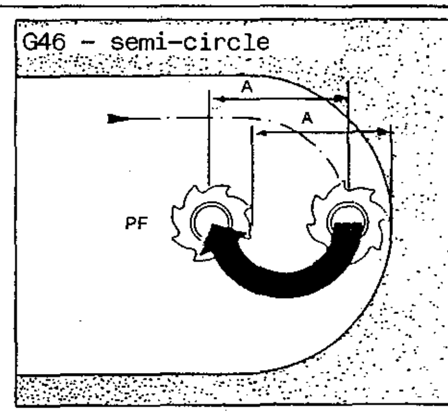


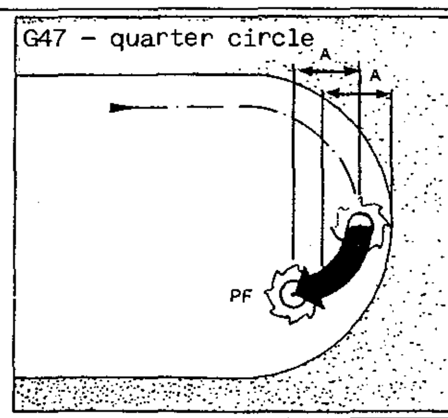
A: Departure distance

When G45 is used,
A = distance between last point on contour and clearance point PF (sign change and A 0 permissible).

When G46 is used,
A = diameter of departure circle (at least 0.02 mm).

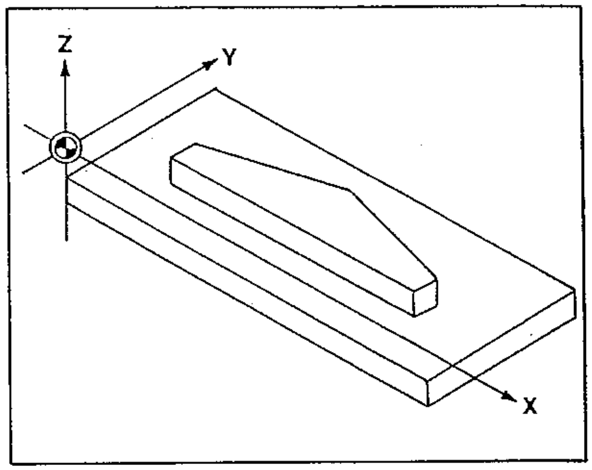
When G47 is used,
A = radius of departure circle (at least 0.02 mm).





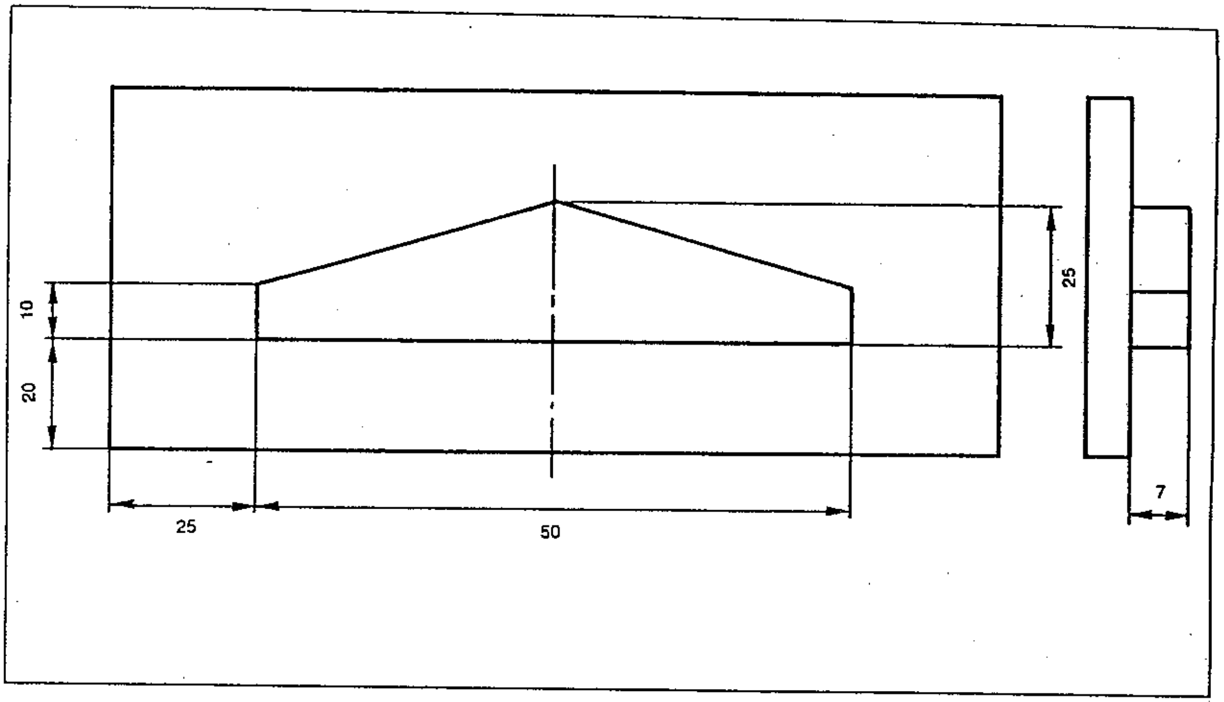
Compensation on contour

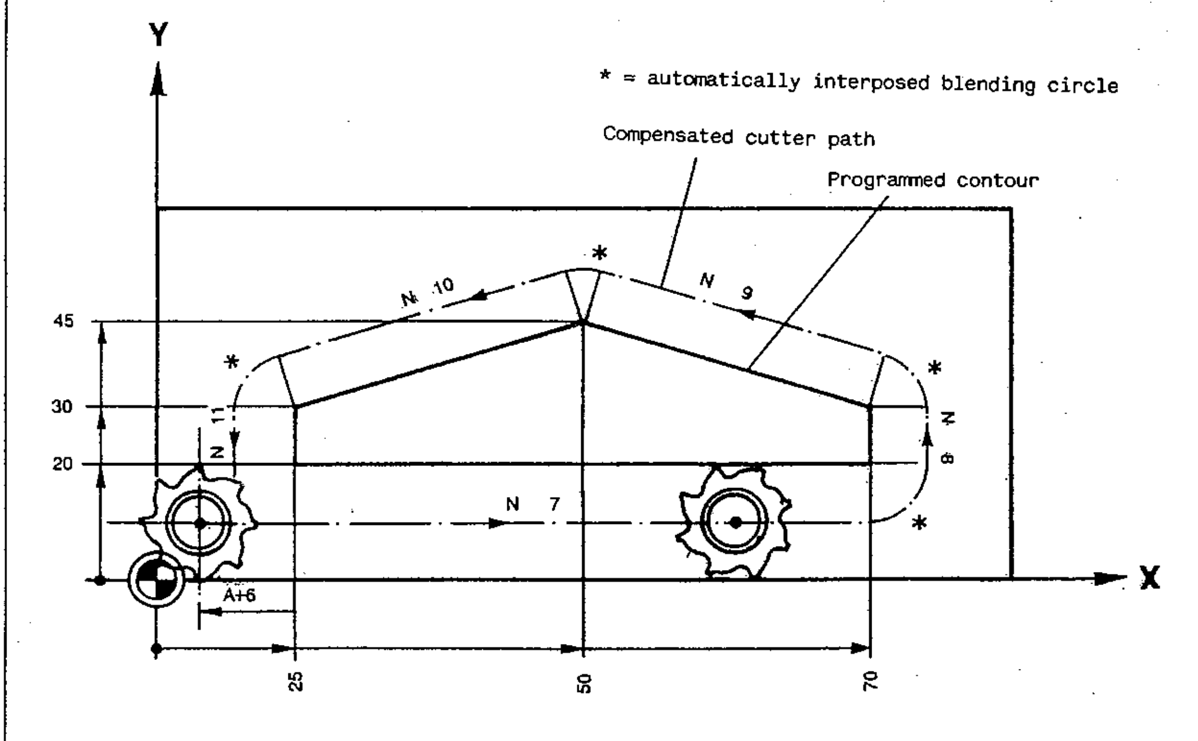
EXAMPLE
External contour
Straight lines
Approach and departure
commmand G45
(parallel to contour)



% 50	Program number
N1 G0 Z100 N2 G17 T1	Tool change. Assign Z axis to length compensation and call tool compensation values.
N3 G0 X0 Y0 Z2 S+500	Positioning in X, Y, Z and spindle start.
N4 Z-7 F100 M70 N5 F200 M70	Blocks skipped temporarily because of M70.
N6 G42 G45 A6 X25 Y20 G0 G60 M62	Call of compensation on contour (G42), tool left of contour. Approach command G45 (parallel to contour). Tool moves to start-off point at rapid rate (G0). Return to N4 to activate blocks with M70. Feed rate in N5 active up to end of program. Contour has no internal corners, so command G60 has no effect. Constant feed rate of cutter axis because of M62.
N7 X75 N8 Y30 N9 X50 Y45 N10 X25 Y30 N11 Y20	Milling of contour.
N12 G40 G45 A3	Deletion of compensation on contour. Departure command G45 (parallel to contour).
N13 T0	Delete tool compensation values.
N14 G0 Z100 M30	Tool retracted in Z, end of program.

Compensation on contour



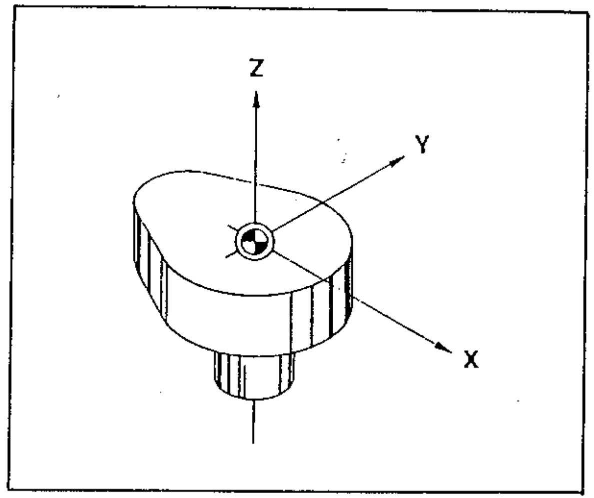


Dimensions in mm

Compensation on contour

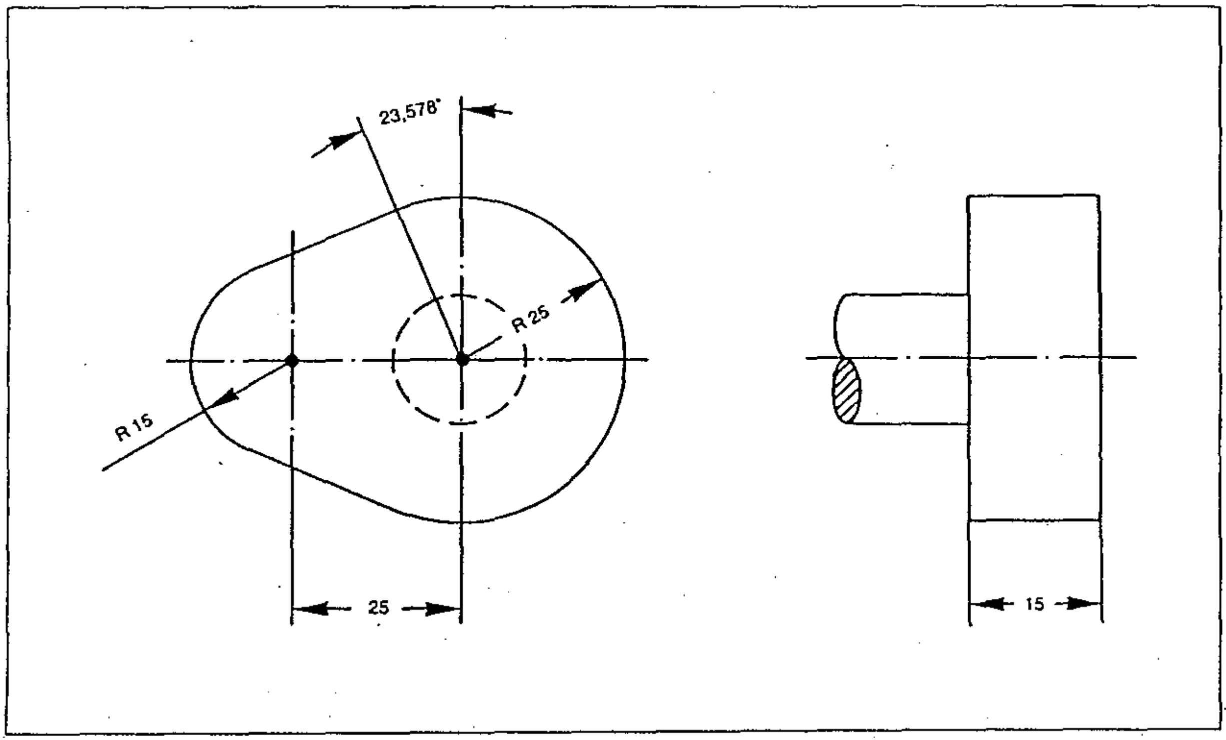
EXAMPLE
External contour
Straight line and circle
Approach and departure
command G47 (quarter circle)
Polar coordinates

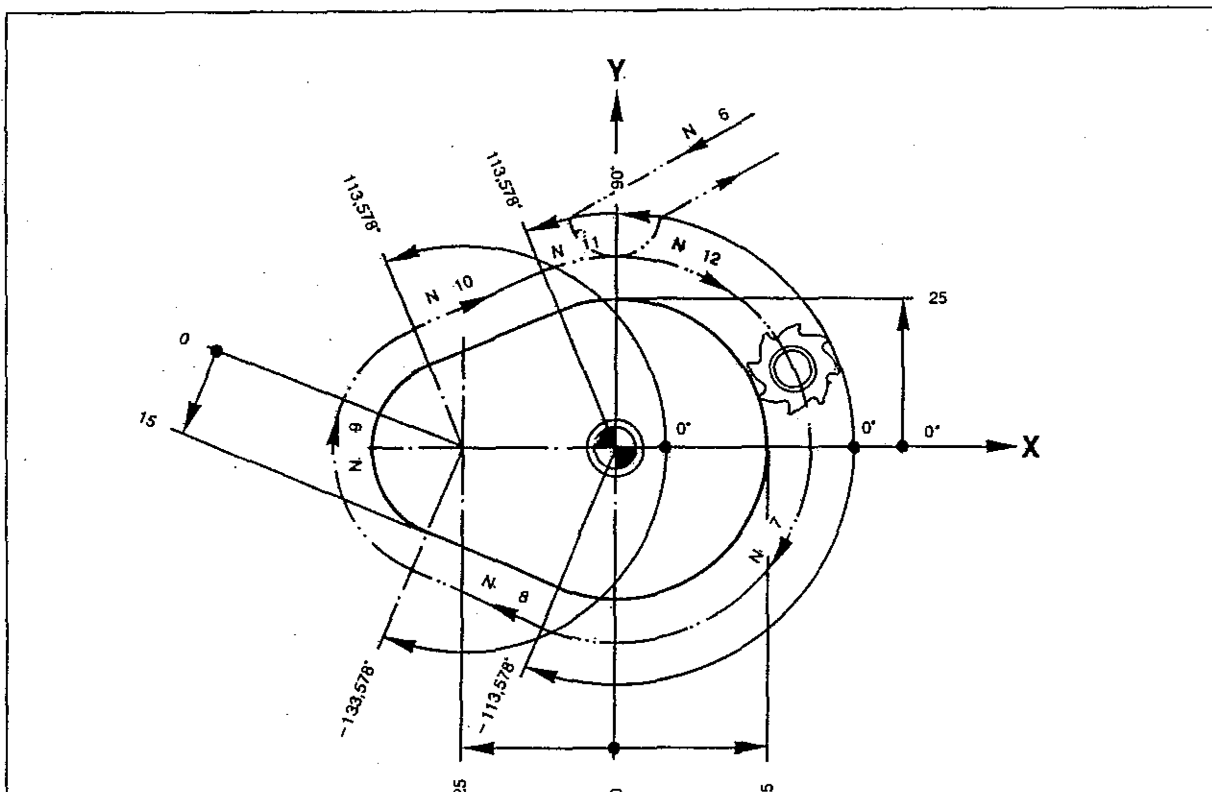
The part drawing is dimensioned in terms of angles and radii: use programming in polar coordinates.



% 55	Program number
N1 G0 Z100 N2 G17 T1	Tool change. Assign Z axis to length compensation and call tool compensation values.
N3 G0 Z2 S+500	Positioning in Z and spindle start.
N4 Z-17 F100 M70 N5 F200 M70	Blocks skipped temporarily because of M70.
N6 G41 G47 A5 X0 Y25 G0 G60 M60	Call of compensation on contour (G41), tool left of contour. Approach command G47 (quarter circle). Tool moves to start-off point at rapid rate (G0). Return to N4 to activate blocks with M70. Feed rate in N5 active up to end of program. Contour has no internal corners, so command G60 has no effect. Constant rate at cutting edges because of M60.
N7 G9 G2 M72 W-113.578 G90 I0 J0 N8 G9 G1 A15 W-113.578 G90 I-25 J0 N9 G9 G2 M72 W113.578 G90 I-25 J0 N10 G9 G1 A25 W113.578 G90 I0 J0 N11 G9 G2 M72 W90 G90 I0 J0	Milling of contour.
N12 G40 G47 A5	Deletion of compensation on contour. Departure command G47 (quarter circle).
N13 T0	Delete tool compensation values.
N14 G0 Z100 M30	Tool retracted in Z, end of program.

Compensation on contour

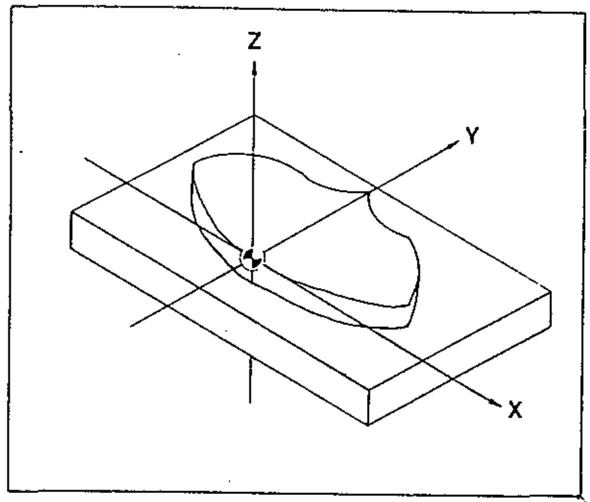




Dimensions in mm

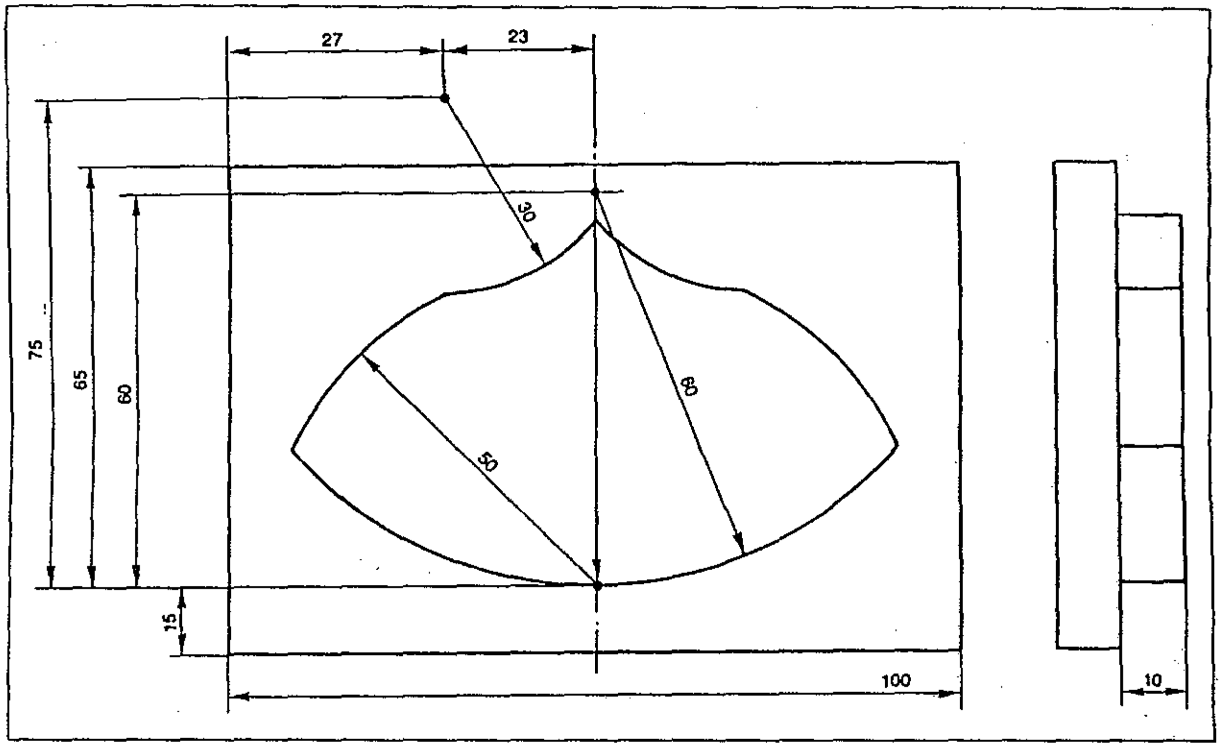
Compensation on contour

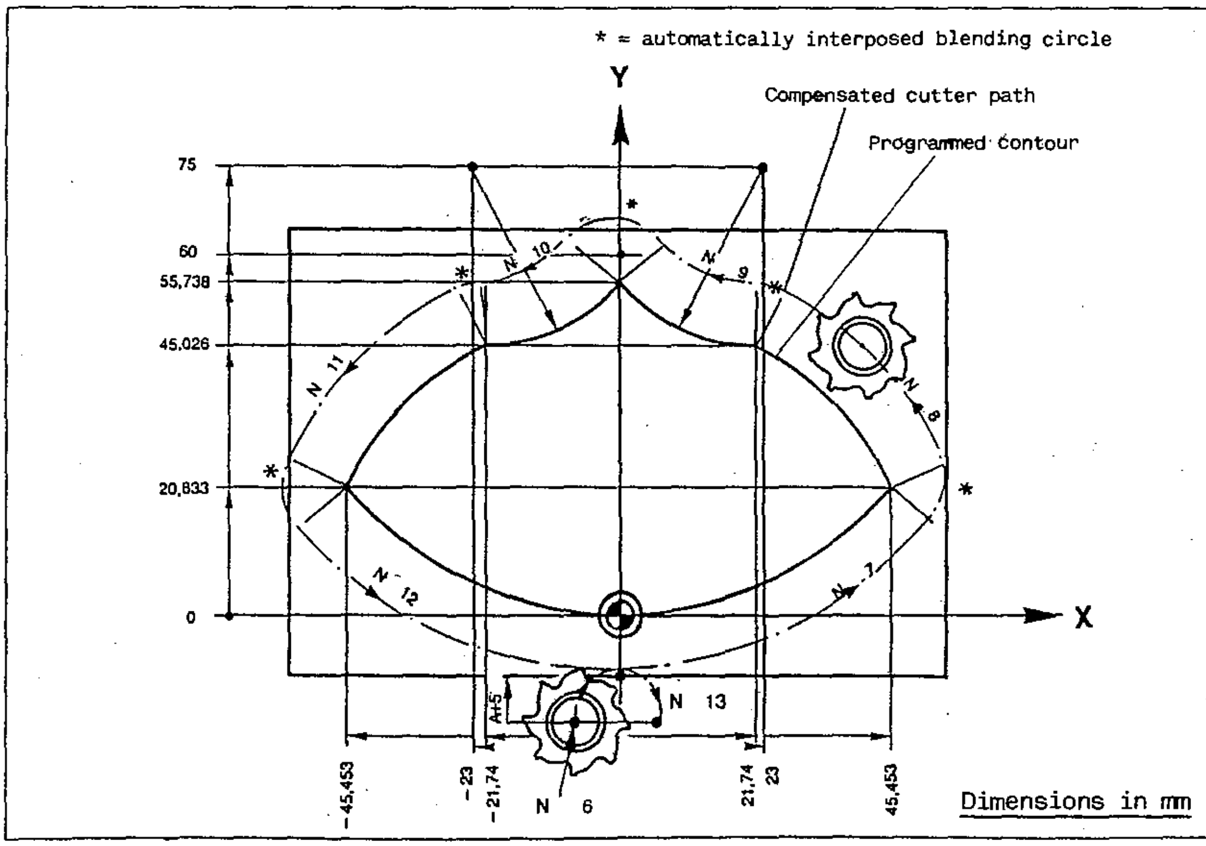
EXAMPLE
External contour
Circle
Approach and departure
command G47 (quarter circle)



% 60	Program number
N1 G0 Z100 N2 G17 T1	Tool change. Assign Z axis to length compensation and call tool compensation values.
N3 G0 Z2 S+500	Positioning in Z and spindle start.
N4 Z-10 F100 M70 N5 F500 M70	Blocks skipped temporarily because of M70.
N6 G42 G47 A5 X0 Y0 G0 G60 M61	Call of compensation on contour (G42), tool right of contour. Approach command G47 (quarter circle). Tool moves to start-off point at rapid rate (G0). Return to N4 to activate blocks with M70. The contour has no internal corners, so command G60 has no effect. Feed rate (from N5) at tool edge remains constant along internal radii because of M61, but is reduced along external radii. Otherwise feed rate along bottom would be too high on the side away from the workpiece centre.
N7 G3 X45.453 Y20.833 I0 J60 N8 G3 X21.74 Y45.026 I-45.453 J-20.833 N9 G2 X0 Y55.738 I1.26 J29.974 N10 G2 X-21.74 Y45.026 I-23 J19.262 N11 G3 X-45.453 Y20.833 I21.74 J-45.026 N12 G3 X0 Y0 I45.453 J39.167	Milling of contour.
N13 G40 G47 A5	Deletion of compensation on contour. Departure command G47 (quarter cirlce).
N14 T0	Delete tool compensation values.
N15 G0 Z100 M30	Tool retracted in Z, end of program.

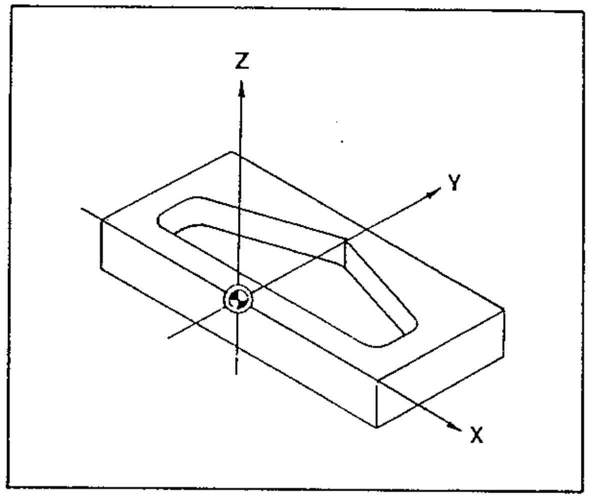
Compensation on contour





Compensation on contour

EXAMPLE
Internal contour
Straight lines and
straight line/circle
Approach and departure
command G47 (quarter circle)



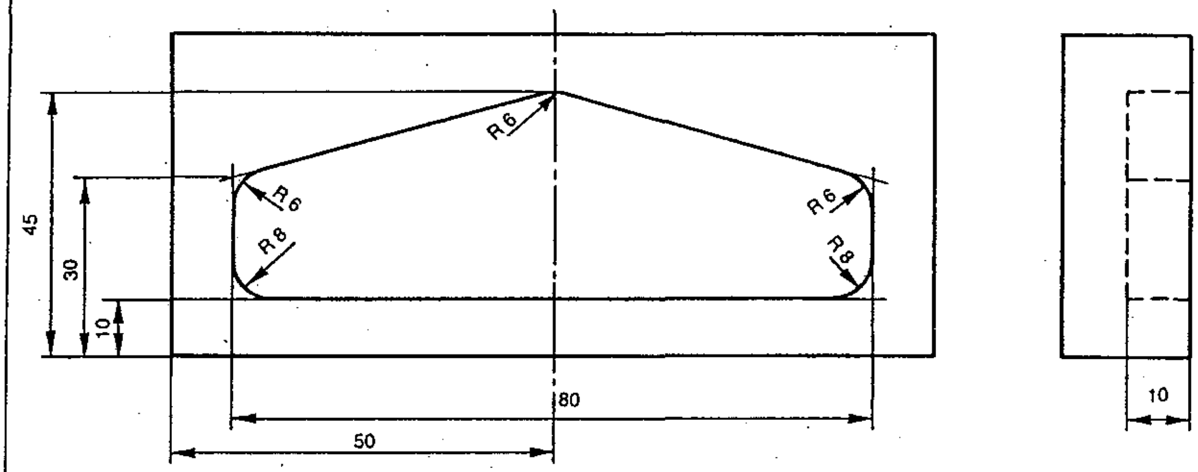
The cutting operation is performed with a cutter of 12 mm diameter (programmed with G7 = rounding of corners).

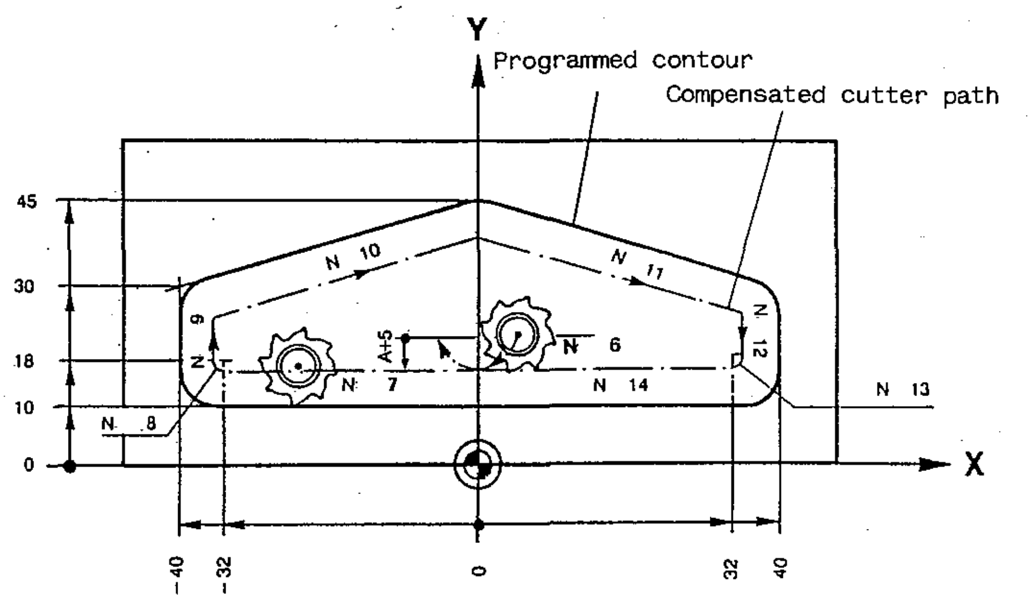
M60 should be programmed together with call of compensation on contour if tool has to machine a pointed corner.

Warning: cutter may be 'pulled' into corner if high stock removal rate is used.

% 65	Program number
N1 G0 Z100 N2 G17 T1	Tool change. Assign Z axis to length compensation and call tool compensation values.
N3 G0 Z1 S+500	Positioning in Z and spindle start.
N4 Z-10 F100 M70 N5 F200 M70	Blocks skipped temporarily because of M70.
N6 G42 G47 A5 X0 Y10 G0 G60 M60	Call of compensation on contour (G42), tool right of contour. Approach command G47 (quarter circle). Tool moves to start-off point at rapid rate (because of G0 in N6). Return to N4 to activate blocks with M70. Tool moves exactly to internal corner because of G60. Feed rate in N5 active up to deletion of compensation on contour.
N7 X-32 N8 G2 X-40 Y18 I0 J8 N9 Y30 N10 X0 Y45 N11 X40 Y30 N12 Y18 N13 G2 X32 Y10 I-8 J0 N14 X0	Milling of contour.
N15 G40 G47 A5	Deletion of compensation on contour. Departure command G47 (quarter circle).
N16 T0	Delete tool compensation values.
N17 G0 Z100 M30	Tool retracted in Z, end of program.

Compensation on contour

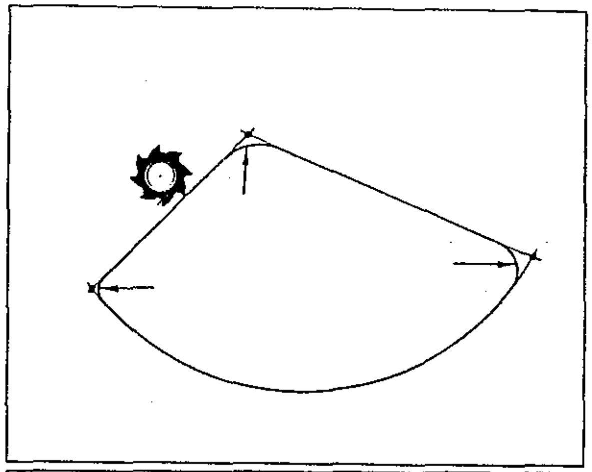




Dimensions in mm

Compensation on contour
Rounding of corners – G7

Contours may be rounded at the points of intersection of
- two straight lines,
- a straight line and a circle,
- two circles.

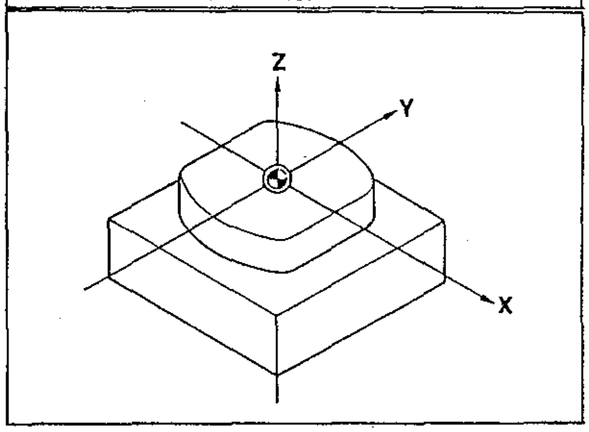


Command G7 (rounding of corners) can be programmed while a compensation on the contour is active. The command is likewise applicable to internal and external contours.
You simply program the point of intersection. The next program block must then contain command G7 and the desired radius of the corner under an R address (at least 0.02 mm).

INPUT DIALOG

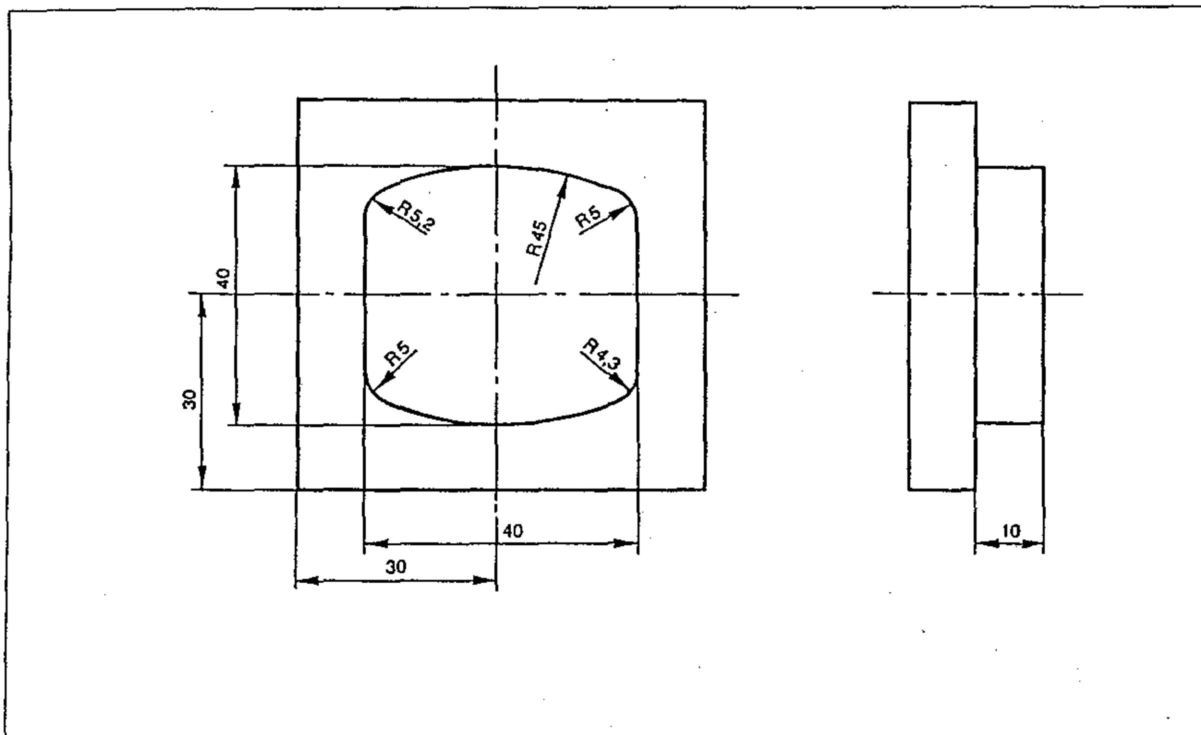
G7	Rounding of corners The control requests:	
R	Radius of corner	

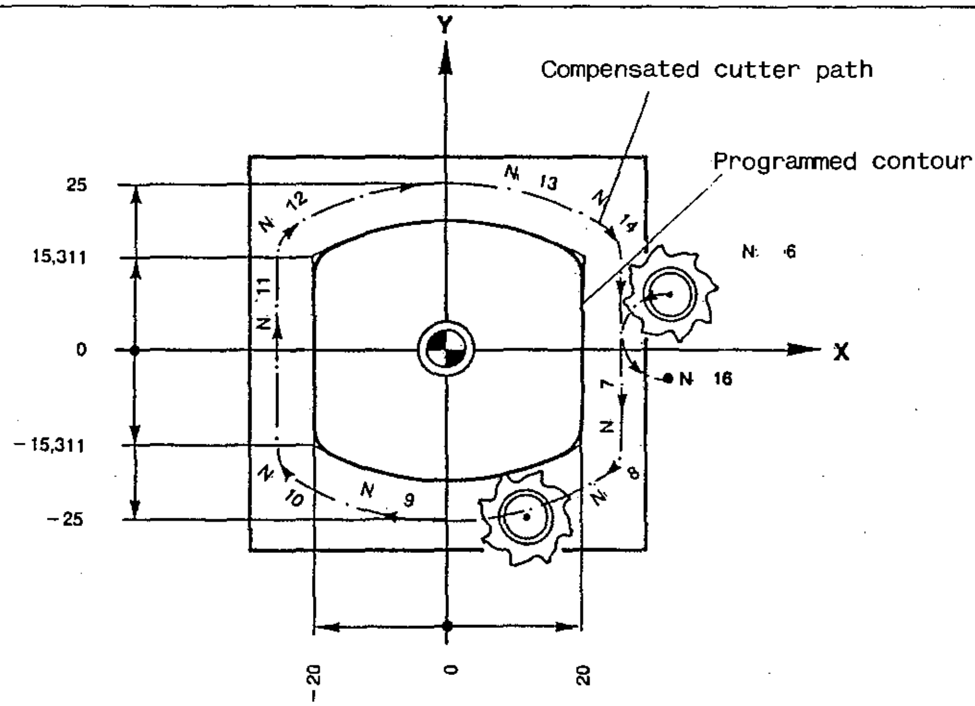
EXAMPLE
External corners



% 70	Program number
N1 G0 Z100 N2 G17 T1	Tool change. Assign Z axis to length compensation and call tool compensation values.
N3 G0 Z2 S+500	Positioning in Z and spindle start.
N4 Z-10 F100 M70 N5 F200 M70	Blocks skipped temporarily because of M70.
N6 G41 G47 A5 X20 Y0 G0 G60 M60	Call of compensation on contour (G41) and approach command (G47).
N7 Y-15.311	Milling of contour.
N8 G7 R4.3 N9 G2 X-20 Y-15.311 I-20 J40.311	Rounding of corners (R 4.3 mm).
N10 G7 R5 N11 Y15.311	Rounding of corners (R 5.0 mm).
N12 G7 R5.2 N13 G2 X20 Y15.311 I20 J-40.311	Rounding of corners (R 5.2 mm).
N14 G7 R5 N15 Y0	Rounding of corners (R 5.0 mm).
N16 G40 G47 A5	Deletion of compensation on contour.
N17 T0	Delete tool compensation values.
N18 G0 Z100 M30	Tool retracted in Z, end of program.

Compensation on contour
Rounding of corners G7

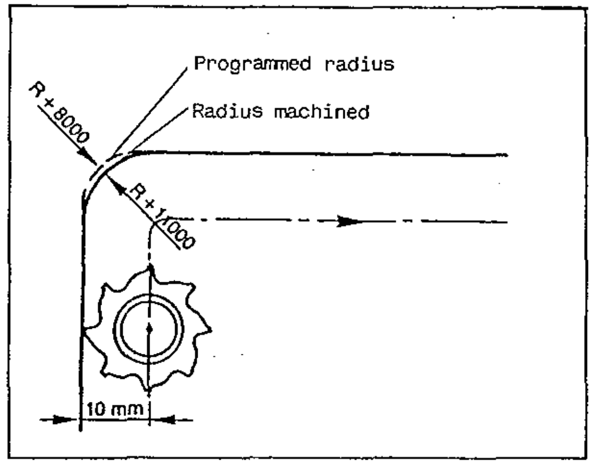




Dimensions in mm

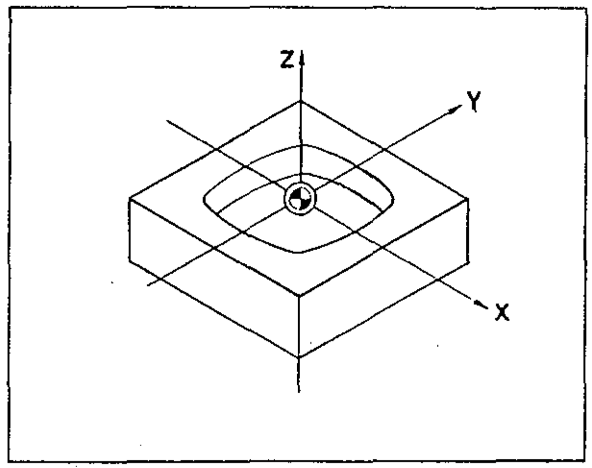
Compensation on contour
Rounding of internal corners – G61

When compensation on the contour is called, the control requests a contouring command.
In G61 the control interposes a blending arc at a radius which is
- 10% larger than the cutter radius for cutters of more than 4 mm radius;
- 0.4 mm larger than the cutter radius for cutters up to 4 mm radius.



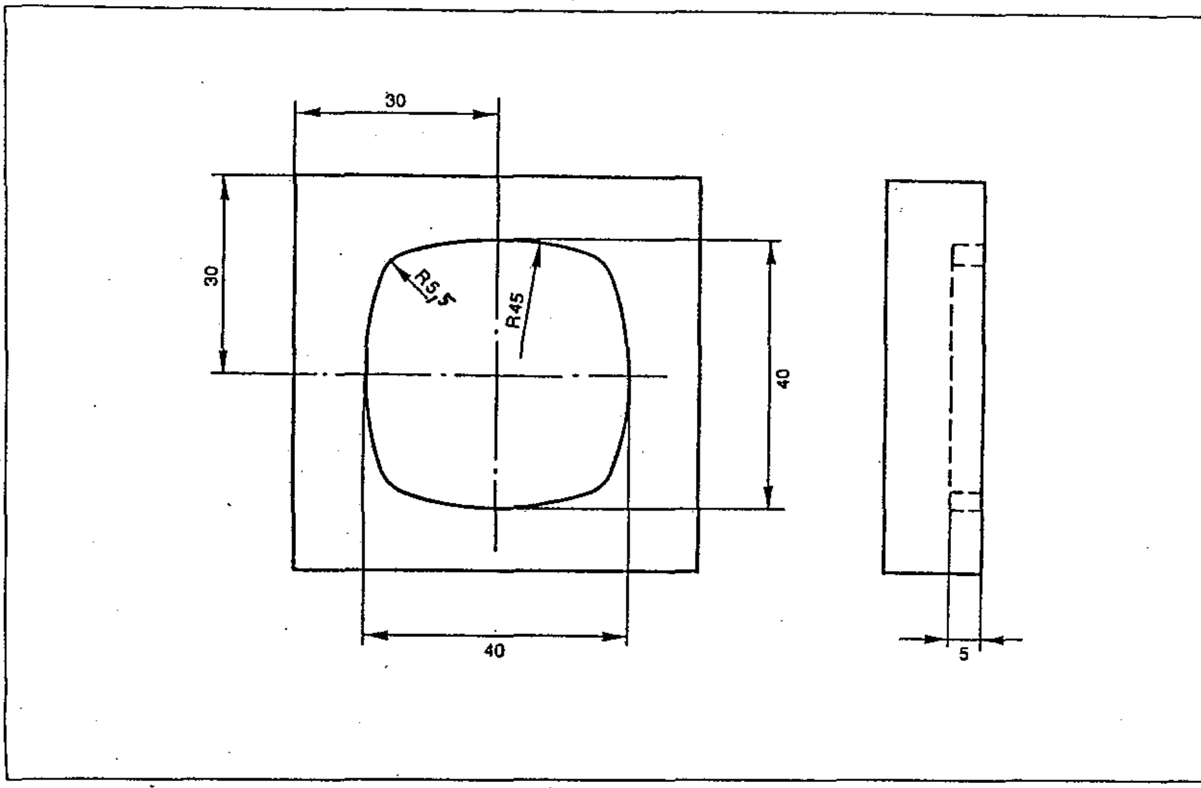
EXAMPLE
Internal contour

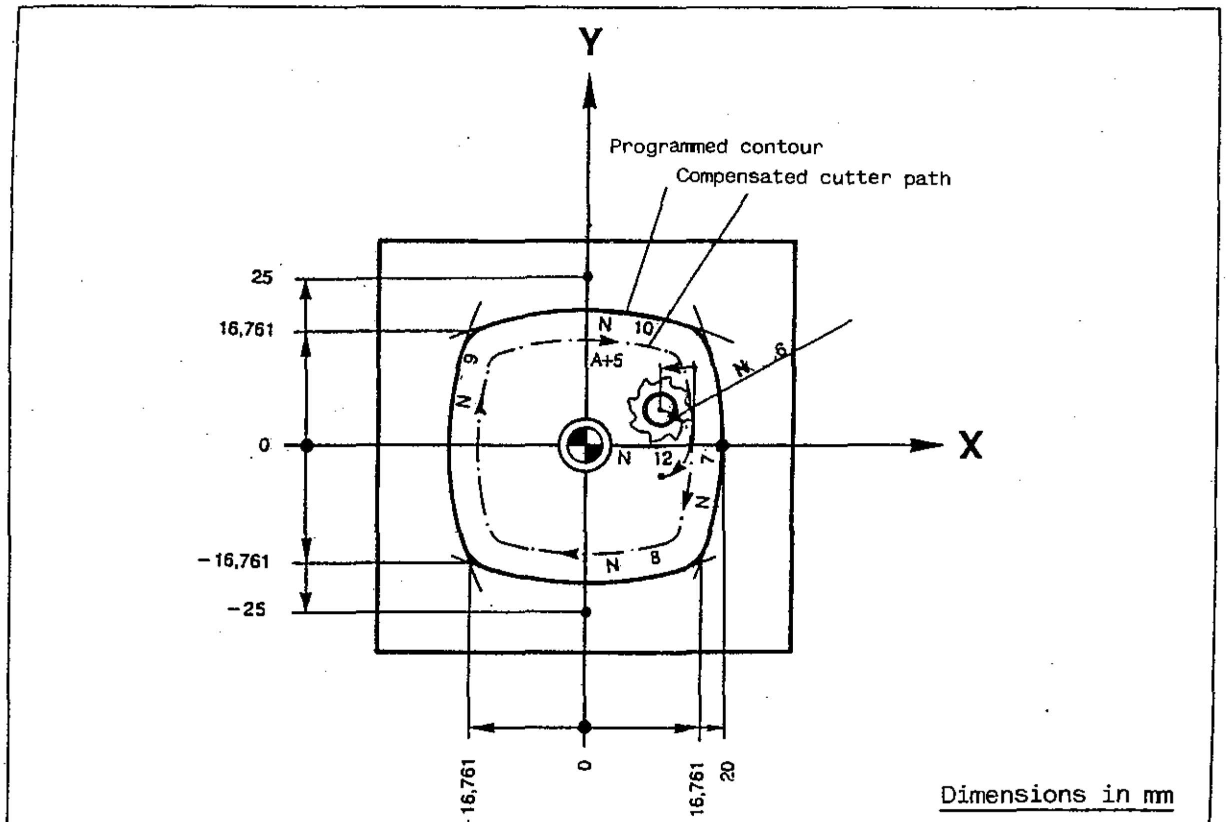
The contour is machined with a cutter of 5 mm diameter.



% 75	Program number
N1 G0 Z100 N2 G17 T1	Tool change. Assign Z axis to length compensation and call tool compensation values.
N3 G0 Z2 S+500	Positioning in Z and spindle start.
N4 Z-5 F60 M70 N5 F200 M70	Blocks skipped temporarily because of M70.
N6 G42 G47 A5 X20 Y0 G0 G61 M61	Call of compensation on contour (G42) and approach command (G47).
N7 G2 X16.761 Y-16.761 I-45 J0 N8 G2 X-16.761 Y-16.761 I-16.761 J41.761 N9 G2 X-16.761 Y16.761 I41.761 J16.761 N10 G2 X16.761 Y16.761 I16.761 J-41.761 N11 G2 X20 Y0 I-41.761 J-16.761	Milling of contour.
N12 G40 G47 A5	Deletion of compensation on contour (G40) and departure command (G47).
N13 T0	Delete tool compensation values.
N14 G0 Z100 M30	Tool retracted in Z, end of program.

Compensation on contour
Rounding of internal corners – G61

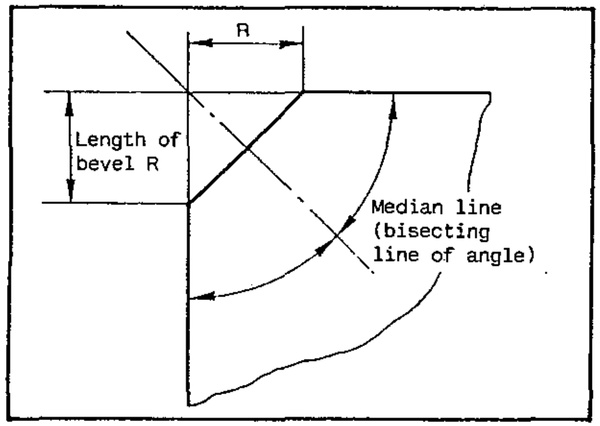




Dimensions in mm

Compensation on contour/Bevelling with programmed length of bevel – G8

A corner may be bevelled at the intersection of two straight lines. You simply program the point of intersection. The next block must contain the command G8 (bevelling of corners) and the length of bevel under an R address (R min 0.02 mm). Bevels may be programmed while a compensation on the contour is active.

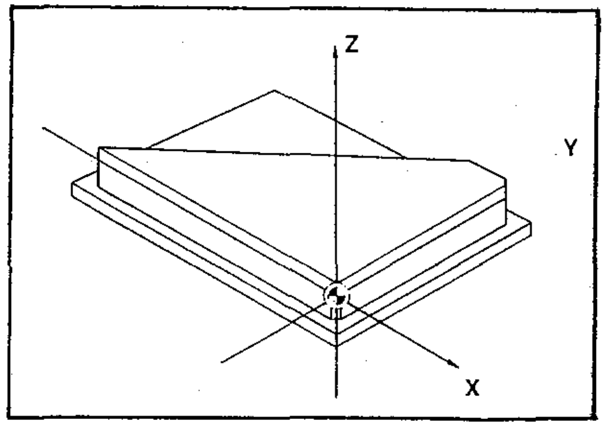


INPUT DIAOLG

The angle between the bevel and the two adjacent surfaces is always the same.

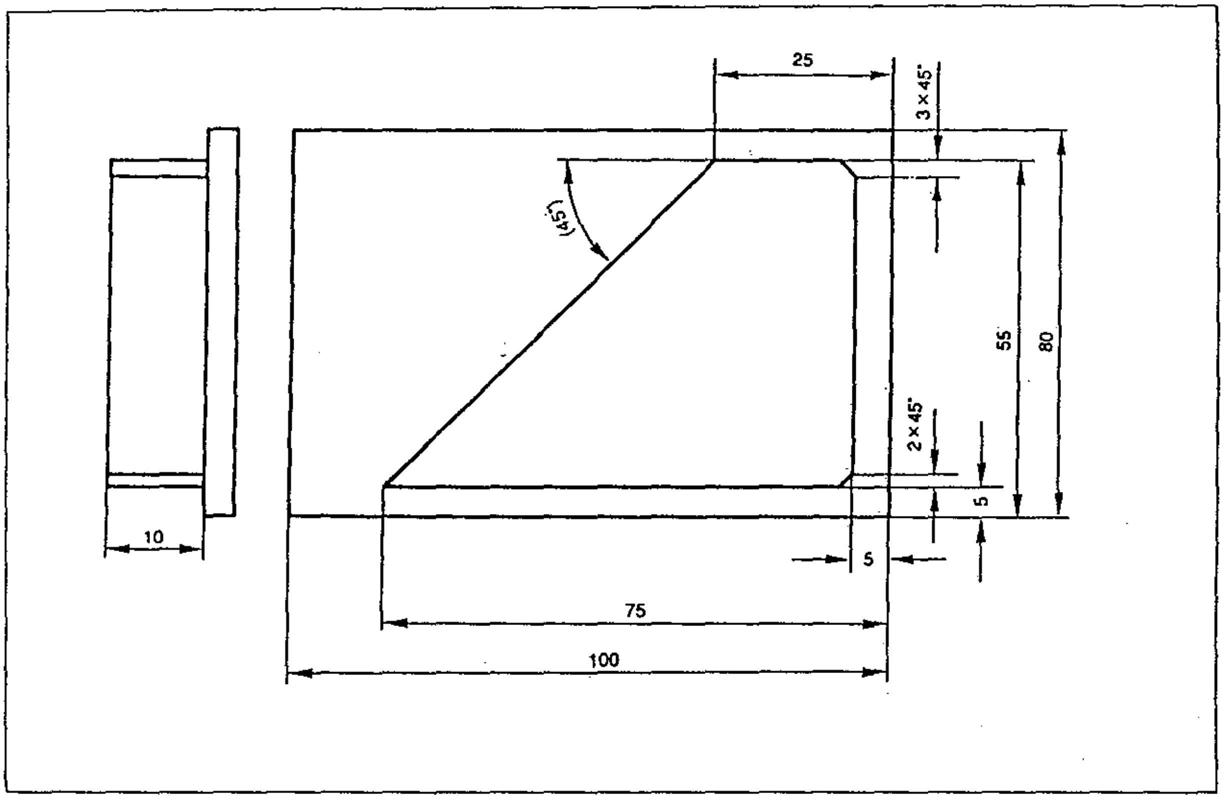
G8	Bevelling of corners The control requests:
R	Length of bevel

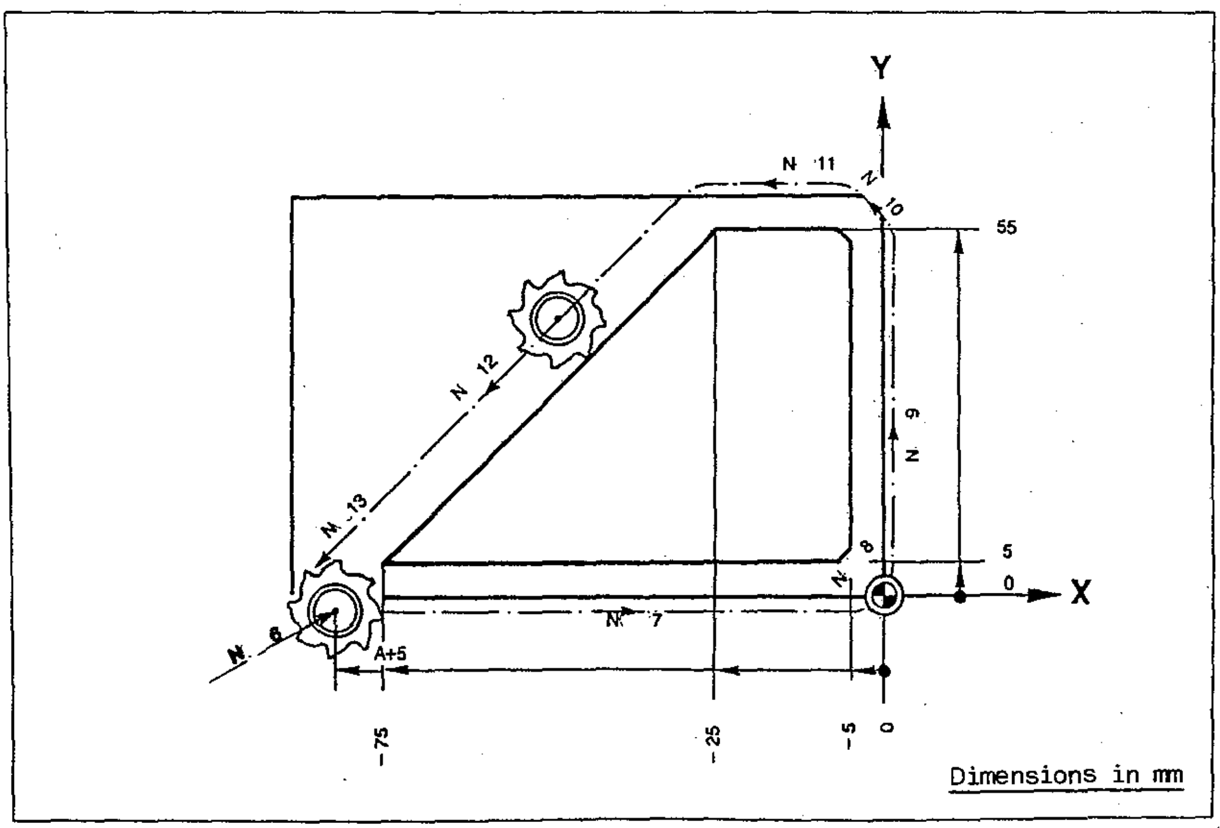
EXAMPLE



% 80	Program number
N1 G0 Z100 N2 G17 T1	Tool change. Assign Z axis to length compensation and call tool compensation values.
N3 G0 Z2 S+500	Positioning in Z and spindle start.
N4 Z-10 F50 M70 N5 F100 M70	Blocks skipped temporarily because of M70.
N6 G42 G45 A5 X-75 Y5 G0 G60 M60	Call of compensation on contour (G42) and approach command (G45).
N7 X-5 N8 G8 R2 N9 Y55 N10 G8 R3 N11 X-25 N12 X-75 Y5	Milling of contour. Bevelling of corner (length of bevel 2 mm). Bevelling of corner (length of bevel 3 mm). Bevelling with programmed length of bevel.
N13 G40 G45 A5	Deletion of compensation on contour.
N14 T0	Delete tool compensation values.
N15 G0 Z100 M30	Tool retracted in Z, end of program.

Compensation on contour/Bevelling with programmed length of bevel – G8

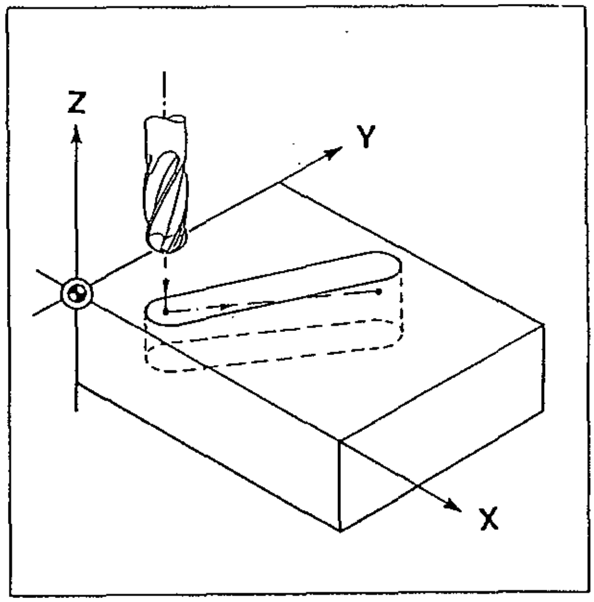




Dimensions in mm

3D interpolation

3 D interpolation means that simultaneous motions in up to 3 of 4 axes are obtainable. Tool length compensation is permissible, radius compensation on the contour (G41/G42) is not permissible.



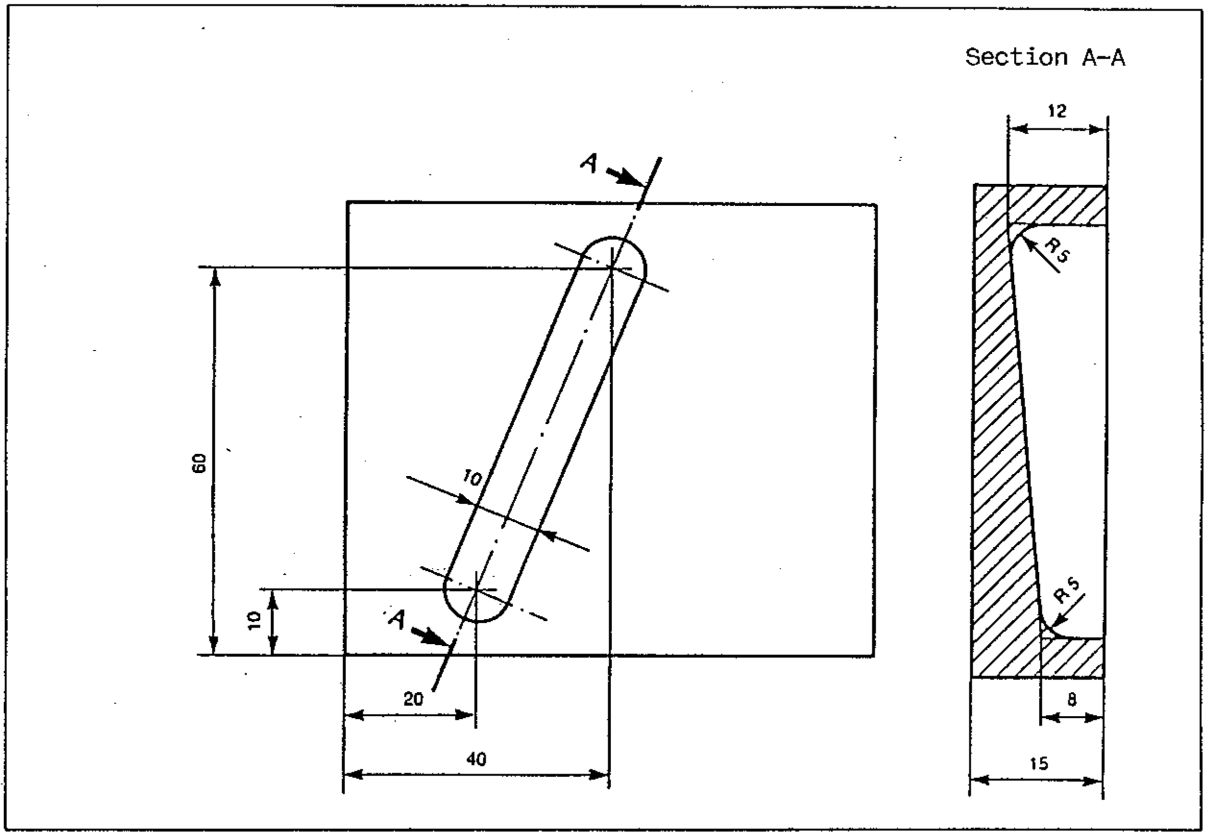
EXAMPLE

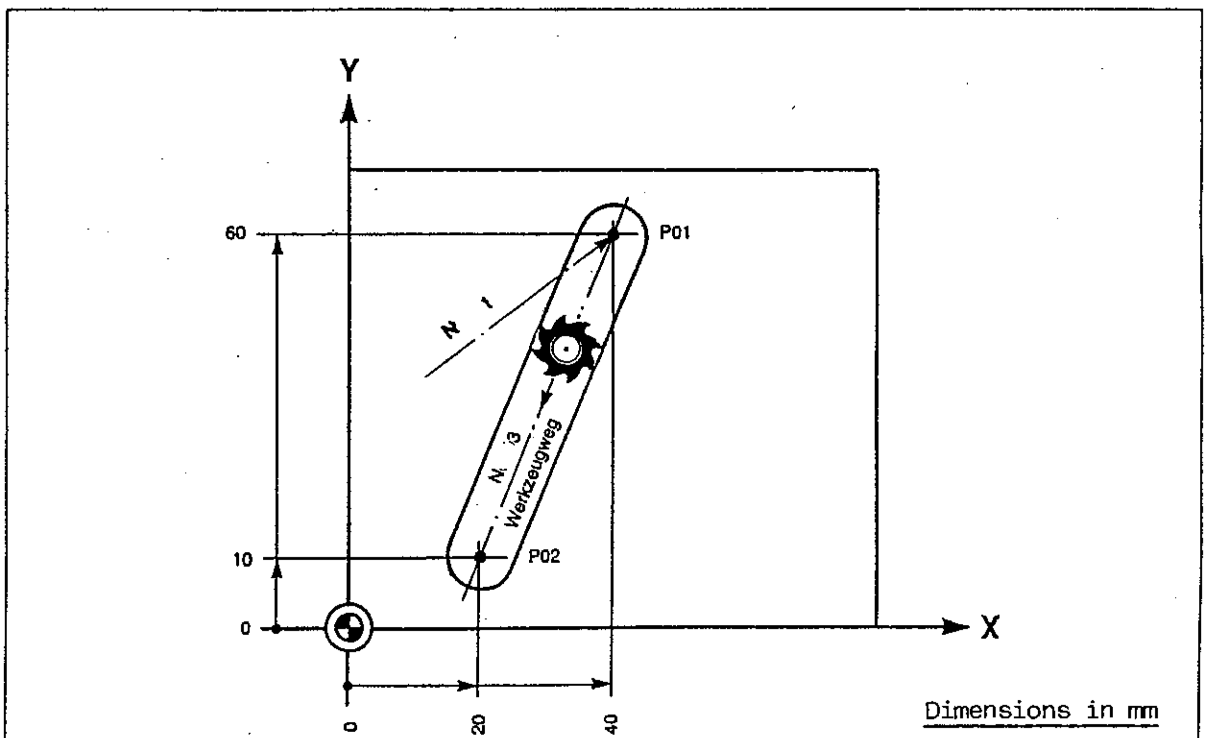
Straight cut

A three-dimensional tool motion along a straight line between two points requires linear interpolation of the control in three axes.

% 85	Program number
N1 G0 Z100	Tool retracted for tool change.
N2 G17 T1	Tool change. Assign Z axis to length compensation and call tool compensation values.
N3 G0 X40 Y60 Z2 S+4000	Tool moves to point P01 (X40, Y60, Z2) at rapid rate; spindle speed S+500.
N4 Z-12 F100	Downfeed to Z-12 (feed rate 100 mm/min).
N5 X20 Y10 Z-8 F200	Linear tool motion from P01 to P02 (feed rate 200 mm/min).
N6 T0	Delete tool compensation values.
N7 G0 Z100 M30	Tool retracted in Z, end of program.

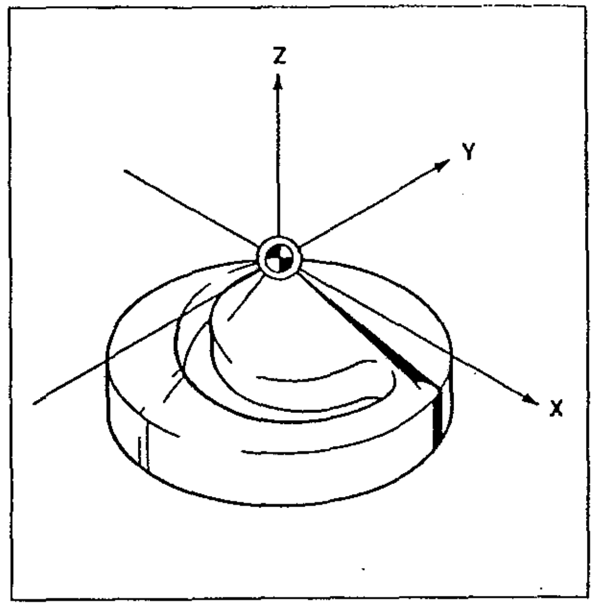
3D interpolation





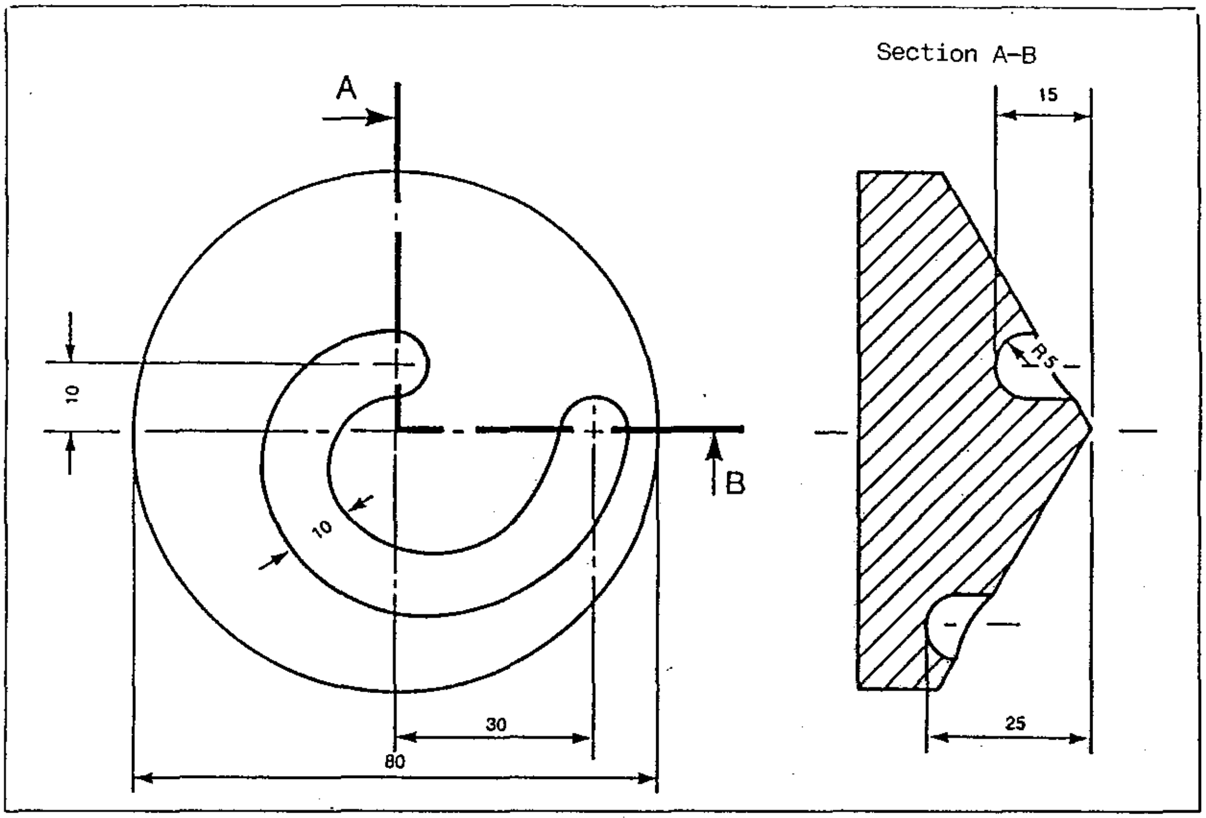
3D interpolation with C axis

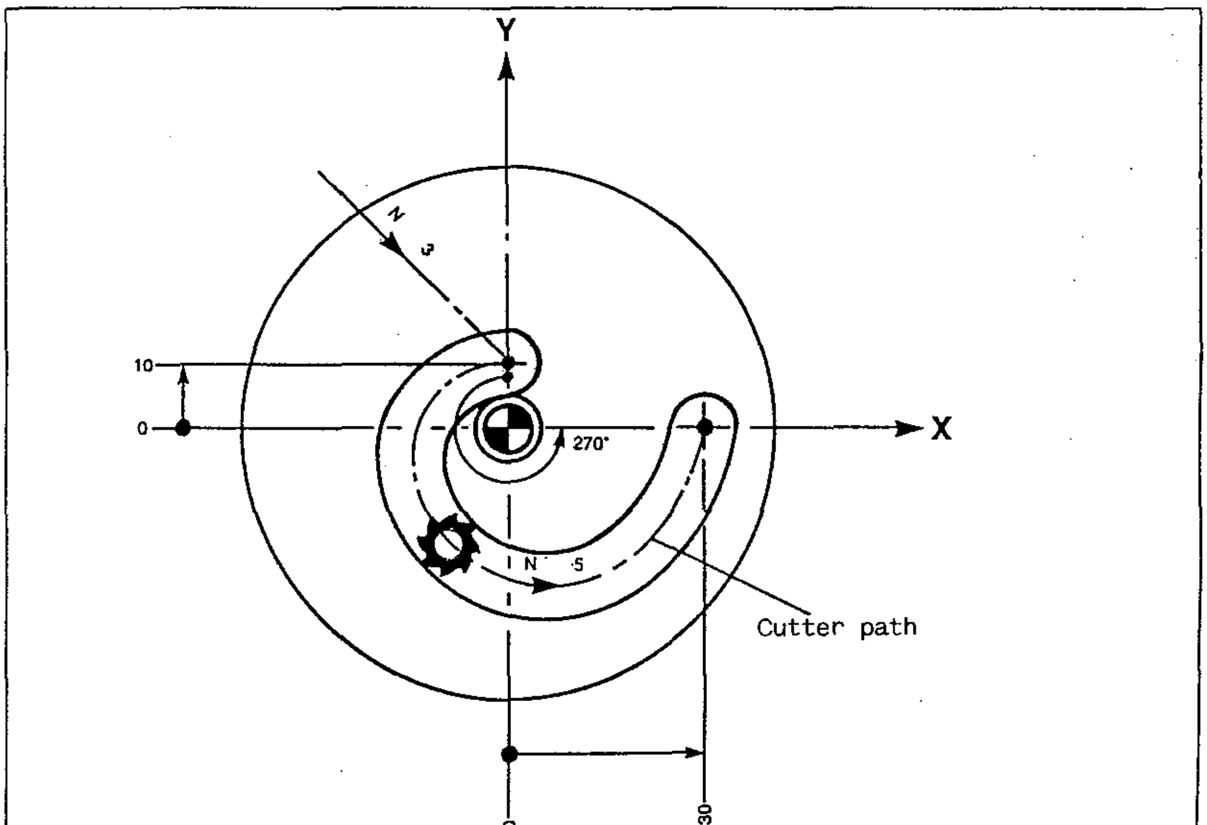
EXAMPLE
Spiral cut



% 90	Program number
N1 G0 Z100 N2 G17 T1	Tool change. Assign Z axis to length compensation and call tool compensation values.
N3 G0 X0 Y10 Z2 C0 S+800	Tool moves to point above workpiece surface at rapid rate.
N4 Z-15 F150	Downfeed in Z.
N5 Y30 Z-25 C270 F250	Milling of spiral slot: NC rotary table motion through 270° with simultaneous downward tool motion along angular straight line.
N6 T0	Delete tool compensation values.
N7 G0 Z100 M30	Tool retracted in Z, end of program.

3D interpolation with C axis



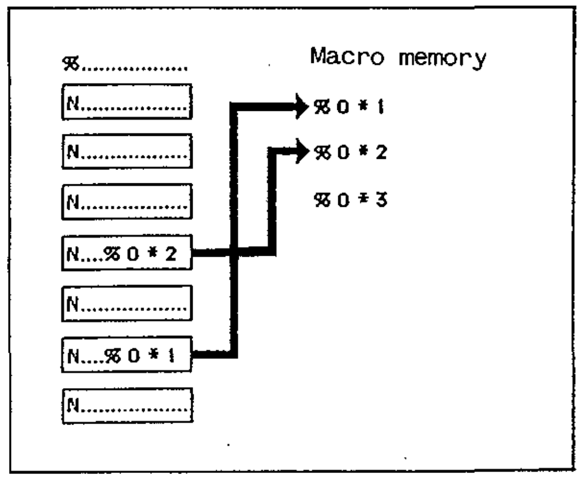


Subroutines

If a workpiece requires several identical machining operations, such operations need be programmed just once in a subroutine.
A differentiation is made between the following:
- macros,
- local subroutines,
- subroutine blocks.

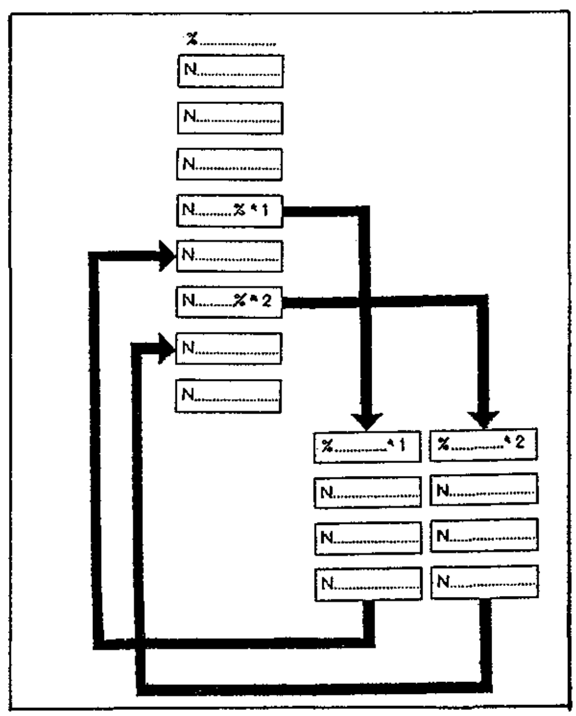
Macros

A macro is a subroutine stored independently of the main program. The control is equipped with a macro memory storing the macros separately of the program memory under a macro number identified by the address %0*.....
To enter a new macro you first select mode 13 and, similar to opening a main program, open a macro number by entering %0*..... (see Chapter 7). Then you change over to mode 11, call the macro number just opened and enter the macro block by block (see Chapter 7).
Such macros can now be called in any main program by entering %0* and the macro number.
Macros must not contain an end-of-program command (M2 or M30), otherwise error code 76 appears on the display screen.



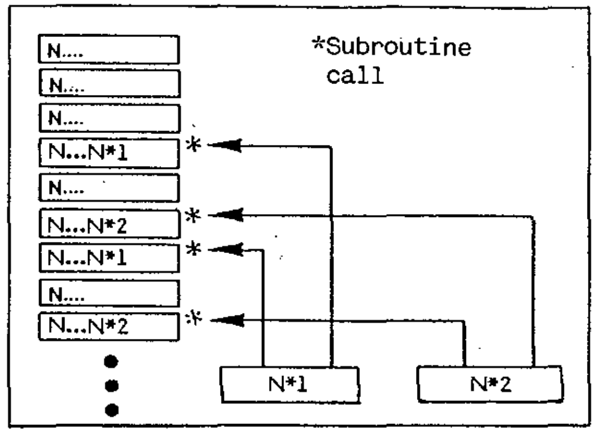
Local subroutines

Local subroutines are always part of a main program and can thus only be called within such main program.
In the program memory, local subroutines are stored under the number of the main program:
% number of main program
% number of main program * number of subroutine.
Within a main program you select mode 11 and enter the local subroutine under % number of main program * number of subroutine.
Local subroutines must not contain an end-of-program command (M2 or M30), otherwise error code 76 appears on the display screen.



Subroutine blocks

Subroutine blocks may be programmed within a main program.
They may be called (= executed) in any desired program block. Subroutine blocks carry the address N* and a subroutine block number.



Subroutine block numbers are freely selectable between N*0 and N*9999, the number having no effect on the sequence of execution within the program.

Subroutine blocks are called by their block numbers. Each number has to be called separately.

Subroutines greatly simplify the programming of drilling and milling cycles.

A program part repetition is not permissible within a subroutine block.

Program part repetition – L

Any part or parts of a program may be repeated between 1 and 99 times at any point within a program.

A program part repetition is called by an L address.

INPUT DIALOG

When an L address with a two-digit number (defining the number of repetitions) has been entered, the control requests the addresses shown on the right.
These should be completed and transferred to the control by pressing the transfer key.

Program part repetition

The control requests:

L Number of repetition desired
N Block number of 1st block of program section to be repeated
N Block number of last block of program section to be repeated

For up to four repetitions:
N* Subroutine block numbers to be substituted in a repetition (if not required: transfer key)

The call of a program part repetition is also valid for macros stored in the macro memory and for local subroutines and subroutine blocks stored in the main program memory.
The block numbers N... requested in the input dialog are in this case simply overwritten by % 0...., %...*... or N*...., respectively.

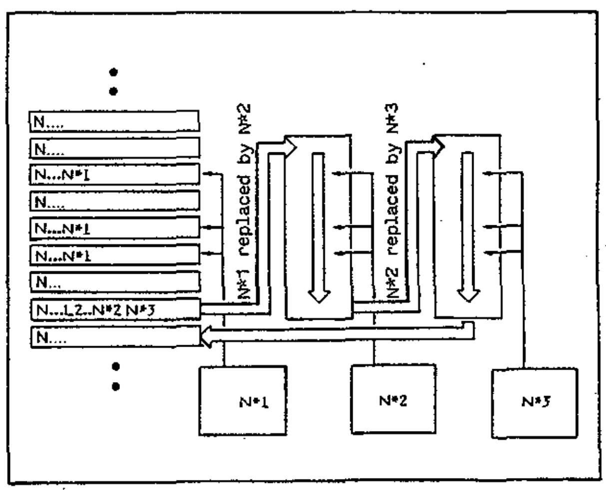
A block with an L address must not contain any words other than the block numbers of the program section to be repeated, and the subroutine blocks, if any.

The first block in a program part repetition must always be a main block.

Program part repetition with subroutine substitution

Program part repetitions can be programmed in such a way that each repetition contains a subroutine for which a different subroutine is substituted in the next following repetition.

Up to 4 different subroutines may be called in this way. If more than 4 subroutines are required, several program part repetitions with subroutine substitution have to be programmed.



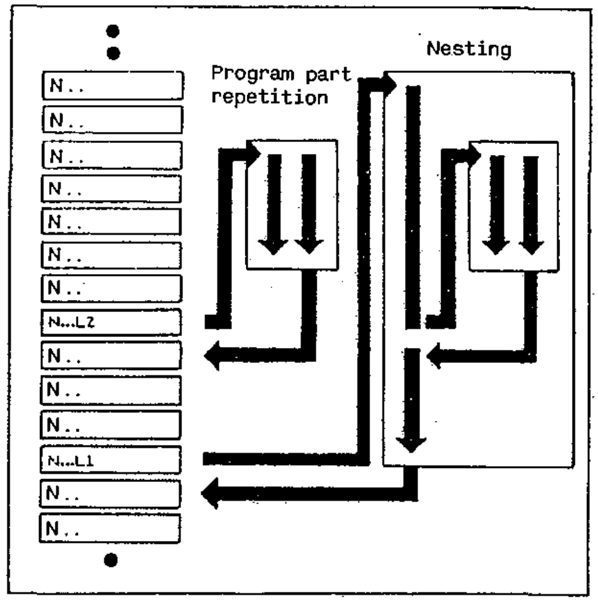
INPUT

See program part repetition

Nesting

It is also possible to repeat a program part in which a program part repetition has already been programmed. This procedure is called 'nesting'.

Nesting is permissible up to 16 times: the 'nesting depth' is then said to be 16. If it is exceeded, error message 76 will appear on the display screen.



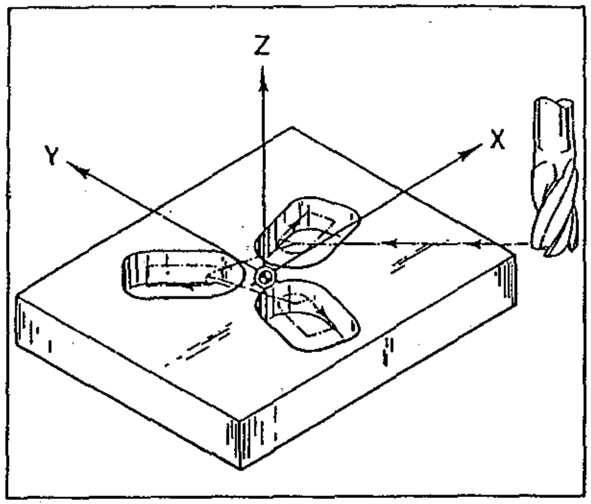
Unconditional jump - LO N...

This command is used for programming an unconditional jump to a certain block: you enter the block number from which the control is to continue the machining operation.

Macros

Example

The milling pattern in this example is to be programmed by rotating the coordinate system three times: once through an angle of 15° and twice through an angle of 120°.
The contour is defined just once in a macro.

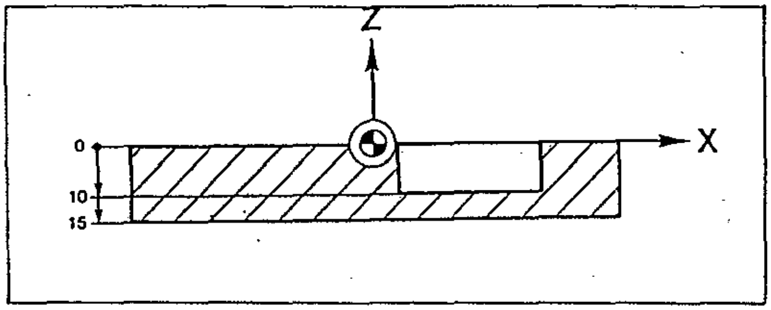


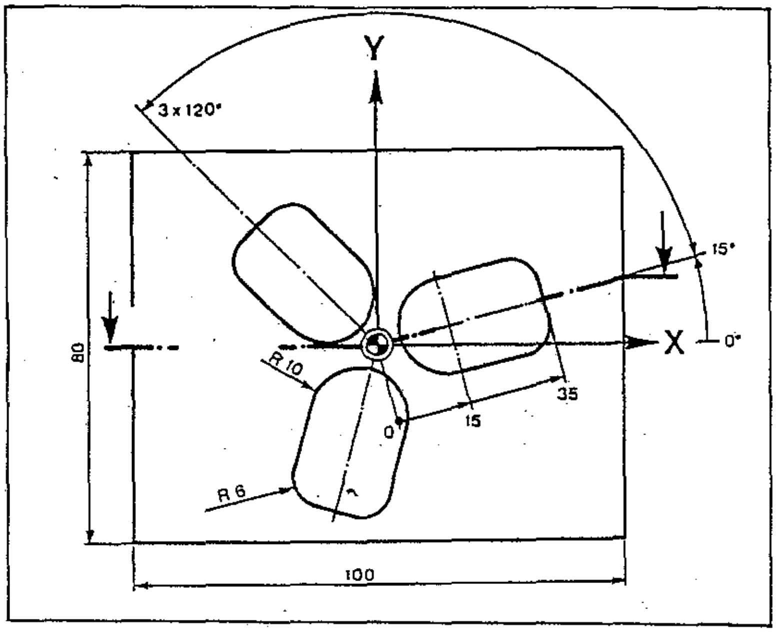
%405	Program number
N1 G0 Z100 N2 G17 T1	Tool change. Assign length compensation to Z axis and call tool compensation values.
N3 G56 W15 I0 J0	Rotate coordinate system through an angle of 15° for first contour (see page 6-74).
N4 %0*1 F250 S+2500	Call macro. Coordinates programmed in macro are now applied to rotated coordinate system.
N5 G55 W120 I0 J0	Rotate coordinate system again through an angle of 120° for second contour (see page 6-74).
N6 %0*1	Call macro. Second contour is machined.
N7 L1 N5 N6	Program part repetition: rotate coordinate system again through an angle of 120° for third contour. Call macro. Third contour is machined.
N8 G53 T0	Return coordinate system to start position (page 6-74).
N9 G0 Z100 M30	Tool retracted in Z axis, end of program.

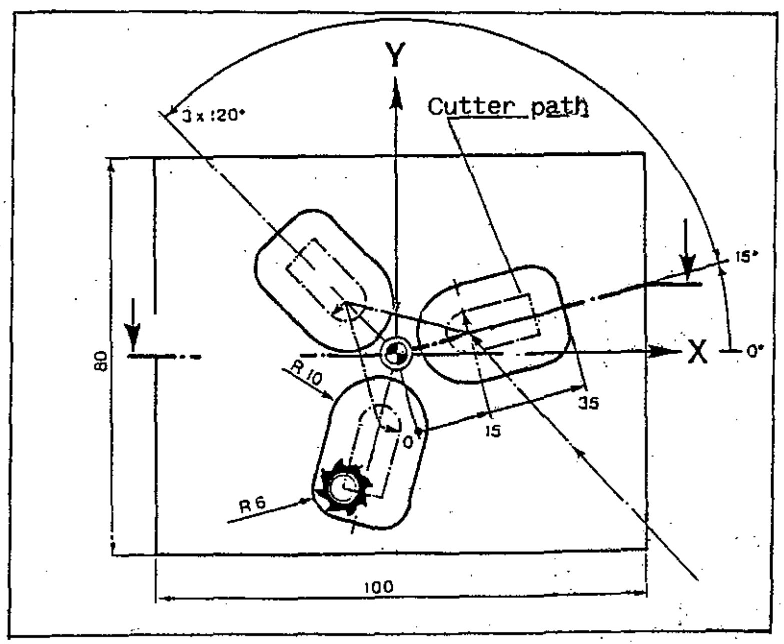
Macro

%0*1	Macro number
N1 G0 X15 Y0 Z2 N2 Z-10 N3 G41 G47 A2 X15 Y10 G1 G60 M61 N4 G3 X15 Y-10 I0 J-10 N5 X35 N6 Y10 N7 X15 N8 G40 G47 A2 N9 G0 Z2	Macro contains program blocks for milling the contour.

Macros







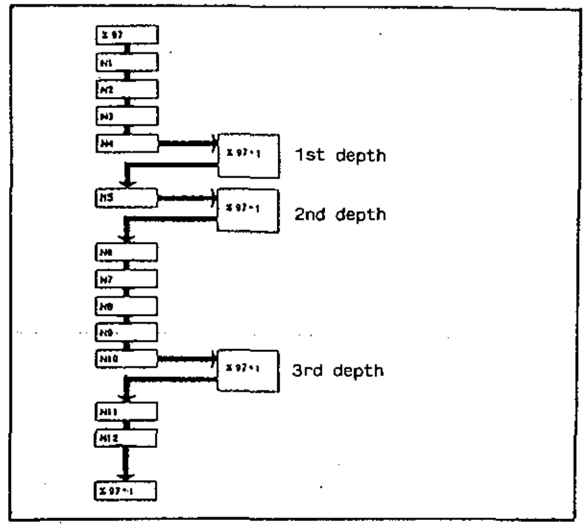
Dimensions in mm

Local subroutines

Example

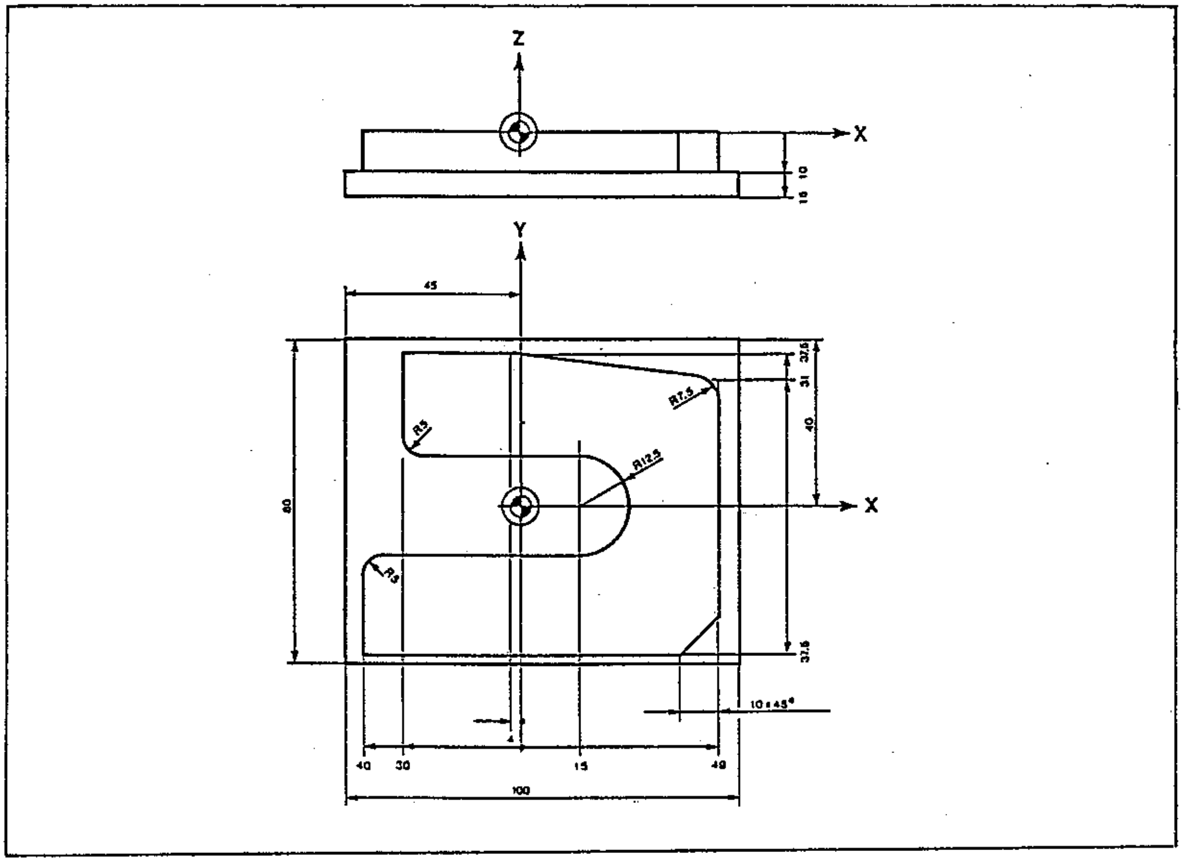
In the following example, the contour is first machined in two roughing passes, using a roughing cutter. This is followed by a tool change for finishing the contour with a finishing cutter at increased feed rate and spindle speed.

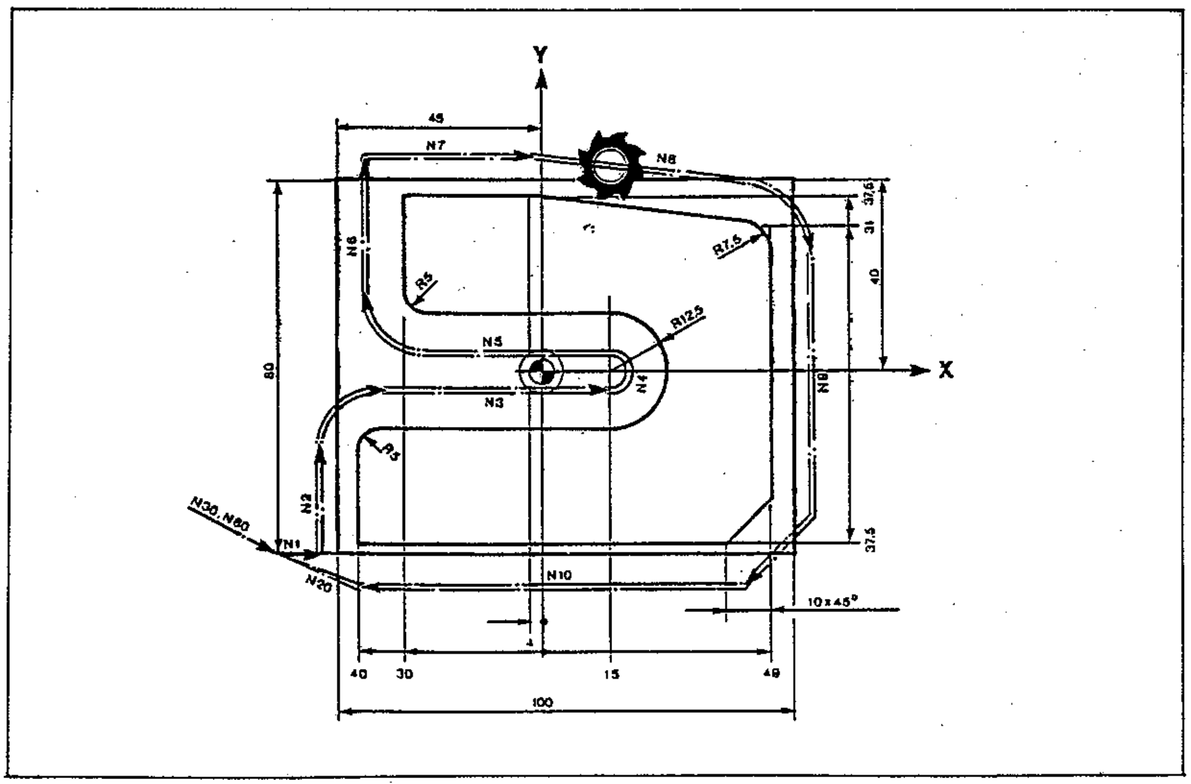
The contour is programmed as a subroutine called once for each cut.
Although the contour is machined three times, it need be programmed just once.



% 97	Program number
N1 G0 Z100 N2 G17 T1	Tool change. Assign length compensation to Z axis and call tool compensation values.
N3 G0 X-55 Y-40 Z2 F125 S+2000	Tool moves to contour at rapid rate.
N4 %*1 Z-5	Rapid downfeed to Z-5 (1st depth). Call subroutine.
N5 %*1 Z-9.8	Rapid downfeed to Z-9.8 (2nd depth). Call subroutine again.
N6 T0	Delete tool compensation.
N7 G0 Z100 N8 T2	Tool change, call new tool compensation values.
N9 G0 X-55 Y-40 Z2 F160 S+2500	New tool moves to contour at rapid rate. Higher feed rate and spindle speed for finishing cut.
N10 %*1 Z-10	Rapid downfeed to Z-10 (3rd depth). Call subroutine.
N11 T0	Delete tool compensation.
N12 G0 Z100 M30	Tool retracted at rapid rate, end of program.
Subroutine	
% 97 *1	Subroutine number
N1 G41 G45 A1 X-40 Y-37.5 G1 G60 M61	Call linear interpolation and contouring control. Call compensation on contour and mode of approach.
N2 Y-12.5 N3 G7 R5 N4 X15 N5 G3 X15 Y12.5 I0 J12.5 N6 X-30 N7 G7 R5 N8 Y37.5 N9 X-4 N10 X49 Y31 N11 G7 R7.5 N12 Y-37.5 N13 G8 R10 N14 X-41	Milling of contour.
N15 G40 G45 A1 N16 G0 X-55 Y-40	Delete compensation of contour. Tool moves to starting point at rapid rate. End of subroutine.

Local subroutines

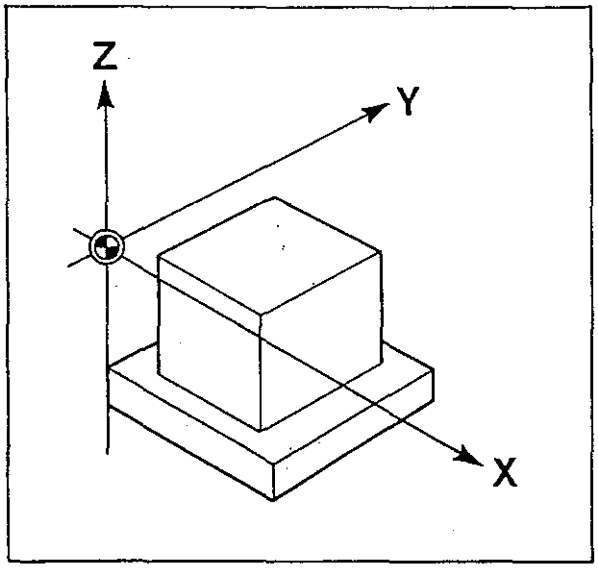




Dimensions in mm

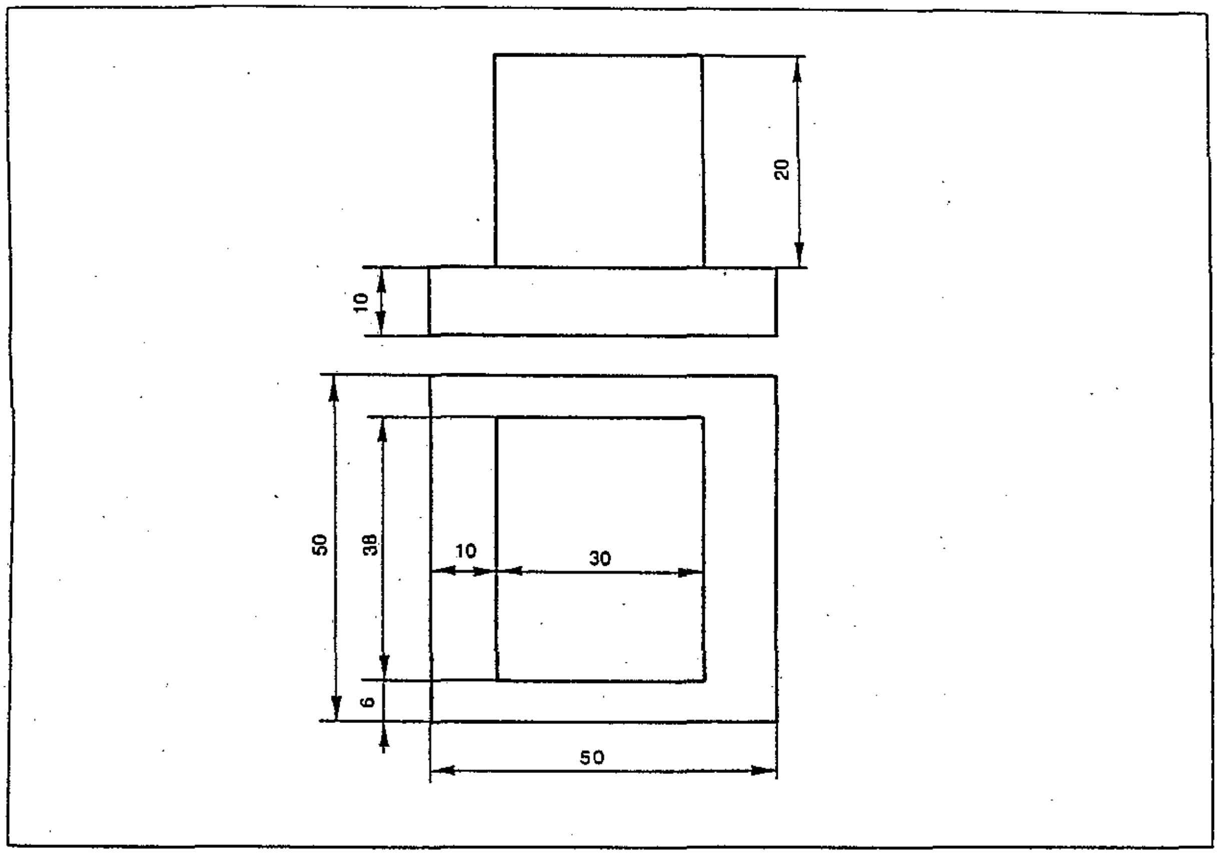
Program part repetition

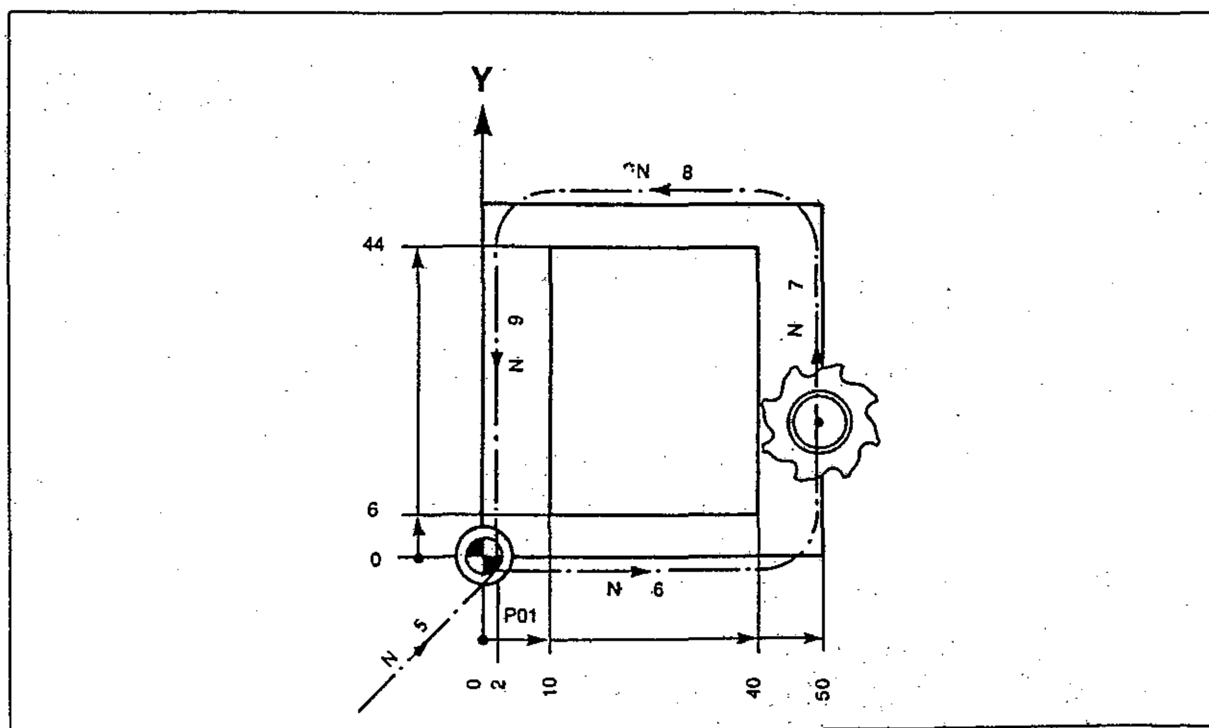
EXAMPLE
Simple program part repetition



% 95	Program number
N1 G0 Z100 N2 G17 T1	Tool change. Assign length compensation to Z axis and call tool compensation values.
N3 G0 Z2 S+500	Positioning in Z and spindle start.
N4 G91 Z-11 F100 M70	Blocks skipped temporarily because of M70.
N5 G42 G45 A1 X10 Y6 G0 G60 M62	Approach command: tool moves parallel to contour to point X10 Y6. Downfeed in Z axis by Z-11 (chain dimension) because of G91 in block N4.
N6 G90 X40 F200 N7 Y44 N8 X10 N9 Y6	Call absolute input; workpiece is machined.
N10 G40 G45 A1	Delete compensation on contour.
N11 L1 N4 N10	Program part repetition from N4 to N10. Tool again downfed by Z-11 because of G91 in block N4. Starting position in block N3 was Z2. 2x (-11) = -22; thus total downfeed = -20.
N12 T0	Delete tool compensation.
N13 G0 Z100 M30	Tool retracted in Z axis, end of program.

Program part repetition

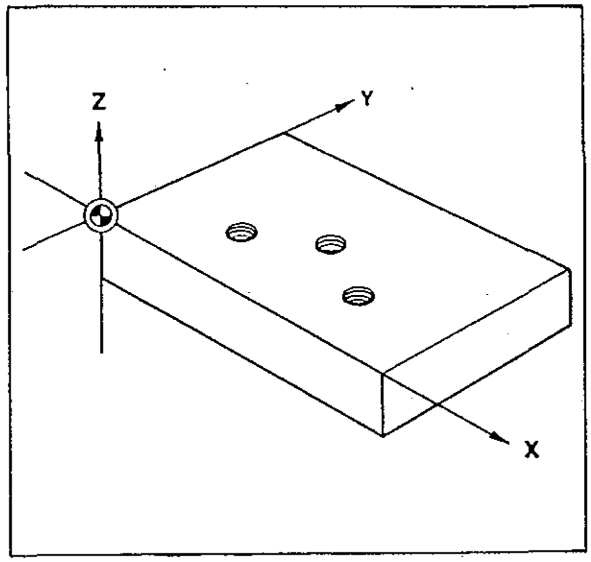




Dimensions in mm

Program part repetition with subroutine substitution

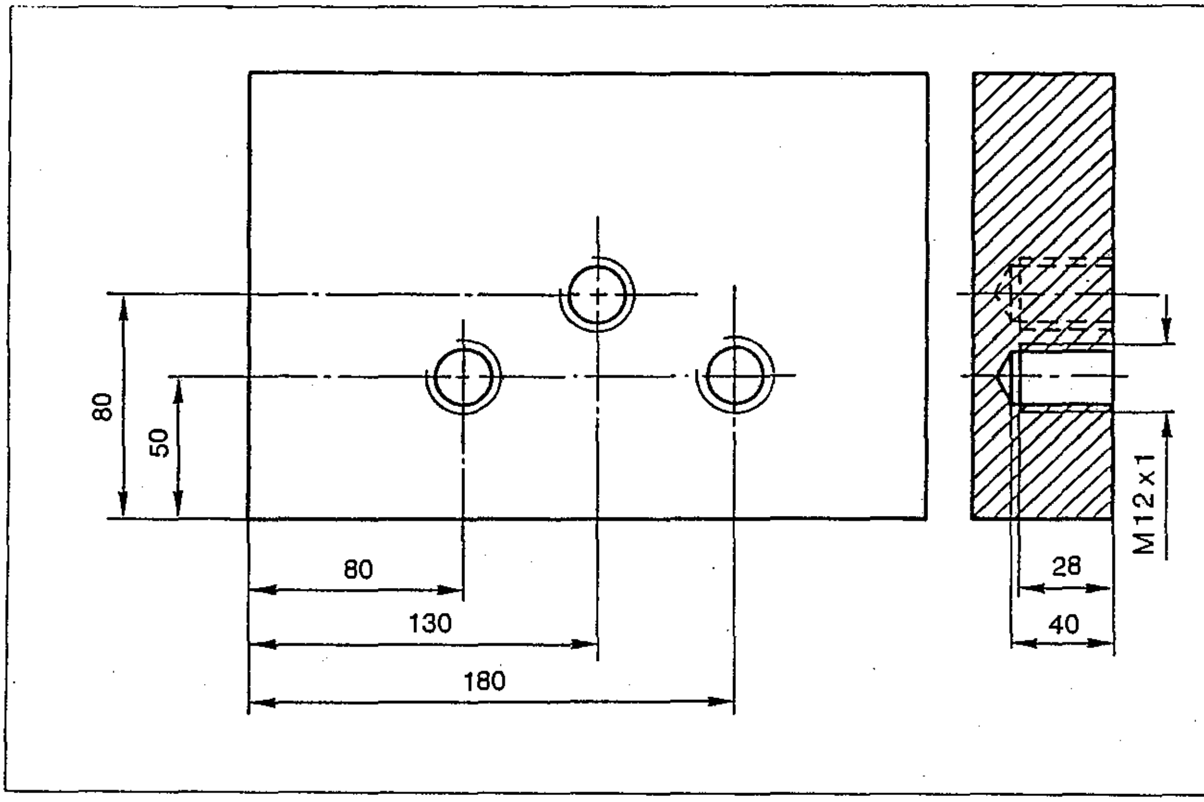
EXAMPLE
Centering
Drilling
Tapping

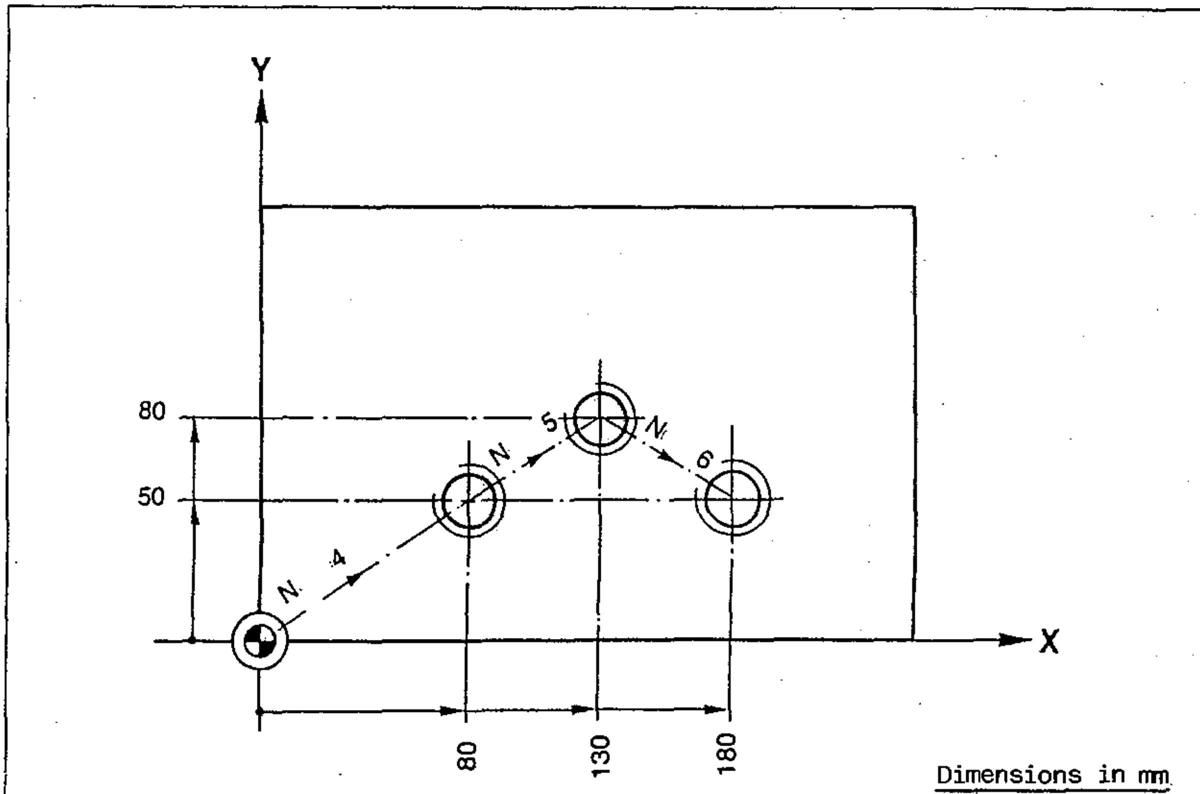


Tool table:
T1 R A L-50 A (centering drill)
T2 R A L0 A (twist drill)
T3 R A L60 A (tap)

%105	Program number
N1 G0 Z100 N2 G17 T1	Tool retracted in Z and tool change. Assign length compensation to Z axis and call tool compensation.
N3 G0 Z2	Positioning in Z.
N4 N*1 G0 X80 Y50 N5 N*1 G0 X130 Y80 N6 N*1 G0 X180 Y50	Tool moves to programmed coordinates; drilling cycle N*1 (centering) performed (for G81 see page 6-88).
N7 T0	Delete tool compensation.
N8 G0 Z100	Tool retracted in Z.
N9 T2	Tool change, call tool compensation.
N10 L1 N3 N8 N*2	Program part repetition of blocks N3 to N8. N*2 substituted for N*1. Cycle N*2 (drilling) performed.
N11 T3	Tool change, call tool compensation.
N12 G0 Z5	Positioning in Z.
N13 L1 N4 N8 N*3	Program part repetition of blocks N4 to N8. N*3 substituted for N*1. Cycle N*3 (tapping - G84) performed.
N14 M30	End of program.
N*1 G81 F200 S+3150 Z-7	Centering
N*2 G81 F160 S+500 Z-42	Drilling
N*3 G84 F200 S+200 Z-33	Tapping

Program part repetition with subroutine substitution

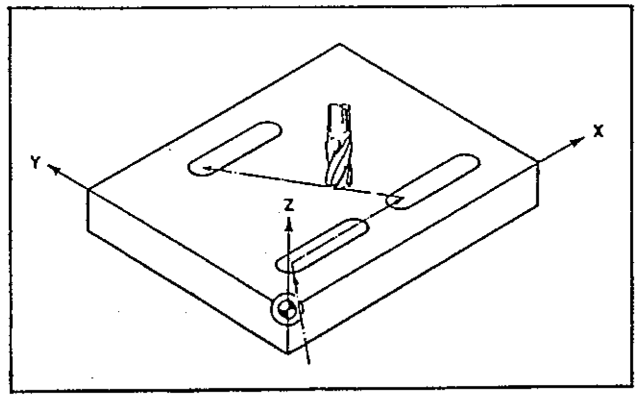




Program part repetition

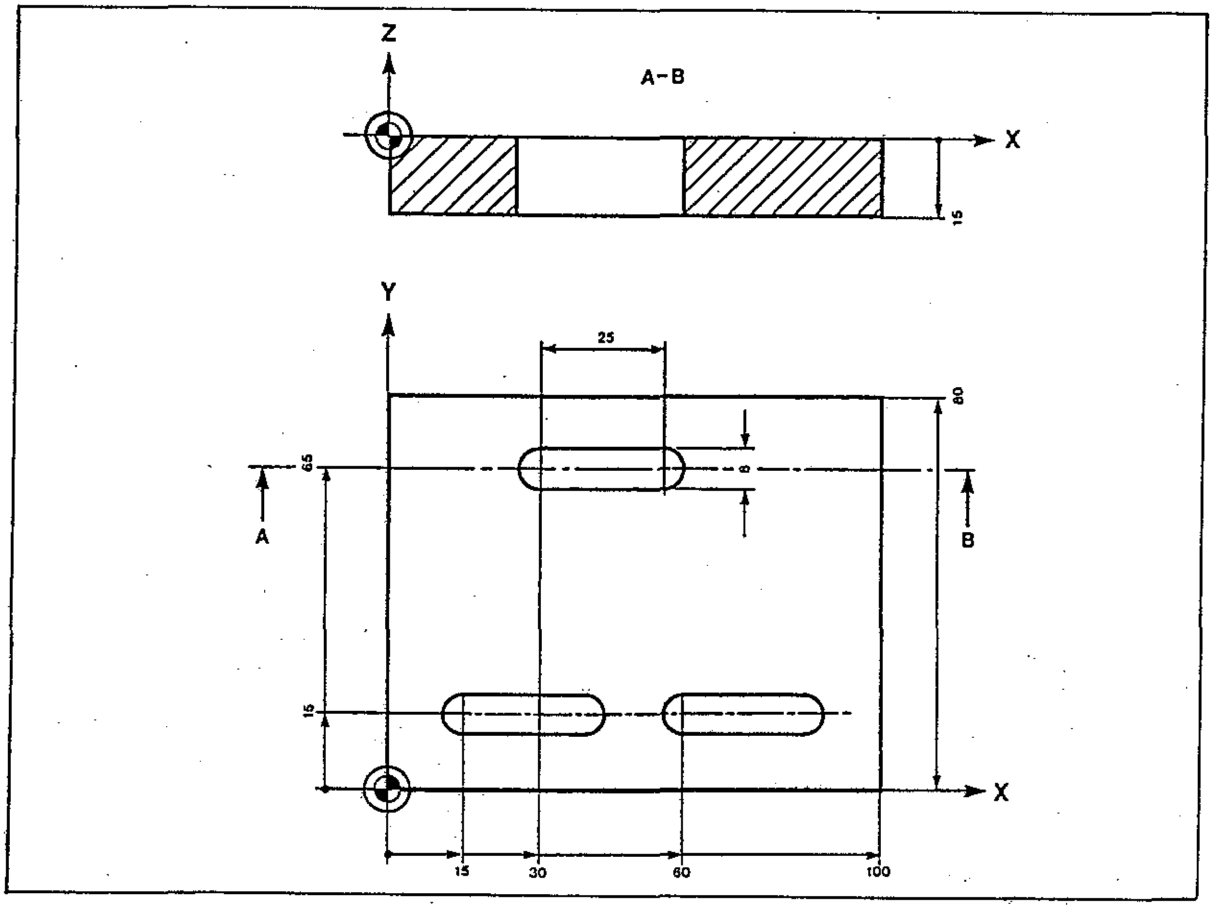
EXAMPLE

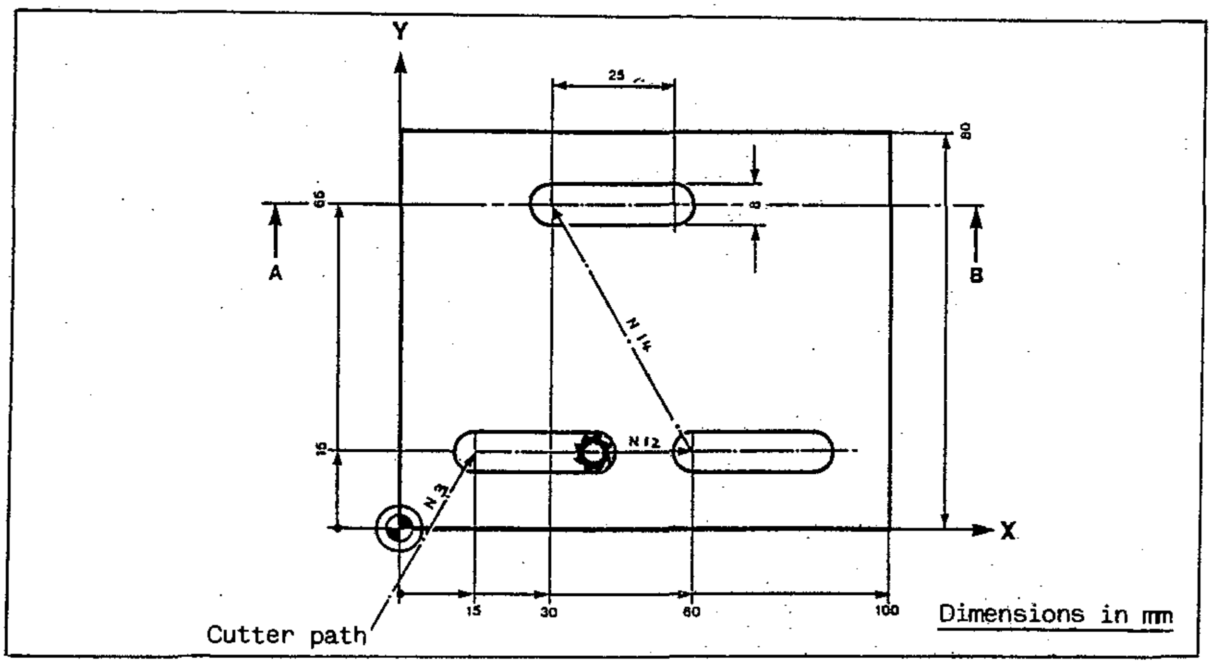
Nesting



% 400	Program number
N1 G0 Z100 N2 G17 T1	Tool change. Assign length compensation to Z axis and call tool compensation.
N3 G0 X15 Y15 Z2 S+2000	Tool moves to start position for machining 1st slot.
N4 Z0 F80	Downfeed in Z axis; from here start of first chain dimension.
N5 G91 Z-2 F80	Downfeed in Z in chain dimension.
N6 X25 F125	Milling at first depth (1st pass).
N7 Z-2 F80 N8 X-25 F125	Milling at second depth (2nd pass).
N9 L3 N5 N8	Program part repetition for further depths.
N10 G90 N11 G0 Z2	Call absolute dimensions; tool retracted in Z axis.
N12 G0 X60 N13 L1 N4 N11 N14 G0 X30 Y65 N15 L1 N4 N11	Milling of 2nd and 3rd slots.
N16 T0	Delete tool compensation.
N17 G0 Z100 M30	Tool retracted in Z axis, end of program.

Program part repetition





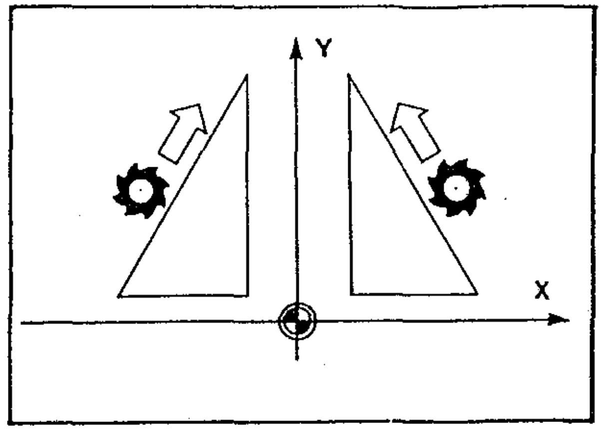
Mirror imaging

Mirror imaging means that the control changes signs of the the mirror-imaged coordinates.

If mirror imaging about one axis is used, the contour is machined

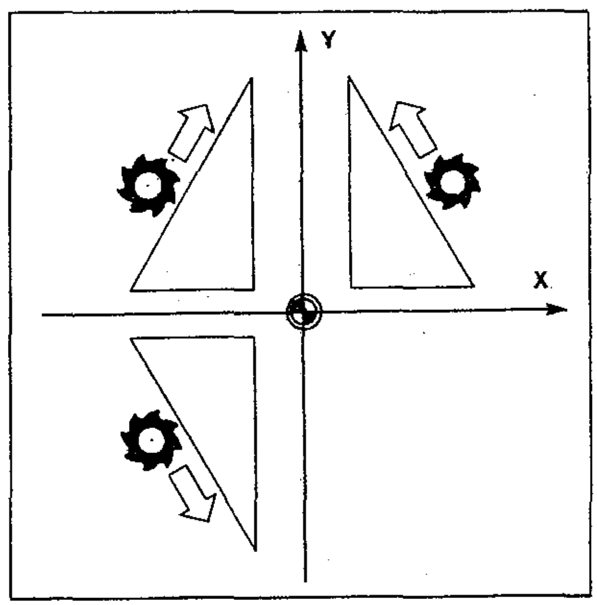
- mirror-inverted,
- on the other side of such axis,
- in the same size and
- at the same distance from the axis.

The cutting direction along the contour is automatically reversed.



If mirror imaging about two axes is used, the procedure described above is performed twice consecutively.
This means the contour is again machined the right way round. The same applies to the cutting direction.

The starting points of drilling and milling cycles can also be mirror-imaged, but the cutting direction in cycles always remains the same.
Zero offsets may be mirror-imaged as well.



Mirror imaging - M81 to M86

Mirror imaging is called by programming any of commands M81 to M86. Command M80 deletes a previously activated mirror imaging operation.

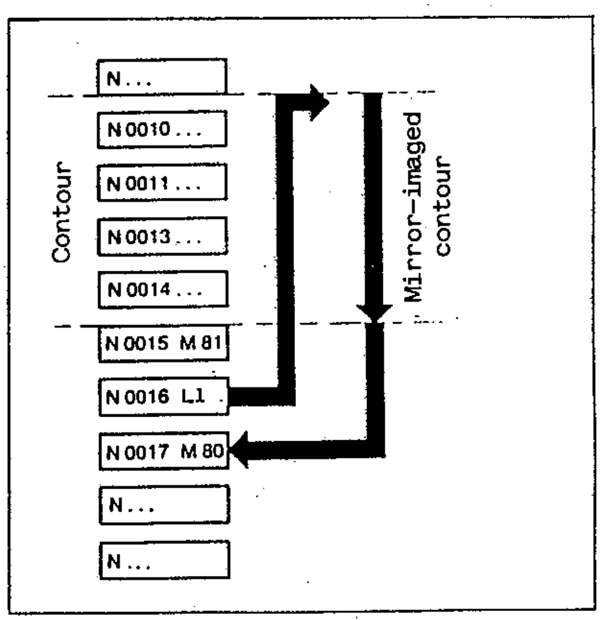
Relationships:

M function	Sign change in
M81	X and I
M82	Y and J
M83	Z and K
M84	X, I, Y and J
M85	X, I, Z and K
M86	Y, J, Z and K
M80	Deletion

Commands M80 to M86 are self-retaining

If, for example, M81 is programmed in a block, the signs of all subsequent X and I coordinates in the program are changed: the contour is machined mirror-inverted.

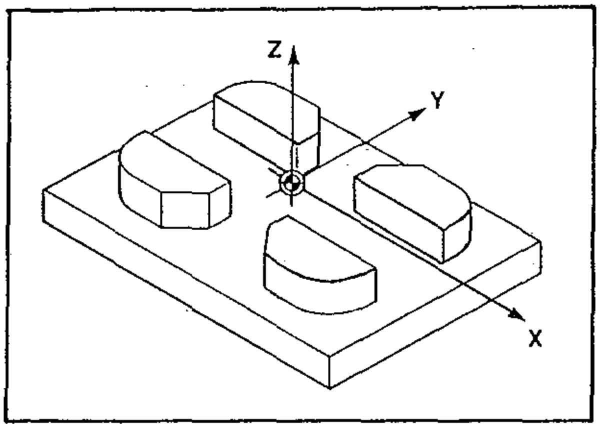
If a repetition of a program section with a contour is programmed after command M81, such contour will be machined once more, but again mirror-inverted (with opposite signs in X and I).



Mirror imaging

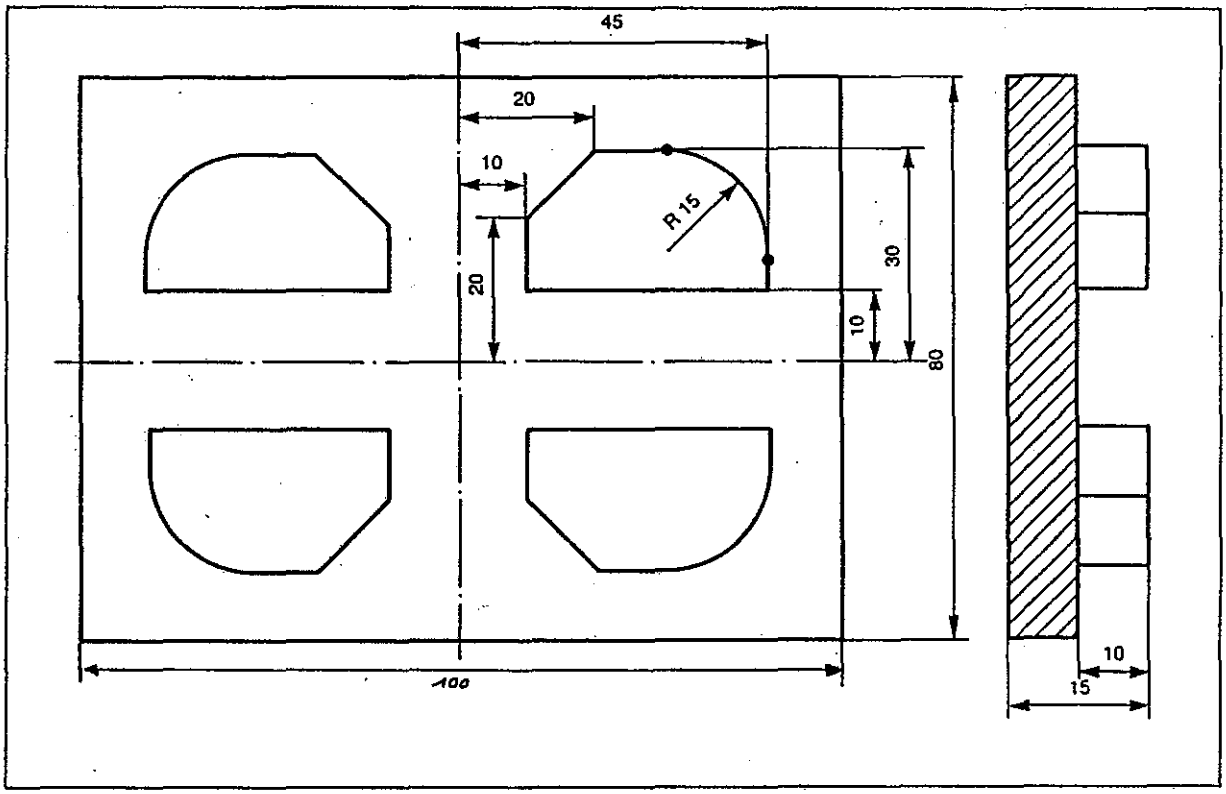
EXAMPLE

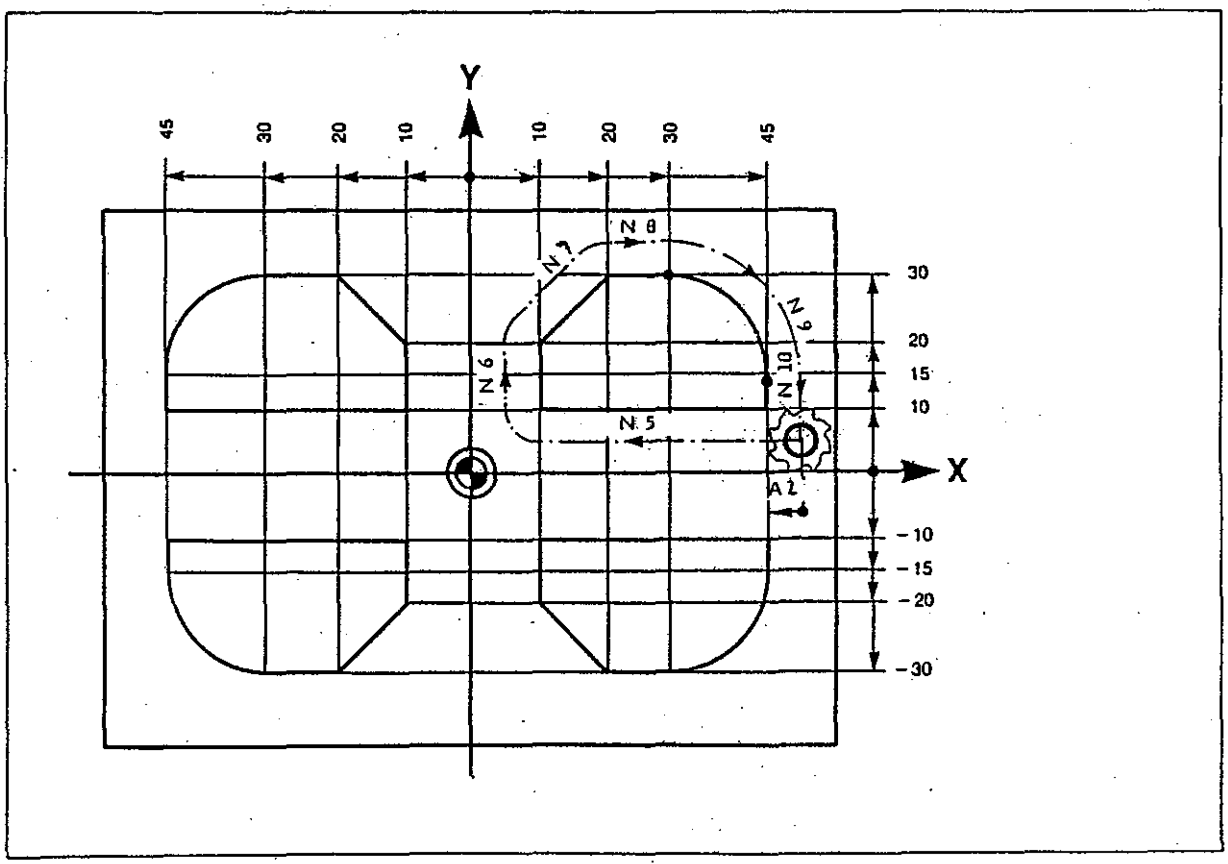
Mirror imaging about one and two axes



% 110	Program number
N1 G0 Z100 N2 G17 T1	Tool change. Assign length compensation to Z axis and call tool compensation values.
N3 %*1	Call subroutine: upper right contour is machined.
N4 %*1 M81	Call subroutine with mirror imaging of X coordinate by change of sign: upper left contour is machined.
N5 %*1 M84	Call subroutine with mirror imaging of X and Y coordinates by change of signs: lower left contour is machined.
N6 %*1 M82	Call subroutine with mirror imaging of Y coordinate by change of sign: lower right contour is machined.
N7 T0	Delete tool compensation.
N8 G0 Z100 M30	Tool retracted in Z axis, end of program.
%110 *1	Subroutine number
N1 G0 X47 Y0 Z2 S+2000 N2 Z-10 F100 N3 F200 N4 G41 G45 A2 X45 Y10 G1 G60 M61 N5 X10 N6 Y30 N7 G8 R10 N8 X45 N9 G7 R15 N10 Y10 N11 G40 G45 A2 N12 G0 Z2	Subroutine contains commands for milling the contour.

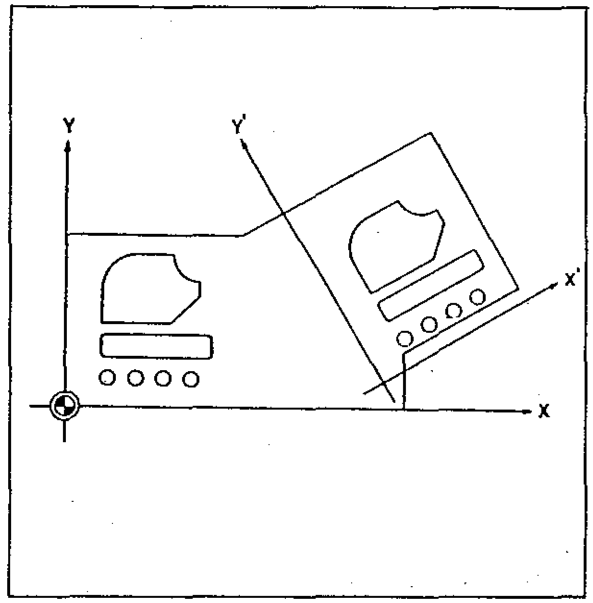
Mirror imaging





Linear and rotary offset of coordinate system

The coordinate system may be relocated, which means moved to, or rotated about, any desired point in the course of a program.
From the block in which such a relocation is programmed, all coordinates are based on the new coordinate system.



If several identical machining operations have to be performed at different points on a workpiece, the coordinate system may be shifted or rotated, as required, and then the cutting operations simply be repeated.

Linear and rotary offset of coordinate system

A relocation of the coordinate system may be composed of

- a linear offset

and

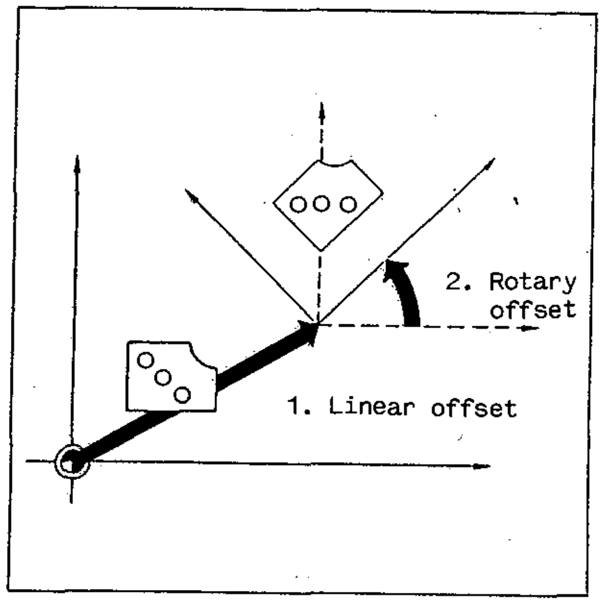
- a rotary offset.

In the same way, the coordinate system may only be shifted in a linear direction or only be rotated about a pole (point of rotation).

If a linear offset and a rotary offset are programmed at the same time, the control will proceed in the following order.

First, the coordinate system is shifted paraxially. This results in the establishment of an 'intermediate coordinate system' which will then be the basis for defining the coordinates of the pole. Finally, the coordinate system is rotated about this pole through the desired angle.

There are two alternatives: absolute or additive offset.
An absolute linear or rotary offset of the coordinate system is always based on the original coordinate system used when the machine was set up.
An additive linear or rotary offset is always based on the current coordinate system.



Additive linear/rotary offset of coordinate system - G55

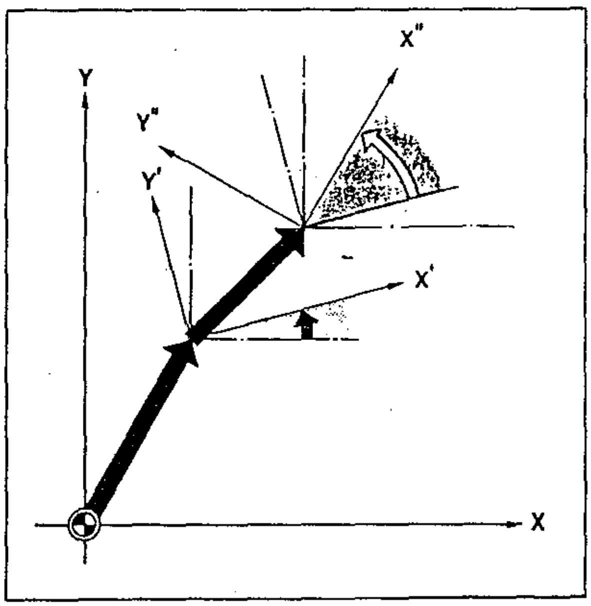
INPUT DIALOG

When command G55 has been entered, the control requests the addresses listed on the right.
The relocation is always based on the zero point of the current paraxial coordinate system.

G55	Additive linear/rotary offset of coordinate system The control requests:
X (Y,Z,C)	Offset in 1st axis (if only rotation without linear offset: transfer key)
Y (Z,C)	Offset in 2nd axis
W	Angle of rotation of coordinate system (if only rotation without linear offset: transfer key)
I(J)(K)	1st coordinate of pole
J(K)(K)	2nd coordinate of pole

The coordinates of a pole are referred to the already displaced 'intermediate coordinate system'.

If a linear/rotary offset is already active, the values refer to the zero point of the last preceding 'intermediate coordinate system'.
Subsequent rotary offsets are only permissible if the pole (referred to the 'intermediate coordinate system') remains the same.



Main relocation plane parallel to ... plane		Pole coordinates with addresses		Angle reference axis parallel to ... axis	
	X,Y		I,J		X+
	Y,Z		J,K		Y+
	Z,X		I,K		X+

Absolute linear/rotary offset of coordinate system – G56

INPUT DIALOG

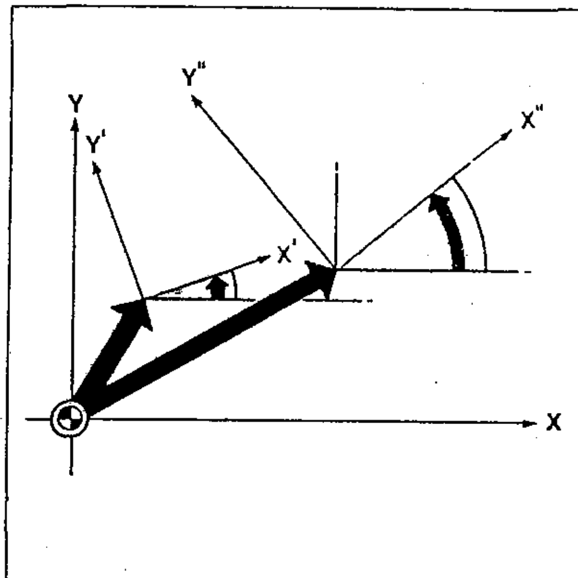
When command G56 has been entered, the control requests the addresses listed on the right.

G56 Absolute linear/rotary offset of coordinate system
The control requests:

X (Y,Z,C)	Offset in 1st axis (if only rotation without linear offset: transfer key)
Y (Z,C)	Offset value in 2nd axis
W	Angle of rotation of coordinate system (if only rotation without linear offset: transfer key)
I(J)(K)	1st coordinate of pole
J(K)(K)	2nd coordinate of pole

The relocation is always based on the absolute program zero point as defined in setting up the machine (see page 4-8) of by G54 (set actual value, page 6-78).

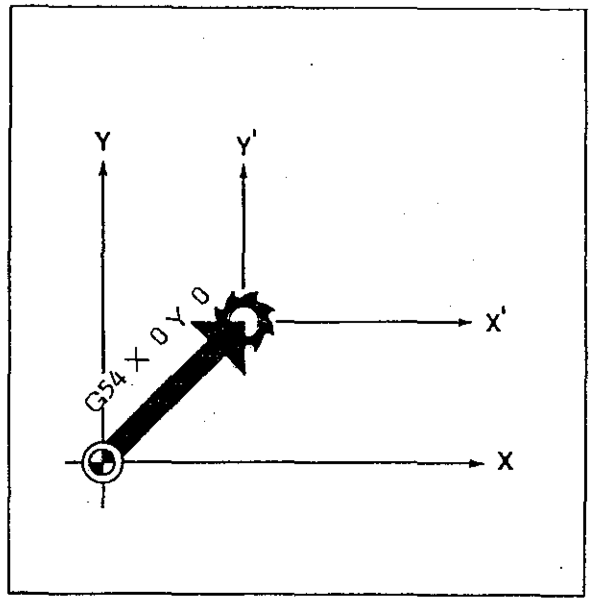
The coordinates of the pole should be based on the already relocated 'intermediate coordinate system'.



Main relocation plane parallel to ... plane	Pole coordinates with addresses	Angle reference axis parallel to ... axis
X,Y	I,J	X+
Y,Z	J,K	Y+
Z,X	I,K	X+

Set actual value – G54

Command G54 assigns new coordinate values to the current tool position.



Command G54 is not permissible as long as G55/G56 (additive or absolute linear/rotary offset of the coordinate system) is active in a program
Remedy: first write command G53 (return to program zero).
Command G54 is not permissible as long as compensation on contour is active (between G41/42 and G40).

EXAMPLE

% 115	Program number
N1 G0 X10 Y5	Tool moves to X10 Y5 (based on program zero as defined in setting up the machine) at rapid rate.
N2 G54 X0 Y0	Coordinates X0 and Y0 are assigned to current tool position which is thus made the new zero point of the coordinate system.
N3 G0 X10 Y5	Tool moves to X10 Y5 (based on new zero point) at rapid rate. This point would have had the coordinates X2 Y10 in the old coordinate system.

Return to program zero - G53

Linear and/or rotary relocations of the coordinate system can be cancelled by command G53.
Command G53 deletes commands G54, G55 and G56: all subsequent coordinates (X, Y, Z, C) in the program are again based on the original program zero defined in setting up the machine.

A block with G53 may contain a feed motion. Rapid traverse motions (G0) are not permissible.

When changing the mode of operation or operating the emergency stop, the program is interrupted and the control 'forgets' any linear or rotary offsets programmed by G54, G55 or G56. For re-entry and continuation of program after program interruptions see Chapter 7.

Moving to reference point - G52

Command G52 (with input of the desired axes) serves to move the machine slides in these axes their respective reference points at rapid rate.

EXAMPLE

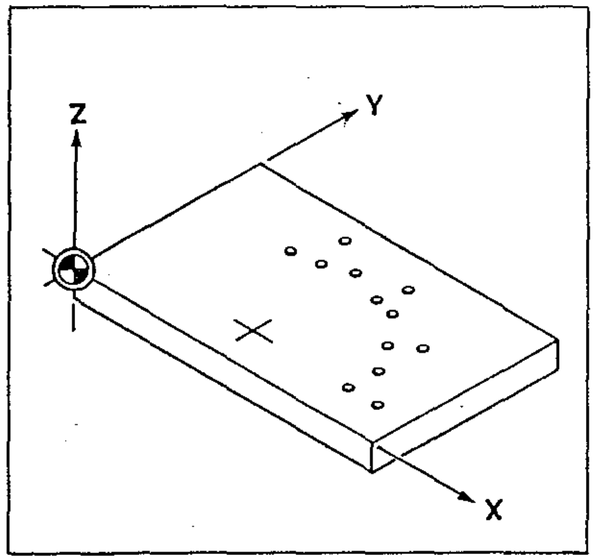
% 120	Program number
N10 G52 X0 Y0 Z0 C0	All machine slides move to their reference points in block N10. (C0 can be used to move the NC rotary table to its reference point, see page 6-145).

Additive linear/rotary offset of coordinate system – G55

EXAMPLE

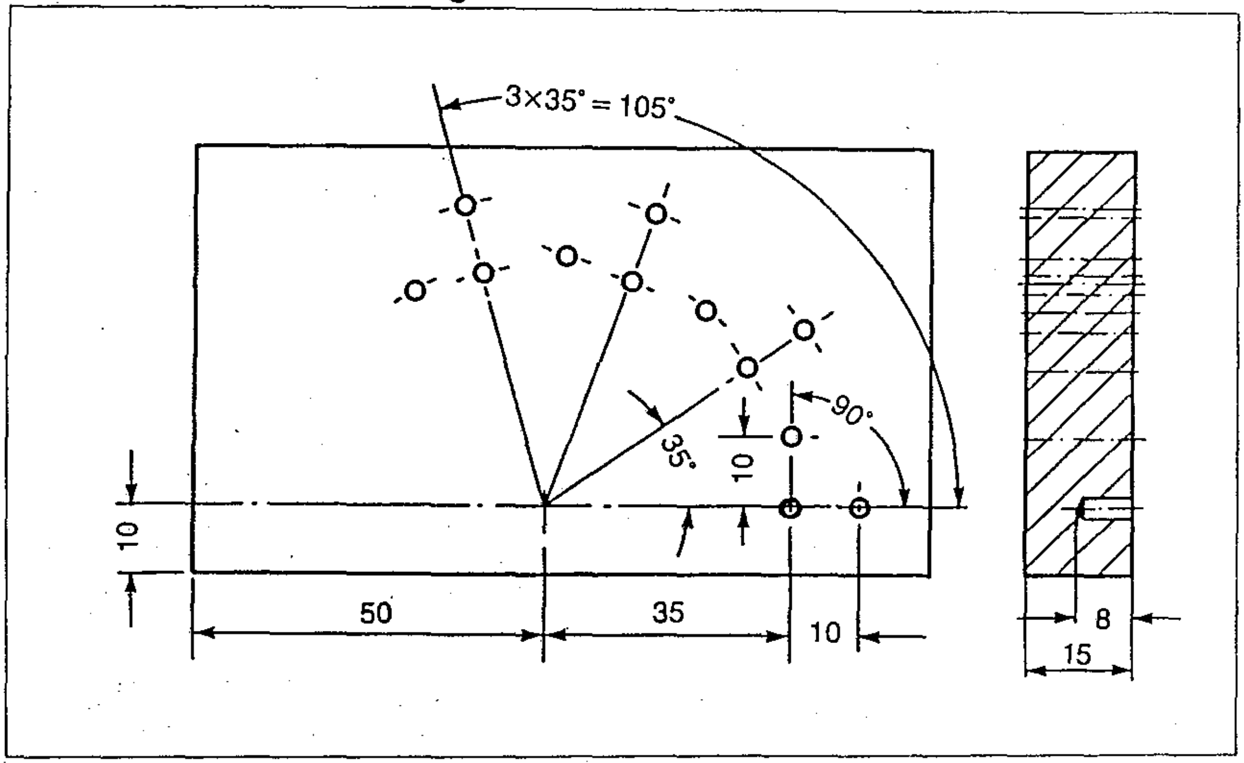
Linear and rotary offset

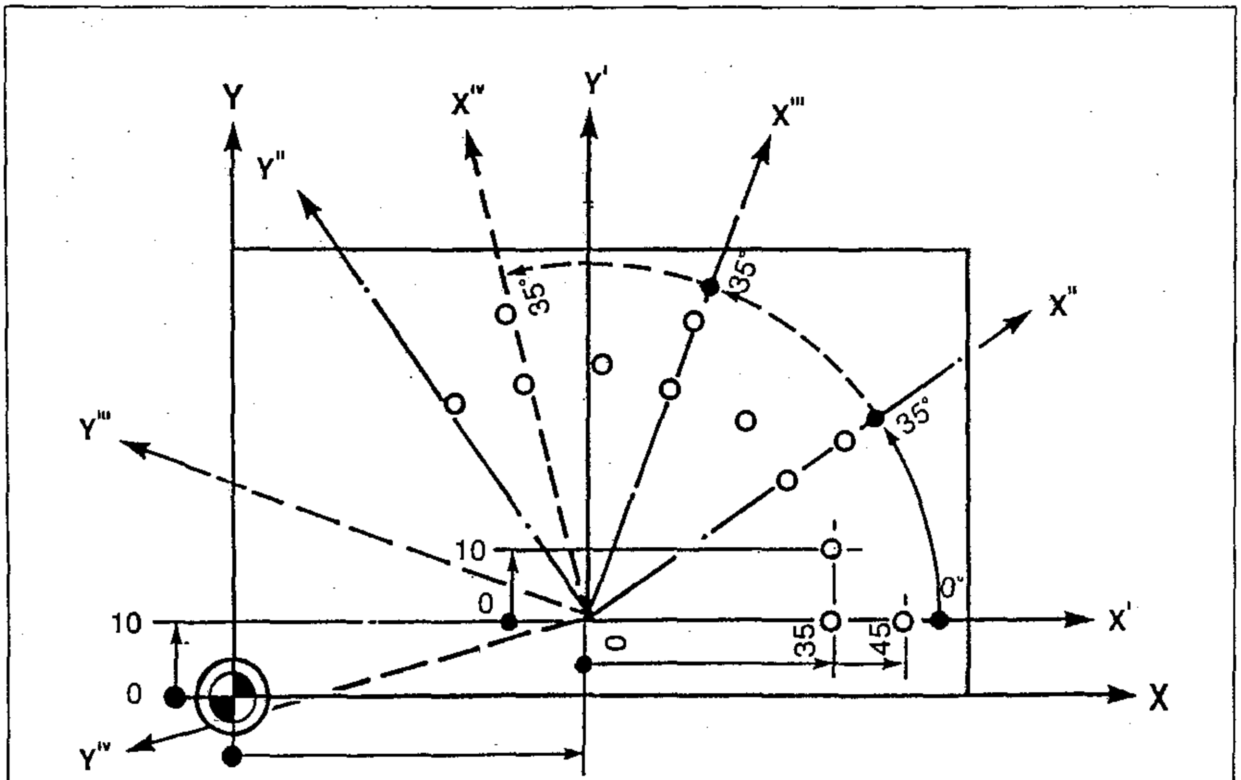
In this example, a drilling pattern is successively rotated three times about the same angle. The zero point of the coordinate system is first placed at the pole (point of rotation) and then the coordinate system is repeatedly rotated by the additive method.



% 125	Program number
N1 G0 Z100 N2 G17 T1	Tool change. Assign length compensation to Z axis and call tool compensation values.
N3 G0 Z2	Positioning above workpiece surface at rapid rate.
N4 G55 X50 Y10	Additive linear offset of coordinate system by 50 mm in X axis and 10 mm in Y axis: all subsequent coordinate data are now based on new (offset) coordinate system.
N5 N*1 G0 X35 Y0 N6 N*1 G0 X45 N7 N*1 G0 X35 Y10	Drilling pattern is machined.
N8 G55 W35 I0 J0	Additive rotary offset of coordinate system about zero point (I 0, J 0) of current paraxial coordinate system; angle of rotation 35°. All subsequent coordinate data are now based on new (rotated) coordinate system.
N9 L3 N5 N8	Blocks N5 to N8 repeated three times, each time with 35° rotary offset (rotary offsets are added), then drilling pattern is machined.
N10 G53	Delete linear and rotary offset: all subsequent coordinate data are again based on the original program zero point.
N11 T0	Delete tool compensation.
N12 G0 X0 Y0 Z100 M30	Tool retracted in Z and positioned above program zero. End of program.
N*1 G81 F100 S+2500 Z-10	Subroutine contains drilling cycle G81.

Additive linear/rotary offset of coordinate system – G55

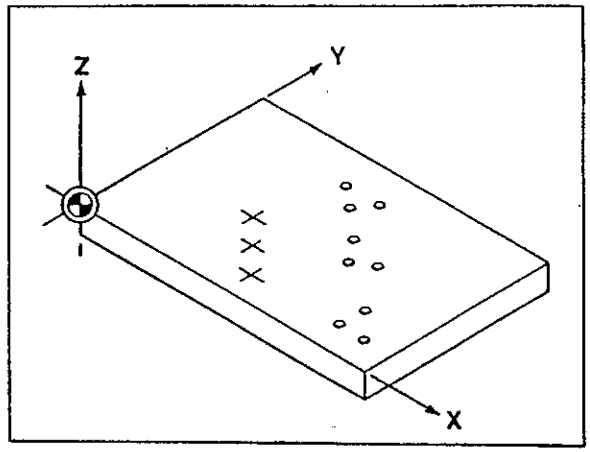




Dimensions in mm

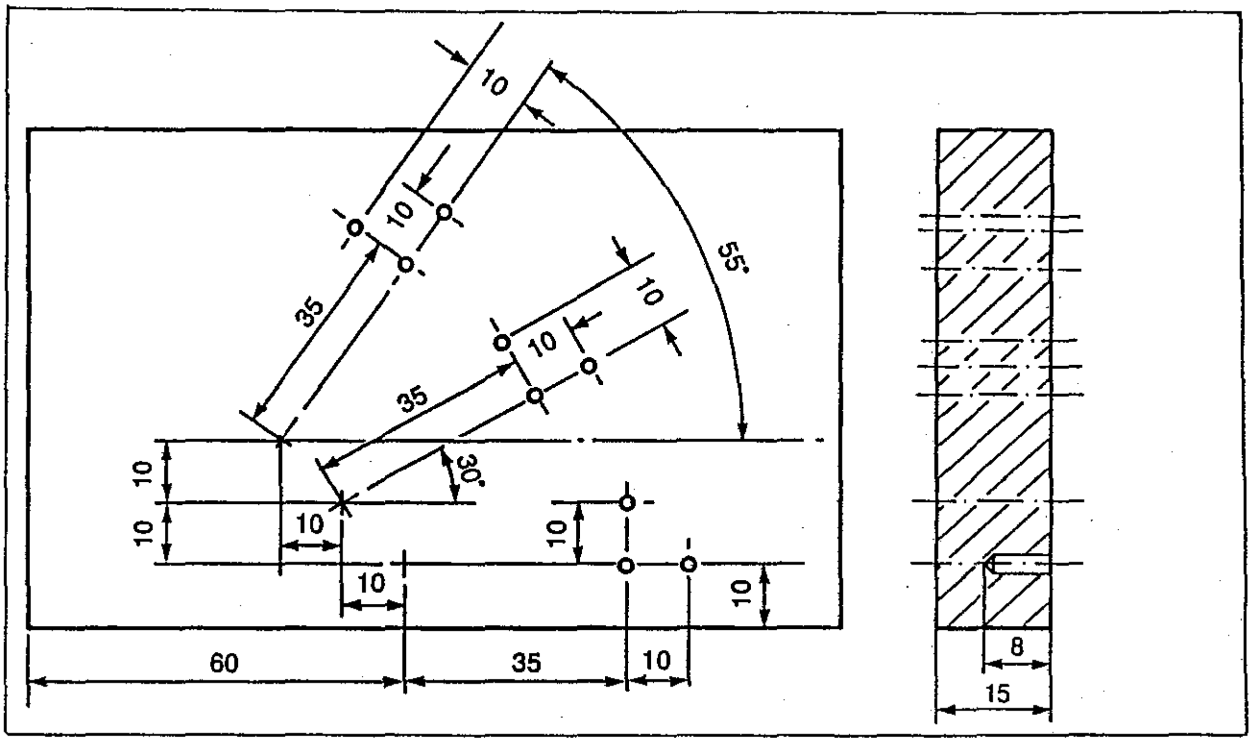
Absolute linear/rotary offset of coordinate system – G56

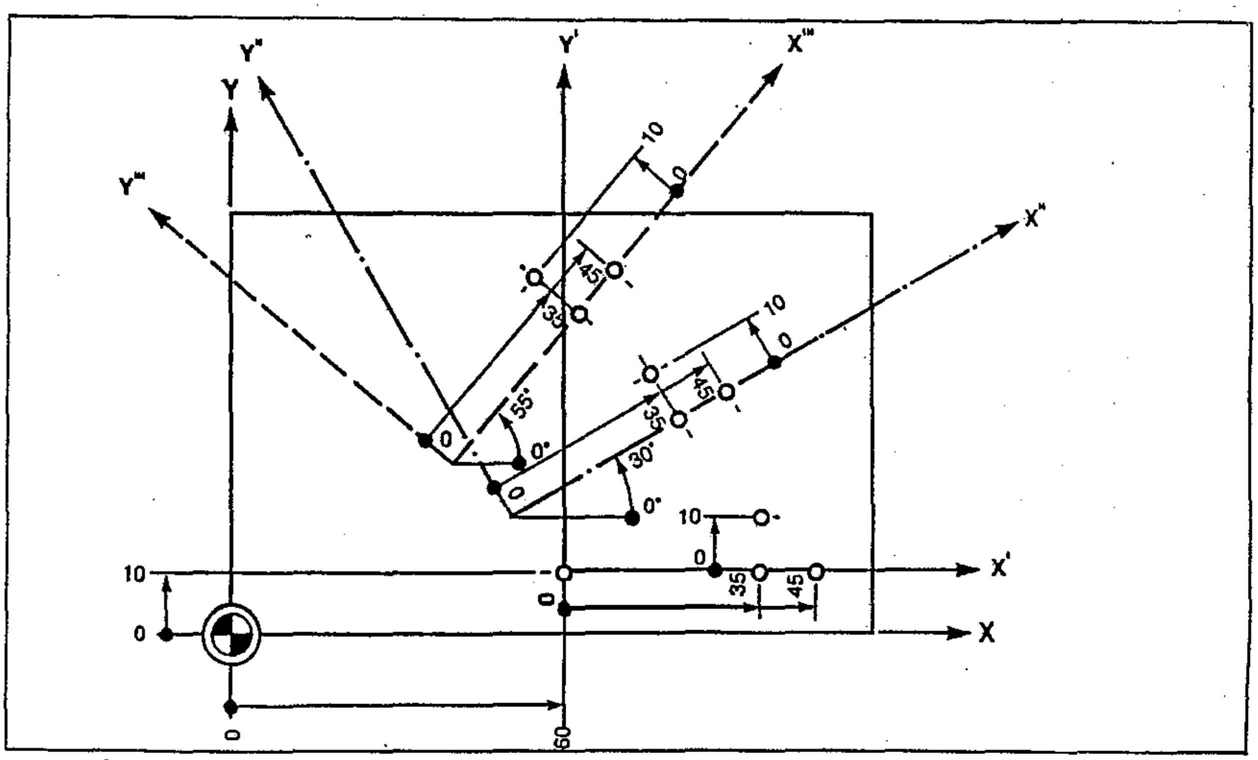
EXAMPLE
Linear and
rotary offset



% 130	Program number
N1 G0 Z100 N2 G17 T1	Tool change. Assign length compensation to Z axis and call tool compensation values.
N3 G0 Z2	Positioning above workpiece surface at rapid rate.
N4 G56 X60 Y10	Linear offset of coordinate system (absolute) by 60 mm in X axis and 10 mm in Y axis: all subsequent coordinate data are now based on new (offset) coordinate system.
N5 N*1 G0 X35 Y0 N6 N*1 G0 X45 N7 N*1 G0 X35 Y10	Drilling pattern is machined.
N8 G56 X50 Y20 W30 I0 J0	Absolute linear and rotary offset of coordinate system, based on program zero (G56): first, linear offset of X50 Y20, then rotary offset about relocated zero point (I0, J0): all coordinate data are now based on the new (offset and rotated) coordinate system.
N9 L1 N5 N7	Program part repetition: drilling pattern machined.
N10 G56 X40 Y30 W55 I0 J0	Renewed absolute linear and rotary offset of coordinate system, based on program zero (G56); see block N8.
N11 L1 N5 N7	Program part repetition: drilling pattern machined.
N12 G53	Delete linear and rotary offset: all subsequent coordinate data are now again based on the original program zero point.
N13 T0	Delete tool compensation.
N14 G0 Z100 M30	Tool retracted in Z axis, end of program.
N*1 G81 F100 S+1250 Z-10	Subroutine contains drilling cycle G81.

Absolute linear/rotary offset of coordinate system - G56





Dimensions in mm

Drilling and milling cycles

Cycles are preprogrammed sequences of commands for frequently used cutting operations.
The necessary tool motions are calculated and performed by the control automatically. Programming of cycles is dialog-assisted by the control: upon entering the cycle call, the control requests all the data to be entered.

Cycle call commands:

Milling cycles

G71	– Milling of rectangular pockets (roughing, conventional) (page 6–102)
G72	– Milling of rectangular pockets (roughing, climb and conventional) (page 6–104)
G72*1	– Milling of rectangular pockets – corner radius programmable (roughing, climb and conventional) (page 6–106)
G73	– Milling of rectangular pockets (finishing to size) (page 6–110)
G74	– Milling of rectangular pockets (finishing to size with intermediate stop) (page 6–112)
G74*1	– Milling of rectangular pockets – corner radius programmable (finishing to size with intermediate stop) (page 6–114)
G75	– Milling of pins (page 6–118)
G76	– Milling of circular pockets (page 6–120)
G77	– Milling of external threads (page 6–124)
G78	– Milling of internal threads (page 6–126)
G79	– Milling of contour pockets (page 6–128)

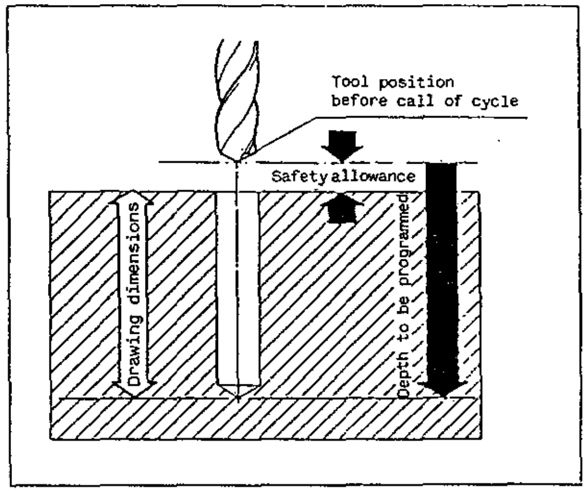
Drilling cycles

Drilling cycles can be performed with either the vertical or the horizontal spindle. In the latter case, the axis addresses requested by the control have to be altered accordingly. This also applies to plus and minus signs.

In the event of input errors proceed as follows:
- when you have entered a wrong word, use delete key on NC keyboard (23), but NOT during graphic display of cycle, and reprogram the entire cycle;
- when you find an error upon completion of input, you may correct each word in the cycle separately. In this case, always check all the signs used in the cycle.

Safety allowance

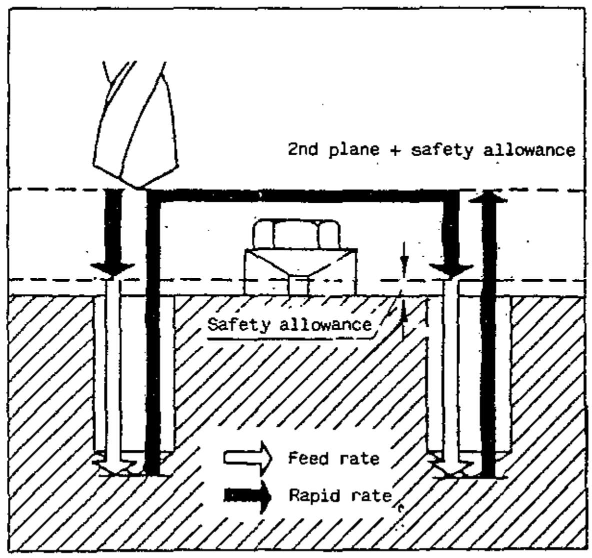
The drilling (boring, milling etc) depth should be programmed in chain dimensions in all cycles. The depth to be programmed is the sum of the drilling or milling depth (as specified in the part drawing) and a safety allowance.
The safety allowance is determined by the position of the tip of the tool before the cycle is called.



2nd plane

In all cycles, a second plane can be programmed. At this level, the tool can move across the workpiece without colliding with an obstacle (see illustration).

At the beginning of a cycle, the tool is advanced at rapid rate by the amount programmed as a second plane. At the end of the cycle it is again retracted to such second plane at rapid rate.

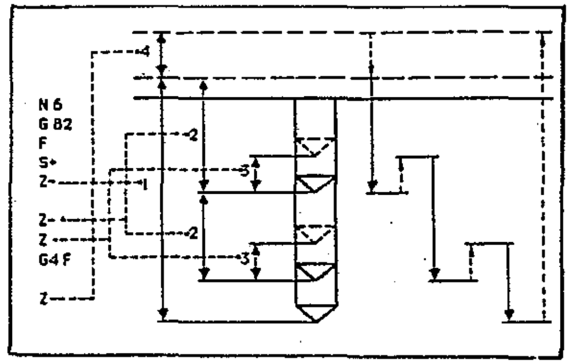


In practice this means that, if a 2nd plane is programmed in a cycle, the tool (in the preceding block) has to be positioned in such a way that its distance from the workpiece is exactly equal to the sum of safety allowance + 2nd plane, otherwise the tool will either jam into the workpiece at rapid rate (if the distance is too small) or fail to move down to the desired drilling depth (if the distance is too large).

Graphics support for input of cycles

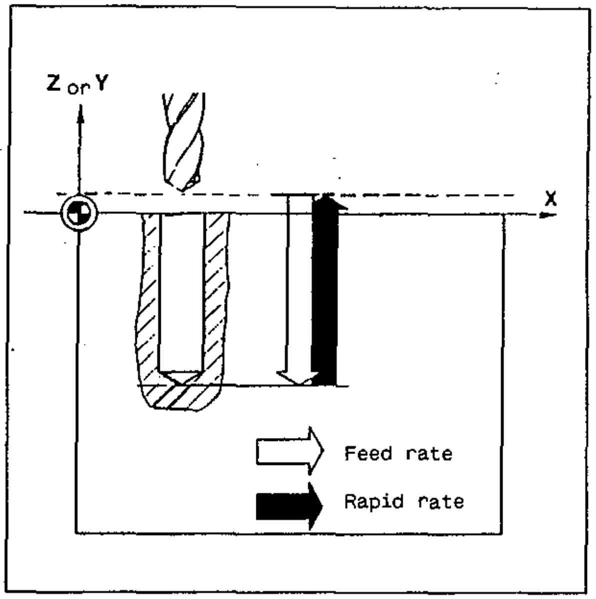
If you press the INFO key on NC keyboard (23) upon calling a cycle, a graphic display of the cutting operation appears on the display screen (see page 7-26).

The screen graphics assists you in the input dialog:
The control requests the cycle parameters in the sequence identified by reference numerals in the display: the first Z value requested is the distance identified by numeral 1, etc. In addition, the respective cycle parameter is shown highlighted.



Drilling - G81

Tool drills to the programmed drilling depth at the programmed spindle speed and infeed rate, then returns to its start position at rapid rate.



INPUT DIALOG

G81 Drilling cycle

The control requests:

F   Feed rate
S+ Spindle speed (sign can be changed)
Z   Sum of drilling depth plus safety allowance (sign can be changed)
G4 F Dwell (transfer key if not req'd)
Z   2nd plane (transfer key if not required)

When programming a 2nd plane do not forget to position the tool before starting the cycle.
Enter tool length compensation.

EXAMPLE

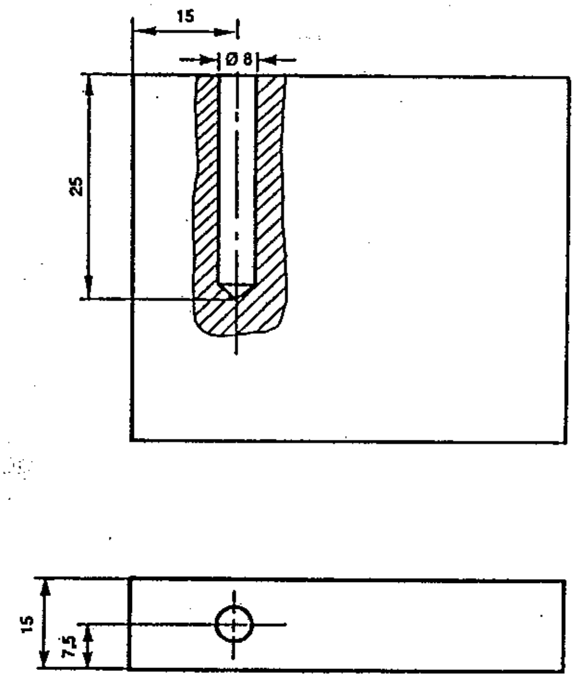
for vertical spindle:

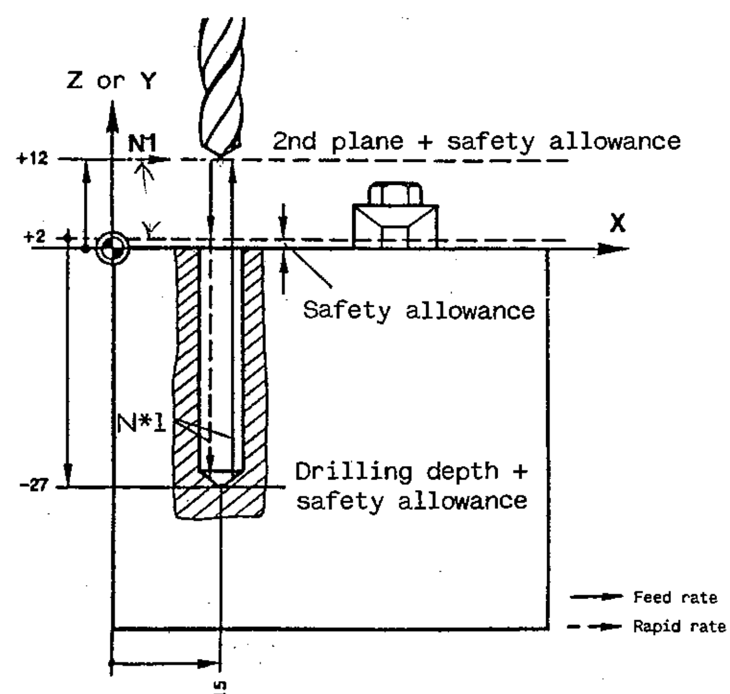
%135	Program number
N1 N*1 G0 X15 Y0 Z12	Tool moves to start position at rapid rate. At that point subroutine N*1 is called and cycle G81 performed.
N*1 G81 F500 S+1000 Z-27 G4 F1 Z10	Subroutine block defines the cycle.

for horizontal spindle:

%140	Program number
N1 N*1 G0 X15 Y12 Z0	Tool moves to start position at rapid rate. At that point subroutine N*1 is called and cycle G81 performed.
N*1 G81 F500 S+1000 Y-27 G4 F1 Y10	Subroutine block defines the cycle.

Drilling - G81

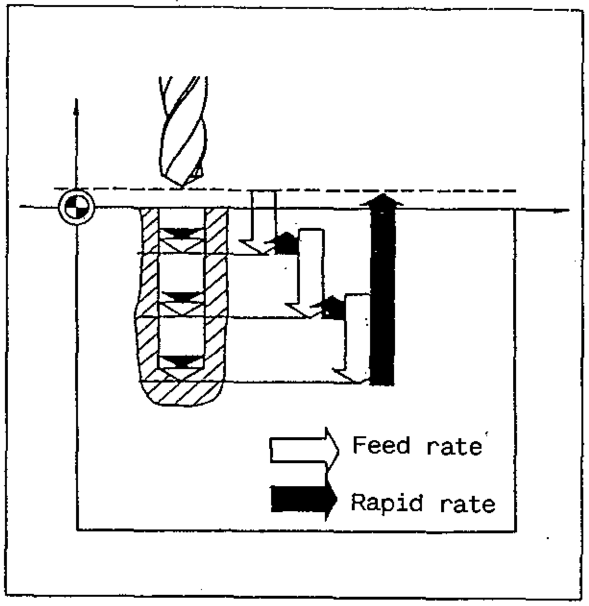




Dimensions in mm

Drilling with intermittent infeed - G82

Tool drills to the programmed drilling depth at the programmed spindle speed and feed rate.
The operation is performed in steps with the tool being advanced by an infeed increment and then retracted by a small amount (lift-off) for breaking the chip.
This procedure is repeated until the full depth has been reached.
At the end of the cycle the tool returns to its start position at rapid rate.



INPUT DIAOLG

G82 Drilling cycle with intermittent infeed

The control requests:

- F Feed rate
- S+ Spindle speed (sign can be changed)
- Z- Sum of drilling feed and safety allowance (sign can be changed)
- Z- Infeed increment
- Z Lift-off
- G4 F Dwell (transfer key if not required)
- Z 2nd plane (transfer key if not required)

When programming a 2nd plane do not forget to position the tool before starting the cycle.
Enter tool length compensation.

EXAMPLE

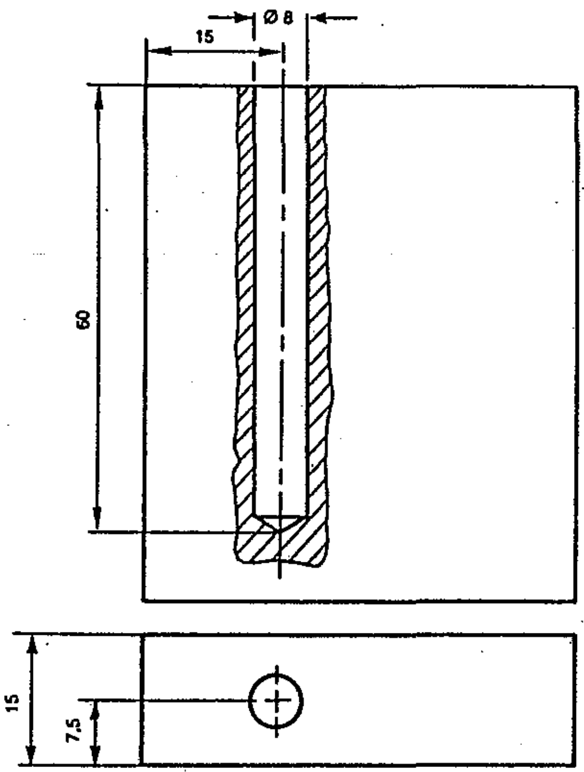
for vertical spindle:

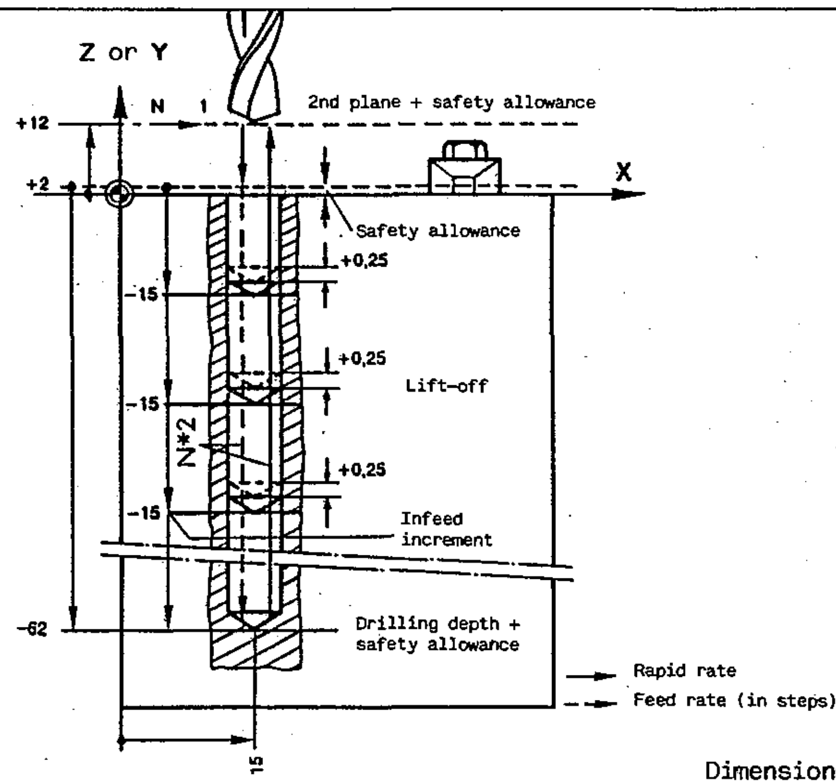
%145	Program number
N1 N*2 G0 X15 Y0 Z12	Tool moves to start position at rapid rate. At that point subroutine N*2 is called and cycle G82 performed.
N*2 G82 F500 S+1000 Z-62 Z-15 Z0.25 G4 F0.5 Z10	Subroutine block defines the cycle.

for horizontal spindle:

%150	Program number
N1 N*2 G0 X15 Y12 Z0	Tool moves to start position at rapid rate. At that point subroutine N*2 is called and cycle G82 performed.
N*2 G82 F500 S+1000 Y-62 Y-15 Y0.25 G4 F0.5 Y10	Subroutine block defines the cycle.

Drilling with intermittent infeed - G82

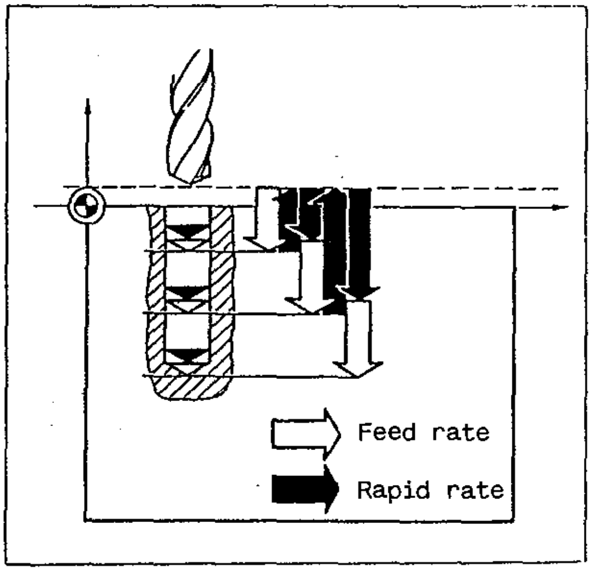




Dimensions in mm

Deep drilling - G83

Tool drills to the programmed drilling depth at the programmed spindle speed and feed rate.
The operation is performed in steps with the tool being advanced by an infeed increment and then retracted to the starting position at rapid rate. The drill is then advanced again at rapid rate, but keeps a programmed distance from the bottom of the hole. This is followed by another drilling step: infeed increment and return to start position.
If a reduction of the infeed increment is programmed, the infeed increment will be reduced by such amount after each cut.
The procedure is repeated until the programmed depth has been reached. At the end of the cycle the tool returns to its start position at rapid rate.



INPUT DIALOG

G83 Deep drilling cycle

The control requests:

F	Feed rate
S+	Spindle speed (sign can be changed)
Z-	Sum of drilling depth and safety allowance (sign can be changed)
Z-	Infeed increment
Z-	Distance from bottom of hole in each step
Z-	Reduction of infeed increment
G4 F	Dwell (transfer key if not requ'd)
Z	2nd plane (transfer key if not required)

Fit Total Depth

Inc

.020

When programming a 2nd plane do not forget to position the tool before starting the cycle.
Enter tool length compensation.

EXAMPLE

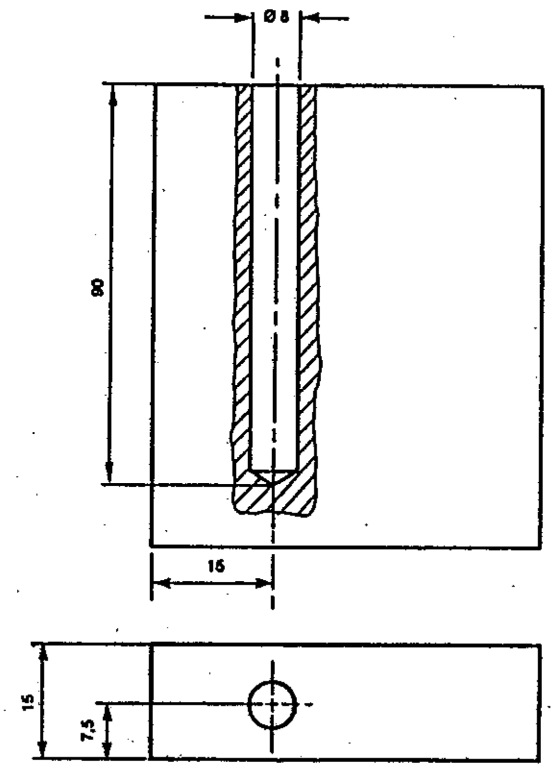
for vertical spindle:

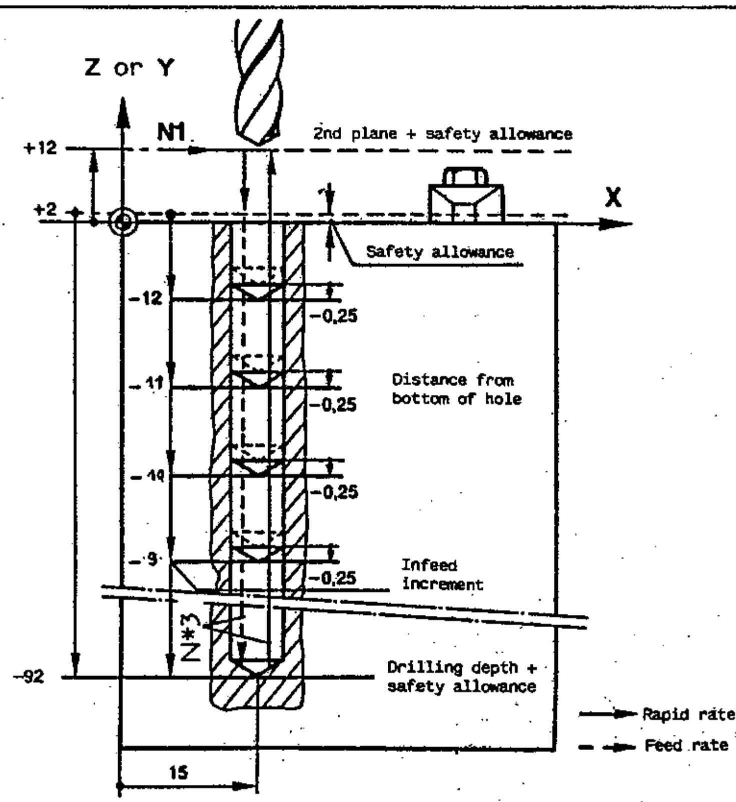
%155	Program number
N1 N*3 G0 X15 Y0 Z12	Tool moves to start position at rapid rate. At that point subroutine N*3 is called and cycle G83 performed.
N*3 G83 F500 S+1000 Z-92 Z-12 Z-0.25 Z-1 Z10	Subroutine block defines the cycle.

for horizontal spindle:

%160	Program number
N1 N*3 G0 X15 Y12 Z0	Tool moves to start position at rapid rate. At that point subroutine N*3 is called and cycle G83 performed.
N*3 G83 F500 S+1000 Y-92 Y-12 Y-0.25 Y-1 Y10	Subroutine block defines the cycle.

Deep drilling - G83

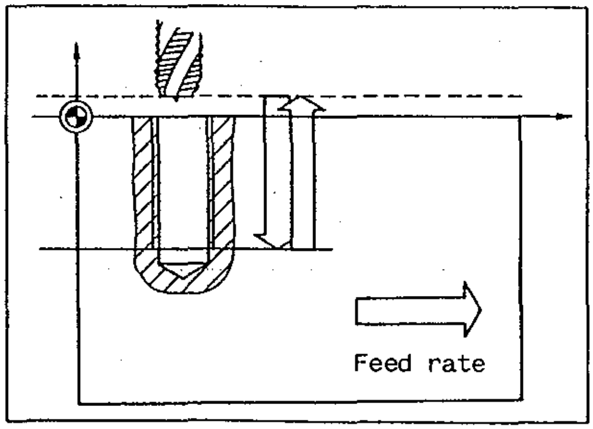




Dimensions in mm

Tapping - G84

Tool taps to the programmed depth of the thread at the programmed spindle speed and feed rate.
Then the spindle rotation is reversed and the tool moves back to start position where the spindle is reversed again.
Feed rate override control (6) is inoperative during a tapping cycle.



INPUT DIALOG

G84 Tapping cycle

The control requests:

F	Feed rate (spindle speed x lead of thread
S+	Spindle speed (sign can be changed)
Z-	Sum of tapping depth and safety allowance (sign can be changed)
Z	2nd plane (transfer key if not required)

When programming a 2nd plane do not forget to position the tool before starting the cycle.
Enter tool length compensation.
A tapping cycle cannot be continued in the event of an interruption, but the spindle rotation is automatically reversed and the tool removed from the thread.
Tapping has to be completed by hand in this case.

EXAMPLE

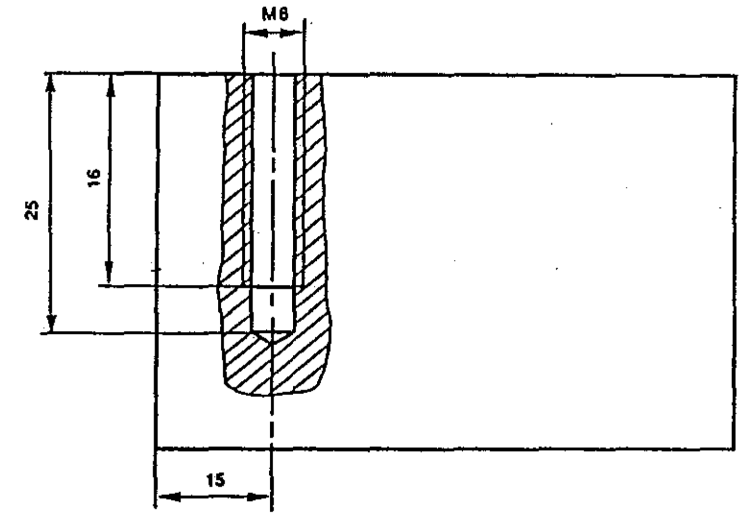
for vertical spindle:

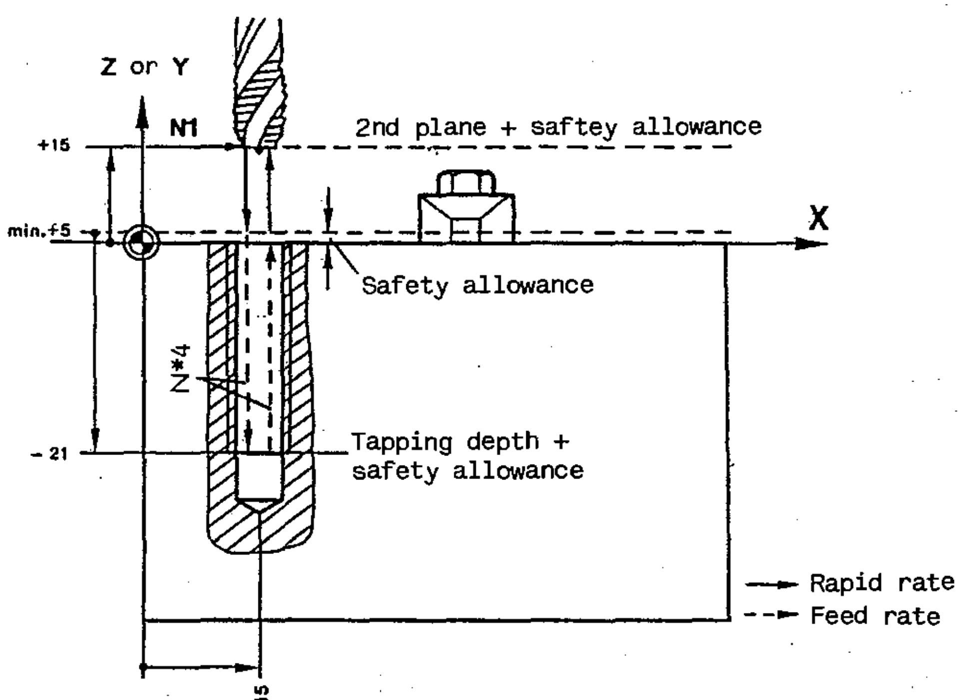
%165	Program number
N1 N*4 G0 X15 Y0 Z15	Tool moves to start position at rapid rate. At that point subroutine N*4 is called and cycle G84 performed.
N*4 G84 F100 S+100 Z-21 Z10	Subroutine block defines the cycle.

for horizontal spindle:

%170	Program number
N1 N*4 G0 X15 Y15 Z0	Tool moves to start position at rapid rate. At that point subroutine N*4 is called and cycle G84 performed.
N*4 G84 F250 S+250 Y-21 Y10	Subroutine block defines the cycle.

Tapping – G84

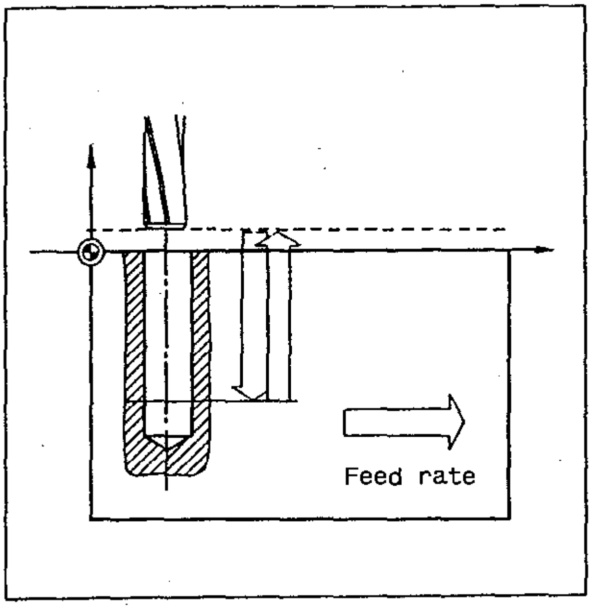




Dimensions in mm

Reaming - G85

The tool reams to the programmed depth at the programmed spindle speed and feed rate, then returns to its start position in the same manner.



INPUT DIALOG

G85 Reaming cycle

The control requests:

F	Feed rate
S+	Spindle speed (sign can be changed)
Z-	Sum of reaming depth and safety allowance (sign can be changed)
G4 F	Dwell (transfer key if not requ'd)
Z	2nd plane (tranfer key if not required)

When programming a 2nd plane do not forget to position the tool before starting the cycle.
Enter tool length compensation.

EXAMPLE

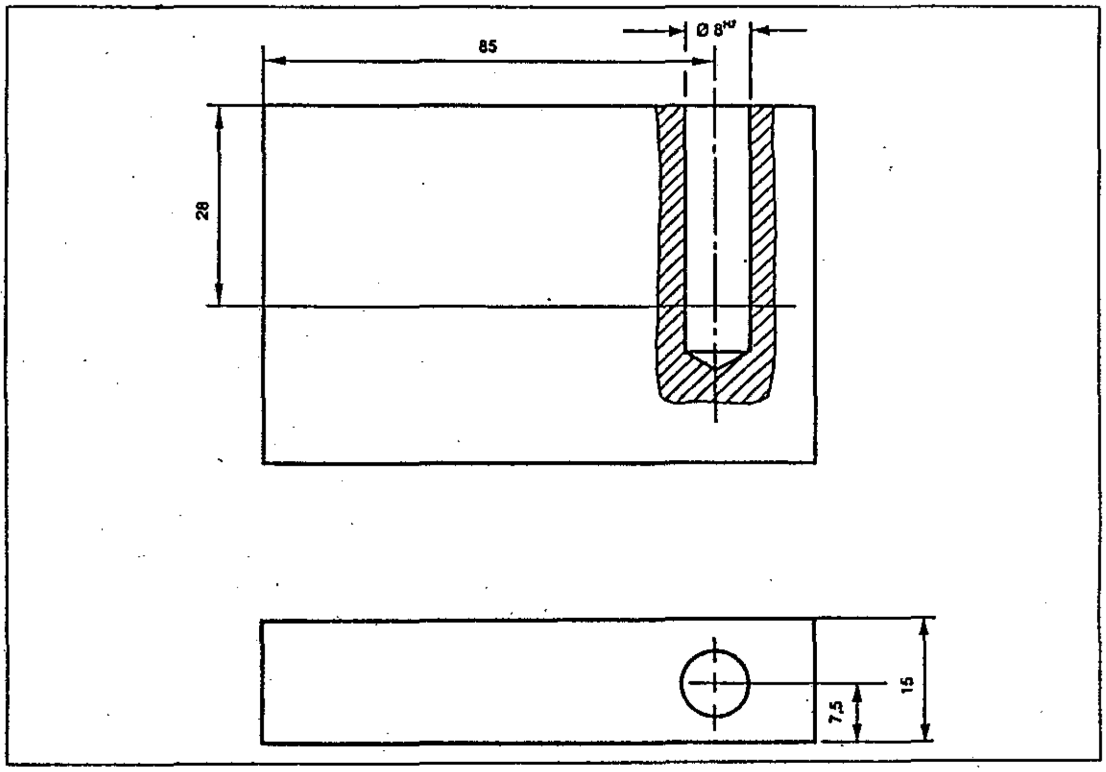
for vertical spindle:

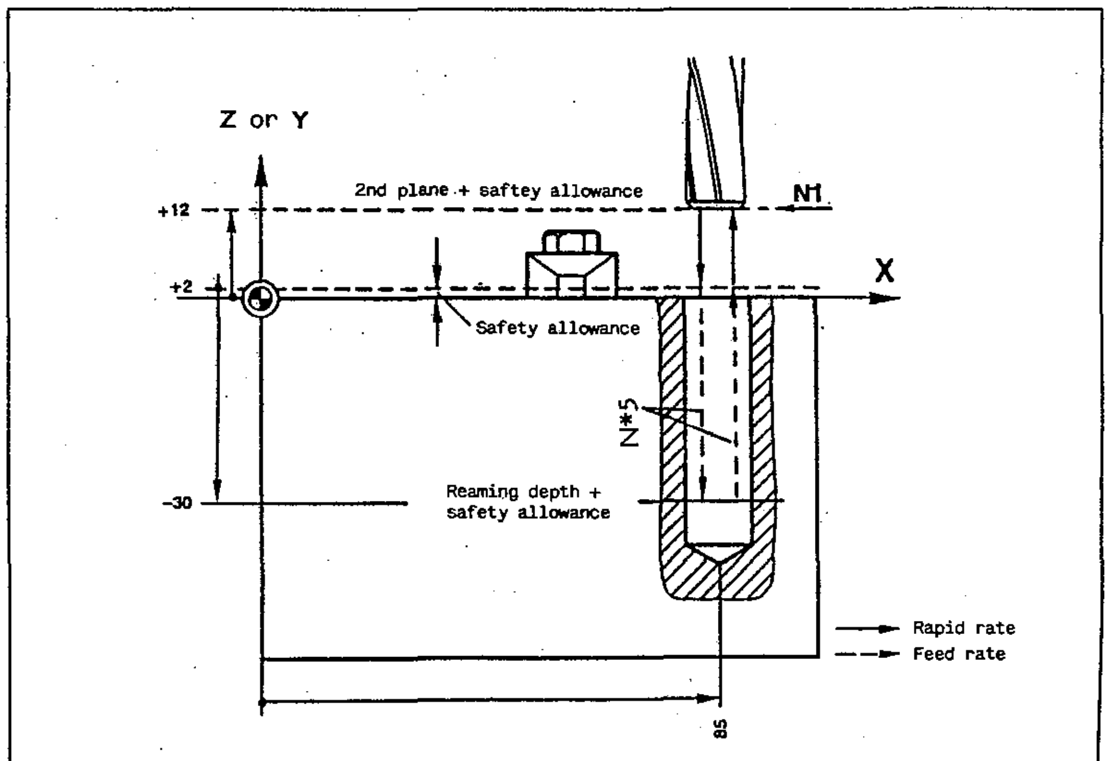
%175	Program number
N1 N*5 G0 X85 Y0 Z12	Tool moves to start position at rapid rate. At that point subroutine N*5 is called and cycle G85 performed.
N*5 G85 F500 S+1000 Z-30 G4 F5 Z10	Subroutine block defines the cycle.

for horizontal spindle:

%180	Program number
N1 N*5 G0 X85 Y12 Z0	Tool moves to start position at rapid rate. At that point subroutine N*5 is called and cycle G85 performed.
N*5 G85 F500 S+1000 Y-30 G4 F5 Y10	Subroutine block defines the cycle.

Reaming – G85

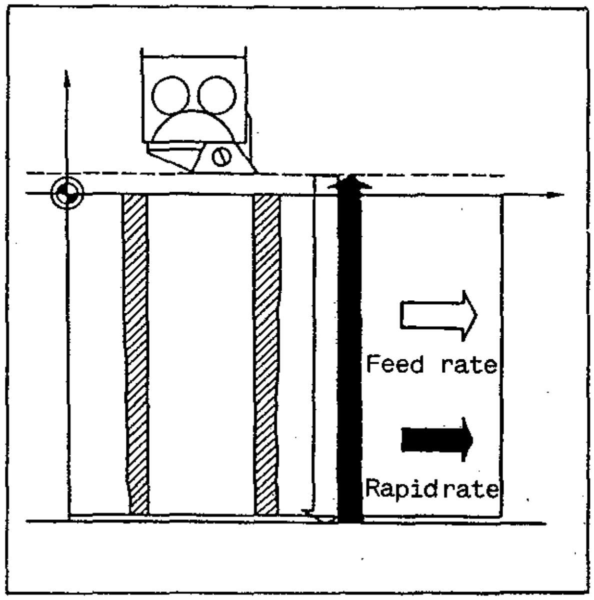




Dimensions in mm

Boring, tool retracted with spindle stopped – G86

Tool bores to the programmed boring depth at the programmed spindle speed and feed rate, then returns to its start position at rapid rate with the spindle standing still.



INPUT DIAOLG

G86 Boring cycle, tool retracted with spindle stopped

The control requests:

F Feed rate
S+ Spindle speed (sign can be changed)
Z– Sum of boring depth and safety allowance (sign can be changed)
G4 F Dwell (transfer key if not requ'd)
Z 2nd plane (tranfer key if not required)

When programming a 2nd plane do not forget to position the tool before starting the cycle.
Enter tool length compensation.

EXAMPLE

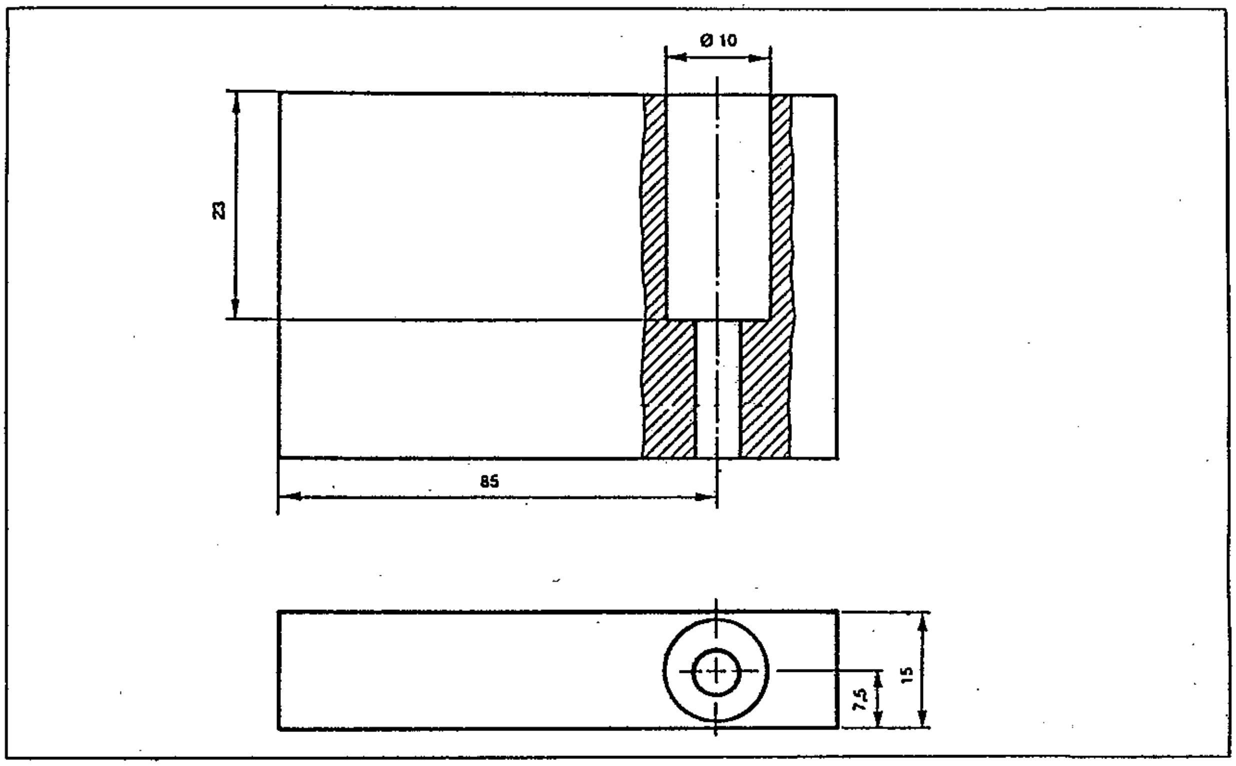
for vertical spindle:

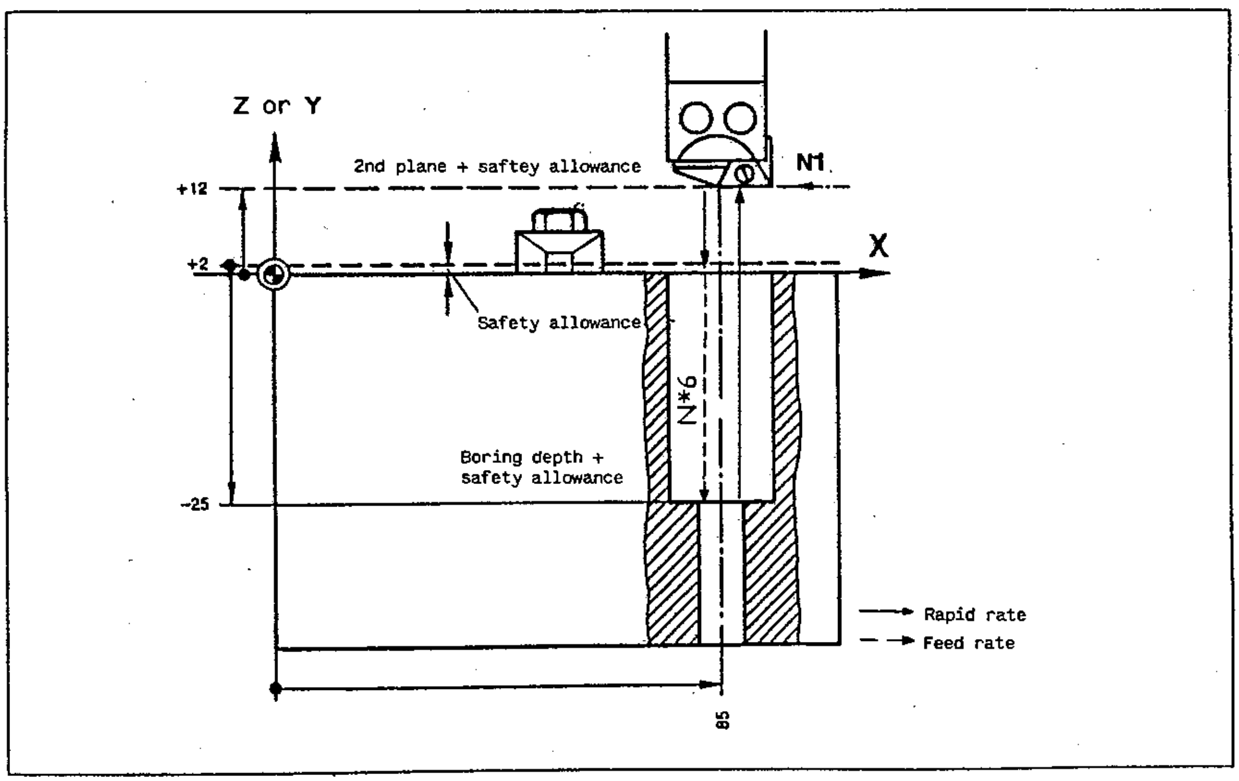
%185	Program number
N1 N*6 G0 X85 Y0 Z12	Tool moves to start position at rapid rate. At that point subroutine N*6 is called and cycle G86 performed.
N*6 G86 F500 S+1000 Z-25 G4 F5 Z10	Subroutine block defines the cycle.

for horizontal spindle:

%190	Program number
N1 N*6 G0 X85 Y12 Z0	Tool moves to start position at rapid rate. At that point subroutine N*6 is called and cycle G86 performed.
N*6 G86 F500 S+1000 Y-25 G4 F5 Y10	Subroutine block defines the cycle.

Boring, tool retracted with spindle stopped - G86



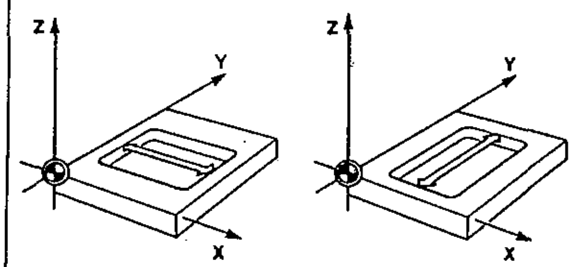


Dimensions in mm

Milling of rectangular pockets

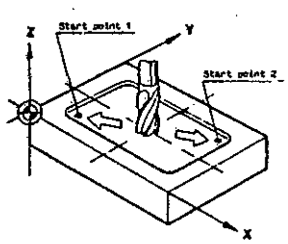
For rectangular pockets, the coordinate system should be placed in such a way that two coordinate axes are parallel to the side walls of the pocket.
The coordinate for the longer side should always be entered first in reply to the input dialog.

In cycles G71, G72, G73 and G74, the signs of the data entered determine the start point of the cycle. In cycles G72*1 and G74*1, the start point is always in the centre of the pocket.
When the first coordinate has been entered, the control requests all other data with the correct sign, even though a sign for the length or width or depth of a pocket may have been changed.

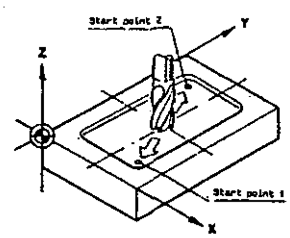


First enter X   First enter Y

X/Y plane (use of vertical spindle)

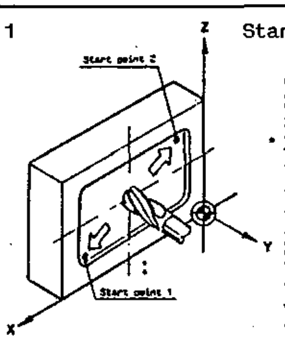


Start point 1	Start point 2
G71	G71
F	F
S+	S+
•X	•X-
D	D
X	X-
Y	Y-
Y	Y-
F	F
Z-	Z-
Z-	Z-
Z-	Z-
Z	Z

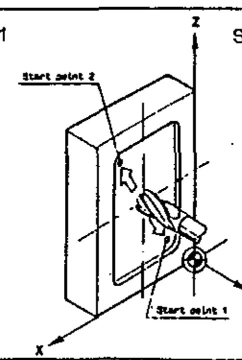


Start point 1	Start point 2
G71	G71
F	F
S+	S+
•Y	•Y-
D	D
Y	Y-
X-	X
X-	X
F	F
Z-	Z-
Z-	Z-
Z-	Z-
Z	Z

X/Z plane (use of horizontal spindle)



Start point 1	Start point 2
G71	G71
F	F
S+	S+
•X-	•X
D	D
X-	X
Z	Z-
Z	Z-
F	F
Y-	Y-
Y-	Y-
Y-	Y-
Y	Y



Start point 1	Start point 2
G71	G71
F	F
S+	S+
•Z	•Z-
D	D
Z	Z-
X	X-
X	X-
F	F
Y-	Y-
Y-	Y-
Y-	Y-
Y	Y

Tool positioning in milling cycles

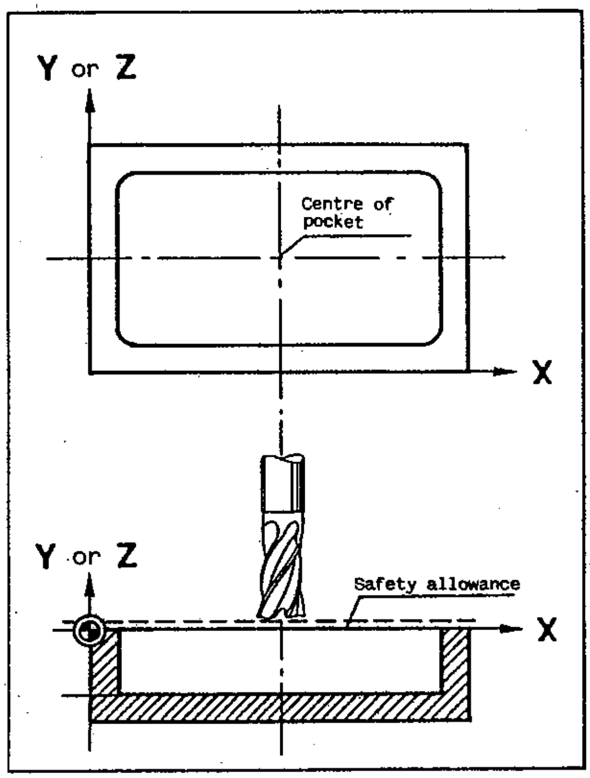
Before a cycle is called, the tool should be positioned as follows:

When the vertical spindle is used,

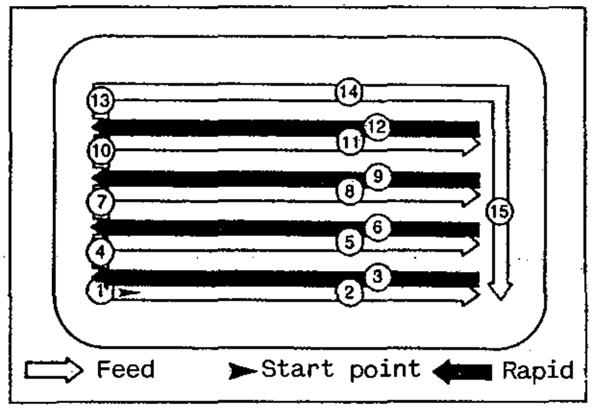
- enter X and Y coordinates of centre of pocket (point of intersection of symmetry axes) and,
- in the Z axis, enter coordinate of the highest point on the workpiece surface plus a safety allowance.

When the horizontal spindle is used,

- enter X and Z coordinates of centre of pocket (point of intersection of symmetry axes and,
- in the Y axis, enter coordinate of that point on the workpiece which projects farthest from the surface, plus a safety allowance.



Rectangular pockets - G71
Roughing, conventional



1 Tool moves to start point.
Tool advanced by infeed increment in Z* at start point.

2–3 Longer side of pocket machined up to finishing allowance, then lift-off and return to start point at rapid rate.

4 Tool advanced by infeed increment in Z* and then by feed increment in Y* (max 0.8 x cutter dia).

5–14 Repetition of steps 2–4 up to finishing allowance in X and Y*.

15 Removal of cutter marks, then return to start position in Z*.

1–15 Steps 1–15 repeated until finishing allowance in Z* has been reached. Finally, tool returns to start position at rapid rate.

INPUT DIALOG

G71	Rectangular pockets, roughing, conventional The control requests:	Y	Feed increment in Y (for square milling: press transfer key)
F	Feed in X and Y	F	Infeed rate in Z
S+	Spindle speed (sign can be changed)	Z-	Final dimension in Z + safety allowance (sign can be changed)
X	Final dimension in X (sign and axis can be changed)	Z-	Infeed increment in Z (number of passes)
D	Tool compensation number (if not required: transfer key)	Z-	Finishing allowance in Z
X	Finishing allowance in X and Y	Z	2nd plane (if not required: transfer key)
Y	Final dimension in Y (axis can be changed)		

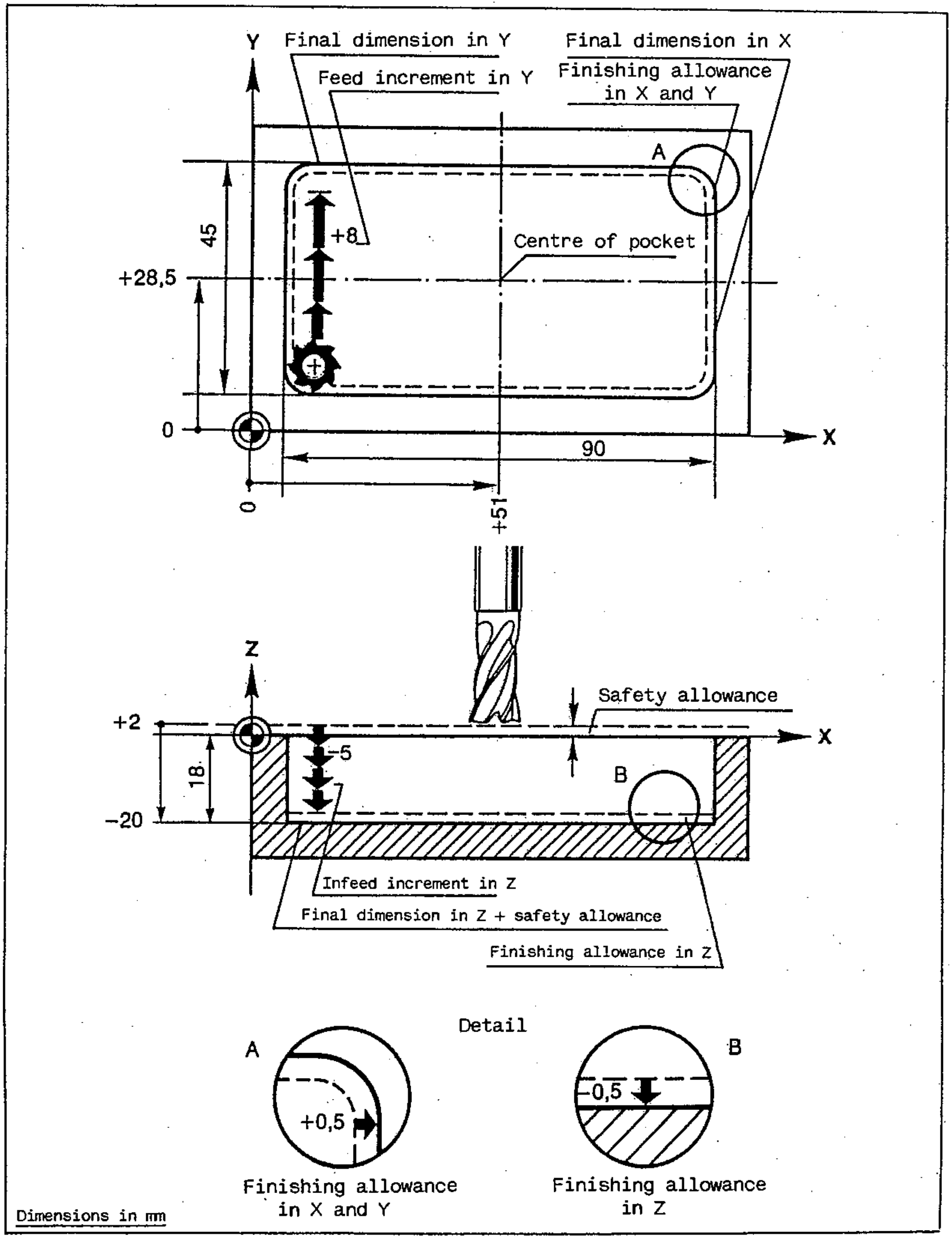
Never forget 2nd plane and positioning of tool before starting the cycle.

EXAMPLE
for use of vertical spindle*

%195	Program number
N0 G17 T1	Assign length compensation to Z axis and call tool compensation.
N1 N*1 G0 X51 Y28.5 Z2	Tool moves to centre of pocket at rapid rate, then N*1 is called and cycle G71 performed.
N*1 G71 F500 S+1000 X90 X0.5 Y45 Y8 F100 Z-20 Z-5 Z-0.5	Subroutine block defines the cycle.

* Use of horizontal spindle: interchange Y and Z axes.

Rectangular pockets - G71
Roughing, conventional



Rectangular pockets – G72
Roughing, climb and conventional

1 Tool moves to start point.
Tool advanced by infeed increment in Z* at start point.

2 Longer side of pocket machined up to finishing allowance

3 Tool advanced by infeed increment in Y* (max 0.8 x cutter dia).

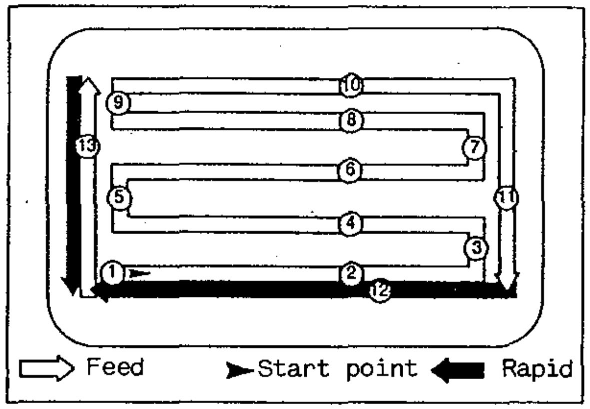
4–10 Repetition of steps 2-3 up to finishing allowance in X and Y*.

11 Removal of cutter marks along one wall.

12 Lift-off, return to start point at rapid rate.

13 Infeed motion and removal of cutter marks on other wall. Return to start point in Z* at rapid rate.

1–13 Steps 1–13 repeated until finishing allowance in Z* has been reached. Finally, tool returns to start position at rapid rate.



INPUT DIAOLG

G72	Rectangular pockets, roughing, climb and conventional The control requests:	Y	Feed increment in Y (for square milling: transfer key)
F	Feed in X and Y	F	Infeed rate in Z
S+	Spindle speed (sign can be changed)	Z–	Final dimension in Z + safety allowance (sign can be changed)
X	Final dimension in X (sign and axis can be changed)	Z–	Infeed increment in Z (number of passes)
D	Tool compensation number (if not required: transfer key)	Z–	Finishing allowance in Z
X	Finishing allowance in X and Y	Z	2nd plane (if not required: transfer key)
Y	Final dimension in Y (axis can be changed)		

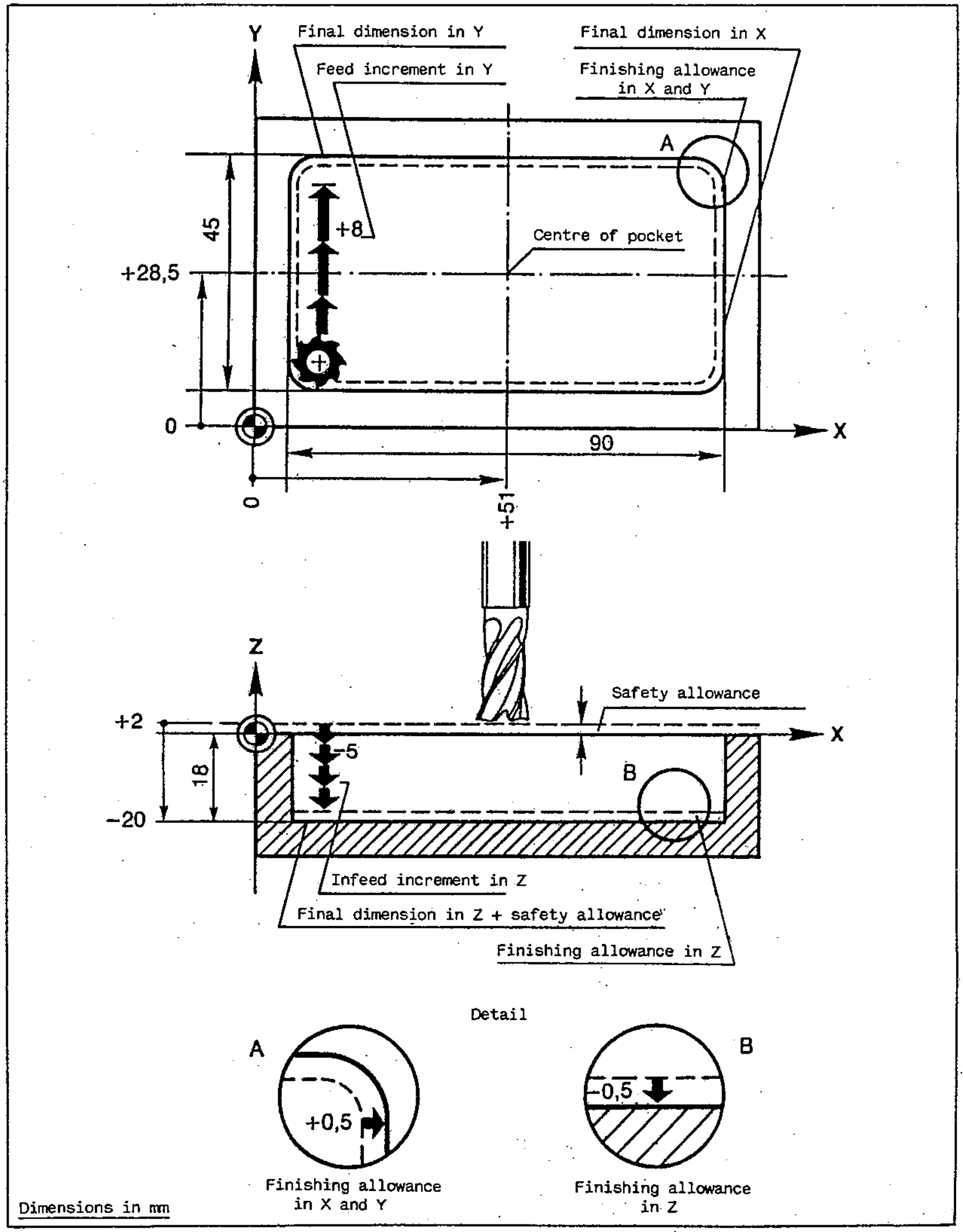
Never forget 2nd plane and positioning of tool before starting the cycle.

EXAMPLE
for use of vertical spindle*

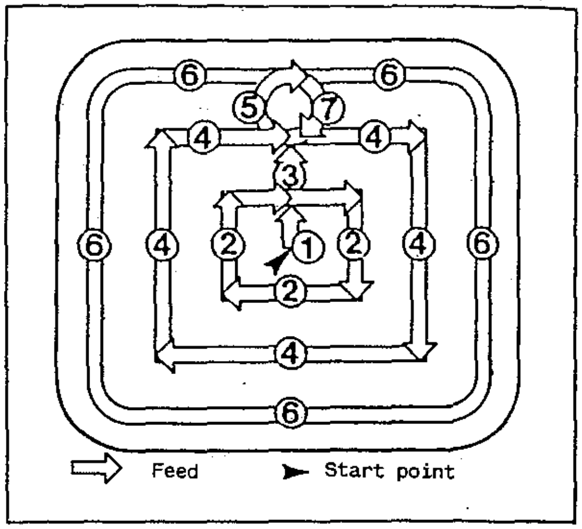
%200	Program number
N0 G17 T1	Assign length compensation to Z axis and call tool compensation.
N1 N*2 G0 X51 Y28.5 Z2	Tool moves to centre of pocket at rapid rate, then N*2 is called and cycle G72 performed.
N*2 G72 F500 S+1000 X90 X0.5 Y45 Y8 F100 Z-20 Z-5 Z-0.5	Subroutine block defines the cycle.

* Use of horizontal spindle: interchange Y and Z axes.

Rectangular pockets – G72
Roughing, climb and conventional



Rectangular pockets – G72 * 1
Roughing, climb and conventional
Corner radius programmable



1 Tool moves to start point.
Rapid downfeed to safety allowance.

2 Tool advanced by infeed increment in Z at start point (centre of pocket) at infeed rate.

3–4 Pocket cleared out. Transverse feed parallel to contour and perpendicular to longitudinal axis of pocket. Infeed by infeed increment in Y (max 0.8 x cutter diameter). Pocket is cleared out in G64 up to last cut but one.

5 In last cut, tool approaches contour in semi-circle (G46) with radius of 0.5 x infeed increment, or 2.5 mm if increment is smaller than 0.5 mm.

6 Last cut performed in mode G60.

7 Departure in semi-circle (G46) with radius of 2.5 mm. (Exception: confined space; in this case radius = 0.25 x smaller pocket wall.)
Lift-off and rapid return to start point (centre of pocket).

1–7 Steps 1–7 repeated until pocket depth (final dimension in Z + safety allowance) minus finishing allowance has been reached.

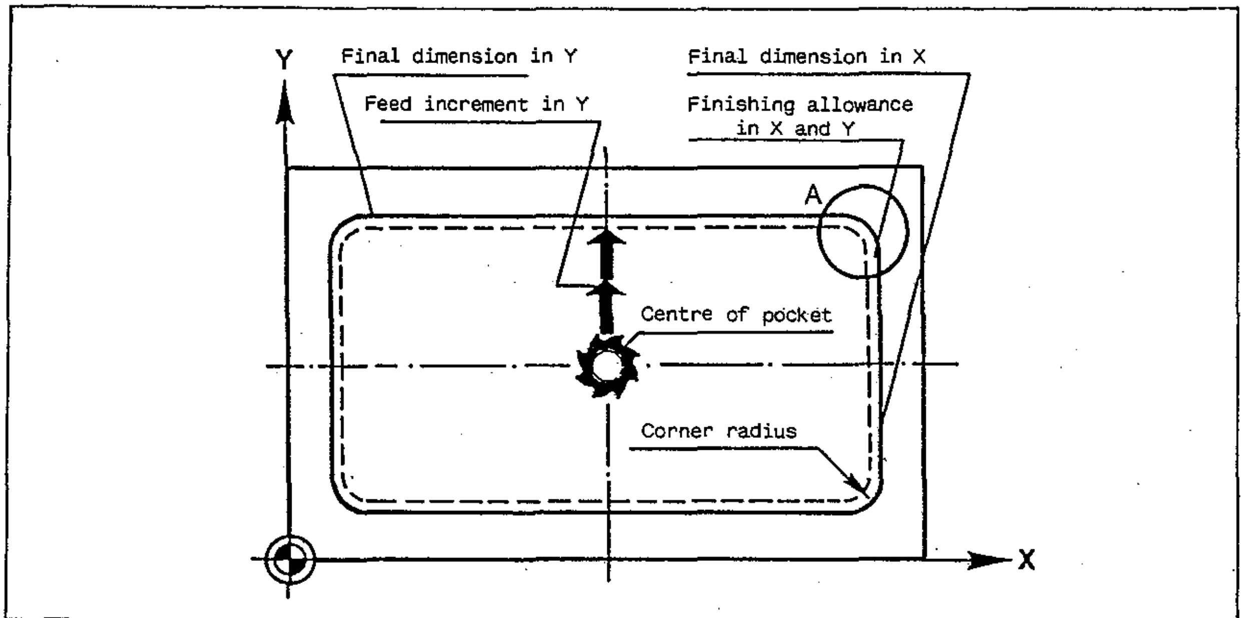
INPUT DIALOG

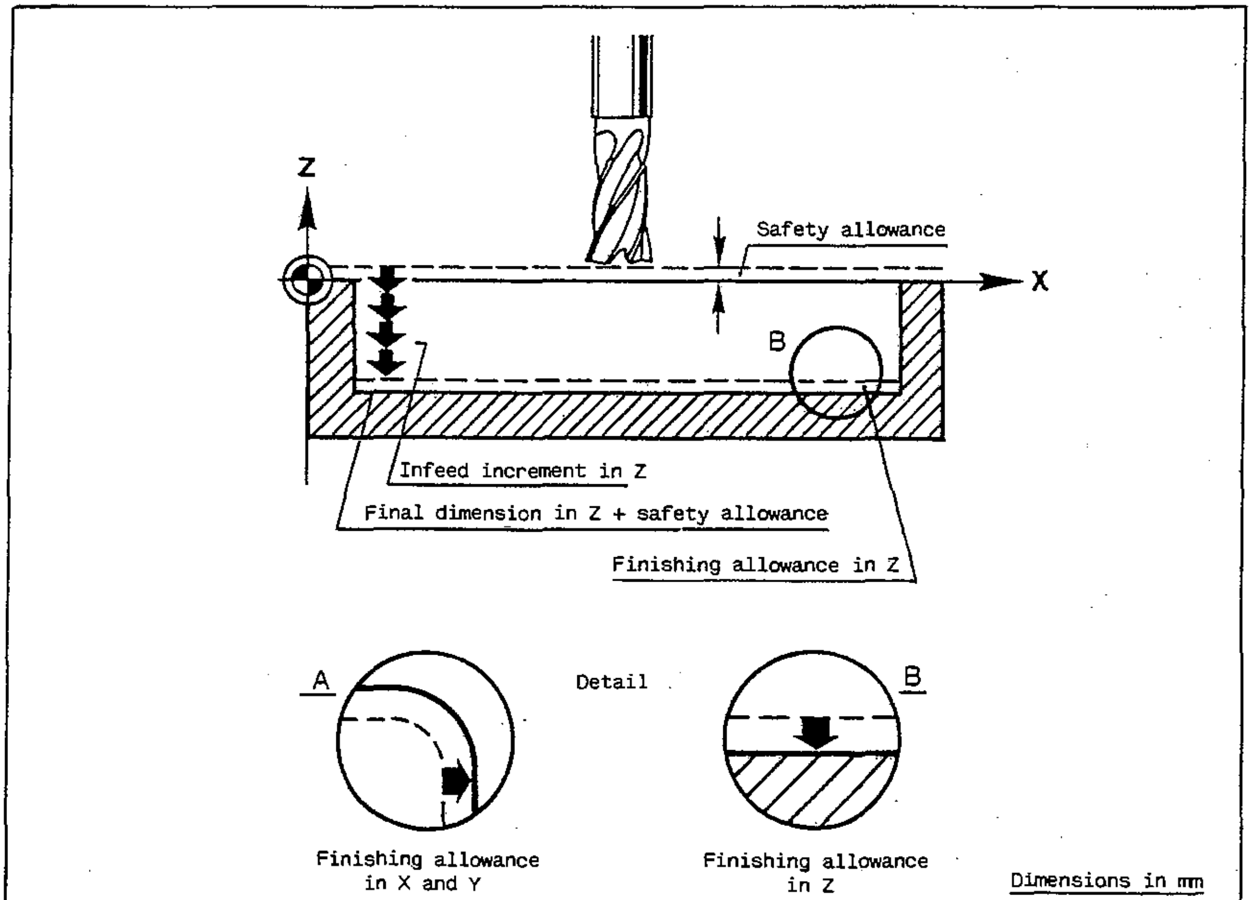
G72*1 Rectangular pockets,
roughing, climb and conventional,
corner radius programmable
The control requests:

F	Feed in X and Y
S+	Spindle speed (sign can be changed)
G	Cutting direction (2 = G2, 3 = G3)
X	Pocket length, final dimension in X (axis can be changed)
X	Finishing allowance in X and Y (note that allowance value entered is applied to each side)
Y	Pocket width, final dimension in Y (axis can be changed)
Y	Feed increment for clearing out in Y (square milling: transfer key)
R	Corner radius
F	Infeed rate in Z
Z–	Final dimension in Z + safety allowance (sign can be changed)
Z–	Infeed increment in Z (number of passes)
Z–	Finishing allowance in Z
Z	2nd plane (if not required: transfer key)

Never forget 2nd plane and positioning of tool before starting the cycle.

Rectangular pockets – G72 * 1
Roughing, climb and conventional
Corner radius programmable

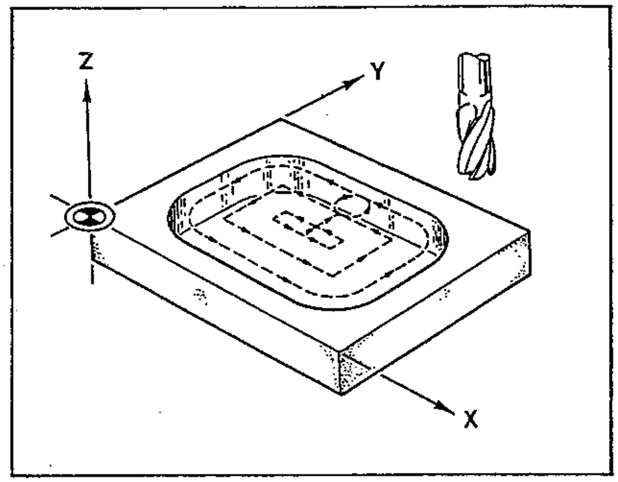




Dimensions in mm

Rectangular pockets – G72 * 1
Roughing, climb and conventional
Corner radius programmable

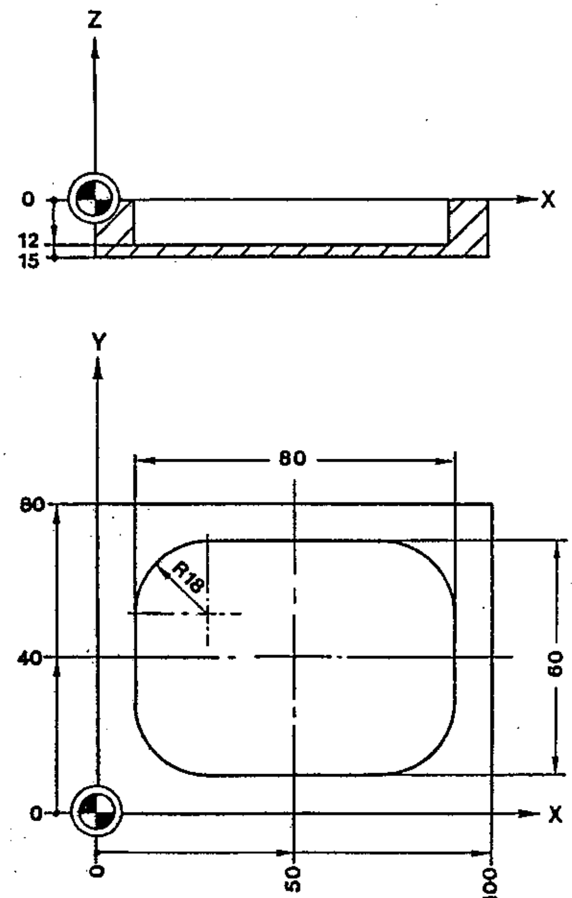
EXAMPLE
for use of vertical spindle

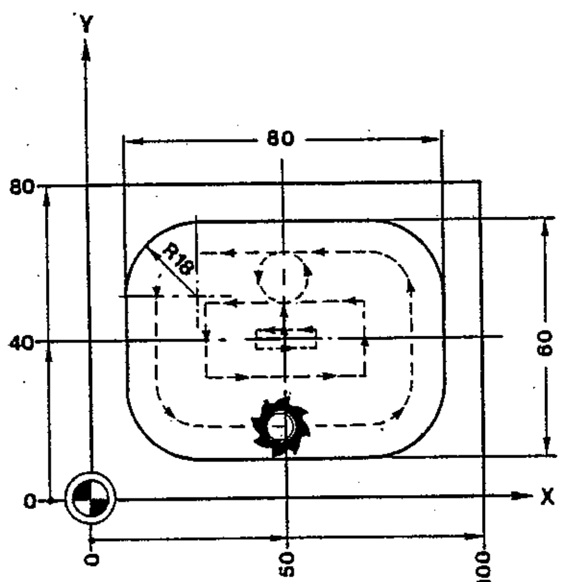


%215	Program number
N1 G0 Z100 N2 G17 T1	Tool change, assign length compensation to Z axis and call tool compensation.
N3 N*1 G0 X50 Y40 Z25	Tool moves to centre of pocket at rapid rate, then N*1 is called and cycle G72 * 1 performed.
N*1 G72*1 F250 S+0 G3 X80 X1 Y60 Y8 R18 F125 Z-14 Z-6 Z-0.1 Z23	Subroutine block defines the cycle.

* Use of horizontal spindle: interchange Y and Z axes.

Rectangular pockets - G72 * 1
Roughing, climb and conventional
Corner radius programmable



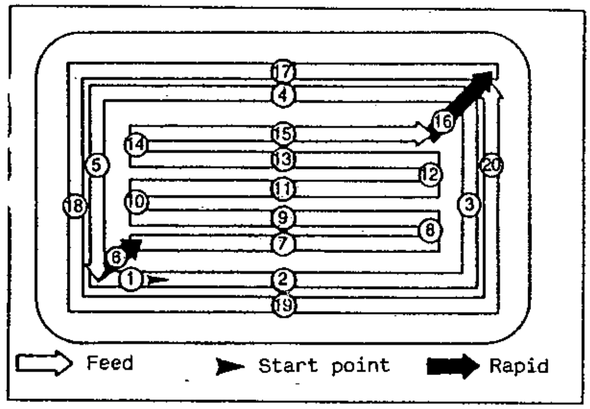


Dimensions in mm

Rectangular pockets - G73
Finishing to size

1 Tool moves to start position. Tool advanced by infeed increment in Z* at start point.
2-5 Pocket contour machined; each complete cycle followed by infeed increment until full depth has been reached.
6 Tool backed away from contour by fixed safety margin of 0.3 mm in X and Y*.
7-15 Pocket cleared out as in G 72.
16 Tool retracted to start position in Z* axis. Infeed in Z* at 1/2 feed rate to final dimension minus programmed distance from bottom of pocket.
17-20 Machining of pocket contour.

16-20 Steps 16-20 repeated until finishing allowance in X and Y* has been reached. Finally tool returns to start position at rapid rate.



INPUT DIALOG

G73 Rectangular pockets, finishing to size
The control requests:

F	Feed in X and Y
S+	Spindle speed (sign can be changed)
X	Final dimension in X (sign and axis can be changed)
D	Tool compensation number (if not required: transfer key)
X	Finishing allowance in X and Y
X	Feed increment in X and Y in finishing
Y	Final dimension in Y (axis can be changed)
Y	Feed increment in Y (for square milling: transfer key)
F	Infeed rate in Z
Z-	Final dimension in Z + safety allowance (sign can be changed)
Z-	Infeed increment in Z (number of passes)
Z-	Finishing allowance in Z
Z	2nd plane (if not required: transfer key)

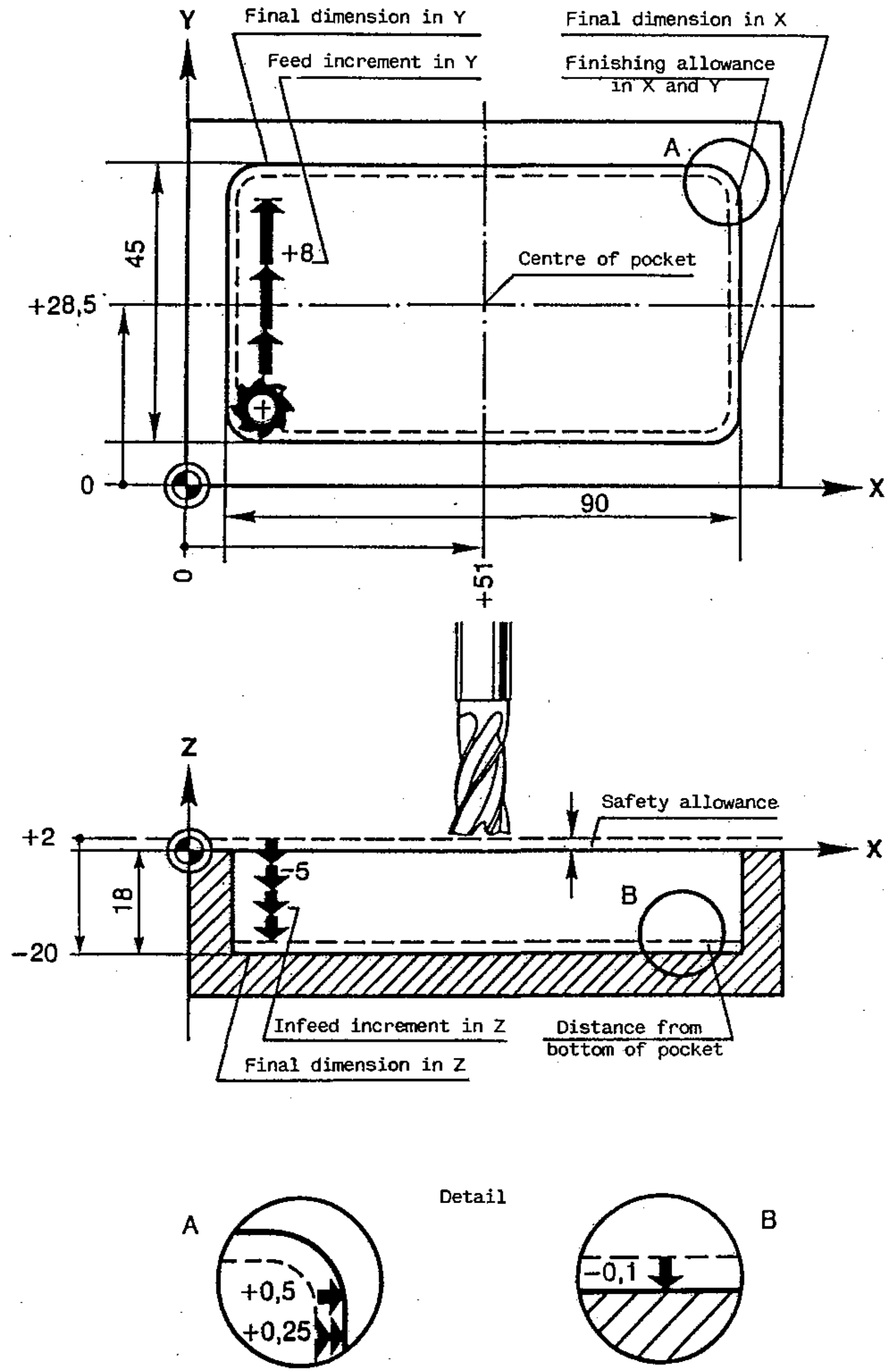
Never forget 2nd plane and positioning of tool before starting the cycle.

EXAMPLE
for use of vertical spindle*

%205	Program number
N0 G17 T1	Tool change, assign length compensation to Z axis and call tool compensation.
N1 N*3 G0 X51 Y28.5 Z2	Tool moves to centre of pocket at rapid rate, then N*3 is called and cycle G73 performed.
N*3 G73 F200 S+1600 X90 X0.5 X0.25 Y45 Y8 F100 Z-20 Z-5 Z-0.1	Subroutine block defines the cycle.

* Use of horizontal spindle: interchange Y and Z axes.

Rectangular pockets - G73
Finishing to size

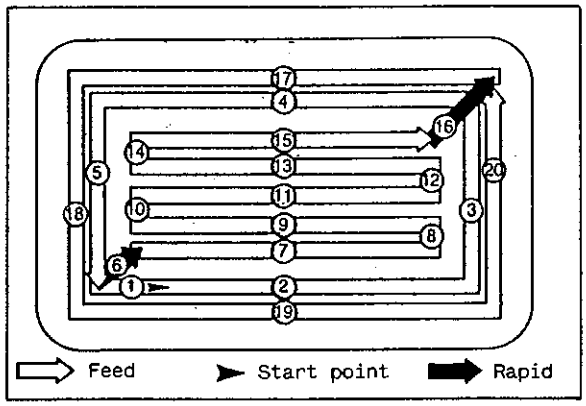


+0.5: finishing allowance in X
+0.25: feed increment in X and Y (number of passes)

Distance from bottom of pocket

Dimensions in mm

Rectangular pockets – G74
Finishing to size with intermediate stop



1–6 Same as G73

7–15 Clearing off pocket as in G72. Return to start position and automatic intermediate stop: tool can be moved by means of push buttons (7) without change of mode. Part can now be measured and new X and Y* feed increments entered. Restart cycle by push button (20). Tool automatically positioned in X and Y* using the new feed increments. Infeed in Z* to final dimensions minus programmed distance from bottom of pocket.

17–20 Machining of pocket contour.

16–20 Steps 16–20 repeated until final dimensions in X and Y* have been reached. Finally, tool returns to start position at rapid rate.

INPUT DIALOG

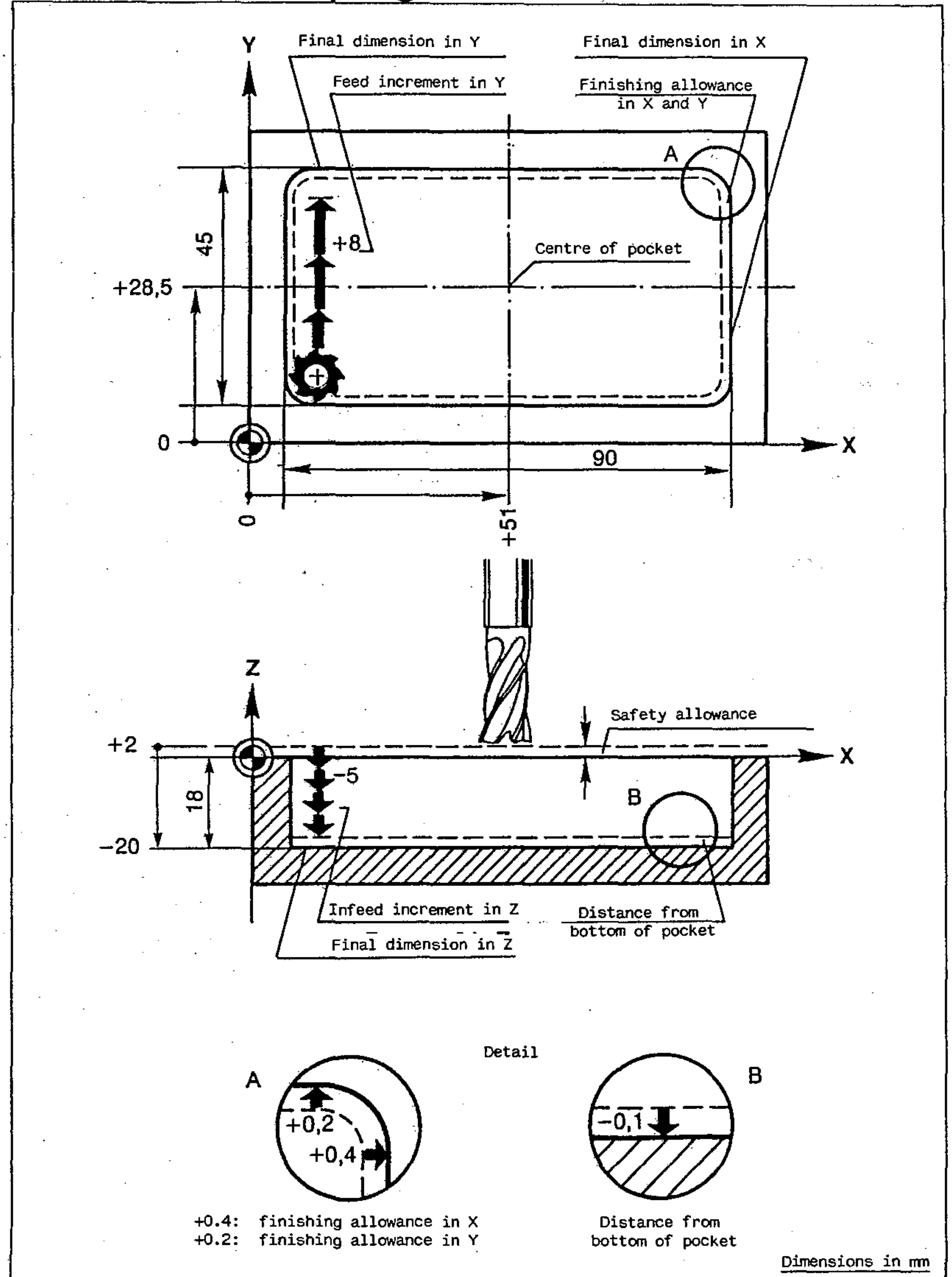
G74	Pocket milling, finishing to size with intermediate stop The control requests:	Y	Finishing allowance in Y when cycle is machined once up to 1st intermediate stop
F	Feed in X and Y	Y	Feed increment in Y (for square milling: transfer key)
S+	Spindle speed (sign can be changed)	F	Infeed rate in Z
X	Final dimension in X (sign and axis can be changed)	Z–	Final dimension in Z + safety allowance (sign can be changed)
D	Tool compensation number (if not required: transfer key)	Z–	Infeed increment in Z (number of passes)
X	Finishing allowance in X when cycle is machined once	Z–	Finishing allowance in Z
Y	Final dimension in Y (axis can be changed)	Z	2nd plane (if not required: transfer key)

EXAMPLE
for use of vertical spindle*

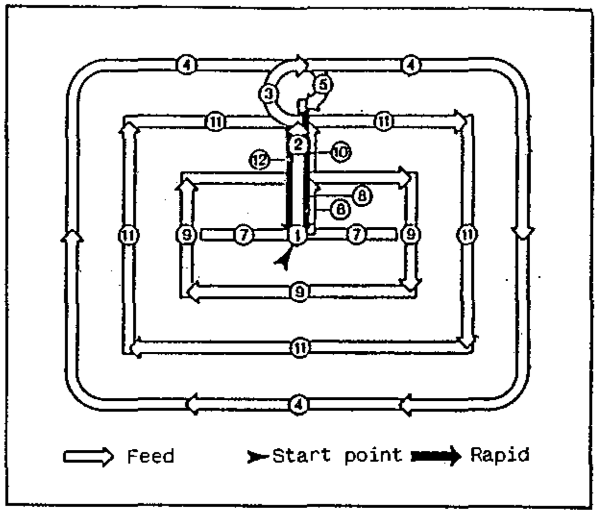
%210	Program number
N0 G17 T1	Tool change, assign length compensation to Z axis and call tool compensation.
N1 N*4 G0 X51 Y28.5 Z2	Tool moves to centre of pocket at rapid rate, then N*4 is called and cycle G74 performed.
N*4 G74 F200 S+1600 X90 X0.4 Y45 Y0.2 Y8 F100 Z-20 Z-5 Z-0.1	Subroutine block defines the cycle.

* Use of horizontal spindle: interchange Y and Z axes.

Rectangular pockets - G74 * 1
Finishing to size with intermediate stop
Corner radius programmable



Rectangular pockets - G74 * 1
Finishing to size with intermediate stop
Corner radius programmable



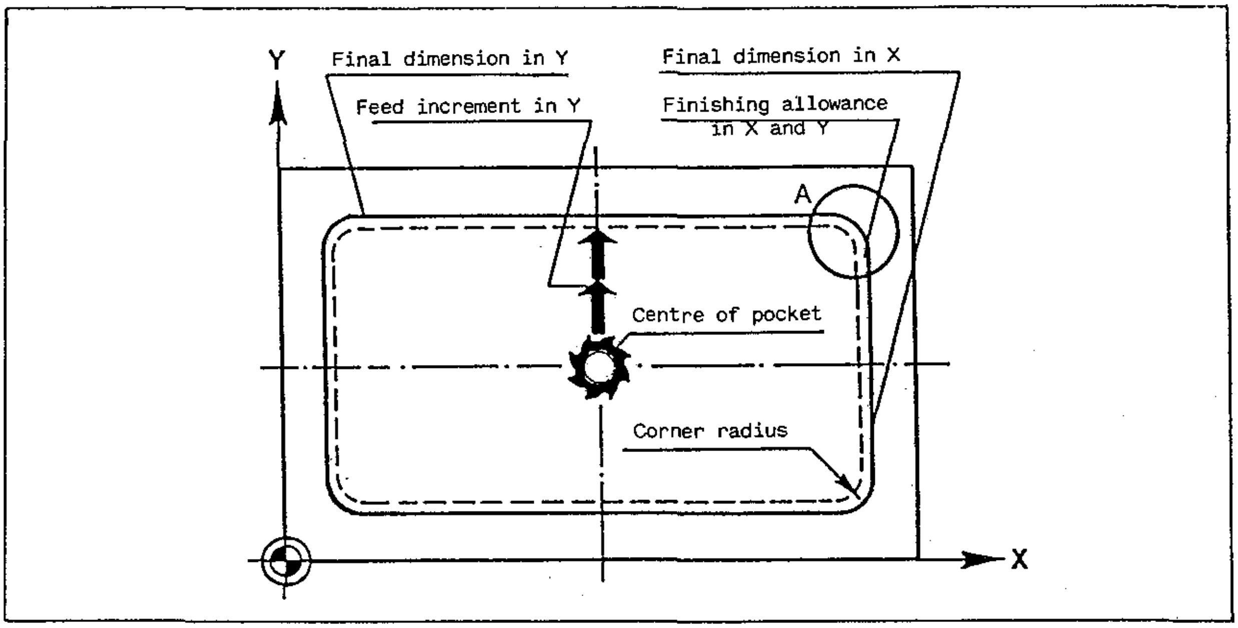
1 Tool moves to start point.
Tool advanced by infeed increment in Z at start point.
2 Transverse feed perpendicular to longitudinal axis of pocket.
3 Semi-circular approach of contour (radius 0.5 x feed increment).
4 Milling of contour (G60).
5 Semi-circular departure from contour (radius 2.5 mm).
6 Tool retracted to safety plane at rapid rate and positioned to centre of pocket.
2-6 Steps 2-6 repeated until pocket depth (final dimension in Z + safety allowance) has been reached.
7-11 Pocket cleared out.
12 Return to start position and automatic <u>intermediate stop</u>: tool can be moved by means of push buttons (7) without change in mode.
Part can now be measured and new X and Y* feed increments entered. Restart cycle by push button (20).
Tool automatically positioned in X and Y* using the new feed increments. Infeed in Z* to final dimensions minus programmed distance from bottom of pocket.
2-12 Steps 2-12 repeated until final dimensions in X and Y* have been reached. Finally, tool returns to start position at rapid rate.

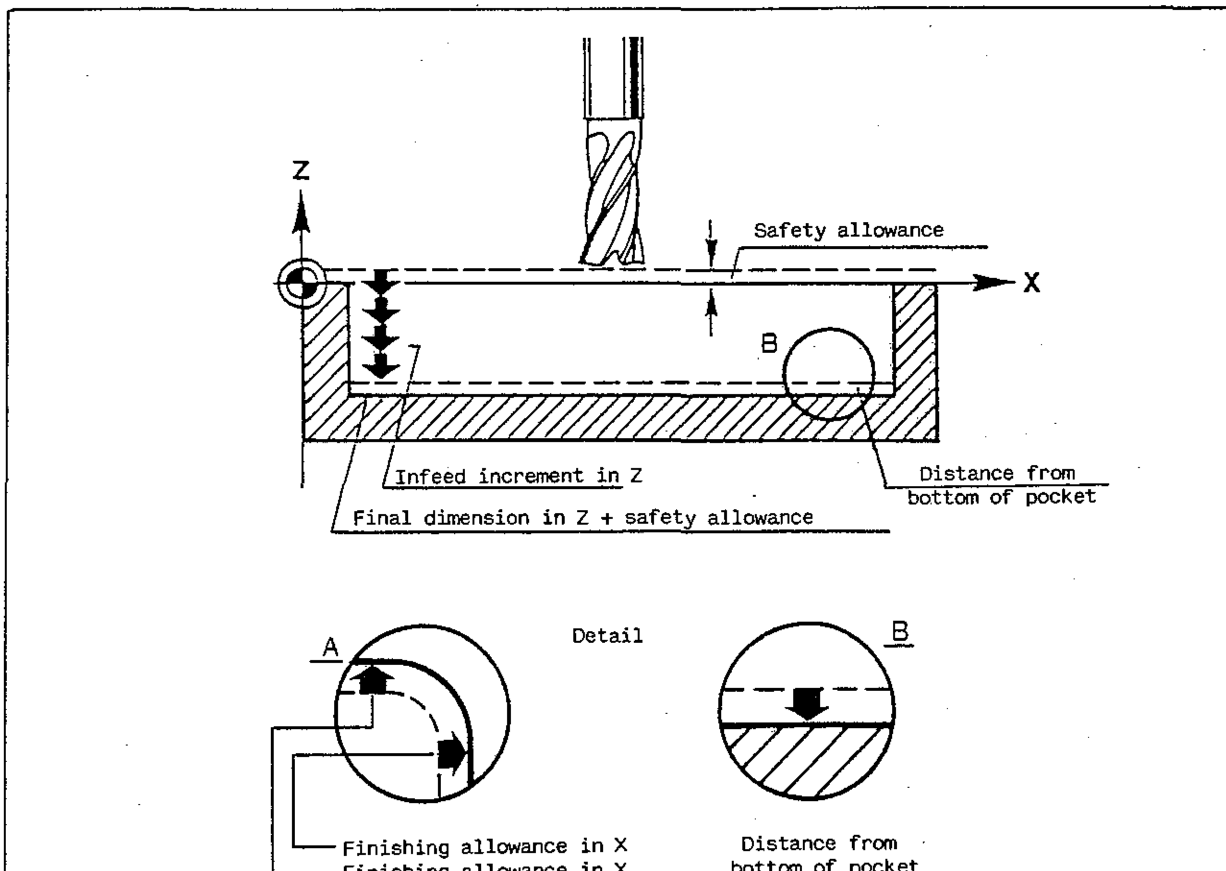
INPUT DIAOLG

G74*1	Rectangular pockets, finishing to size with intermediate stop, corner radius programmable
	The control requests:
F	Feed in X and Y
S+	Spindle speed (sign can be changed)
G	Cutting direction (2 = G2, 3 = G3)
X	Pocket length, final dimension in X (axis can be changed)
D	Tool compensation number (if not required: transfer key)
X	Finishing allowance in X when cycle is machined once up to 1st intermediate stop
Y	Pocket width, final dimension in Y (axis can be changed)
Y	Finishing allowance in Y when cycle is machined once up to 1st intermediate stop
Y	Feed increment for clearing out in Y (square milling: transfer key)
R	Corner radius
F	Infeed rate in Z
Z-	Pocket depth, final dimension in Z + safety allowance (sign can be changed)
Z-	Infeed increment in Z (number of passes)
Z-	Distance from bottom of pocket after immediate stop
Z	2nd plane (if not required: transfer key)

Never forget 2nd plane and positioning of tool before starting the cycle.

Rectangular pockets - G74 * 1
Finishing to size with intermediate stop
Corner radius programmable

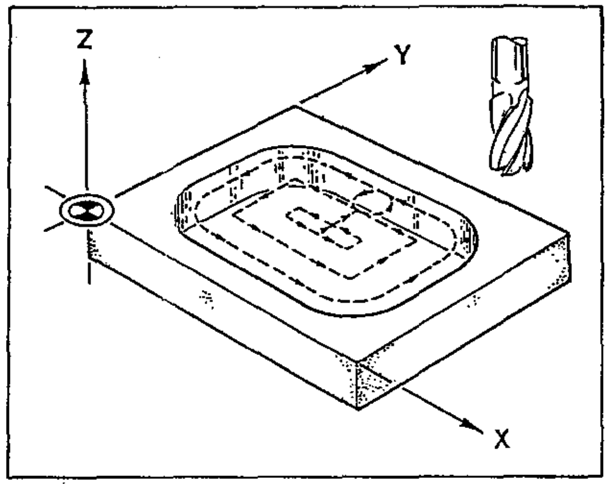




Dimensions in mm

Rectangular pockets - G74 * 1
Finishing to size with intermediate stop
Corner radius programmable

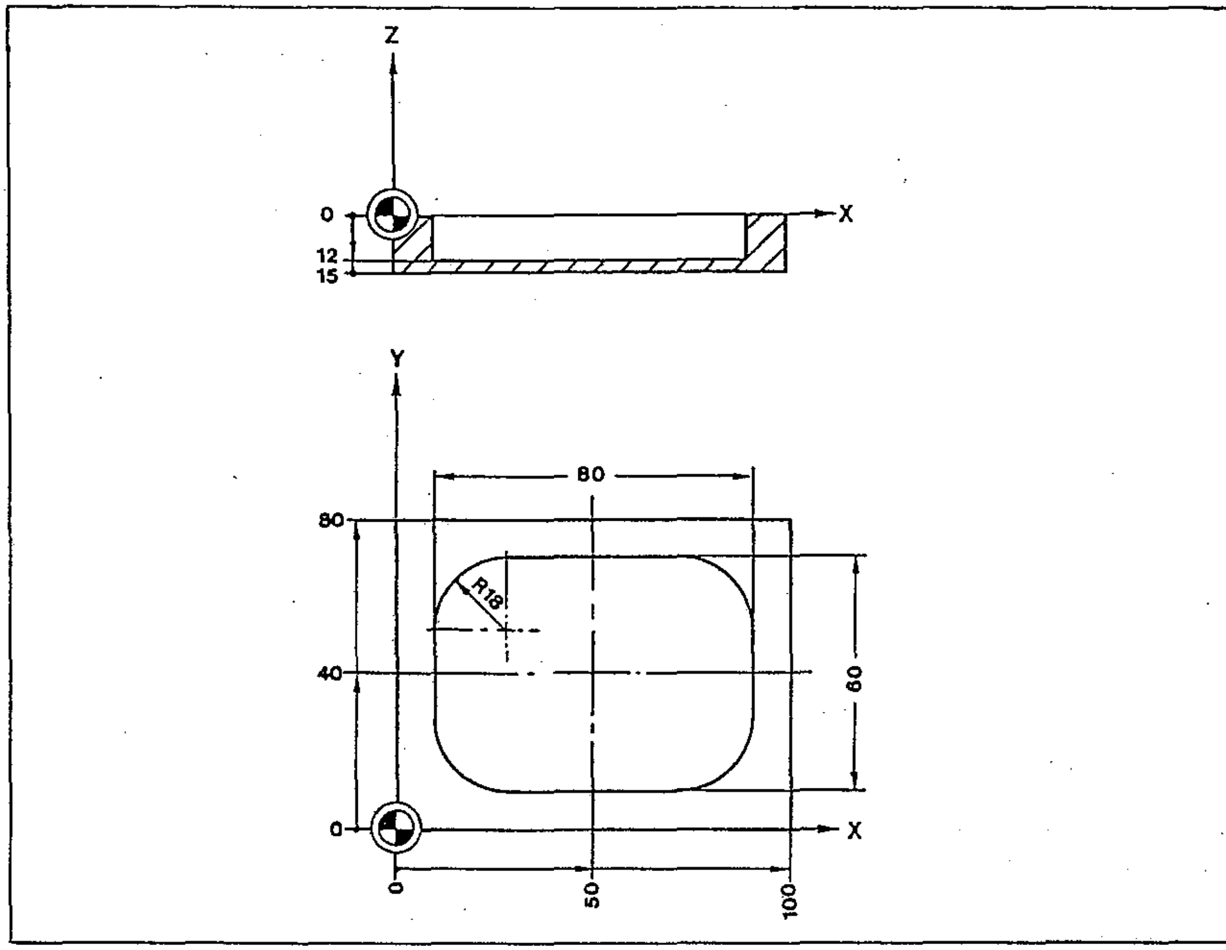
EXAMPLE
for use of vertical spindle

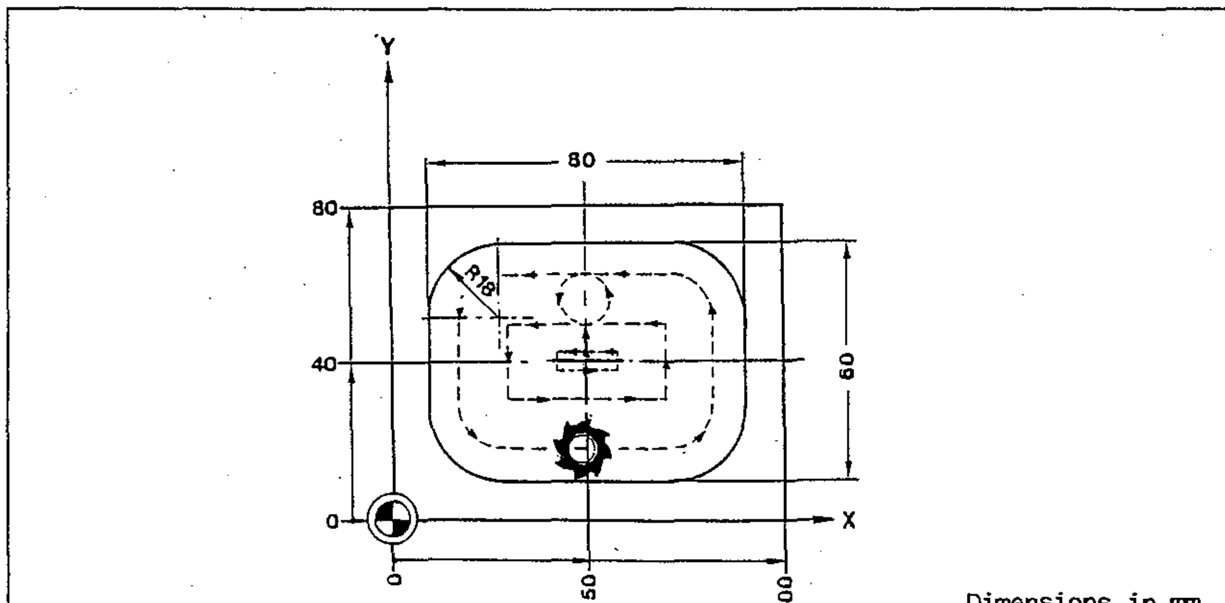


%220	Program number
N1 G0 Z100 N2 G17 T2	Tool change, assign length compensation to Z axis and call tool compensation.
N3 N*2 G0 X50 Y40 Z25	Tool moves to centre of pocket at rapid rate, then N*4 is called and cycle G74 * 1 performed.
N*2 G74*1 F315 S+0 G3 X80 X0.5 Y60 Y0.5 Y12 R18 F200 Z-14 Z-10 Z-0.1 Z23	Subroutine block defines the cycle.

* Use of horizontal spindle: interchange Y and Z axes.

Rectangular pockets - G74 * 1
Finishing to size with intermediate stop
Corner radius programmable



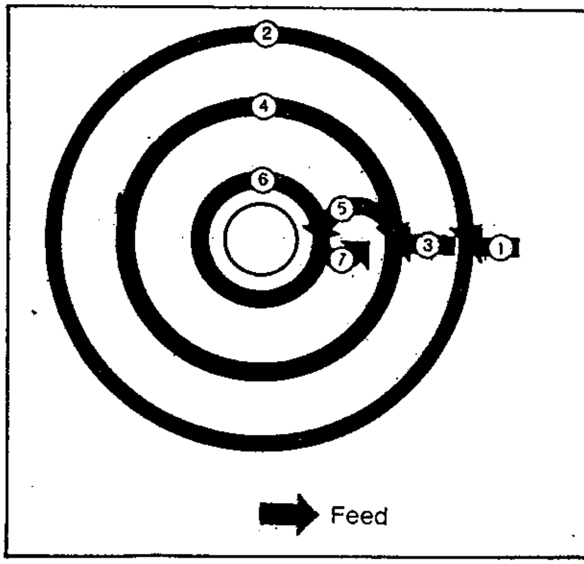


Dimensions in mm

Milling of pins - G75

1 Tool moves to starting point. Tool advanced by infeed increment in Z* at starting point.
2-4 Pin rough-machined; each cut followed by infeed increment in X.
5 Semi-circular approach to last cut. Radius = 1/2 infeed increment, at least 2.5 mm.
6 Milling of pin to final size in X*
7 Semi-circular departure, radius 2.5 mm.

1-7 Steps 1-7 repeated until final dimension in Z has been reached.



INPUT DIALOG

G75	Milling of pins The control requests:	Y	Radius of pin
		Y	Finishing allowance of pin
F	Feed at cutting edge of tool	F	Downfeed rate in Z
S+	Spindle speed (sign can be changed, if not required: transfer key)	Z-	Final dimension in Z + safety allowance
G	Cutting direction - enter G2 or G3.	Z-	Downfeed increment in Z (number of passes)
X	External radius + safety allowance (axis change permissible)	Z-	Finishing allowance in Z
D	Tool compensation number (if not required: transfer key)	Z	2nd plane (if not required: transfer key)
X	Radial infeed (max: cutter diameter)		

Never forget 2nd plane and positioning of tool before starting the cycle.

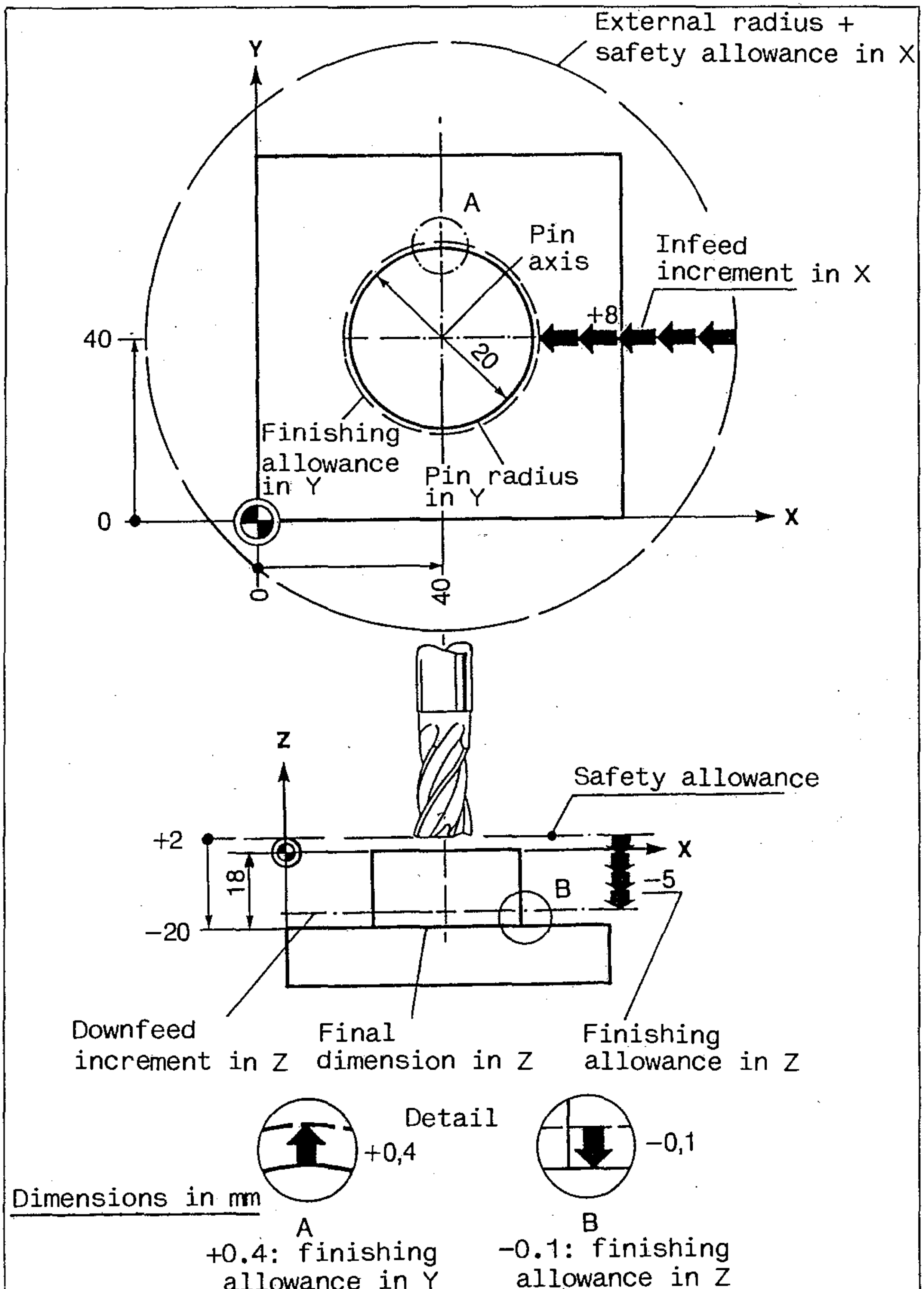
EXAMPLE

for use of vertical spindle*

%225	Program number
N0 G17 T1	Assign length compensation to Z axis and call tool compensation.
N1 N*5 G0 X40 Y40 Z2	Tool moves to centre of pocket at rapid rate, then N*5 is called and cycle G75 performed.
N*5 G75 F125 S+1600 G2 X50 X8 Y20 Y0.4 F100 Z-20 Z-5 Z-0.1	Subroutine block defines the cycle.

* Use of horizontal spindle: interchange Y and Z axes.

Milling of pins - G75



Milling of circular pockets - G76

1 Tool advanced by downfeed increment in Z*, then infeed to 1st cut. If programmed internal radius = 0, or smaller than or equal to tool compensation value: tool motion in X and Z* at feed rate. If internal radius = larger than tool compensation value: tool motion to internal radius at rapid rate.

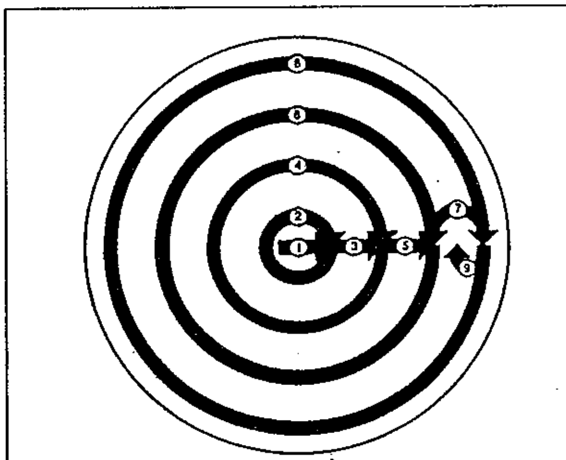
2-6 Pocket rough-machined; each cut followed by infeed increment in X*.

7 Semi-circular approach to last cut. Radius = 1/2 infeed increment, at least 2.5 mm. If there is no room for this approach, control will reduce radius accordingly.

8 Milling of circular pocket to final size.

9 Semi-circular departure, radius = 2.5 mm. If not enough room, control will reduce radius.

1-9 Steps 1-9 repeated until final dimension in Z* has been reached.



INPUT DIAOLG

G76	Circular pockets The control requests:	X	Radial finishing allowance
		X	Radial infeed increment (max = cutter diameter)
F	Feed rate in circular cuts	Y	Internal radius -safety allowance
S+	Spindle speed (sign can be changed, if not required: transfer key)	F	Downfeed rate in Z
G	Cutting direction - use G2 or G3.	Z-	Final dimension + safety allowance
X*	External radius + safety allowance (axis change permissible)	Z-	Downfeed increment in Z (number of passes)
		Z-	Finishing allowance in Z
D	Tool compensation number (if not required: transfer key)	Z	2nd plane (if not required: transfer key)

Never forget 2nd plane and positioning of tool before starting the cycle.

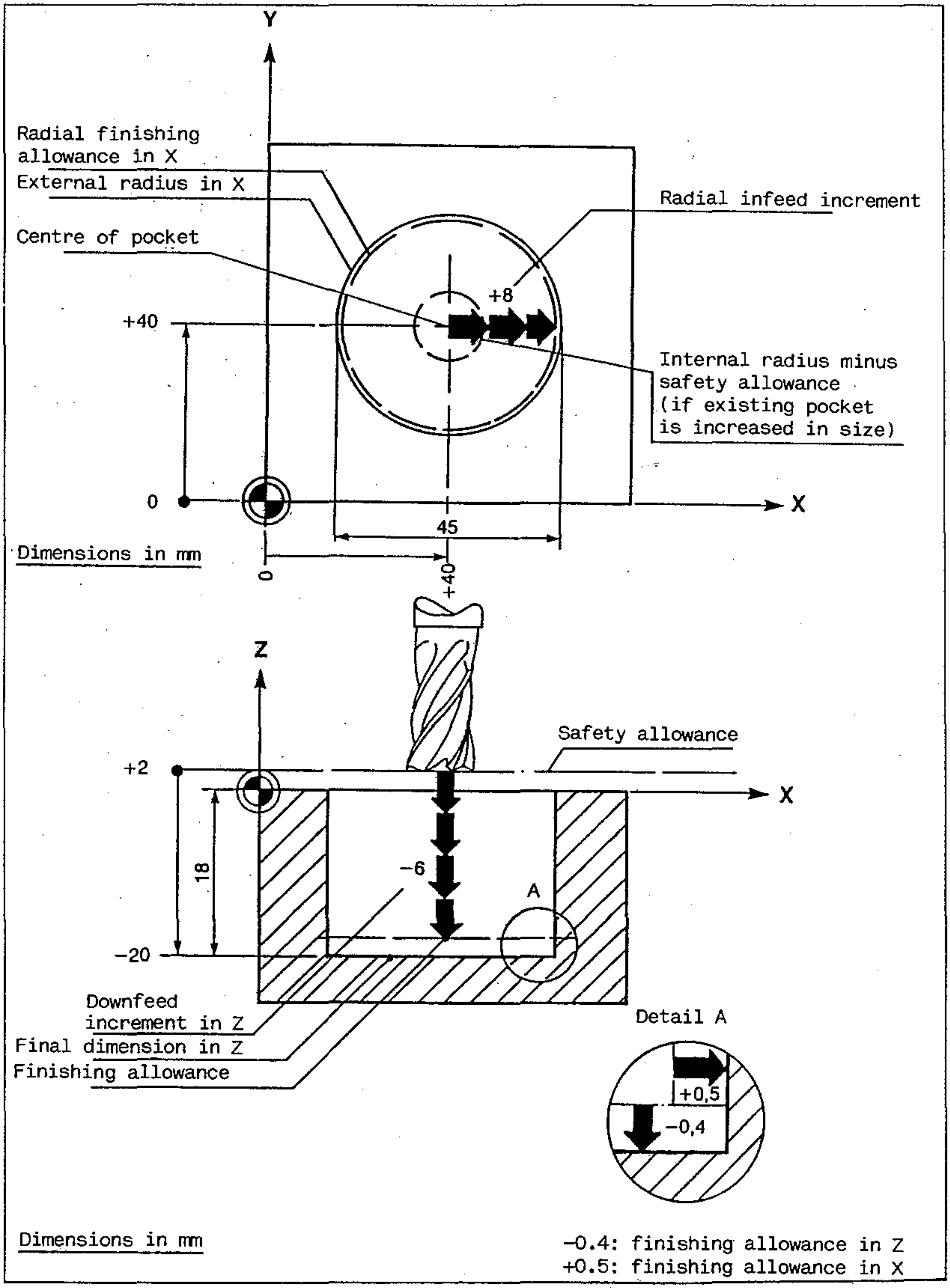
EXAMPLE

for use of vertical spindle*

%230	Program number
N0 G17 T1	Assign length compensation to Z axis and call tool compensation.
N1 N*6 G0 X40 Y40 Z2 S+1000	Tool moves to centre of pocket at rapid rate, then N*6 is called and cycle G76 performed.
N*6 G76 F200 G2 X22.5 X0.5 X8 Y0 F100 Z-20 Z-6 Z-0.4	Subroutine block defines the cycle.

* Use of horizontal spindle: interchange X and Y axes.

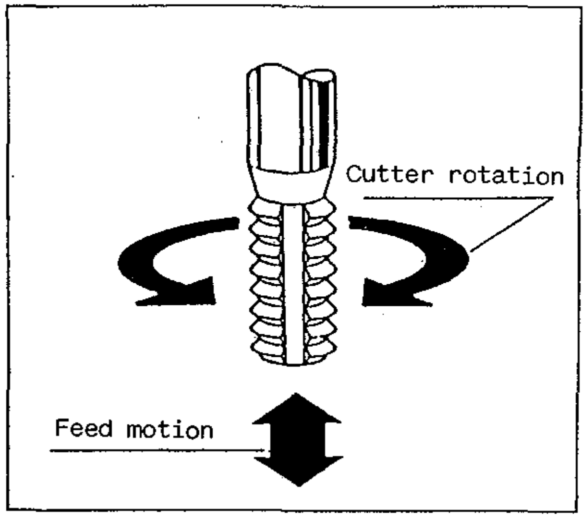
Milling of circular pockets - G76



Thread milling

This cycle permits the machining of single-start or multi-start threads by means of thread milling cutters.

Right-hand and left-hand threads are programmed by linking the cutter rotation with the feed motion as required (see illustrations).



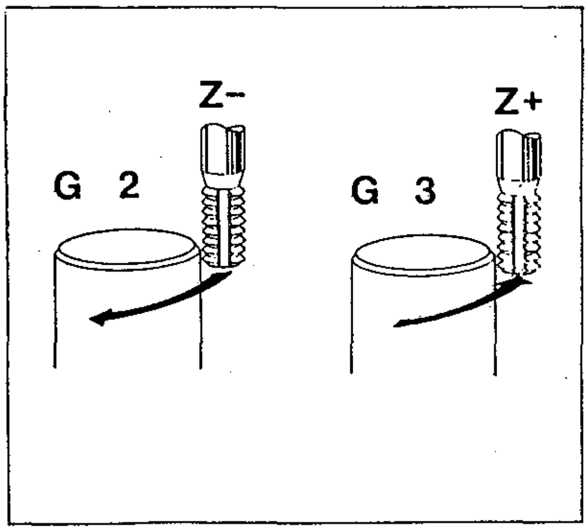
Right-hand threads are obtained when the following combinations are used:

G2 — clockwise rotation and

Z- or Y- — feed motion in negative axis direction)

G3 — counterclockwise rotation and

Z+ or Y+ — feed motion in positive axis direction



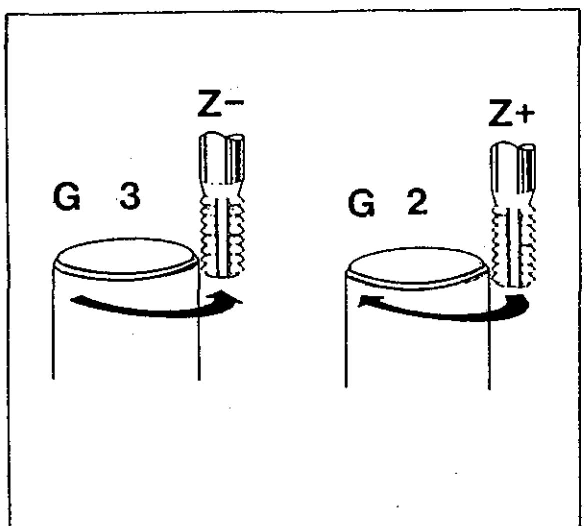
Left-hand threads are obtained when the following combinations are used:

G2 — clockwise rotation and

Z+ or Y+ — feed motion in positive axis direction

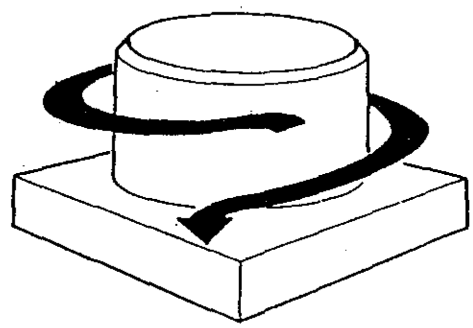
G3 — counterclockwise rotation and

Z- or Y- — feed motion in negative axis direction

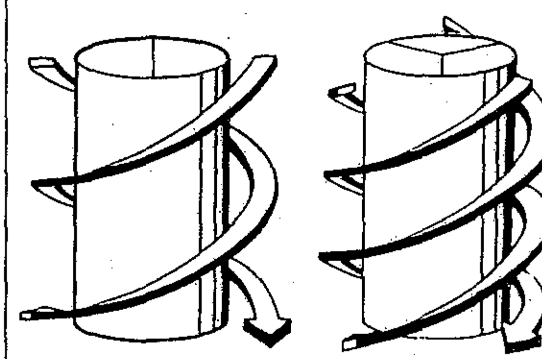


Thread milling

The approach of, and departure from, the contour is by means of a quarter circle or semi-circle.
The radius is freely selectable or can automatically be calculated by the control on the basis of thread diameter, lead and cutter diameter.
The smallest possible radius is 1 mm.
The control automatically allows for the amount of infeed required for the circular arc at the bottom of the hole. In blind holes, the tool will thus not move deeper than to the level of the programmed cutting depth.



Multi-start threads are obtained by using pitch x number of starts as the programmed lead value. The cycle will then be machined several times, depending on the number of starts, with the coordinate system being rotated through the required angle after each cut.



Max lead obtainable:

$$\text{Max lead} = \frac{X^* - 2D}{4}$$

where D = compensation value

Max tool compensation value usable:

$D_{max} = 0.25 \times X^*$

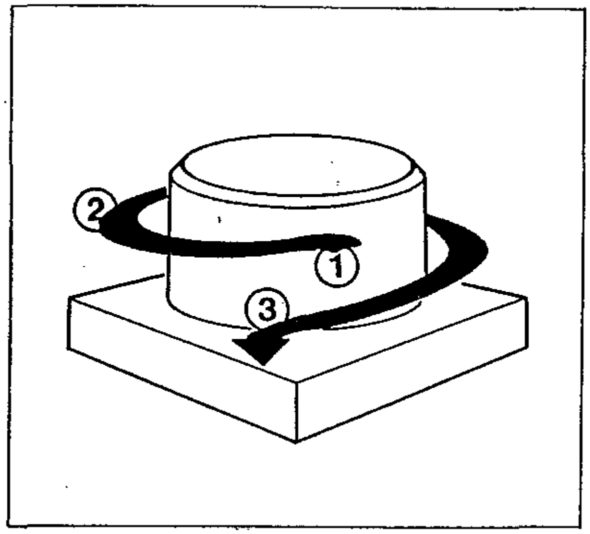
* X = minor or nominal diameter

External threads - G77

1 Quarter-circular approach of thread cutting position.

2 Milling of external thread.

3 Quarter-circular departure from contour.



INPUT DIALOG

G77 External threads

The control requests:

F	Feed rate at cutting edge
S+	Spindle speed (if not required: transfer key; sign can be changed)
G	Cutting direction: use G2 or G3
X*	Minor diameter of thread (axis can be changed)
D	Tool compensation number (if not required: transfer key)
X	Allowance on thread diameter (sign can be changed)
A	Appproach radius (if to be calculated by control: transfer key)
Z-	Lead of thread (sign can be changed)
Z-	Final dimension in Z (+ safety allowance)
Z	2nd plane (if not required: transfer key)

Never forget 2nd plane and positioning of tool before starting the cycle.

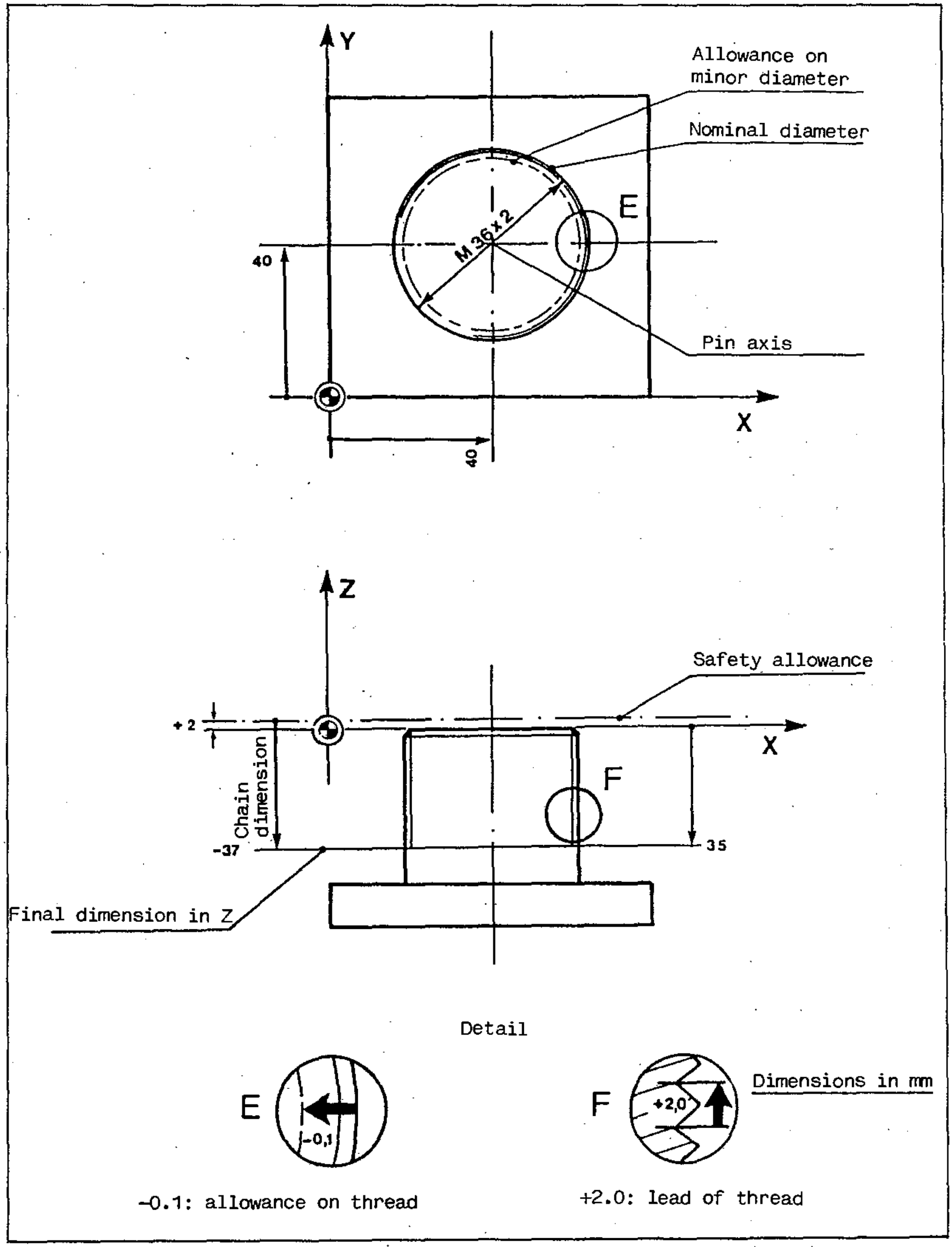
EXAMPLE

for vertical spindle*

%235	Program number
N0 G17 T1	Assign length compensation to Z axis and call tool compensation.
N1 N*7 G0 X40 Y40 Z2 S+1000	Tool moves above pin axis at rapid rate, then N*7 is called and cycle G77 performed.
N*7 G77 F160 G3 X33.546 X-0.1 A1 Z2 Z-37	Cycle containing subroutine block.

* Use of horizontal spindle: interchange X and Z axes.

External threads – G77

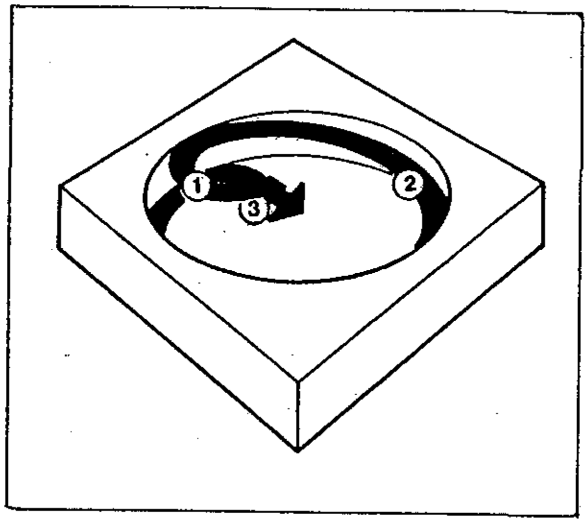


Internal threads - G78

1 Quarter-circular approach of thread cutting position.

2 Milling of internal thread.

3 Quarter-circular departure from contour.



INPUT DIALOG

G78	Internal threads The control requests:	X–	Allowance on thread diameter (sign can be changed)
F	Feed rate at cutting edge	A	Appproach radius (if to be calculated by control: transfer key)
S+	Spindle speed (if not required: transfer key; sign can be changed)	Z–	Lead of thread (sign can be changed)
G	Cutting direction: use G2 or G3	Z–	Final dimension in Z (+ safety allowance)
X*	Nominal diameter of thread	Z	2nd plane (if not required: transfer key)
D	Tool compensation number (if not required: transfer key); max cutter dia: 0.5 x thread nominal dia		

Never forget 2nd plane and positioning of tool before starting the cycle.

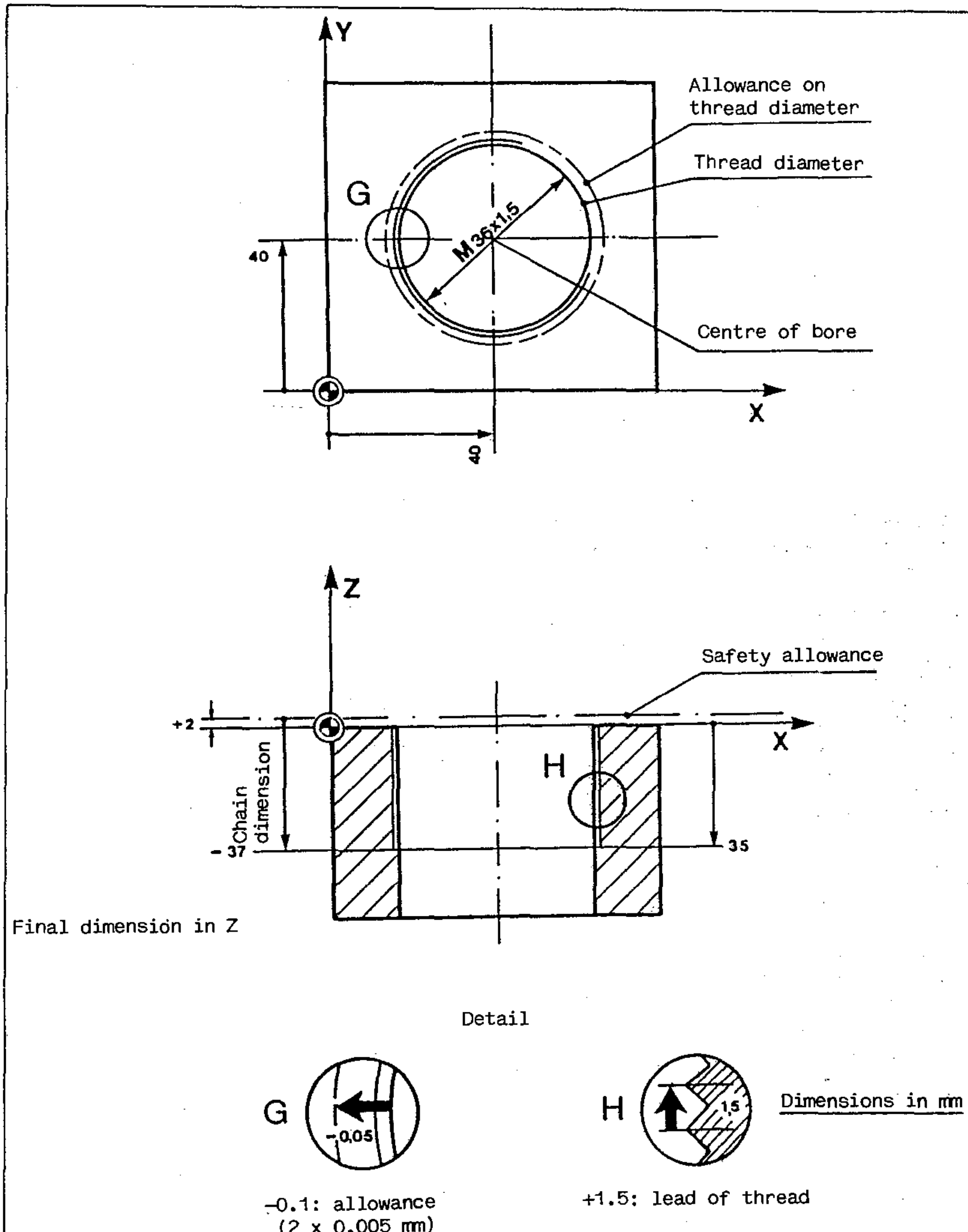
EXAMPLE

for vertical spindle*

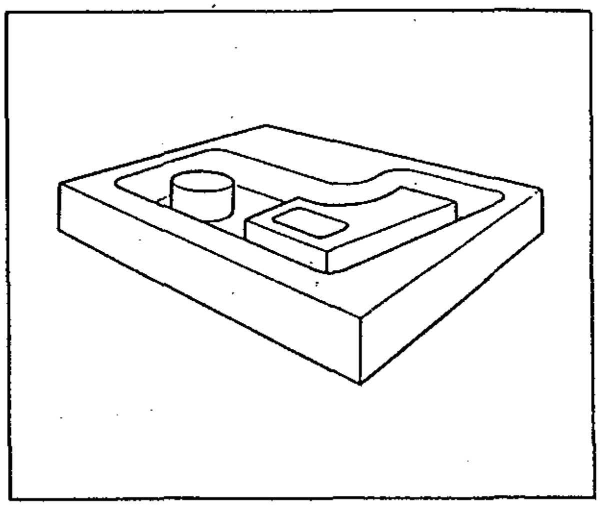
%240	Program number
N0 G17 T1	Assign length compensation to Z axis and call tool compensation.
N1 N*8 G0 X40 Y40 Z2 S+1000	Tool moves above centre of bore at rapid rate, then N*8 is called and cycle G78 performed.
N*8 G78 F160 G3 X36 X0.1 A1 Z1.5 Z-23	Cycle containing subroutine block.

* Use of horizontal spindle: interchange X and Z axes.

Internal threads - G78

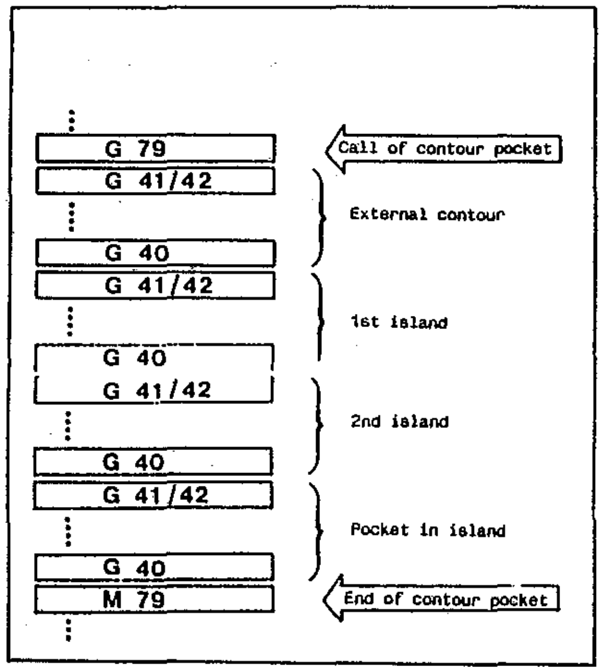


Contour pockets - G79

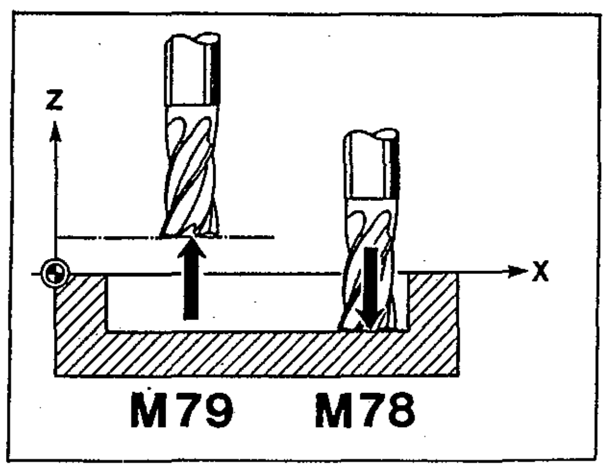


The control permits the programming of pockets

- of any desired external contour and
- with up to 7 'islands'.



The individual contours are programmed consecutively, for example - first the pocket, then islands in the pocket (if any), then pockets in the islands (if any), etc.
Such program blocks are preceded by a block with the address G79. This block contains information on how the stock inside the pocket is to be cleared out.
The program blocks with the pockets and islands are followed by a block with the address M78/M79. This informs the control that all contours between the block with G79 and the block with M78/M79 are elements of a contour pocket.
The following points should be observed in writing the program:
- the outline of the pocket should be programmed as an internal contour,
- the islands as external contours and
- additional pockets in the islands again as internal contours.



M78/M79

If a contour pocket is concluded by command M79, the tool will move back to the retraction plane when the pocket has been cleared out.
Command M78 is used for clearing out a contour pocket in several passes.
If M78 is used, the tool will again move above the starting point of the first contour when the pocket has been cleared out, and will then be downfed to the last preceding cutting depth.
This facilitates the repetition of the operation at a new cutting depth.

Contour pockets – G79

INPUT DIALOG

Data input for calling a contour pocket is dialog-assisted by the control. Upon calling G79 and pressing the transfer key on NC keyboard (23), the control requests the addresses listed on the right. These should be completed as required and transferred to the control by pressing the transfer key.

Feed increment for clearing out pocket

The coordinate determines the axis of incremental feed, the sign controls the feed direction.

Cutting direction

The sign determines at what point and in what direction the tool starts clearing out the pocket (value always to be other than 0).

Tool length compensation number

If the workpiece is machined with a cutting tool for which tool length compensation has been programmed, this compensation must be programmed here as well. If no length compensation is required, the tool table must not contain any entry for the tool length. For D... press transfer key.

Distance from next pocket bottom

Tool moves from retraction plane to programmed distance at rapid rate, and from there to full pocket depth at feed rate. Transfer key without numerical input: tool moves at rapid rate all the way.

Feed rate

Feed rate used for clearing out pocket.

Retraction plane

This is the Z value to which the tool is retracted in clearing out a pocket.

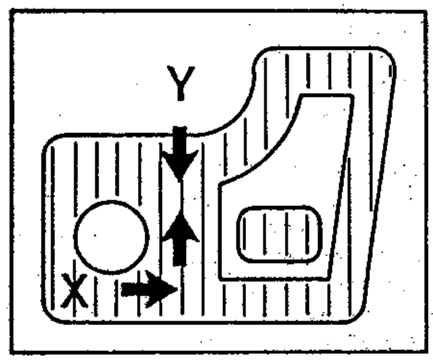
End of contour pocket – M78/79

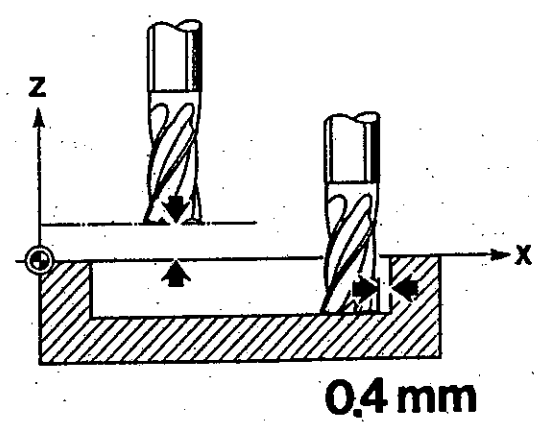
A block with M78 or M79 indicates the end of a contour pocket.

*Use of horizontal spindle: interchange Y and Z axes.

G79: Call of contour pocket
The control requests:

X(Y)(Z)*	Feed increment and axis direction for clearing out pocket
Y(X)(Z)	Starting point and cutting direction for clearing out pocket
F	Feed rate for clearing out
Z	Lift-off in Z
D	Tool length compensation number (if not required: transfer key)
Z	Distance from botton of pocket





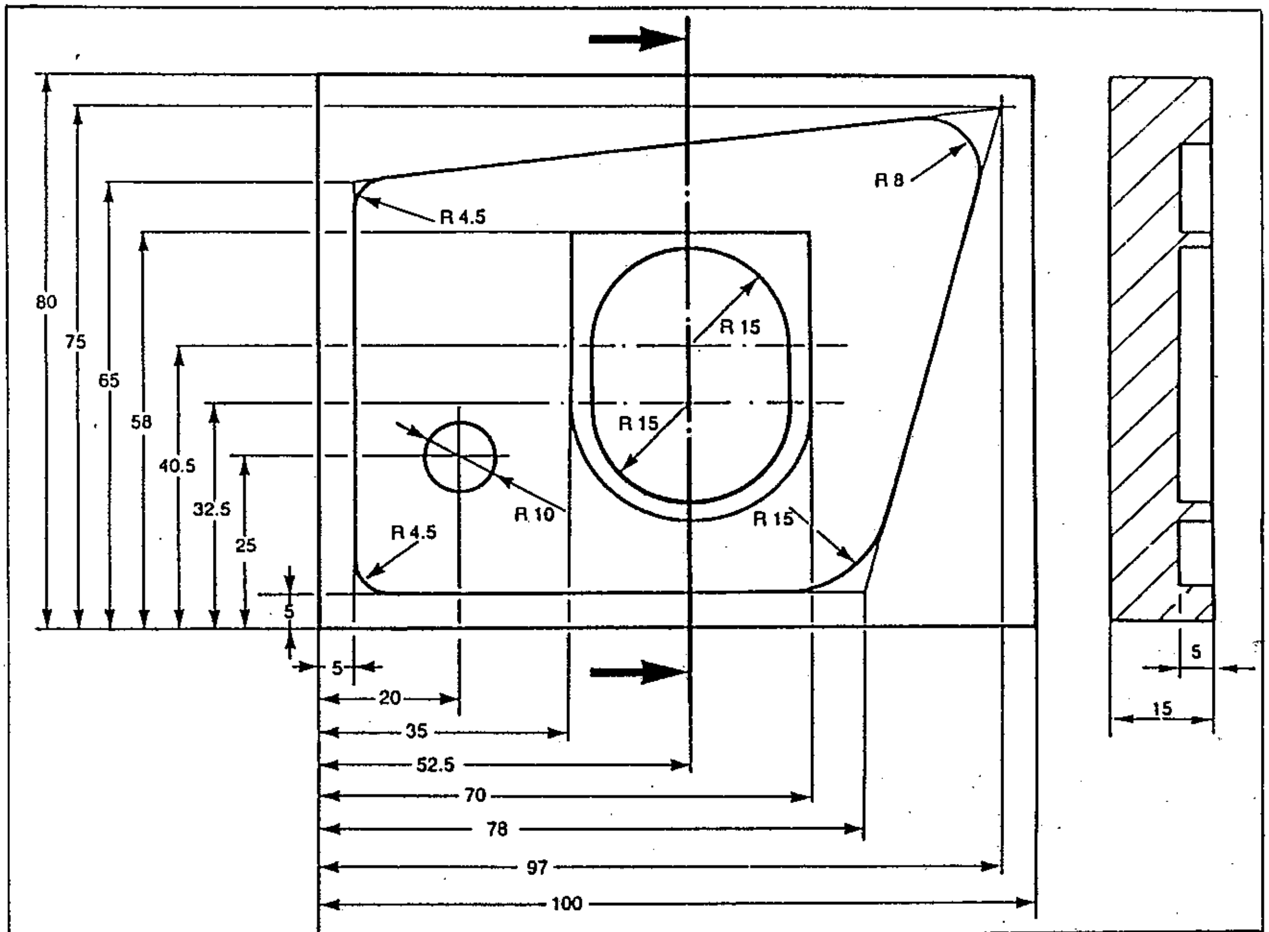
Contour pockets

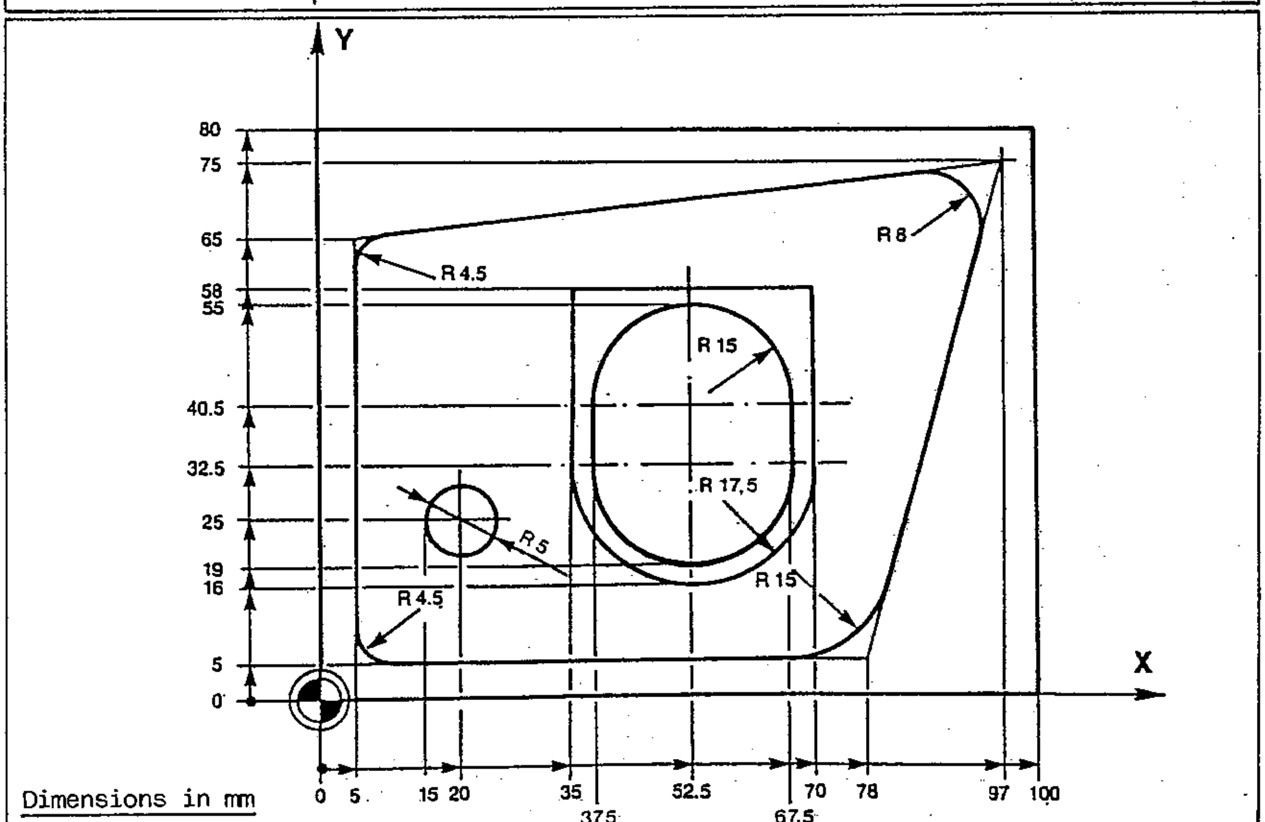
EXAMPLE

Clearing out in a single pass

%245	Program number
N1 G0 Z100 N2 G17 T1	Tool change. Assign length compensation to Z axis and call tool compensation.
N3 G79 Y5 X1 F100 Z2	Definition of pocket: feed 100 mm/min and pocket cleared out in X axis, starting in lower left corner. Feed increments 5 mm in Y axis after each cut. Islands cleared at a distance of 2 mm from top surface.
N4 G0 Z2 S+2500 M70	Tool is positioned at rapid rate to point above workpiece surface upon call of compensation.
N5 Z-5 F100 M70	Downfeed to cutting depth upon call of compensation.
N6 G41 G47 A2 X5 Y25 G0 G60 M61 N7 Y5 N8 G7 R4.5 N9 X78 N10 G7 R15 N11 X97 Y75 N12 G7 R8 N13 X5 Y65 N14 G7 R4.5 N15 Y25 N16 G40 G47 A2	Milling of external contour.
N17 G41 G47 A2 X15 Y25 G1 G61 M61 N18 G2 I5 J0 N19 G40 G47 A2	Milling of circular island.
N20 G0 Z2 N21 Z-5 M70	Lift-off and renewed downfeed to cutting depth upon call of compensation on contour.
N22 G41 G45 A1 X35 Y58 G0 G61 M61 N23 X70 N24 Y32.5 N25 G2 X35 Y32.5 I-17.5 J0 N26 Y58 N27 G40 G45 A1	Milling of island.
N28 G0 Z2 N29 Z-5 M70	Lift-off and renewed downfeed upon call of compensation on contour.
N30 G41 G47 A2 X37.5 Y32.5 G0 G61 M61 N31 G3 X67.5 Y32.5 I15 J0 N32 Y40.5 N33 G3 X37.5 Y40.5 I-15 J0 N34 Y32.5 N35 G40 G47 A2	Milling of pocket in island.
N36 M79	End of pocket; clearing-out called.
N37 T0	Delete compensation of contour.
N38 G0 Z100 M30	Tool retracted; end of program.

Contour pockets - G79



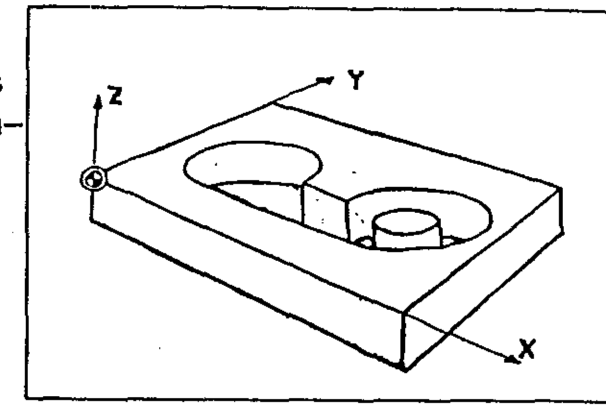


Dimensions in mm

Contour pockets - G79

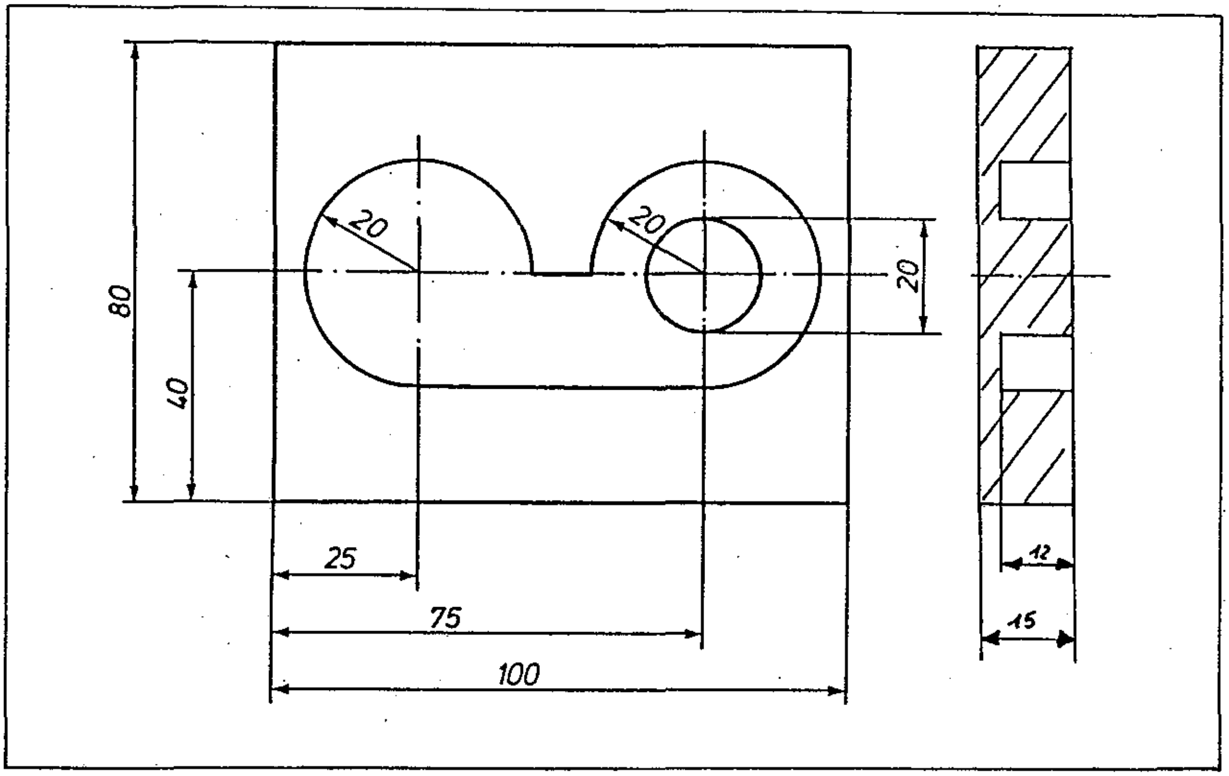
EXAMPLE

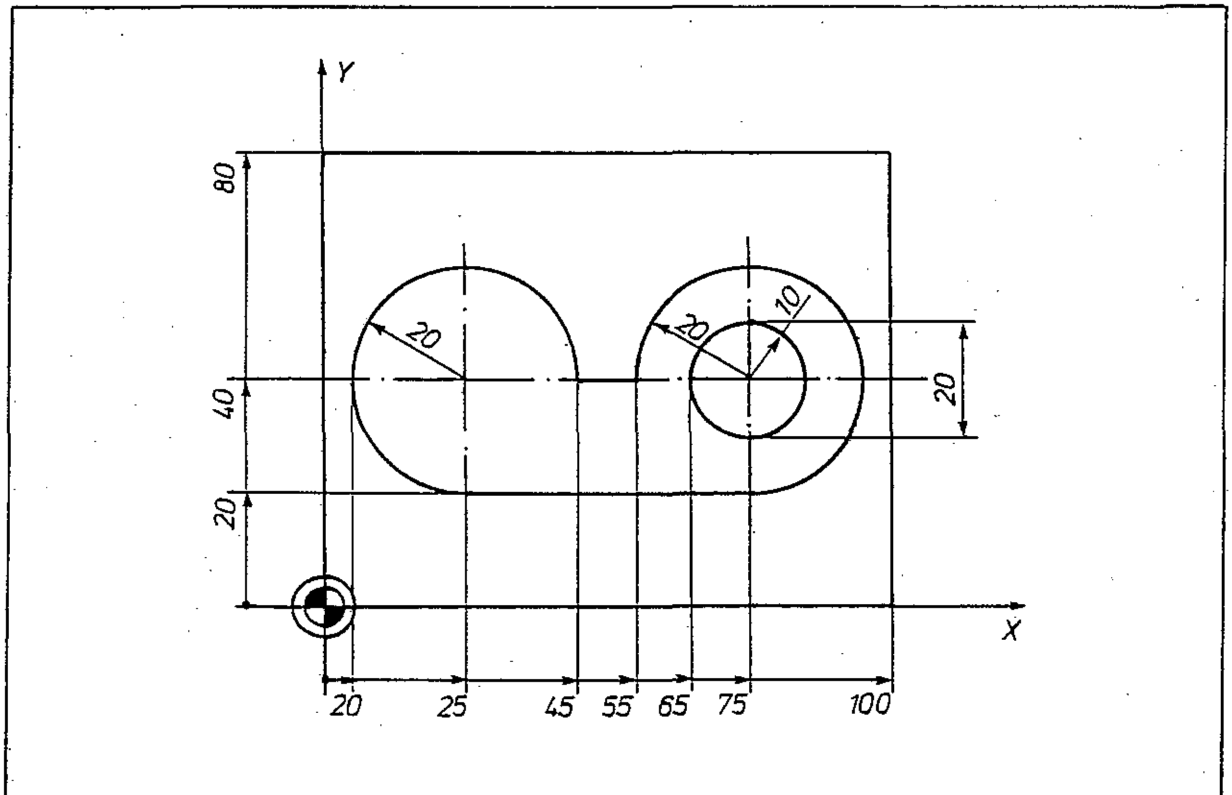
Clearing out in several passes
Tool not retracted between contour sections



%250	Program number
N1 G0 Z100 N2 G17 T1	Tool change. Assign length compensation to Z axis and call tool compensation.
N3 G0 X55 Y30 Z2 S+6300	Tool moves to safety distance at rapid rate.
N4 Z0 F100	Tool advanced to workpiece surface at feed rate: this is the starting point for three program part repetitions (downfeed in chain dimensions).
N5 G79 Y7 X1 F100 Z2 Z6	Definition of contour pocket: tool advanced at feed 100 mm/min. Pocket cleared out in X axis, starting in lower left corner. For each cut, tool is fed 7 mm in Y axis. Above islands, tool is retracted to 2 mm above top surface each time and then downfed again to a point 6 mm above pocket bottom at rapid rate.
N6 G91 Z-4	Tool downfed 4 mm in Z axis (chain dimension).
N7 G90	Call of absolute input.
N8 G41 G45 A1 X55 Y40 G0 G60 M61 N9 X45 N10 G3 X25 Y20 I-20 J0 N11 X75 N12 G3 X55 Y40 I0 J20 N13 G40 G45 A1	Milling of external contour.
N14 G41 G45 A1 X65 Y40 G1 G60 M61 N15 G2 I10 J0 N16 G40 G45 A1	Milling of island.
N17 M78	End of pocket, clearing out called. After clearing out pocket, tool moves back to retraction plane, then above starting point of external (1st) contour, and is then downfed to pocket bottom.
N18 L2 N5 N17	Two program part repetitions (N5 to N17). At start, tool is downfed 4 mm in Z axis (chain dimension) and at that level repeats the first cut of the contour pocket. 3 x 4 mm = full depth of 12 mm.
N19 T0	Delete compensation of contour.
N20 G0 Z100 M30	Tool retracted in Z; end of program.

Contour pockets – G79





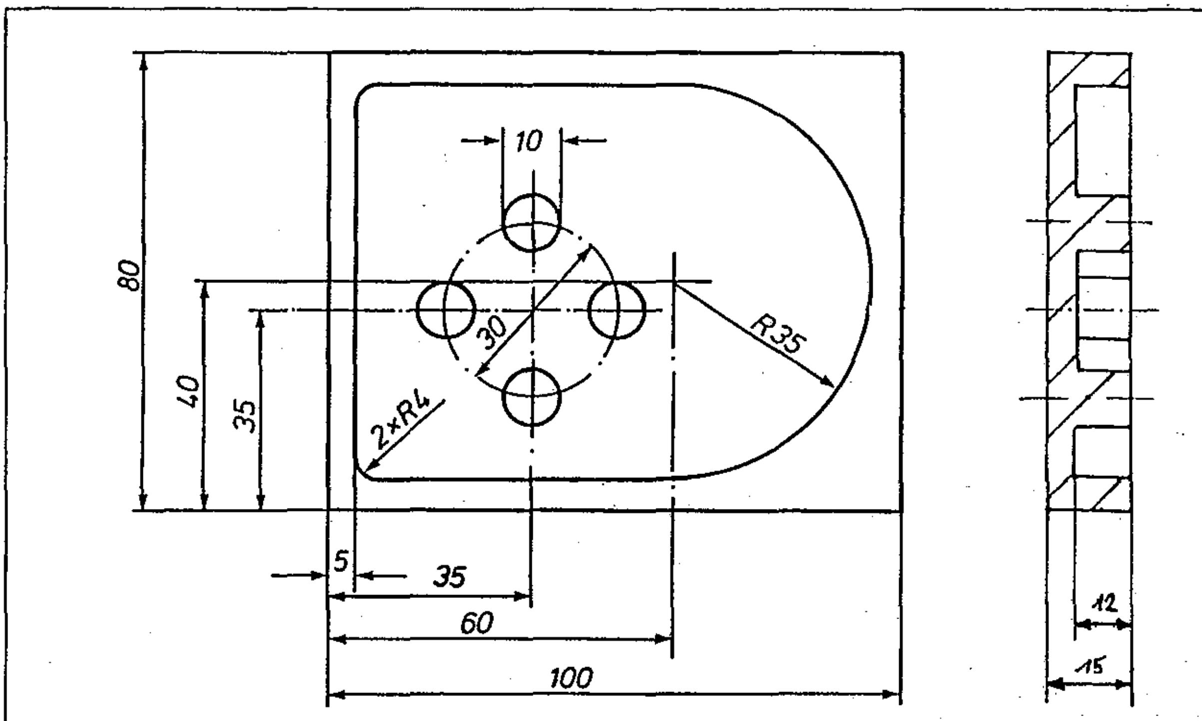
Dimensions in mm

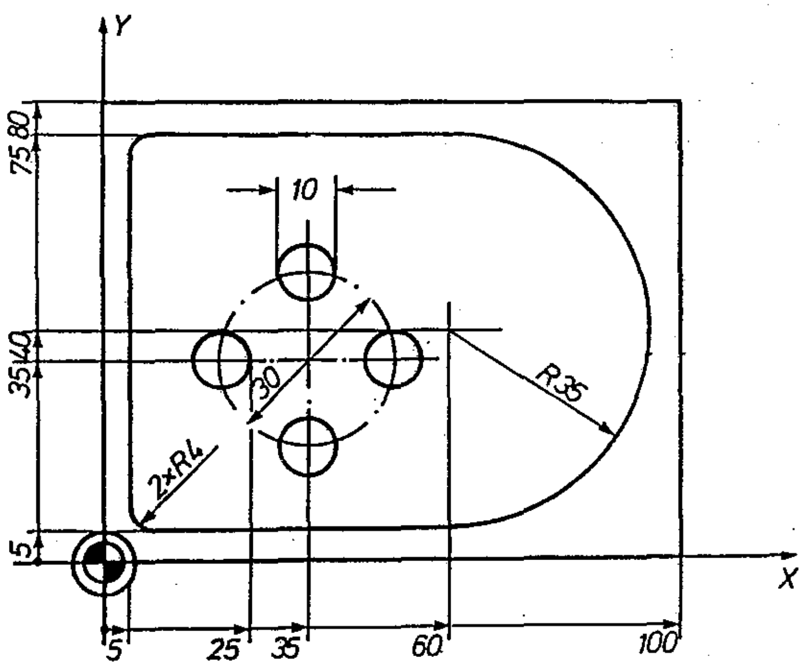
Contour pockets - G79

EXAMPLE
Tool retracted between contour sections

%255	Program number
N1 G0 Z100 N2 G17 T1	Tool change. Assign length compensation to Z axis and call tool compensation.
N3 G0 X60 Y65 Z2 F200 S+3150	Tool moves to safety distance at rapid rate.
N4 G79 X7 Y1 F100 Z2 Z6	Definition of contour pocket: tool advanced at feed 100 mm/min. Pocket cleared out in Y axis, starting in lower left corner. For each cut, tool is fed 7 mm in X axis. Above islands, tool is retracted to 2 mm above top surface each time and then downfed again to a point 6 mm above pocket bottom at rapid rate.
N5 N*1	Call of subroutine defining cutting depth.
N6 G41 G47 A5 X60 Y75 G1 G60 M61 N7 X5 N8 Y5 N9 X60 N10 G3 X60 Y75 I0 J35 N11 G40 G45 A5	Milling of external contour. Call of incremental input.
N12 G0 Z2	Tool retracted to Z2 at rapid rate.
N13 G0 X35 Y35	Tool moves to centre of pocket at rapid rate.
N14 N*1	Call of absolute input; tool downfed to cutting depth defined in subroutine.
N15 G41 G47 A2 X25 Y35 G1 G61 M61 N16 G2 I-5 J0 N17 G40 G47 A2	Milling of first pin.
N18 G55 W90 I35 J35	Rotation of coordinate system about centre of pin.
N19 L3 N15 N18	Three program part repetitions of blocks N15 to N18: milling of other three pins.
N20 G53	Deletion of zero offset.
N21 M78	End of contour pocket, clearing out called. After clearing out, tool is downfed to pocket bottom above starting point of external contour.
N22 L2 N4 N21 N*2 N*3	Two program repetitions (blocks N4 to N21) with subroutine substitution. Subroutines define the cutting depths for the three cuts.
N23 T0	Delete compensation of contour.
N24 G0 Z100 M30	Tool retracted in Z; end of program.
N*1 Z-4 N*2 Z-8 N*3 Z-12	Subroutine blocks for downfeed motions in Z.

Contour pockets – G79





Dimensions in mm

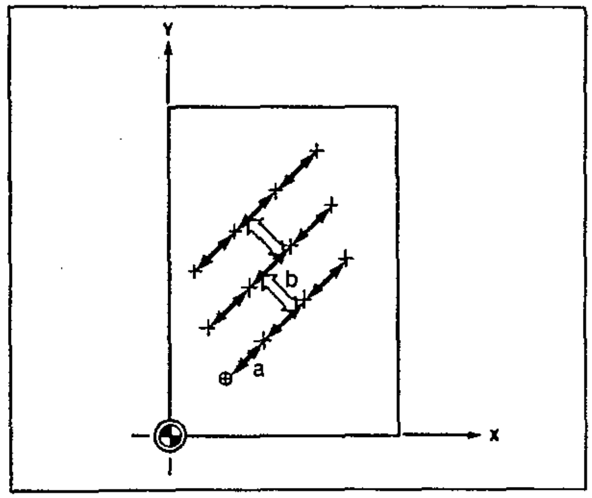
Hole patterns - G87

If the tool is to be positioned to a number of holes of equal spacing and configuration for the purpose of performing the same operation (eg drilling holes) at each point, such points can be programmed as hole patterns in a single block.
The cutting operation is programmed in a subroutine block which can then, by program part repetition with subroutine substitution (page 6-58), be replaced by another subroutine block containing a different cutting operation. In this way, several cutting operations can be performed consecutively at each of the points programmed.

Point patterns can be programmed if:

- the points are equally spaced (a) in one or several parallel rows, and
- these parallel rows are equally spaced (b) as well.

The distances (a) of the points in a row may be different from the distances (b) between the rows.



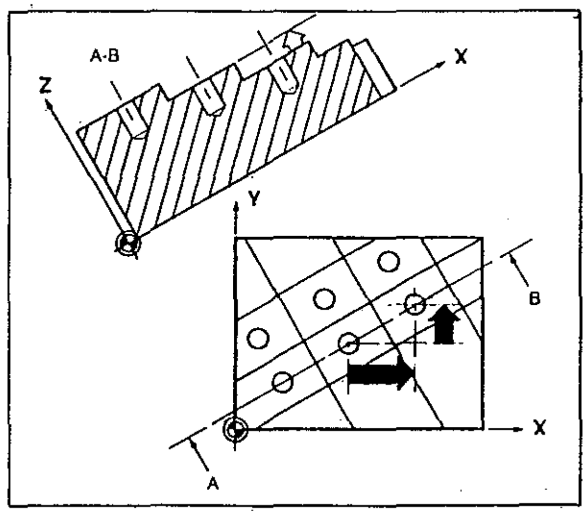
INPUT DIALOG

When command G87 has been entered, the control requests the addresses shown on the right.
These should be completed and transferred to the control by pressing the transfer key.

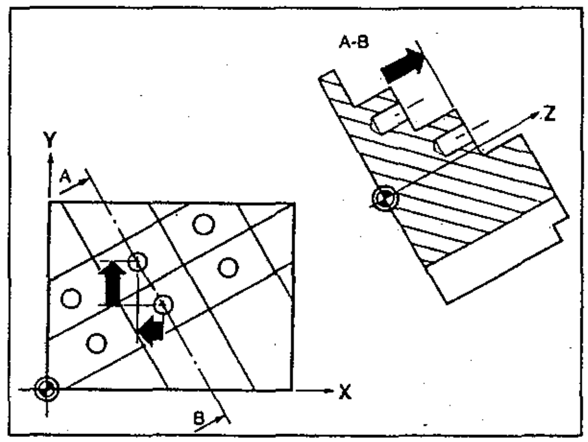
G87	Point patterns The control requests
N*	Subroutine block used for machining all holes of pattern
L	Number of holes in a row (max 999)
X	Distance between holes in a row in X axis
Y	Distance between holes in a row in Y axis
Z	Distance between holes in a row in Z axis
C	Spacing angle of holes in a row in C axis
L	Number of rows (max 999; if only one: transfer key)
X	Distance between rows in X axis
Y	Distance between rows in Y axis
Z	Distance between rows in Z axis
C	Spacing angle of rows in C axis

Hole patterns - G87

Dimension data of holes in each row (axes not required: transfer key).



Dimension data of rows (axes not required: transfer key).

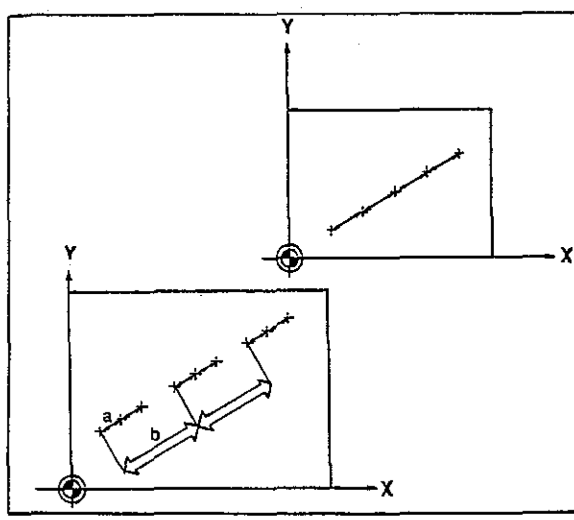


Tool moves from hole to hole along straight lines. Make sure tool clearance is sufficient to prevent collisions.

Special hole patterns obtainable:

If tool has to be positioned to holes in only one row, press transfer key in response to request for number of rows.

If tool has to be positioned to holes in only one row, but at two different, regularly recurring distances between such holes, the distance between the rows (b) has to be programmed as shown in the drawing (bottom right).

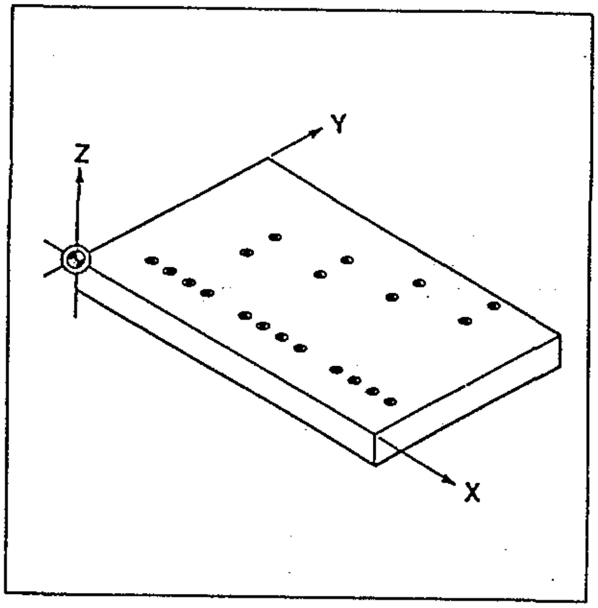


Spindle rotation must be started in the block preceding the call of command G87.

Hole patterns – G87

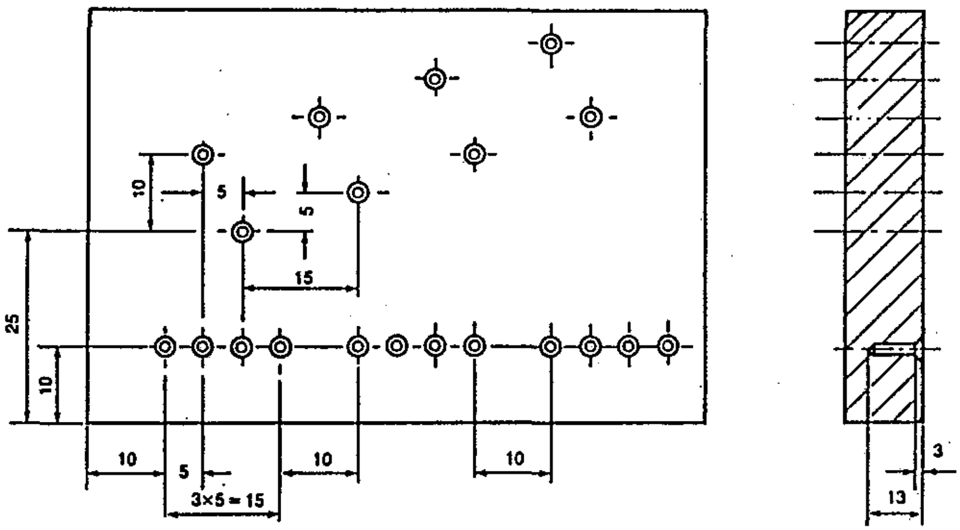
EXAMPLE

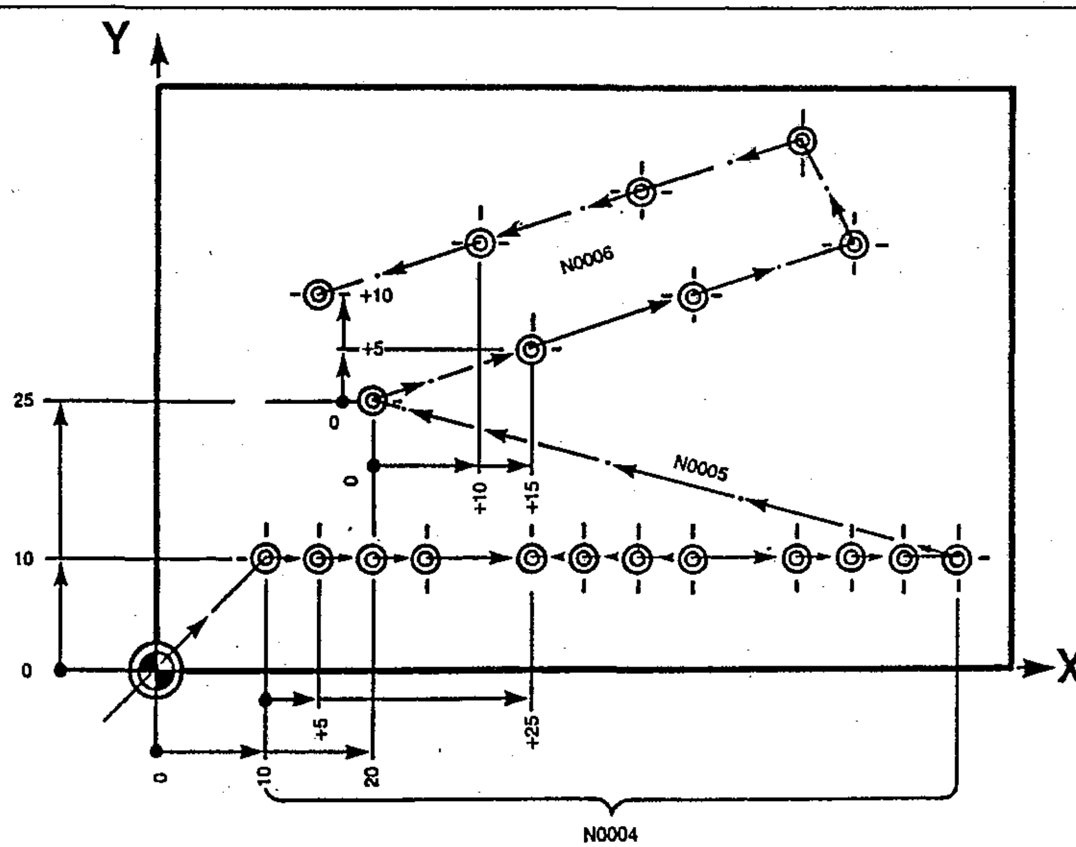
A workpiece with a hole pattern is to be machined in two steps: first, spot drilling and chamfering, then tool change and drilling to nominal diameter.
The program for the second operation includes a program part repetition with subroutine call (see page 6-58).



%260	Program number
N1 G0 Z100 N2 G17 T1	Tool change. Assign length compensation to Z axis and call tool compensation.
N3 G0 X10 Y10 Z2 S+2500	Tool moves above first hole at rapid rate.
N4 G87 N*1 L4 X5 L3 X25	Machining of lower row of holes using the drilling cycle programmed under N*1.
N5 G0 X20 Y25	Tool moves above upper hole pattern at rapid rate.
N6 G87 N*1 L4 X15 Y5 L2 X-5 Y10	Machining of upper hole pattern.
N7 T0 N8 G0 Z100 N9 T2	Delete compensation of contour. Tool retracted to tool change position. Tool change, call new tool compensation.
N10 L1 N3 N8 N*2	Program part repetition with tool positioning block and subroutine call of blocks N3 to N8, but now N*2 is performed instead of N*1.
N11 M30	End of program.
N*1 G81 F250 S+2500 Z-5	Drilling cycle for first cutting operation.
N*2 G81 F160 S+3150 Z-15	Drilling cycle for second cutting operation.

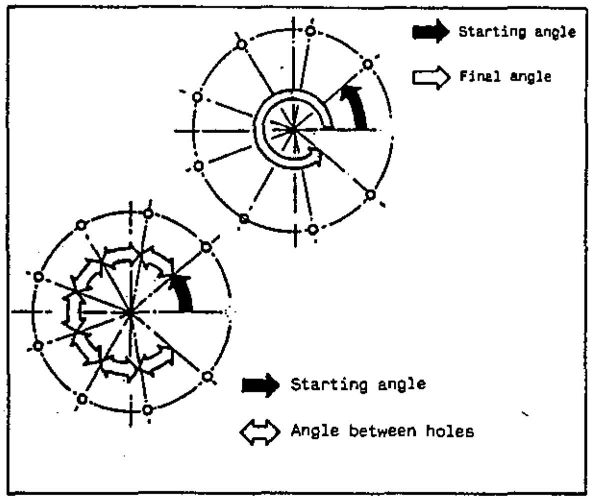
Hole patterns - G87





Patterns on circular arcs - G88

If the tool is to be positioned to a number of holes of equal angular spacing on a circular arc for the purpose of performing the same operation (for example, drilling holes) at each point, such points can be programmed as hole patterns in a single block.
The cutting operation is programmed in a subroutine block which can then, by means program part repetition with subroutine substitution, be replaced by another subroutine block containing a different cutting operation. Several cutting operations can thus be performed consecutively at each point.



To determine the coordinates of the holes, the control requires the centre point and the radius of the circular arc.

The positions of the holes on the cicular arc are defined by the following data:

- number of holes, instruction to space holes equally on circular arc, and definition of starting angle;
- number of holes, definition of starting and final angles: holes will be equally spaced;
- number of holes, definition of starting angle and final angle.

Hole patterns are arranged at intersections of circular arc with starting and final angles and equally spaced in between.

INPUT DIALOG

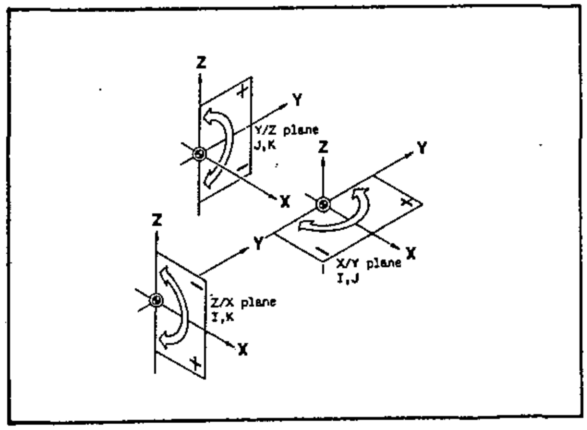
When command G88 has been entered, the control requests the addresses shown on the right.
These should be completed and transferred to the control by pressing the transfer key.

G88: Patterns on circular arcs
The control requests:

N*	Subroutine to be machined at all points
L	Number of holes on circular arc (max 999)
R	Radius fo circular arc
G	Cutting direction clockwise/ counterclockwise (G2/G3)
W	Starting angle
M7	Definition of holes on circular arc (M71/M72)
W	Final angle/intermediate angle
I(J)I	1st coordinate of centre of circular arc
J(K)K	2nd coordinate of centre of circular arc
G	Tool travel from hole to hole with linear or circular interpolation (G1/G2/G3)

Patterns on circular arcs – G88

Starting angle W is the angle between a line from the pole to the starting point and the angle reference axis.

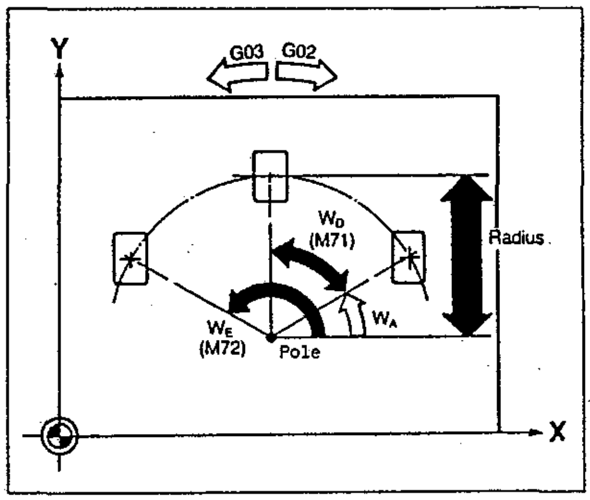


Location of holes on circular arcs:

M7 and transfer key (without numerical input): full circle;

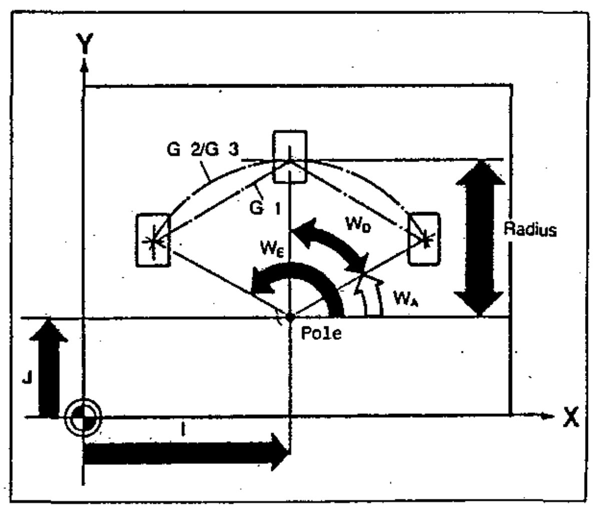
M72 definition by entering number of holes and final angle (absolute input of angle);

M71 definition by entering number of holes and angle between holes (incremental input of angle).



Commands M72 or M71 determine whether the final angle or the angle between the holes is to be entered under address W.

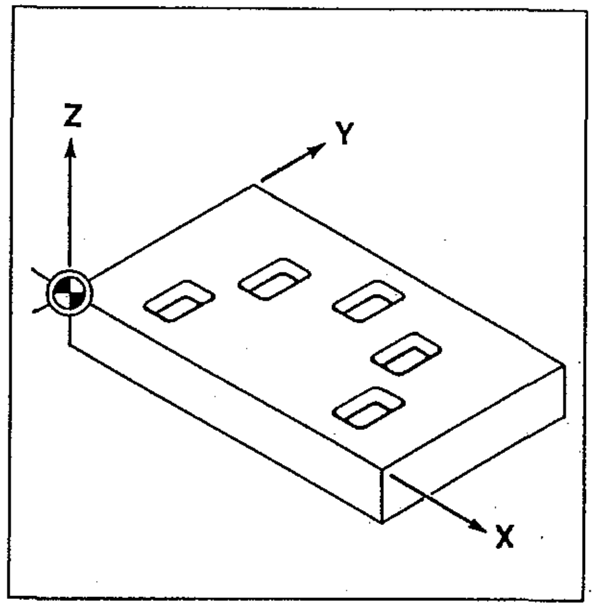
Tool may move from hole to hole along either a straight line (use G1) or a circular arc (use G2 or G3).



Spindle rotation must be started in the block preceding the call of command G88.

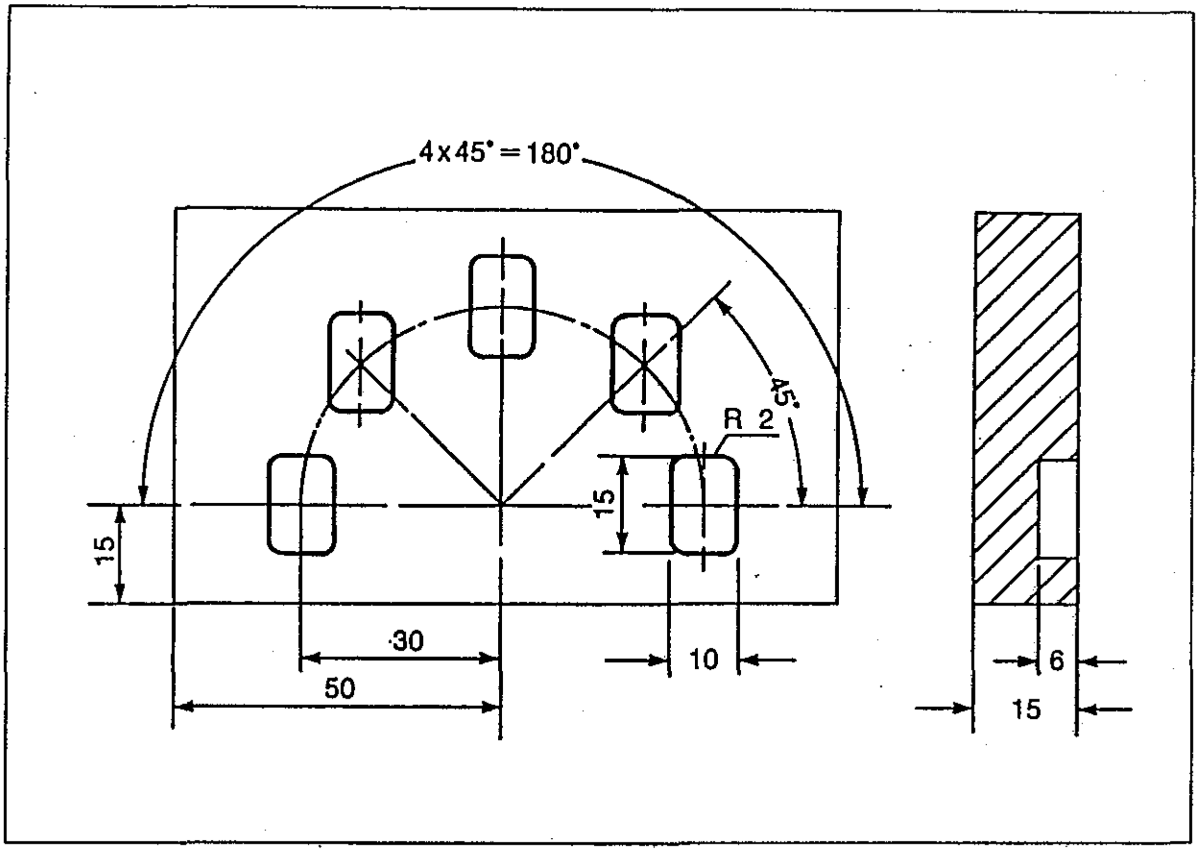
Patterns on circular arcs - G88

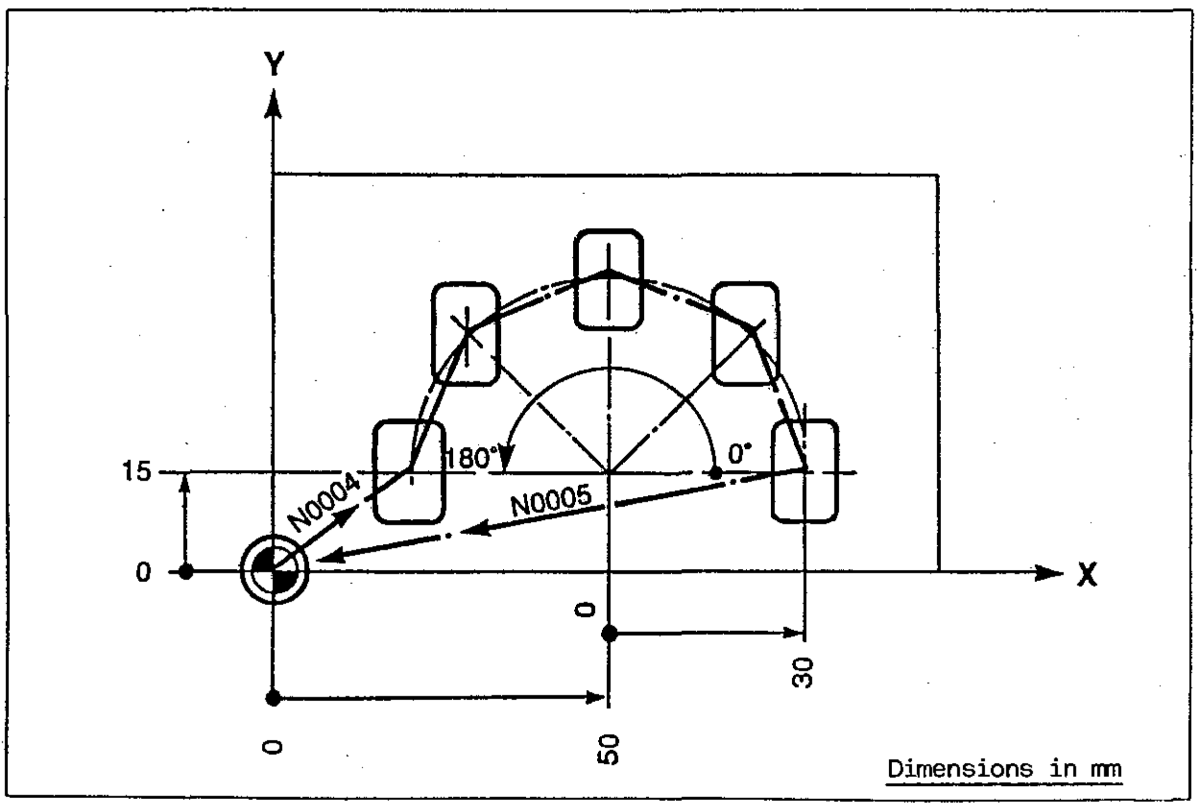
EXAMPLE
Rectangular pockets
on circular arc



%265	Program number
N1 G0 Z100 N2 G17 T1	Tool change. Assign length compensation to Z axis and call tool compensation.
N3 G0 Z2 S+1600	Downfeed to workpiece surface at rapid rate. Spindle started because following block contains G88 (patterns on circular arc).
N4 G88 N*1 L5 R30 G2 W180 M72 W0 I50 J15 G1	Machining of 5 pockets on circular arc, pockets programmed in subroutine N*1. Clockwise direction (G2), starting and final angle (M72).
N5 T0	Delete tool compensation.
N6 G0 X0 Y0 Z100 M30	Tool retracted in Z and positioned to point above program zero. End of program.
N*1 G72 F500 Y15 Y0 X-10 X-3 F100 Z-8 Z-3 Z-0	Subroutine defines the pocket milling cycle.

Patterns on circular arcs - G88



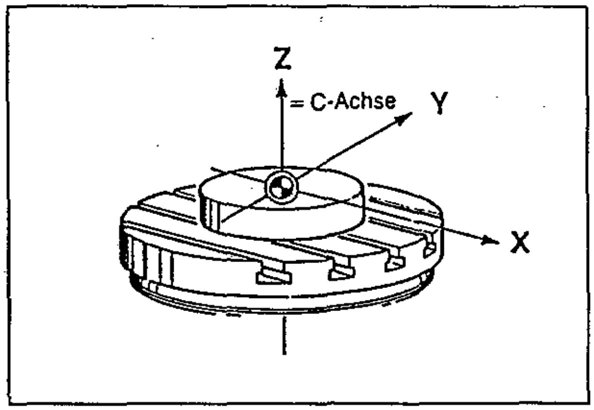


Coordinate system

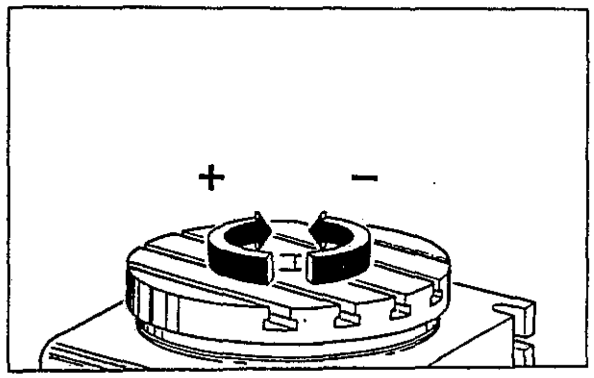
All machines of the FP-NC line can be used with an NC rotary table.

The rotary table can move at rapid traverse rate and at a programmed feed rate.

The axis of rotation is called C axis. The Z axis and the C axis are parallel (page 4-1). It is good practice to place the coordinate system in such a way that the Z axis coincides with the C axis.



The travel of the rotary table about the C axis is expressed in degrees, either in absolute dimensions or in chain dimensions. The angle of rotation is programmed under a C address.
The direction of rotation is always defined as viewed from above.



When chain dimensions (G91) are used:

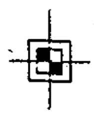
- positive sign indicates clockwise rotation,

- negative sign indicates counterclockwise rotation.

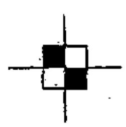
When absolute dimensions are used, the table rotates clockwise if the programmed angle is larger than the actual value, and counterclockwise if the programmed angle is smaller than the actual value.

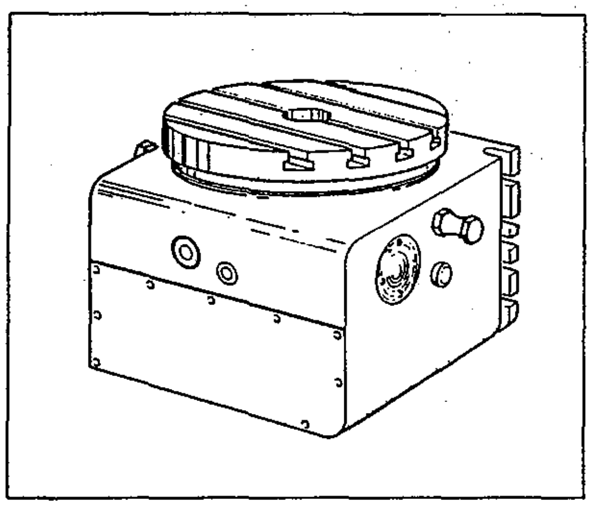
Setting up the rotary table

The reference point for the C axis (= 0° position) is defined during set-up of the machine (page 4-6), in the same manner as for all other axes.
Preferably use the angular position in which the rotary table is at its reference point. You are perfectly free, however, to choose any other position as a zero point of the C axis. The following symbol is used for the rotary table zero point:



To move the rotary table to its reference point, select mode 3 and press plus button (10) for table rotation on the control console: rotary table moves to its reference position with the T-slots parallel to the X axis.
The following symbol is used for the rotary table reference point:





Feed rate and radius in rotary milling

The NC rotary table can be rotated at the following rates:

- at rapid rate (2700 deg/min) and
- at feed rate (2 to 2000 deg/min).

At rapid rate, the tool can move simultaneously in all 4 axes.

At feed rate, the tool can move simultaneously about the C axis and in two linear axes (3 D interpolation).

Feed motions should always be accompanied by 'Spindle ON' in the program, otherwise the program will be interrupted.

The actual feed rate of the tool depends on the distance between the cutting edge of the tool and the C axis. This distance is the radius used for the milling operation on the rotary table (see illustration).

The actual feed (v) of the tool (mm/min) and the programmed feed rate F (mm/min) can be calculated as follows:

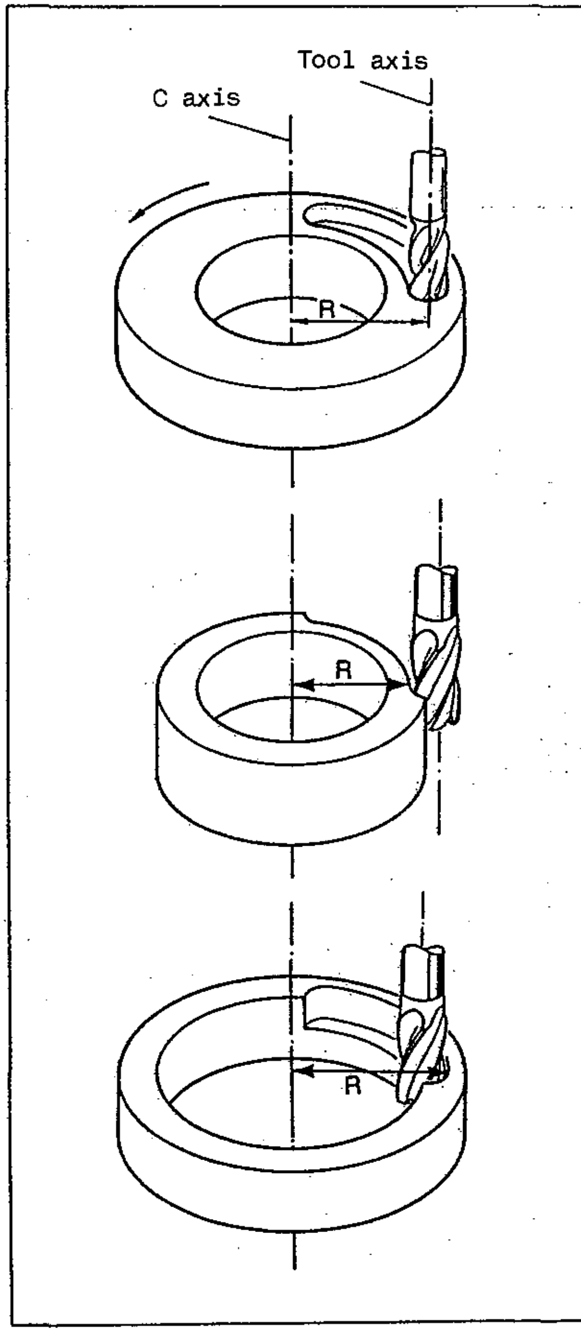
$$U = 2 \times R \times \pi$$

$$v = \frac{F \times U}{360°}$$

$$F = \frac{v \times 360°}{U}$$

where R = radius to be milled (mm)
U = circumference of circle to be milled (mm)

When milling spirals, the actual feed rate varies with the radius. To be on the safe side, always use the largest radius as a basis for calculating the feed rate.



Feed rate and radius in rotary milling

EXAMPLE

Calculating the feed rate

What is the feed rate F to be programmed for milling a circle with a radius of 100 mm, when the actual feed is to be 125 mm/min?

Solution: $U = 2 \times R \times \pi$

$U = 2 \times 100 \text{ mm} \times \pi = 628.3 \text{ mm}$

Insert into formula:

$$F = \frac{v \times 360°}{U}$$

$$F = \frac{125 \text{ mm/min} \times 360°}{628.3}$$

$$F = 71.6°/\text{min} \quad 72°/\text{min}$$

Feed rate to be programmed: F72.

EXAMPLE

Conversion of minutes and seconds of arc into decimals of degrees

12°14'12"

12°	: 1	=	12°
14'	: 60	=	0.2333°
12"	: 3600	=	0.0033°
			12.2366°

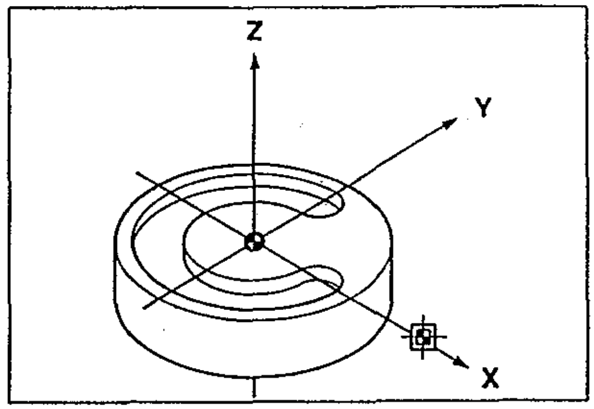
Value to be programmed: C+12237.

Milling operations on rotary table

EXAMPLE
Vertical spindle
Groove

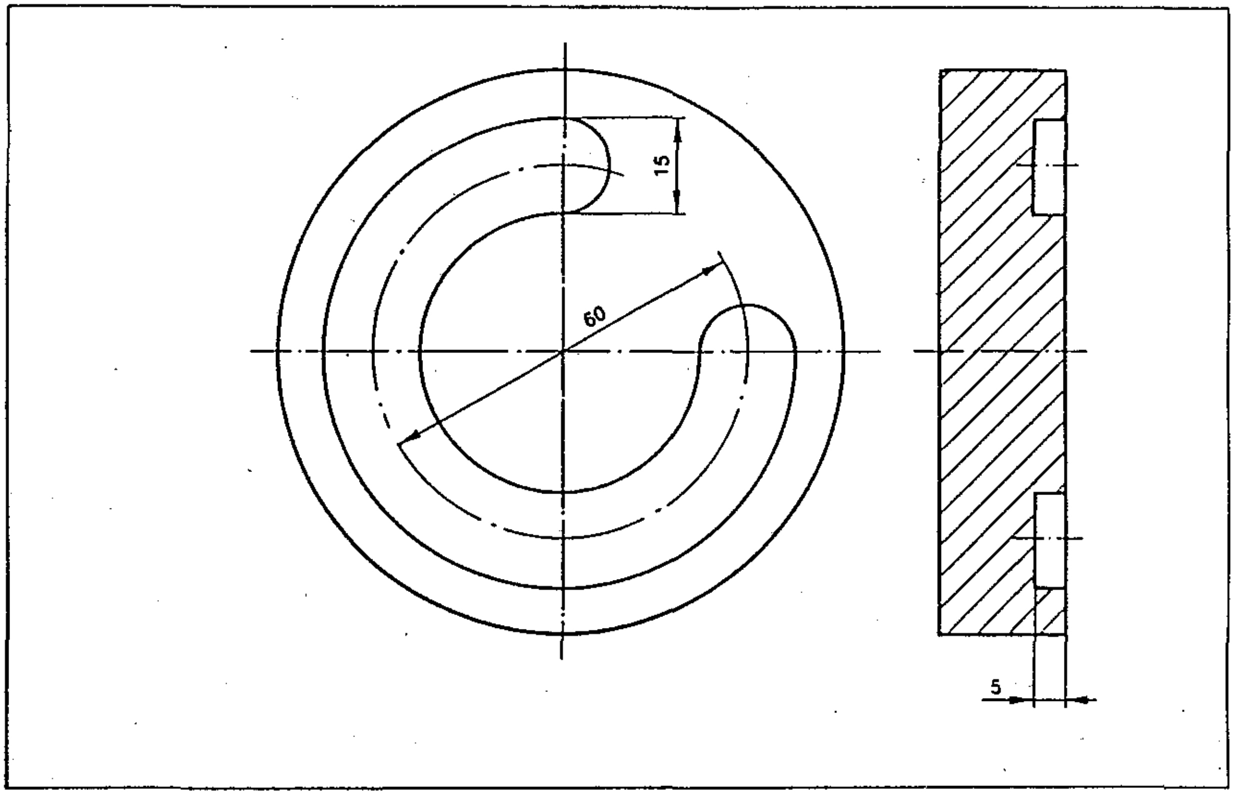
Align workpiece axis with rotary table axis.

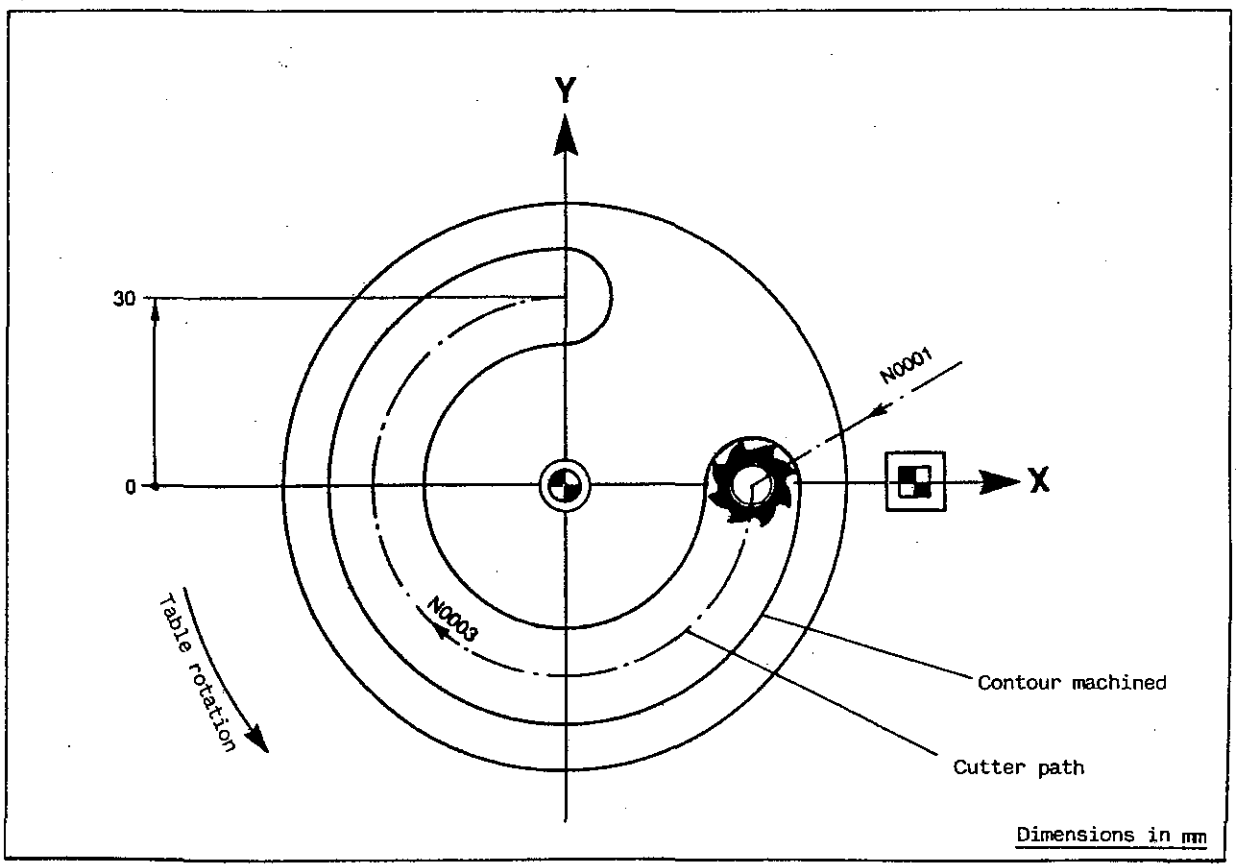
Use reference point as program zero.



%270	Program number
N1 G0 X30 Y0 Z2 C0 S+500	Tool moves to programmed position at rapid rate; rotary table simultaneously rotates to C 0 at rapid rate.
N2 Z-5 F35	Infeed in Z.
N3 C-270 F100	Table rotates clockwise through 270°: milling of groove.
N4 G0 Z100 S0	Tool retracted in Z.
N5 M30	End of program.

Milling operations on rotary table



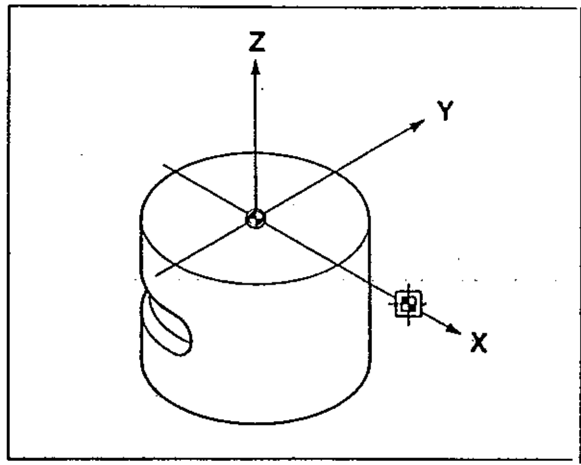


Milling operations on rotary table

EXAMPLE
Horizontal spindle
Groove

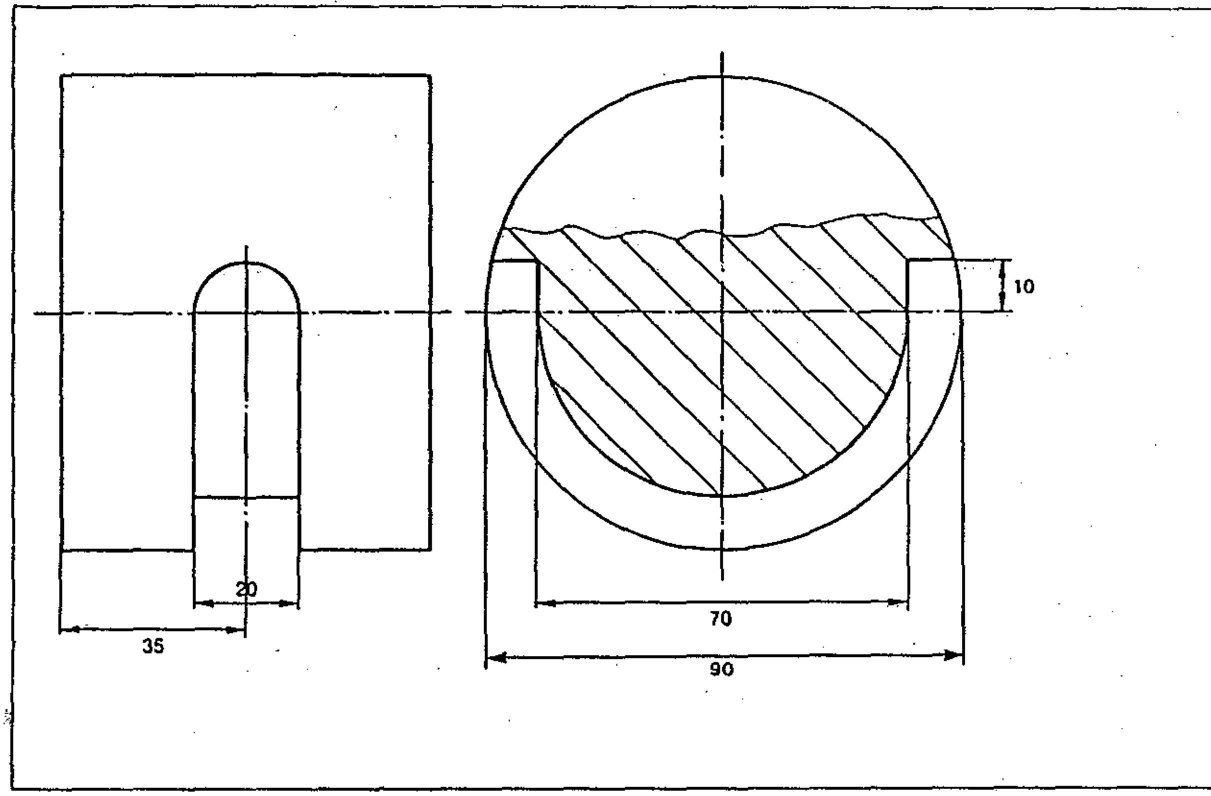
Align workpiece axis with rotary table axis.

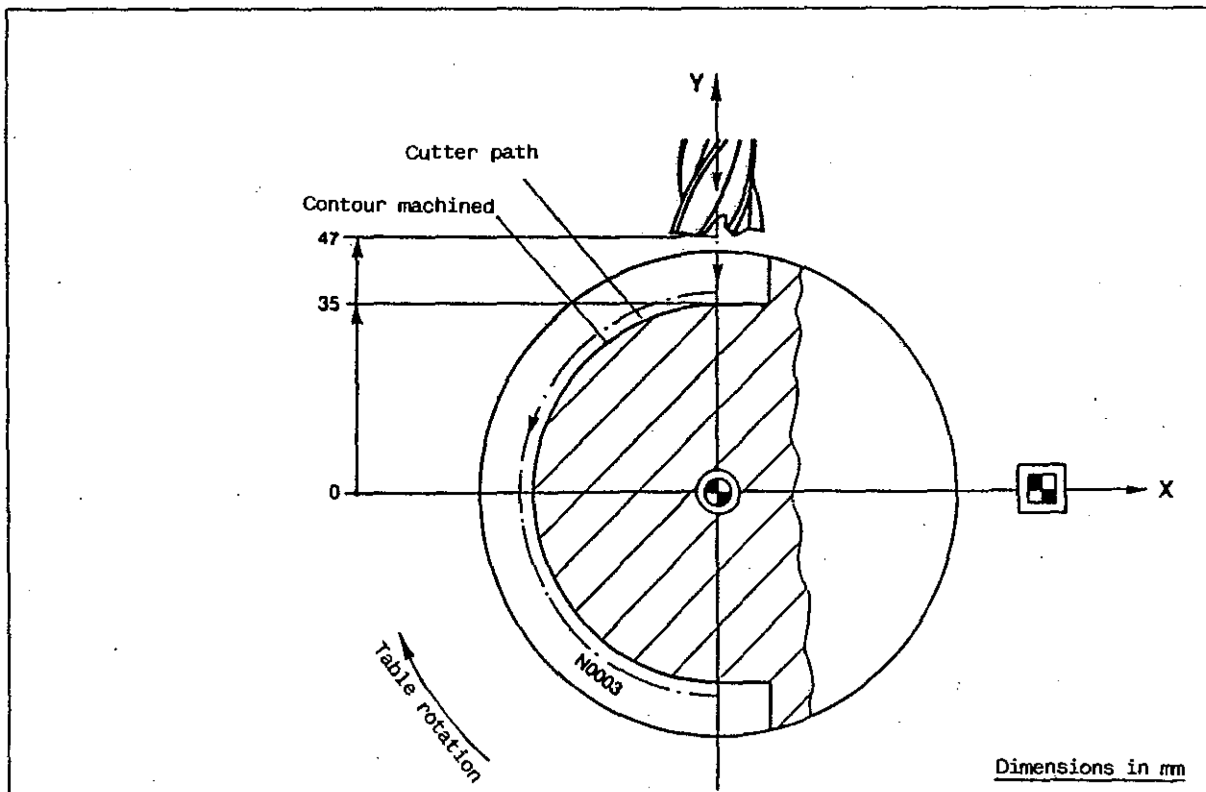
Use reference point as program zero.



%275	Program number
N1 G0 Y150 N2 G18 T1	Tool change. Assign length compensation to Y axis and call tool compensation.
N3 G0 X0 Y47 Z-35 C0 S+1000	Tool moves to programmed position at rapid rate; rotary table simultaneously rotates to C 0 at rapid rate.
N4 Y35 F50	Infeed in Y.
N5 C180 F125	Table rotates clockwise through 180°: milling of groove.
N6 T0	Delete tool compensation.
N7 G0 Y150 M30	Tool retracted in Y (horizontal spindle). End of program.

Milling operations on rotary table





Milling operations on rotary table

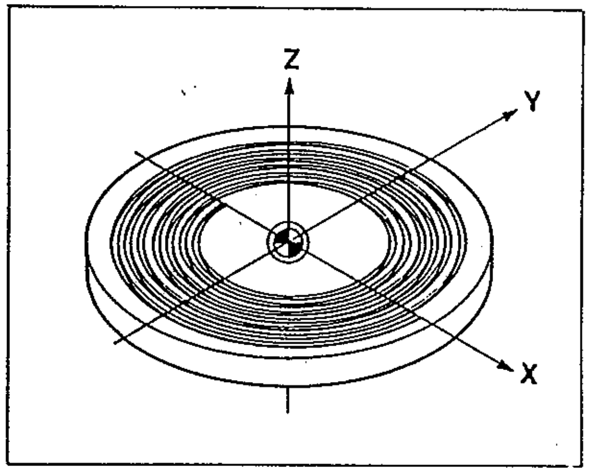
EXAMPLE
Vertical spindle
Spiral groove

Spiral lead	5.6 mm/360°
Outside diameter	165.0 mm
Starting point angle	0°
Inside diameter	92.2 mm
End point angle	180°
Number of rotations	6.5

For reasons of clarity the contour is not shown in the drawing (next page).

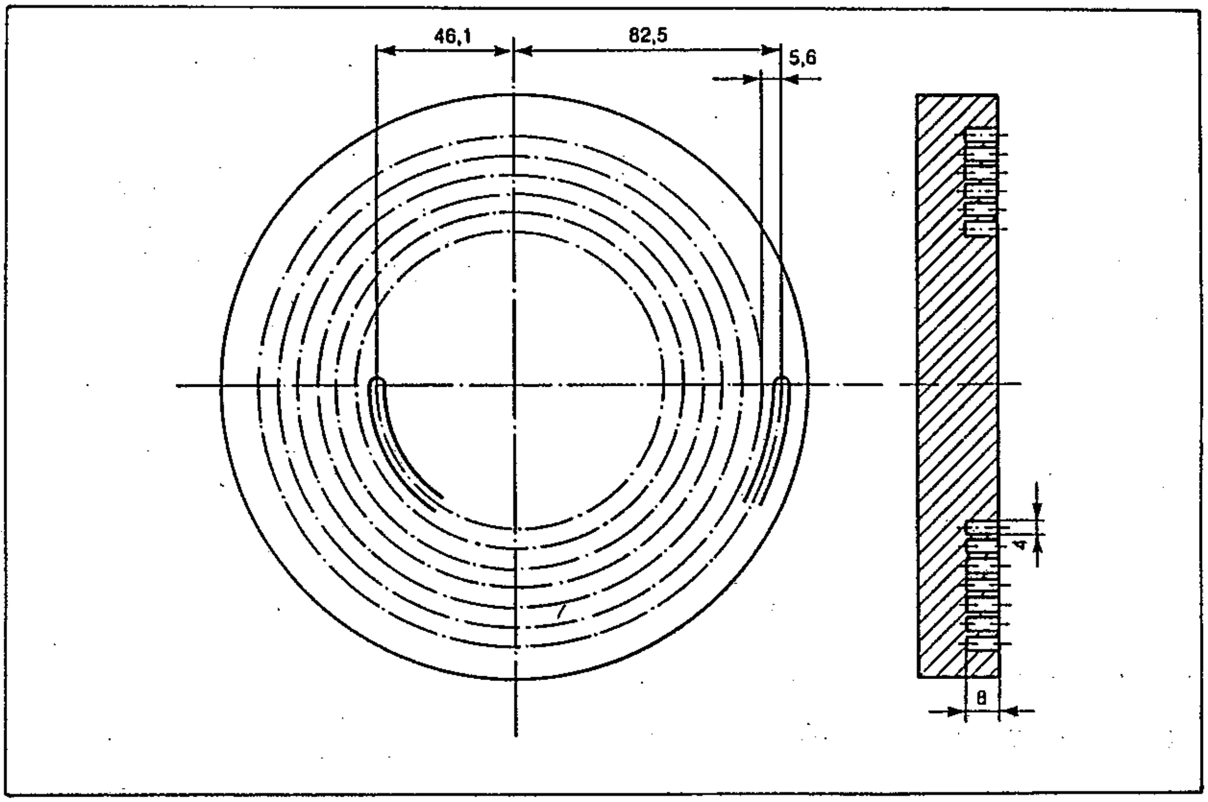
Align workpiece with rotary table axis.

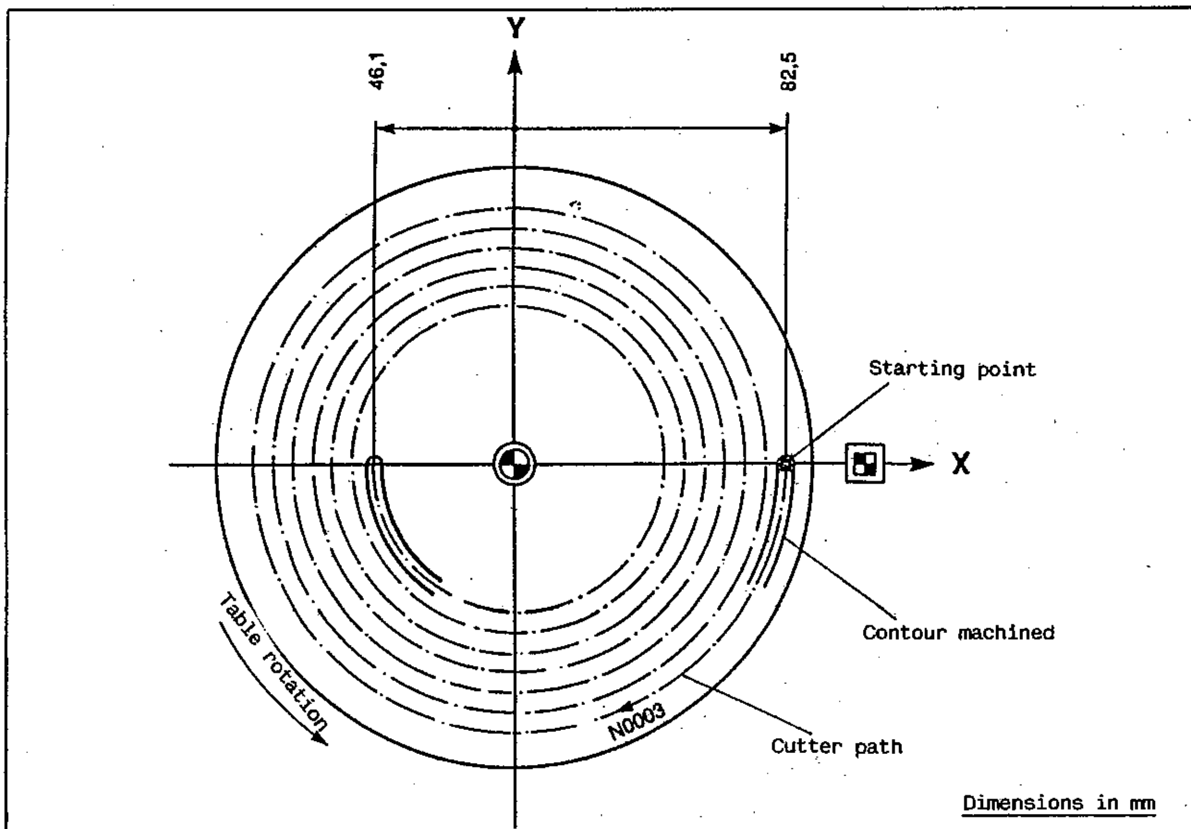
Use reference point as program zero.



% 280	Program number
N1 G0 X82.5 Y0 Z2 C0 S+2000	Tool moves to X82.5, Y0 and Z2 at rapid rate; rotary table simultaneously rotates to C0 at rapid rate.
N2 Z-8 F200	Infeed in Z.
N3 X46.1 C-2340 F300	Programming of spiral: end point coordinate Y46.1; angle of rotation C-2340, corresponding to 6.5 rotations. Table rotates counterclockwise (negative sign) to C-2340 (based on rotary table zero position): spiral groove is machined. $\frac{\text{OD radius} - \text{ID radius}}{\text{spiral lead}}$ = number of rotations $\frac{82.5 - 46.1}{5.6} = 6.5$
N4 G0 Z2	Tool retracted in Z.
N5 G0 X0 Y0 Z100 C0	Tool moves to Z100, while table rotates back to C 0. This means 6.5 rotations in a clockwise direction – the same number as previously in a counterclockwise direction.
N6 M30	End of program.

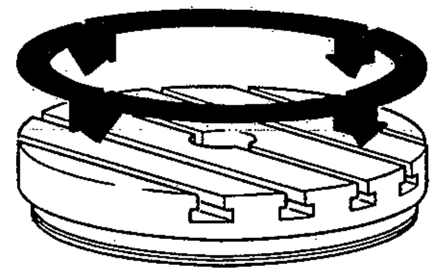
Milling operations on rotary table





Rotary index table

Rotary index table (FP2/3/4NC only)



The rotary index table (optionally available for FP2/3/4NC machines) rotates through a programmable angle between machining operations, but is standing still while cutting is in progress.

For the rotary motion of the table, the table top is lifted approx 5 mm out of engagement of the crown gear teeth.

Before rotating the table, always move the tool to a position where it cannot collide with the table or workpiece.

The indexing increments and indexing rate can be set on the rotary index table (page 10–18).
The following indexing increments are obtainable: 5°, 15°, 30°, 45°, 90°, 180°.
If the indexing increment (angle) is to be changed during a program cycle, a program interruption (M0) has to be programmed at the desired point.

Programming a rotary motion

The rotary index table can be rotated in only one direction: all angles programmed under a C address have to be positive.

Such angles can only be programmed in absolute dimensions (G90) and at rapid traverse rate (G0); they must be between 0° and 360°.

Programming in absolute dimensions (G90)

When absolute programming is used, the angles programmed under a C address refer to a fixed zero point.
Make it a rule always to use the reference point of the rotary index table as zero point.

Zero offsets (G54, G55. G56) are not permissible in operations with the rotary index table.

Rotary index table

Emergency stop during indexing operations

It may happen that an angle has been programmed under the C address, which is not obtainable with the previously set indexing increments. In this case, the table top rotates at the preset increments until it moves beyond the programmed angular position.
At this point, the control automatically initiates an emergency stop: the machine is switched off.

To complete the indexing operation, proceed as follows:

Select mode 5 and press plus button (10).

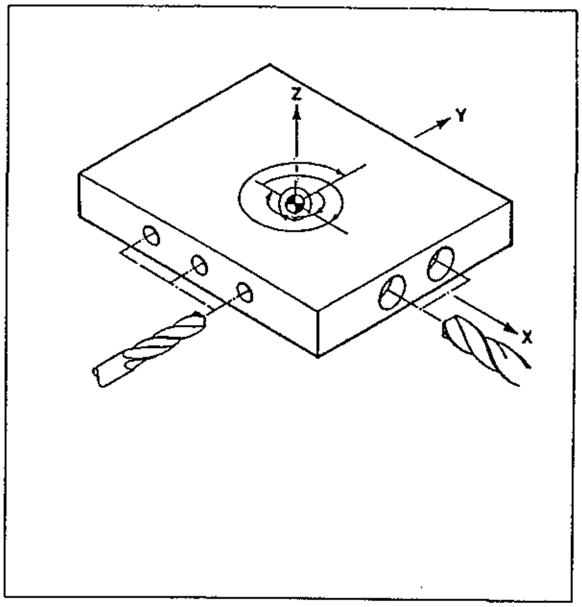
Always move to reference point (page 4–6) before resuming the work cycle after an 'Emergeny stop'.

Milling operations on rotary index table

EXAMPLE

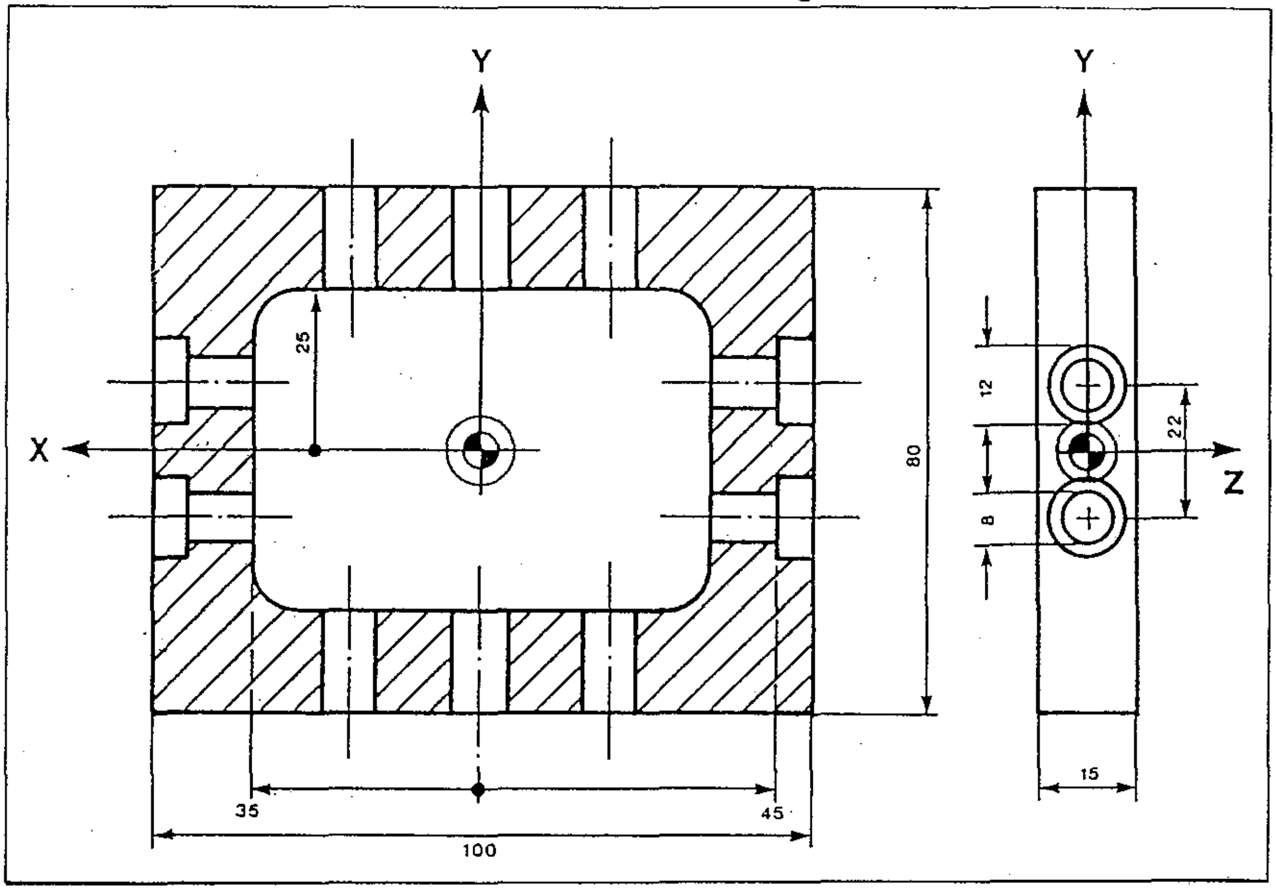
Align workpiece with table axis and with zero point of C axis.

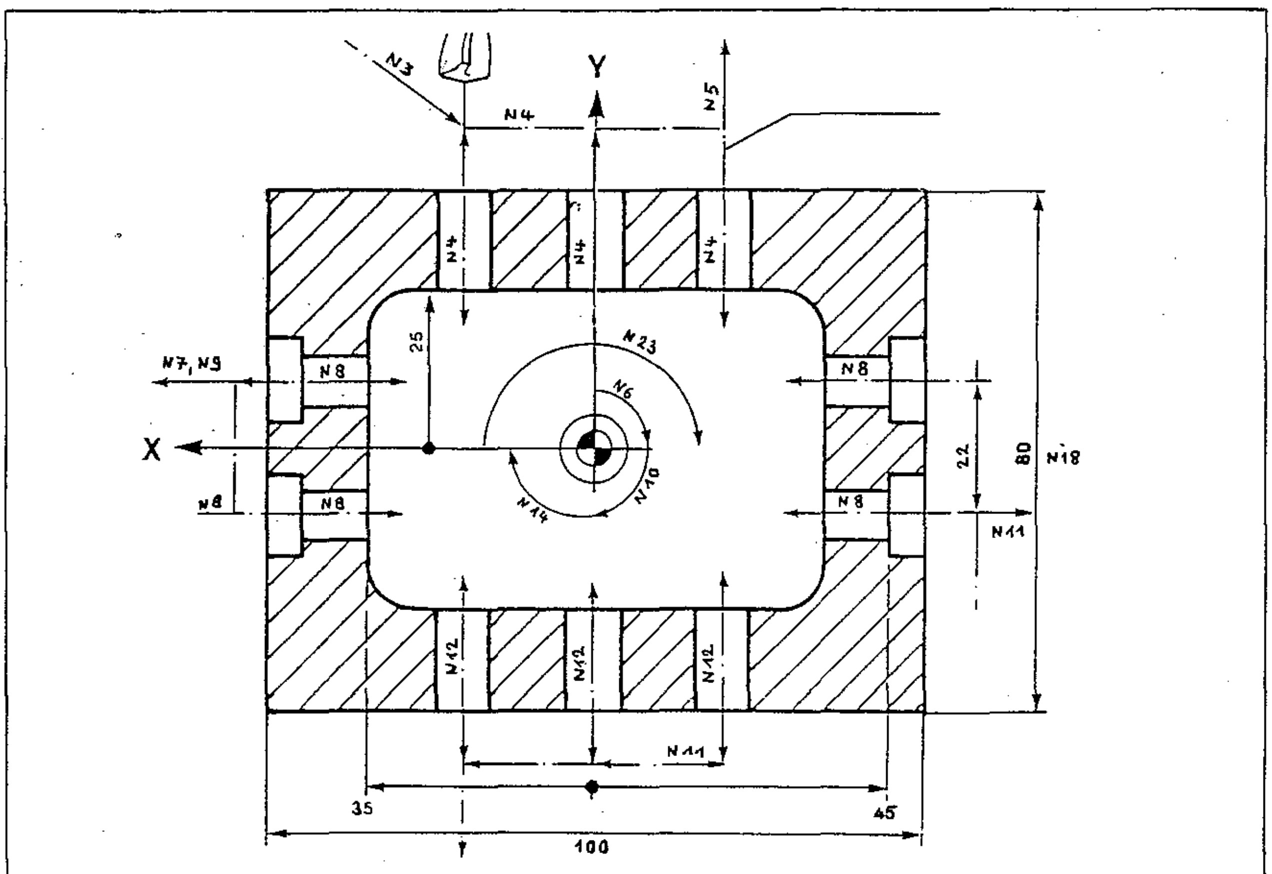
Set rotary index table for indexing increments of 90°.



%285	Program number
N1 G0 Y200 C 0 N2 G18 T1	Tool change to twist drill 8 mm dia. Assign length compensation to Y axis and call tool compensation.
N3 G0 X20 Y42 Z0 S+3150	Tool moves to 1st drilling position.
N4 G87 N*1 L3 X-20	Tool drills the first three holes of 8 mm dia.
N5 G0 Y100 N6 G0 C90	Tool retracted in Y; table rotates 90° clockwise.
N7 G0 X-11 Y52	Table moves to new drilling position (coordinate system does not rotate).
N8 G87 N*1 L2 X22	Tool drills the next two holes shown on left side of part drawing.
N9 G0 Y100 N10 G0 C180	Tool retracted in Y; table rotates 90° clockwise.
N11 G0 X20 Y42 N12 L1 N4	Tool drills the next three holes of 8 mm dia.
N13 G0 Y100 N14 G0 C270	Tool retracted in Y; table rotates 90° clockwise.
N15 G0 X-11 Y52 N16 L1 N8	Tool drills the last two holes shown on right side of part drawing.
N17 T0 N18 G0 Y200 N19 T2	Tool change to counterbore 12 mm dia.
N20 G0 X-11 Y52 Z0 S+2500	Tool moves to position.
N21 L1 N8 N8 N*2	Counterboring 12 mm dia.
N22 G0 Y100 N23 G0 C180	Table rotates 2 x 90° clockwise.
N24 G0 X-11 Y52 N25 L1 N8 N8 N*2	Tool moves to position; counterboring 12 mm dia.
N26 T0	Delete tool compensation.
N27 G0 Y200 M30	Tool retracted in Y axis; end of program.
N*1 G81 F300 S+3150 Y-22 N*2 G81 F160 S+2500 Y-7	Subroutine block: drilling. Subroutine block: counterboring.

Milling operations on rotary index table





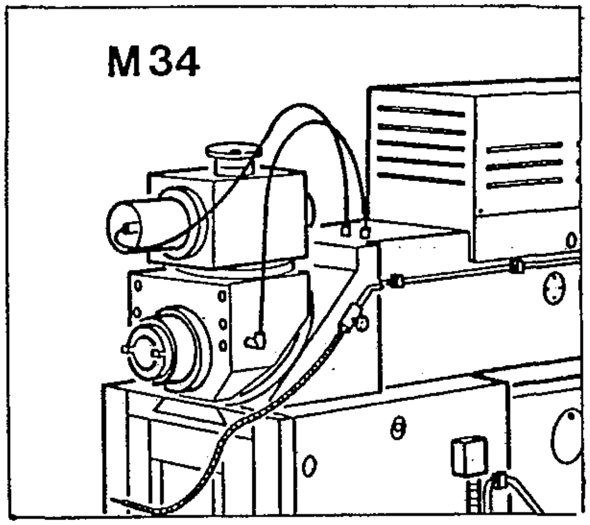
Tilting the turret head - M34, M35 (FP6/7NC only)

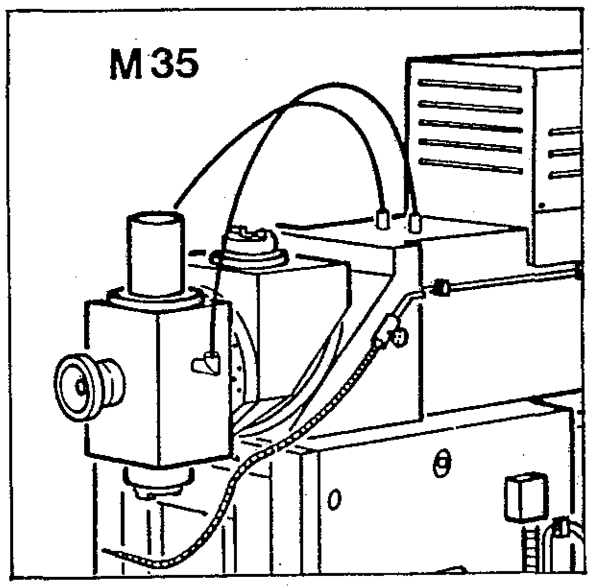
Commands M34 and M35 can be used for obtaining tilting motions of the turret head in program-controlled operation on FP6/7NC machines

M34: tilting the turret head to the position for horizontal milling;

M35: tilting the turret head to the position for vertical milling.

A block with command M34 or M35 must also contain the command spindle stop (S0).



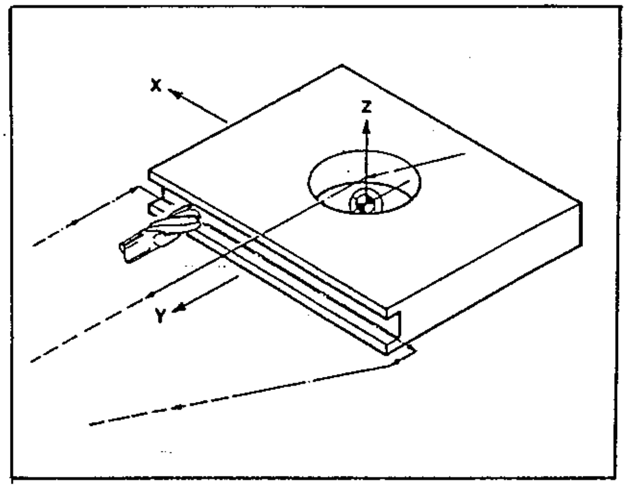


Notes & comments

Tilting the turret head – M34, M35

EXAMPLE

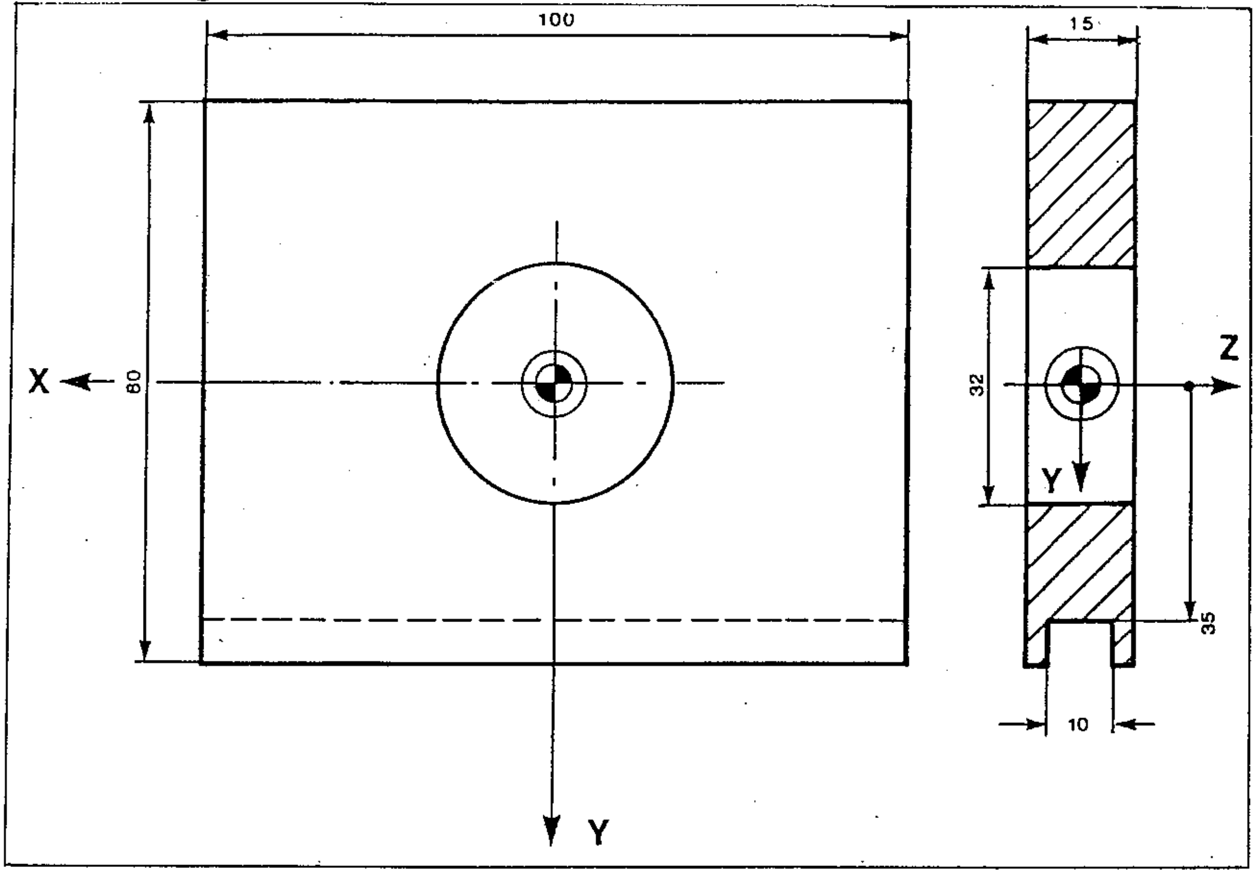
For the following example, clamp the two cutting tools in the two spindles and set up the machine with each of the two spindles properly aligned.
The machining operation begins with the vertical spindle.

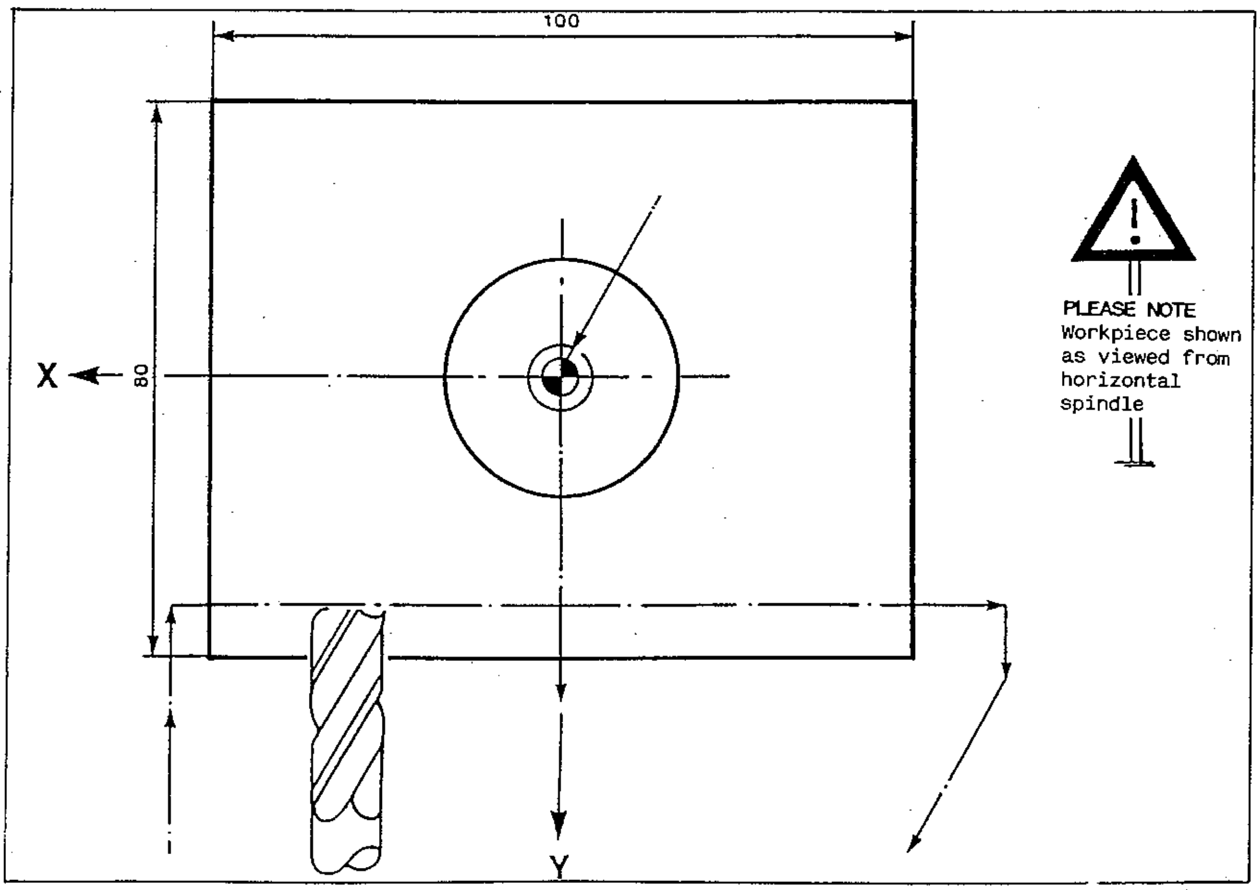


PLEASE NOTE
Workpiece shown as viewed from horizontal spindle

%290	Program number
N1 G0 Z200 N2 M35	Retract tool, move turret head to position for vertical milling.
N3 G0 X0 Y0 Z9.5 D1	Tool (end mill 32 mm dia) moves to workpiece zero point, spindle clockwise start.
N4 G81 F100 S+1000 Z-19	Call drilling cycle G81: tool drills to specified depth.
N5 G0 Y200 Z200 S0	Tool moves to tilting position.
N6 M34	Tilt turret head to position for horizontal milling.
N7 G56 Y160 Z-130 N8 G0 X57 Y42 D2 Z0 S+1800	Call new zero offset, tool moves to starting point for milling the groove.
N9 G0 Y35 D2 F200 N10 X-57 F180	Infeed in Y axis, milling of groove.
N11 G0 Y42 D2	Retract tool at rapid rate.
N12 G53	Delete zero offset.
N13 L1 N1	Repeat N1.
N14 M30	End of program.

Tilting the turret head





Addresses

A	Distance in approach and departure commands/ distance of polar coordinates
C	Angle of rotation of NC rotary table and NC index head
D	Tool compensation number
F	Feed rate
G	Preparatory function
G4F	Dwell
I	Centre-of-circle coordinate/polar coordinate in X
J	Centre-of-circle coordinate/polar coordinate in Y
K	Centre-of-circle coordinate/polar coordinate in Z
L	Program part repetition/number of positions
L0 N	Unconditional jump
M	Miscellaneous or switching functions
N	Block number
N*	Subroutine block number
P	Program number
R	Rounding radius/length of bevel/radius of circular arc
S	Spindle speed
T	Tool change, tool store
T0	Deletion of tool compensation
W	Angle
X	X coordinate
Y	Y coordinate
Z	Z coordinate
%*	Local subroutine number
%0*	Macro number

G functions (Preparatory functions)

G0	Rapid traverse rate
G1	Linear interpolation
G2	Circular interpolation clockwise
G3	Circular interpolation counterclockwise
G7	Rounding of corners
G8	Bevelling of corners
G9	Input in polar coordinates
G17	Length compensation in +Z axis
G18	Length compensation in +Y axis
G19 * 1	Length compensation in +X axis
G19 * 2	Length compensation in +X axis
G40	Deletion of compensation on contour
G41	Call of compensation on contour (tool left of contour)
G42	Call of compensation on contour (tool right of contour)
G45	Approach and departure command (parallel to contour)
G46	Approach and departure command (semi-circle)
G47	Approach and departure command (quarter circle)
G52	Moving to reference point
G53	Return to original coordinate system
G54	Zero set of actual value
G55	Linear and rotary offset of coordinate system (additive)
G56	Linear and rotary offset of coordinate system (absolute)

G functions
(Preparatory functions)

* G60	Exact stop
* G61	Automatic rounding of internal corners (larger radius)
* G64	Blending of contours (smooth transitions)
G71	Milling of rectangular pockets (roughing, conventional)
G72	Milling of rectangular pockets (roughing, climb and conventional)
G72 * 1	Milling of rectangular pockets (roughing, climb and conventional), corner radius programmable
G73	Milling of rectangular pockets (finishing to size)
G74	Milling of rectangular pockets (finishing to size with intermediate stop)
G74 * 2	Milling of rectangular pockets (finishing to size with intermediate stop), corner radius programmable
G75	Milling of pins
G76	Milling of circular pockets
G77	Milling of external threads
G78	Milling of internal threads
G79	Milling of contour pockets
G81	Drilling
G82	Drilling with intermittent infeed
G83	Deep drilling
G84	Tapping
G85	Reaming
G86	Boring, tool retracted with spindle stopped
G87	Hole patterns
G88	Patterns on circular arcs
* G90	Absolute dimensions
* G91	Chain dimensions

* MODAL

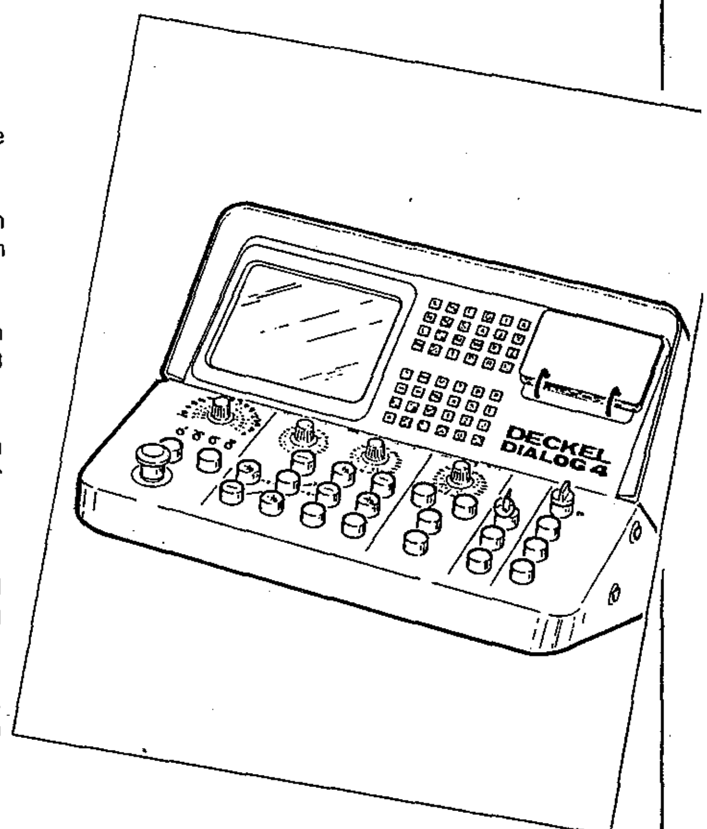
M functions
(Miscellaneous and switching functions)

M0	Program stop
M2	End of program
M7	Lube pulse
M8	Coolant on
M9	Coolant off
M10	Clamps in all axes applied
M11	Clamps in all axes released (= position control)
M20	Clamps in X axis applied
M21	Clamps in X axis released (= position control)
M22	Clamps in Y axis applied
M23	Clamps in Y axis released (= position control)
M24	Clamps in Z axis applied
M25	Clamps in Z axis released (= position control)
M26	Clamps in C axis applied
M27	Clamps in C axis released (= position control)
M30	End of program with rewind to program start
M34	Tilt turret head to position for horizontal milling (FP6/7NC only)
M35	Tilt turret head to position for vertical milling (FP6/7NC only)
* M60	Constant feed rate along contour (tool edge)
* M61	Constant feed rate along contour (tool edge) on internal corners and straight lines, deceleration on external corners
* M62	Constant feed rate of cutter axis
M70	Block skipped and activated after compensation on contour
M71	Angles, incremental
M72	Angles, absolute
M78	End of contour pocket with downfeed to last cutting depth
M79	End of contour pocket with return to retraction plane
M80	Delete mirror imaging
M81	Mirror imaging (sign change X/I)
M82	Mirror imaging (sign change Y/J)
M83	Mirror imaging (sign change Z/K)
M84	Mirror imaging (sign change X/I and Y/J)
M85	Mirror imaging (sign change X/I and Z/K)
M86	Mirror imaging (sign change Y/J and Z/K)

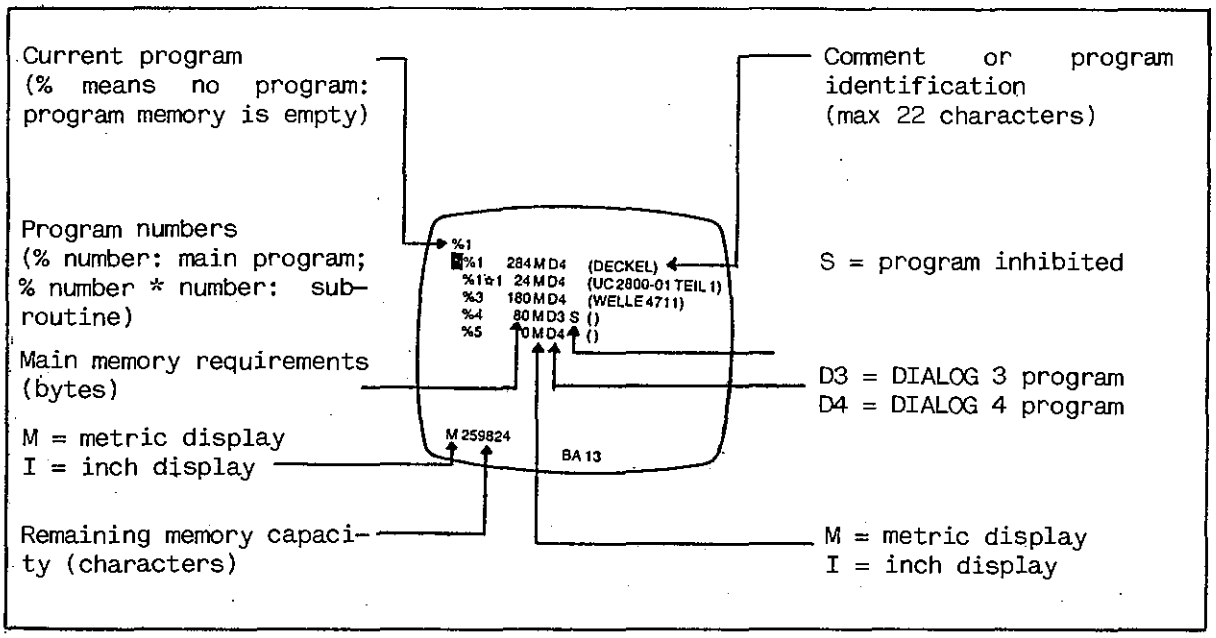
7. Handling the control unit – program controlled operation

In this chapter you will find:

- how to manage your programs in the control unit (pages 7-1 to 7-12);
- how to program a machining operation without using the coordinate system ('Playback', see pages 7-13 to 7-17);
- how to enter a program or change an already existing program (pages 7-18 to 7-34);
- how to enter tool compensations in tool store T (pages 7-35 to 7-40) or tool compensation store D (pages 7-41 to 7-45);
- how to determine the spindle speed and feed rate with the aid of the cutting data calculator (pages 7-46 to 7-48);
- how to store your programs and tool compensations outside the control, on cassettes or punched tape (pages 7-49 to 7-61);
- how to run a program and what to do if a machining operation is interrupted (pages 7-62 to 7-71);
- how to check your programs with screen graphics (pages 7-72 to 7-86);
- how to adjust the measuring system, interfaces and spindle speed indication (pages 7-87 to 7-91).



Explanatory notes on program list

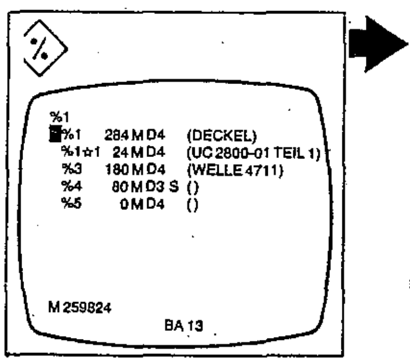


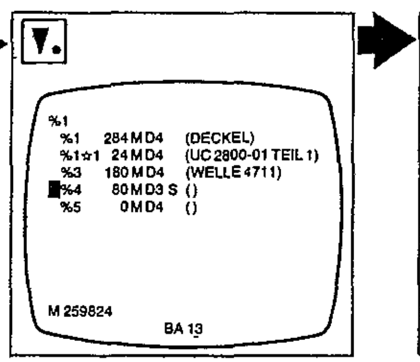
Checking the program contents
- Program list

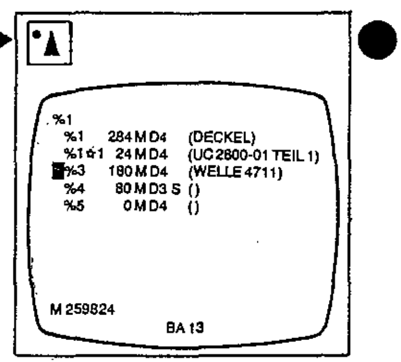
Select mode 13.

Moving down the program list:
use 'Cursor down' key.

Moving up the program list:
use 'Cursor up' key.







Display screen shows program list.

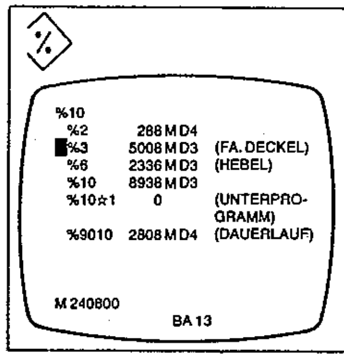
Cursor moves one step down each time key is pressed.
If more programs are stored than can be shown on screen, display will move on when cursor has reached screen bottom.

Cursor moves one step up each time key is operated.

Checking the program contents
- Macro list

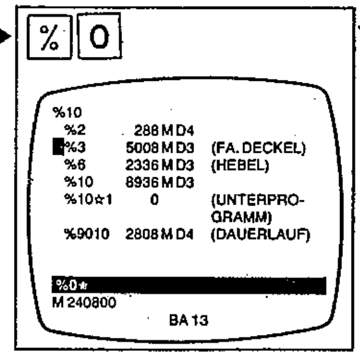
DIALOG 4 differentiates between macros and local subroutines. Macros can be called from any program.
Local subroutines belong to one program and can only be called within such program (page 6-56).

Select mode 13.



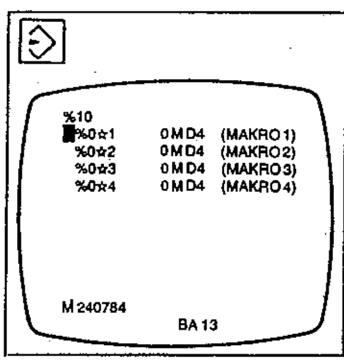
Display screen shows program list.

Press '%' and '0' keys.



'%0*' appears on input line.

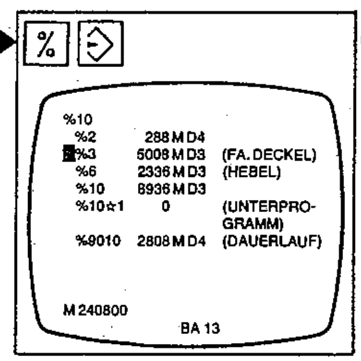
Press 'Transfer' key.



Display screen shows macro list.

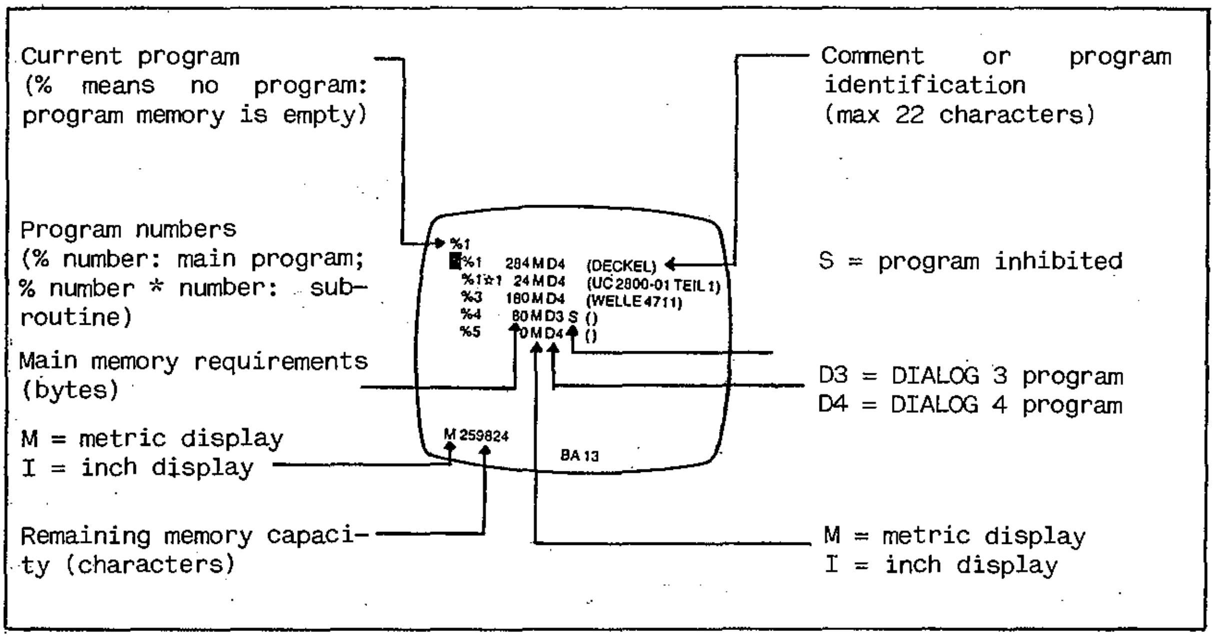
Moving up and down the macro list: same procedure as described for program list (page 7-1).

To return to program list, press '%' and 'Transfer' keys.



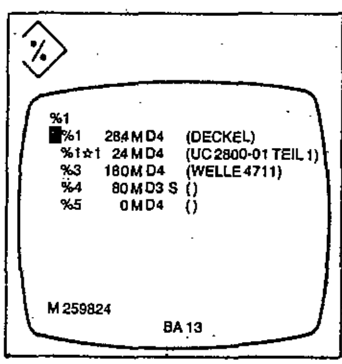
Display screen again shows program list.

Explanatory notes on program list



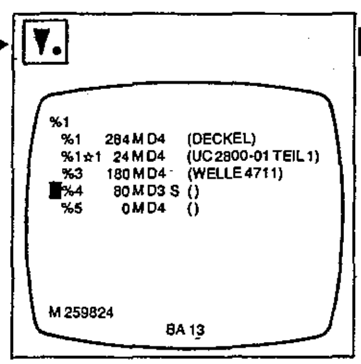
Checking the program contents
– Program list

Select mode 13.



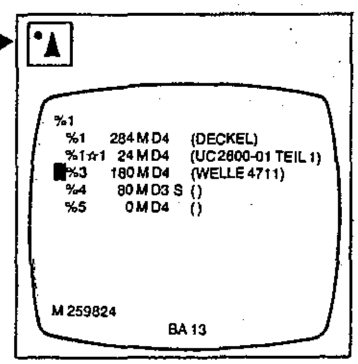
Display screen shows program list.

Moving down the program list:
use 'Cursor down' key.



Cursor moves one step down each time key is pressed.
If more programs are stored than can be shown on screen, display will move on when cursor has reached screen bottom.

Moving up the program list:
use 'Cursor up' key.

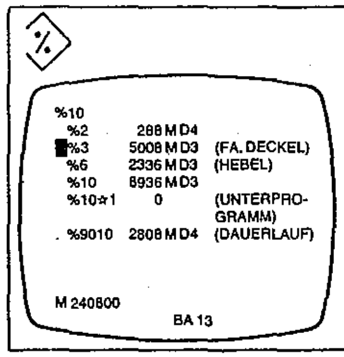


Cursor moves one step up each time key is operated.

Checking the program contents
- Macro list

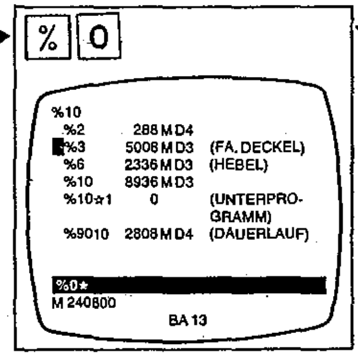
DIALOG 4 differentiates between macros and local subroutines. Macros can be called from any program.
Local subroutines belong to one program and can only be called within such program (page 6-56).

Select mode 13.



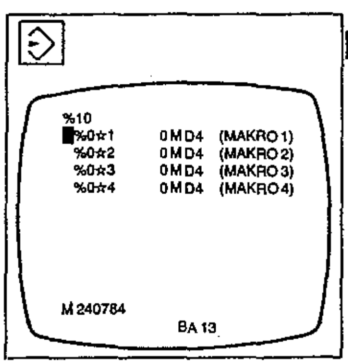
Display screen shows program list.

Press '%' and '0' keys.



'%0*' appears on input line.

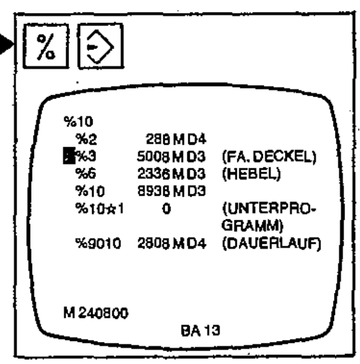
Press 'Transfer' key.



Display screen shows macro list.

Moving up and down the macro list: same procedure as described for program list (page 7-1).

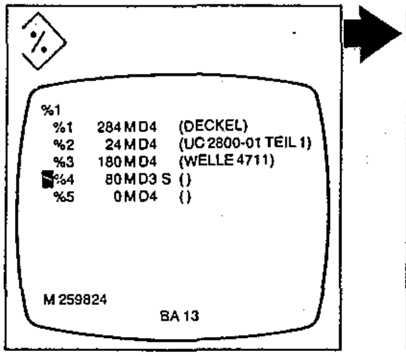
To return to program list, press '%' and 'Transfer' keys.



Display screen again shows program list.

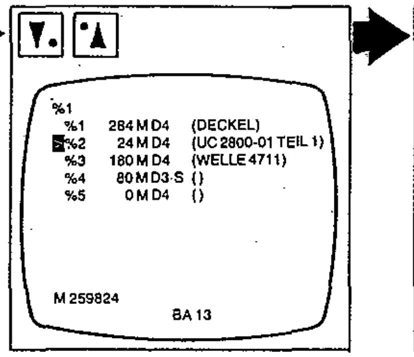
Selecting a program

Select mode 13.

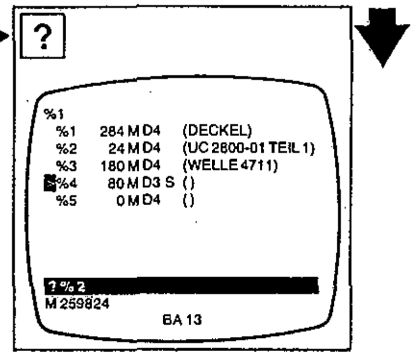


Display screen shows program list.

Either:
use 'Cursor up/down' key until cursor is placed at desired program.

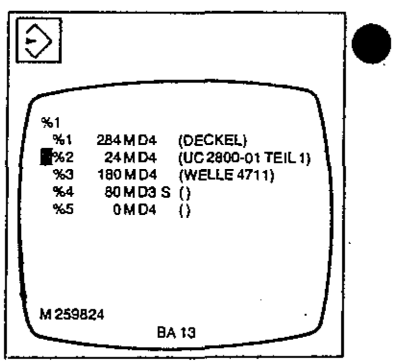


Or:
Press '?' key and enter desired program number.



Program number entered appears on input line.

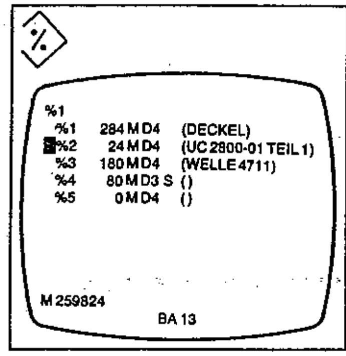
Press 'Transfer' key.



Cursor moves to desired program number.

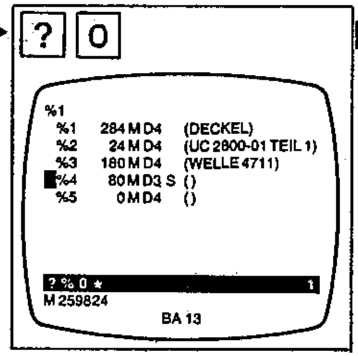
Selecting a macro

Select mode 13.



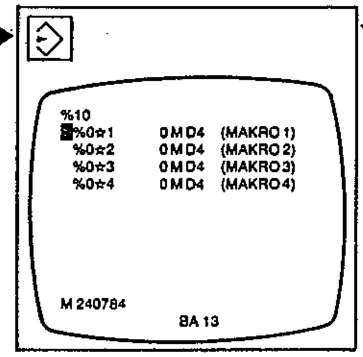
Display screen shows program list.

Either:
press '?' and '0' keys and enter number of desired macro.



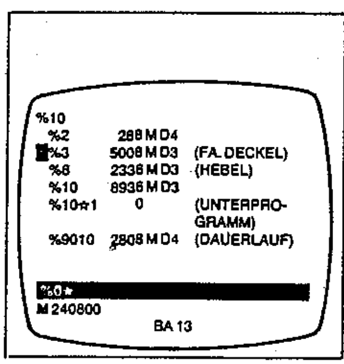
Macro number entered appears on input line.

Press 'Transfer' key.



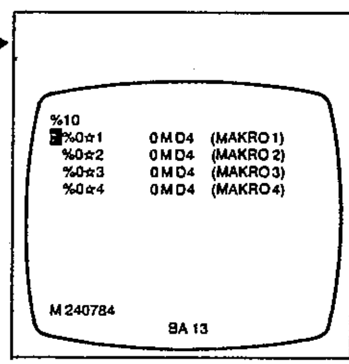
Cursor moves to desired macro number.

Or:
Press '%' and '0' keys.



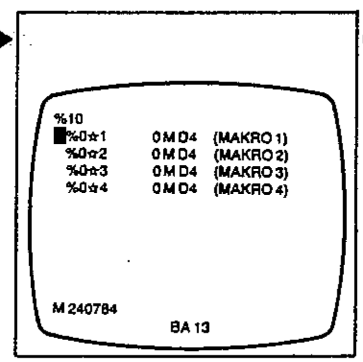
'%0*' appears on input line.

Press 'Transfer' key.



Display screen shows macro list.

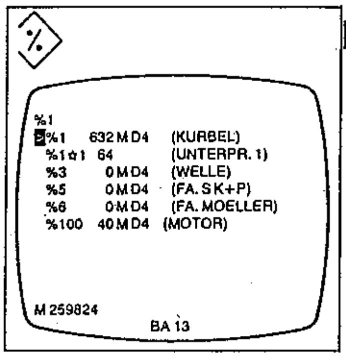
Use 'Cursor up/down' key until cursor is placed at desired macro number.



Cursor moves to desired macro number.

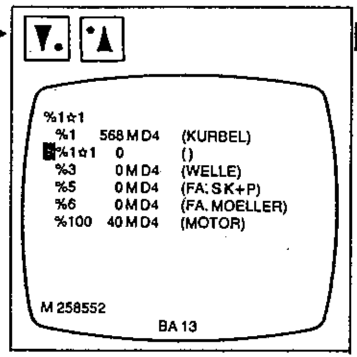
Selecting a local subroutine

Select mode 13.

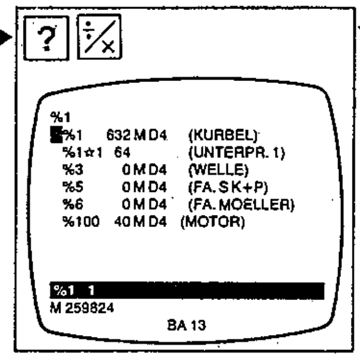


Display screen shows program list.

Either:
use 'Cursor up/down' key until cursor is placed at desired subroutine number.

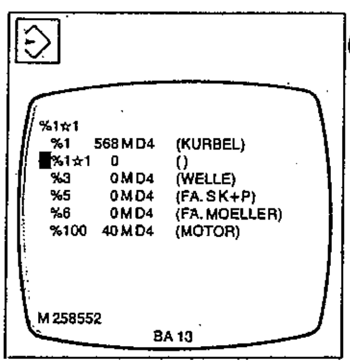


Or:
press '?' and 'x' keys and enter number of desired subroutine.



Subroutine number entered appears on input line.

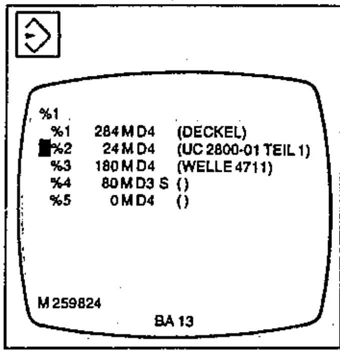
Press 'Transfer' key.



Cursor moves to desired subroutine number.

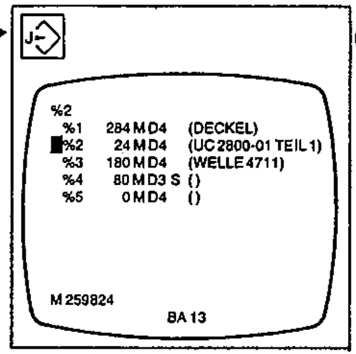
Transferring a stored program to the main memory

In mode 13, select program (page 7-3).



Cursor moves to desired program number.

To retrieve program marked by cursor, press 'Acknowledgment' key.

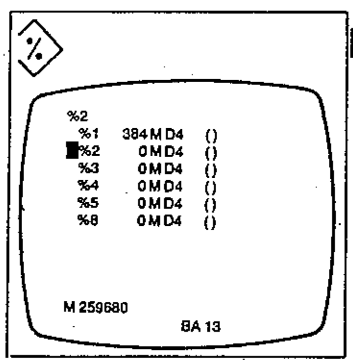


Program transferred to main memory appears in lower left.

In exchange, the program previously contained in the main memory is transferred to the program store. The main memory of the control now only contains the program just called.

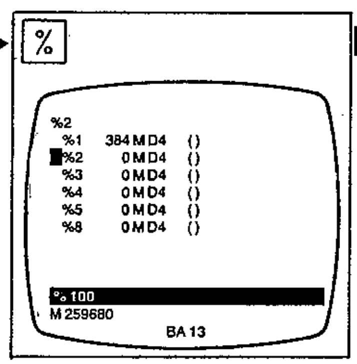
Opening a new program

Select mode 13.



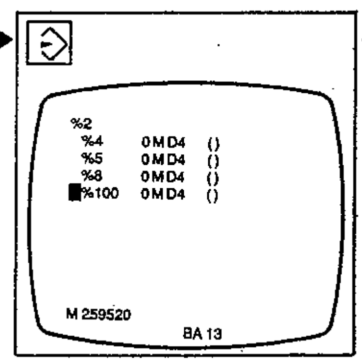
Display screen shows program list.

Press '%' key and enter new program number.



Program number entered appears on input line.

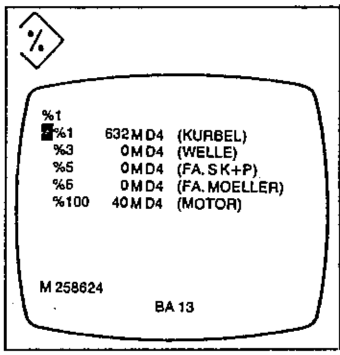
Press 'Transfer' key.



Program number appears in program list. Should a program already have been stored under that number, a message will appear in lower right of screen. Old program will not be cancelled.

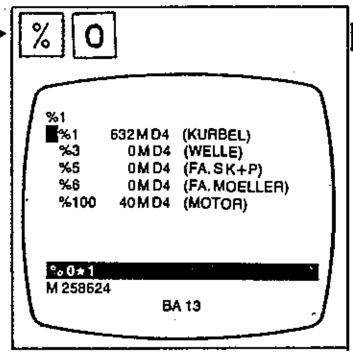
Opening a macro

Select mode 13.



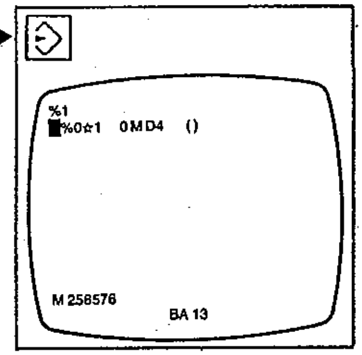
Display screen shows program list.

Press '%' and '0' keys and enter macro number.



Macro number entered appears on input line.

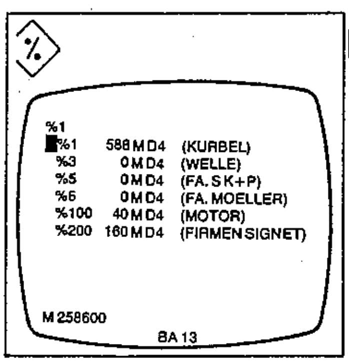
Press 'Transfer' key.



Macro number appears in macro list.

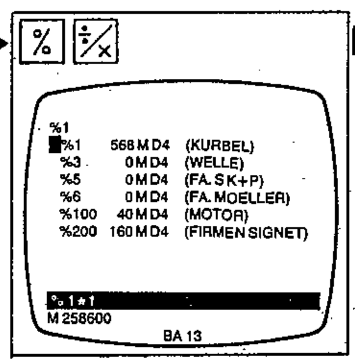
Opening a local subroutine

Select mode 13.



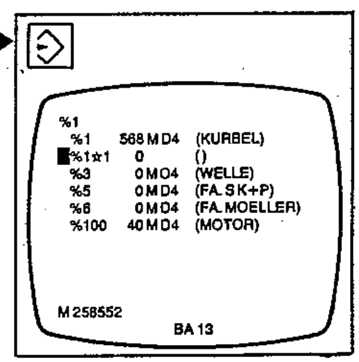
Display screen shows program list.

Press '%' and 'x' keys and enter subroutine number.



Subroutine number appears on input line.

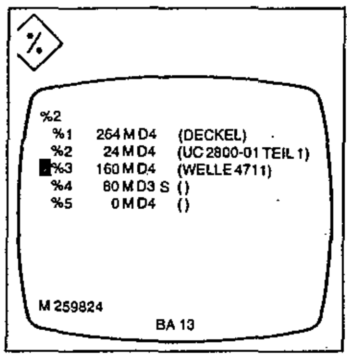
Press 'Transfer' key.



Subroutine number appears in program list underneath associated program number.

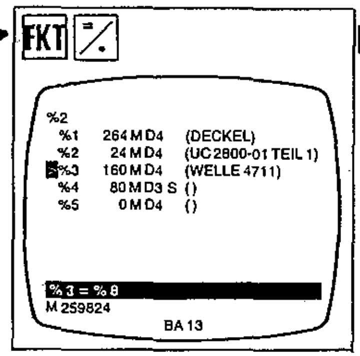
Changing a program number

In mode 13, select program (page 7-3).



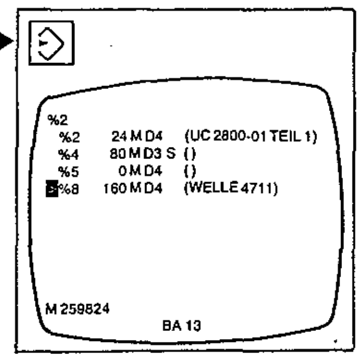
Cursor moves to desired program number.

Press '=' key and enter new program number.



Old and new numbers, connected by '=', appear highlighted on display screen.

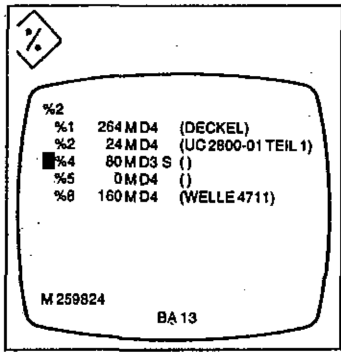
Press 'Transfer' key.



New number is now shown in program list.

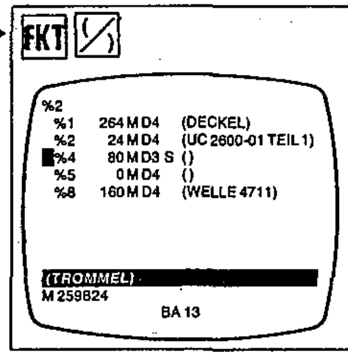
Entering a comment

In mode 13, select program or macro or subroutine (page 7-3).



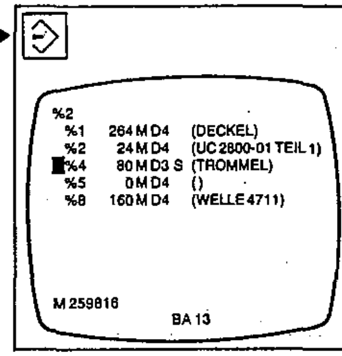
Cursor moves to number where comment is to be added.

Press '(' key, enter comment, then press ')' key.



Comment appears on input line.

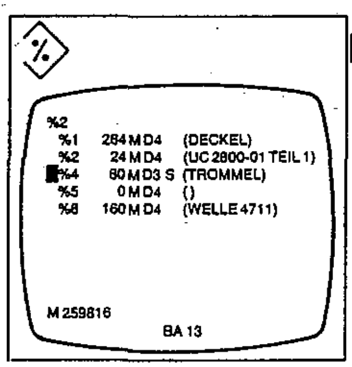
Press 'Transfer' key.



Program list now shows comment adjacent to number selected.

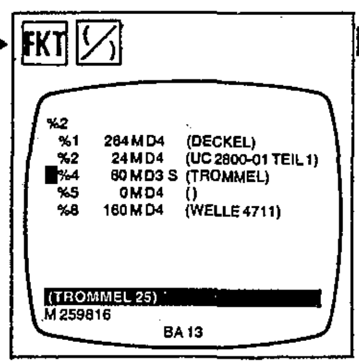
Changing a comment

In mode 13, select program or macro or subroutine (page 7-3).



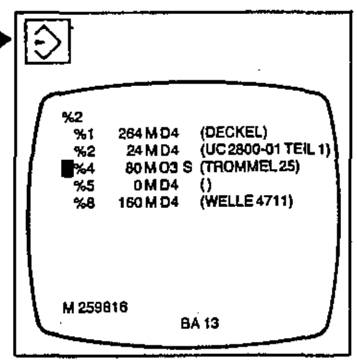
Cursor moves to number where comment is to be changed.

Press '(' key, enter new comment, then press ')' key.



New comment appears on input line.

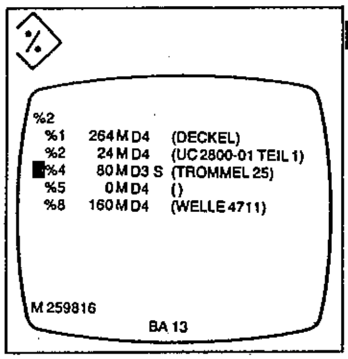
Press 'Transfer' key.



Program list now shows new comment.

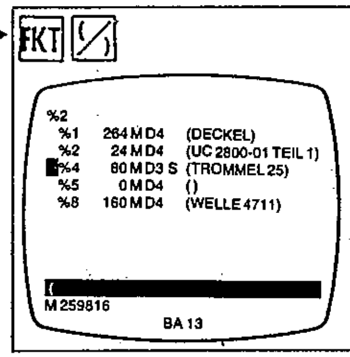
Deleting a comment

In mode 13, select program or macro or subroutine (page 7-3).



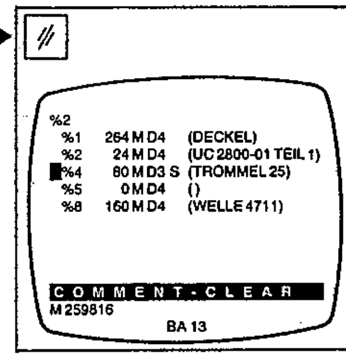
Cursor moves to number where comment is to be deleted.

Press '(' key.



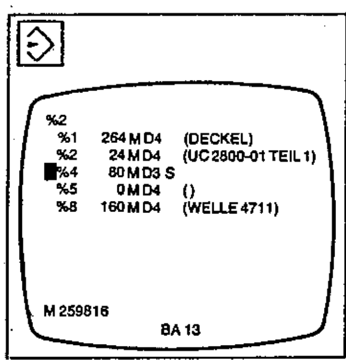
'(' appears on input line.

Press 'Delete' key.



'COMMENT CLEAR' appears on input line.

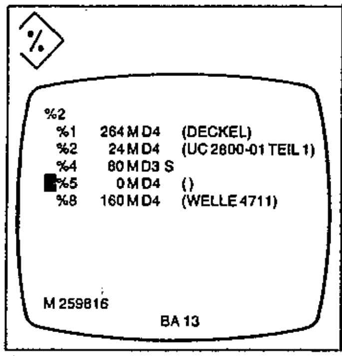
Press 'Transfer' key.



Program list now shows program number without comment.

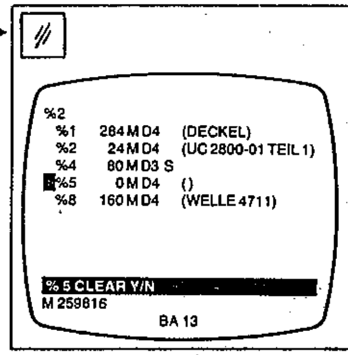
Cancelling a program

In mode 13, select program (page 7-3). Set key lock switch 'Program lock' (16) on NC keyboard to vertical position.



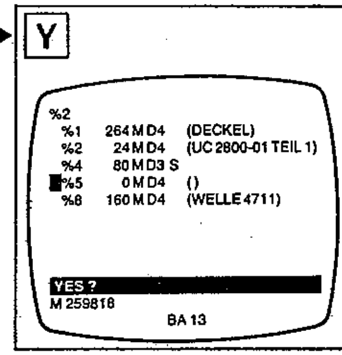
Cursor moves to program number selected.

To cancel program, press 'Delete' key.



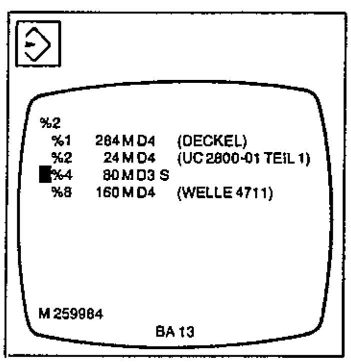
'CLEAR Y/N' appears adjacent to program number on input line, but program is not yet cancelled.

If you are certain you wish to cancel the program, press 'Y' for YES (otherwise 'N' for NO).



'YES' appears on input line, but program is not yet cancelled.

Press 'Transfer' key.



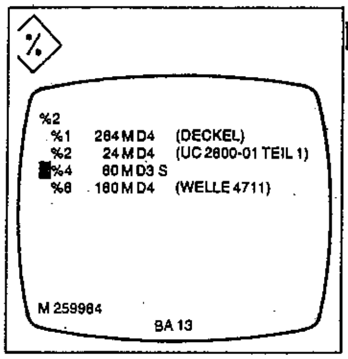
Program disappears from program list.

The program, together with local subroutines (if any) and tool compensations contained in the program, has now been erased from the program memory.

To cancel a macro, or a local subroutine alone, select the item concerned (pages 7-4 or 7-5) and proceed in the same manner.

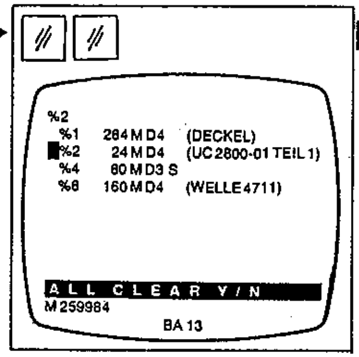
Cancelling all programs

Select mode 13.
Set key lock switch 'Program lock' (16) on NC keyboard to vertical position.



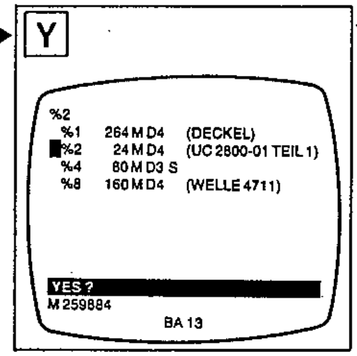
Display screen shows program list.

To cancel all programs, press 'Delete' key 2x.



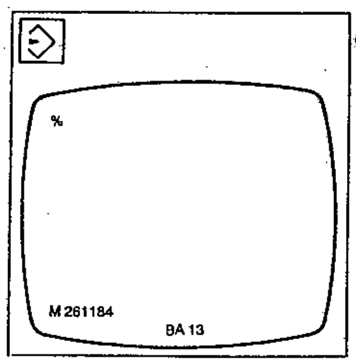
'ALL CLEAR Y/N' appears on input line, but programs are not yet cancelled.

If you are certain you wish to cancel all programs, press 'Y' for YES (otherwise 'N' for NO).



'YES' appears on input line, but programs are not yet cancelled.

Press 'Transfer' key.



All program numbers disappear from program list.

All programs, together with local subroutines (if any) and tool compensations contained in the programs, have now been erased from the program memory.

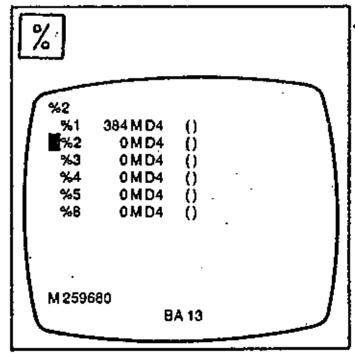
To cancel all macros, call macro list (page 7-2) and proceed in the same manner.

Playback

Playback is a method of indirect, stepwise program generation on the machine. A workpiece is machined step by step in modes 1, 4, 5, 6 or 7. After each step the control records the values and functions entered by the operator and converts them into a program block.

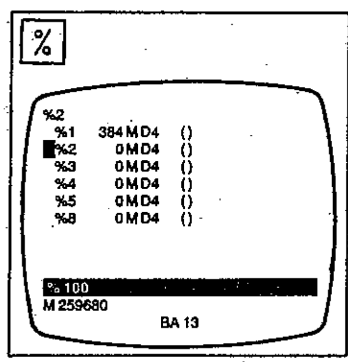
Thus, as a sample workpiece is being machined, the control at the same time generates the NC program for such part. Playback is especially used if no exact coordinate data are available, for example in line-by-line milling, clearing out pockets, or for producing simple programs. The playback procedure may be used either for creating a new program or for supplementing an existing program.

Select mode 13.
Set key lock switch 'Program lock' (16) on NC keyboard to vertical position.



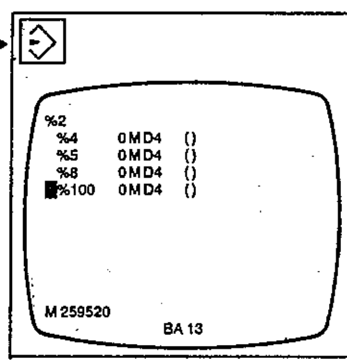
Display screen shows program list.

Press '%' key and enter desired program number.



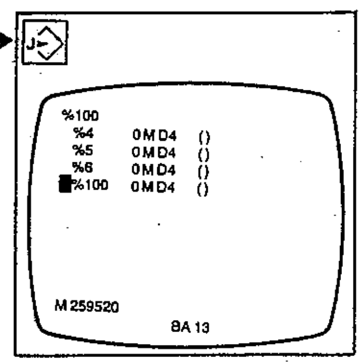
Program number entered appears on input line.

Press 'Transfer' key.



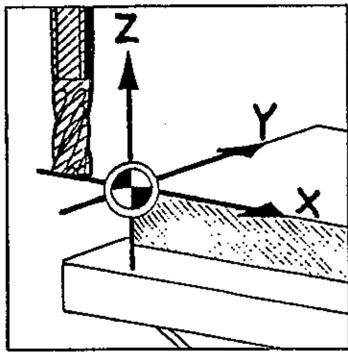
Program number appears in program list.

To transfer program to main memory of control, press 'Acknowledgment' key.

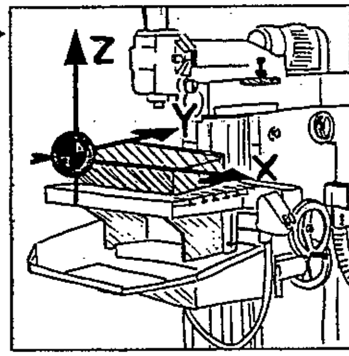


Program number appears in the upper left.

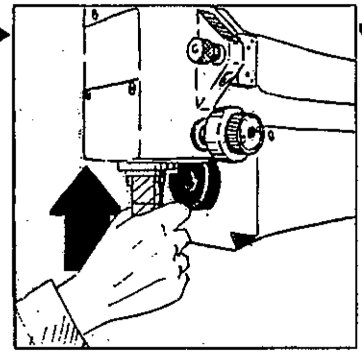
Determine program zero on workpiece.



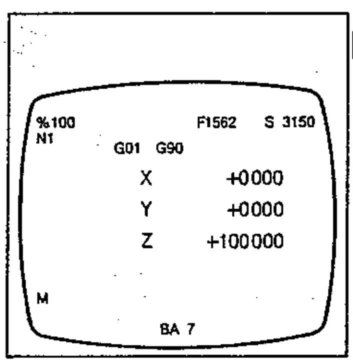
Set up machine in desired machining plane.



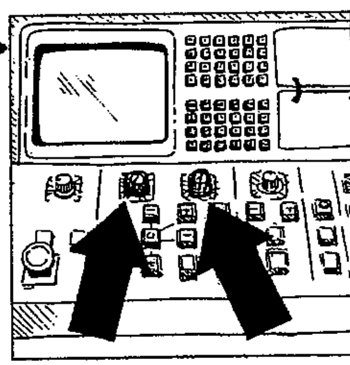
Mount cutting tool and set up machine in cutter axis.



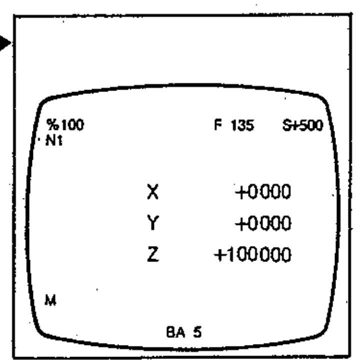
Set mode switch to mode 1, 4, 5, 6 or 7.



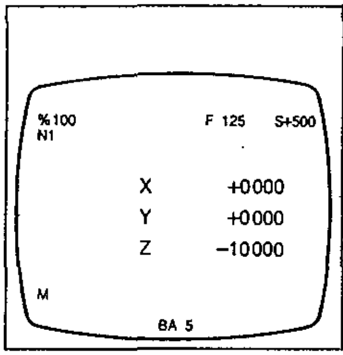
Set feed rate override (6) to 100 % and use selector (9) to set the desired feed rate.



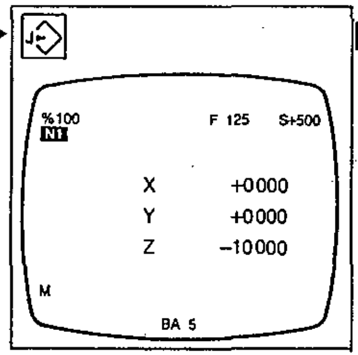
Select spindle speed and set on selector (12), then start spindle.



Move machine slides to desired position, as required for mode used.

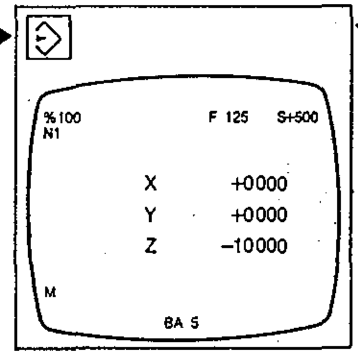


To record new position as a program block in main memory, press 'Acknowledgment' key.



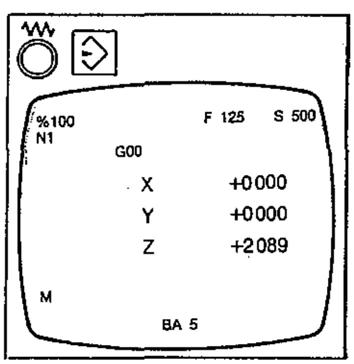
Block number is shown highlighted in upper left of display screen.

If block is to be stored as a feed rate block in modes 1, 4, 5 or 6: press 'Transfer' key with spindle running.



Inverse display returns to normal; block is recorded in main memory under the block number shown and with the feed rate selected.

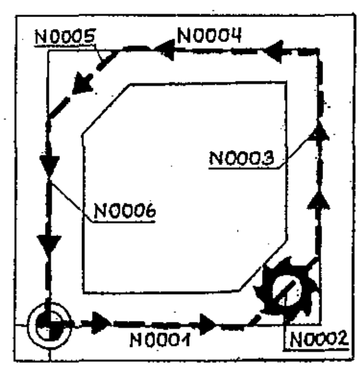
If block is to be stored as a rapid traverse block in modes 1, 4, 5 or 6:
simultaneously press 'Rapid traverse override' button (8) and 'Transfer' key.



Inverse display returns to normal; block is recorded as a rapid traverse block in main memory under the block number shown.

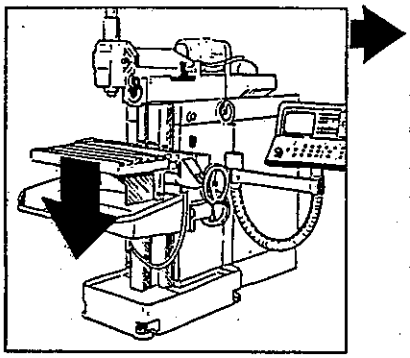
In mode 7, the block is automatically stored correctly as a feed rate or rapid traverse block, depending on the input.

Repeat this procedure until the workpiece has been completed.

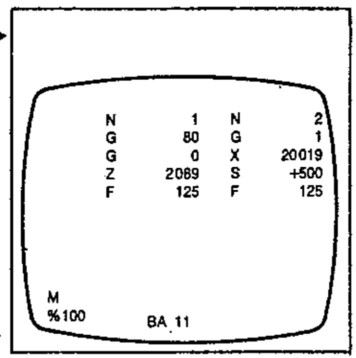


The control stores each machining step as a program block.

Retract tool.



Select mode 11.



First program block is shown on display screen.

● Move to last block number (page 7-29) and add M2 or M30 to last block (page 7-31).

The following addresses and functions are stored by the control in playback operation:

- feed rate
- spindle speed
- direction of spindle rotation
- linear interpolation
- rapid traverse G0 with spindle stopped
- absolute input G90
- axis addresses X/Y/Z/C

Correct and edit program if necessary (page 7-30) and add missing words (such as M commands).

After a trial run (page 7-62) or a screen graphics test (page 7-72), the program is obtainable for machining (page 7-65).

If a different tool is used, remember to add the necessary tool compensations (page 7-45).

Playback on machines with C-axis control

In playback operation the control, when taking over the first program block, will store all actual positions as a program block. On machines with an active C axis, this means that 4 axis positions will be stored: X, Y, Z and C. In machining, however, the control can only handle 3 axes simultaneously. To ensure that your program is handled correctly (without an error code appearing on the screen), delete one of the axis positions upon entering the first block.

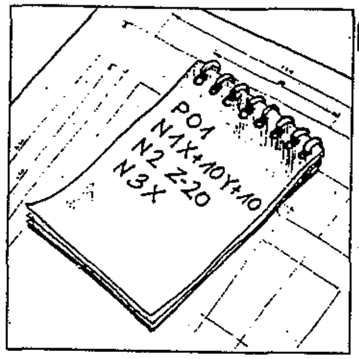
Blocks with chain dimensions in playback mode

Blocks entered in chain dimensions in the playback mode (mode 7) are correctly processed, but stored in the control as blocks with absolute dimension.

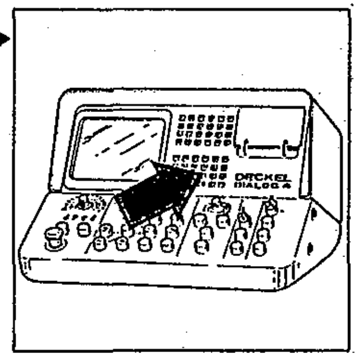
Direct generation of a program

Direct programming means planning all the machining steps in advance.
The program is written down and then entered word by word using the keys on the NC keyboard.

Write down your program block by block, using the part drawing as a reference (see Chapter 6, Operator's Manual).

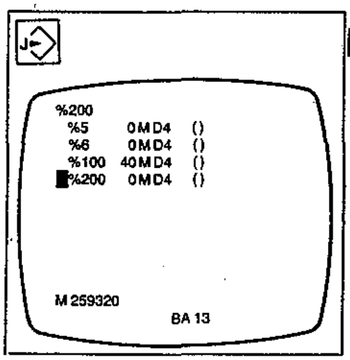


Enter program by keying in all the data on the NC keyboard.



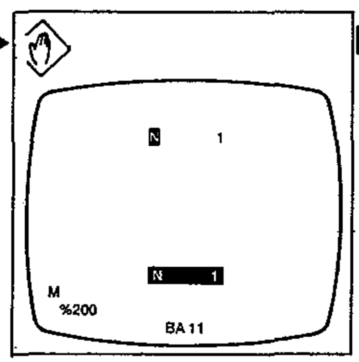
Entering a program

Select mode 13, open program (page 7-6) and press 'Acknowledgment' key.



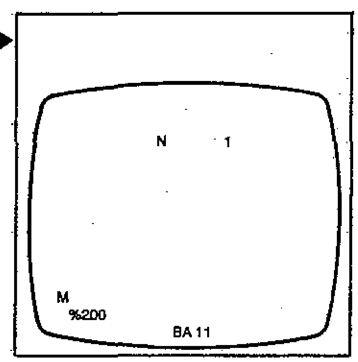
Program number is transferred to main memory of control.

Select mode 11.



Block number N1 appears on input line.

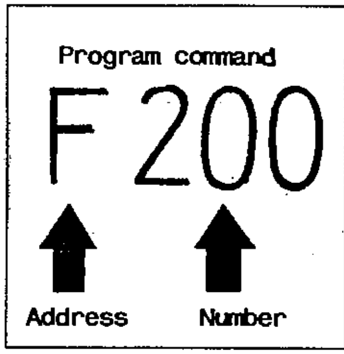
Press 'Transfer' key to transfer block number to memory (or enter a different block number, if desired).



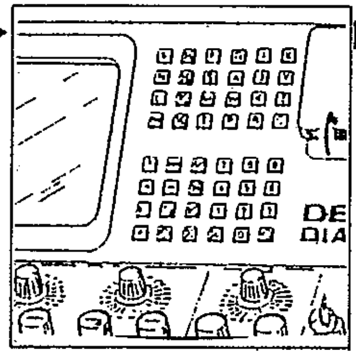
Input line disappears; block number is now contained in main memory.
Proceed by entering program blocks word by word (page 7-19).

Entering program commands

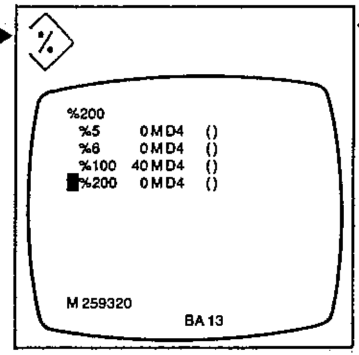
Each program command (word) consists of a letter (address code) and a number.



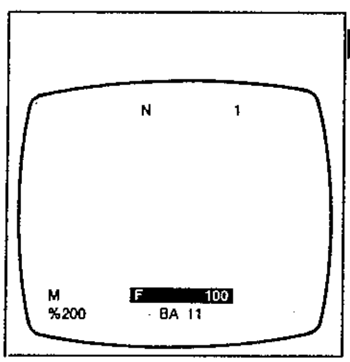
For the entering program commands, use keyboard (23) on control console. The keyboard has two sections, one for letters, one for numerals.



Open program (see page 7-6).

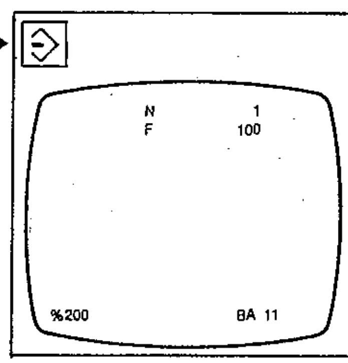


Enter program command, for example F100.



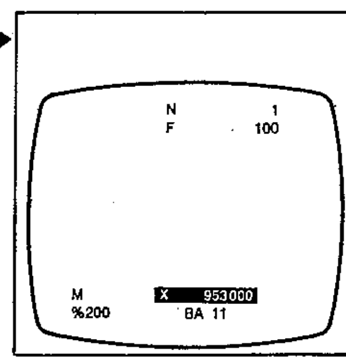
Command entered appears in input line.

Press 'Transfer' key.



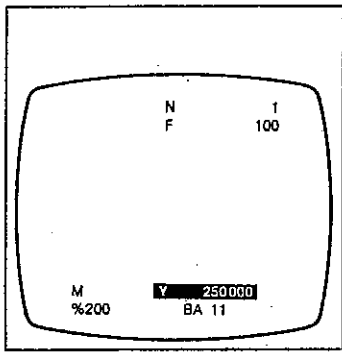
Command disappears from input line and is transferred to main memory of control.

Enter next program command, for example X953.



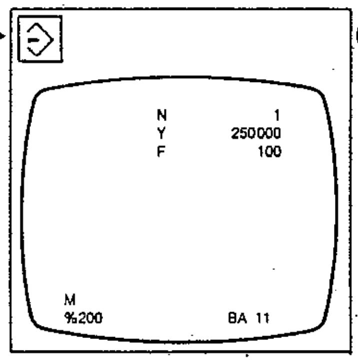
Command entered appears on input line.

If you make a mistake, enter the correct command, for example Y250.



Corrected command appears on input line.

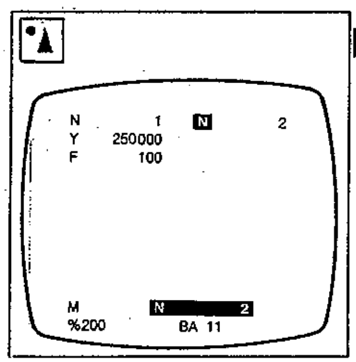
Press 'Transfer' key.



Command disappears from input line and is transferred to main memory of control.

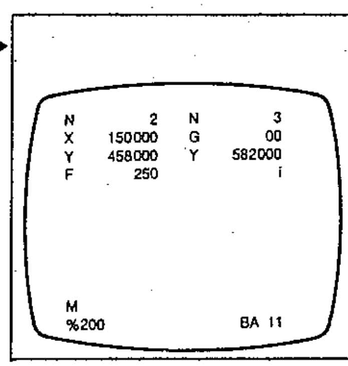
Entering the next block

When all words of a block have been entered, press 'Cursor up' key.



Block just entered moves to the left; next higher block number appears on input line.

Enter next block word by word, then proceed in the same manner with all other blocks.

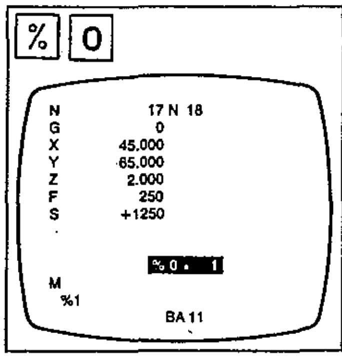


Blocks are transferred to main memory.

The memory of the control has a capacity of 256 Kbytes (option: 512 Kbytes).
The remaining memory capacity (in bytes) is indicated under the program number in mode 13.

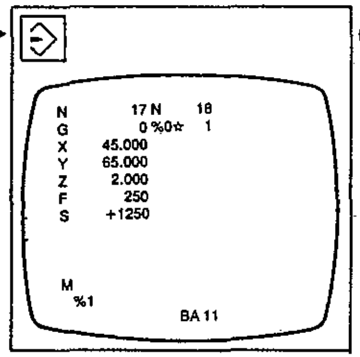
Entering a macro call

In mode 11, press '%' and '0' keys and enter macro number.



Macro number entered appears on input line.

Press 'Transfer' key.



Macro call appears in block.

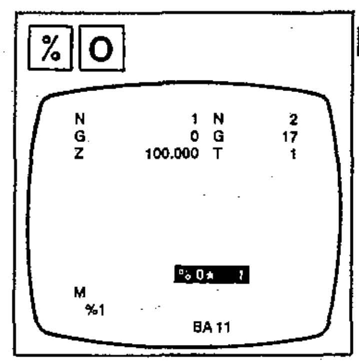
Generating a macro

To generate a macro, it has first to be opened (page 7-7) and transferred to the main memory of the control by pressing the 'Acknowledgment' key. Then select mode 11 and enter macro like a main program.

A macro (which has previously been opened in mode 13) may also be transferred to the main memory directly while generating a program in mode 11.

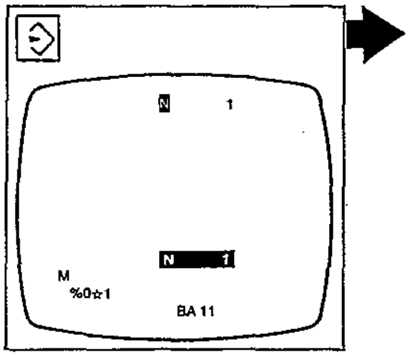
Proceed as follows:

In mode 11, press '%' and '0' keys and enter macro number.



Macro number appears on input line.

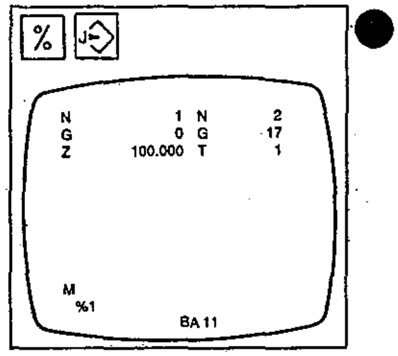
Press 'Transfer' key.



First block number of macro appears on input line.

Macros are generated in the same manner as programs.

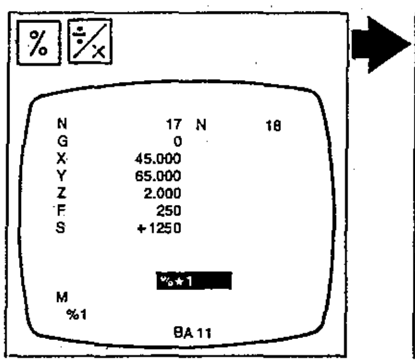
To return to main program, press '%' and 'Acknowledgment' keys.



Display screen shows main program block last previously worked on.

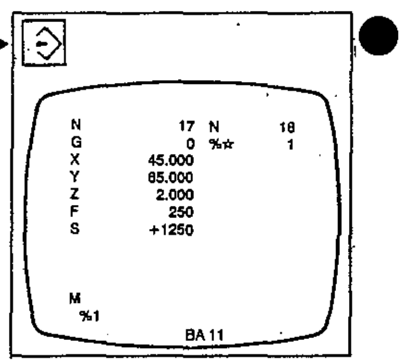
Entering a local subroutine call

Press '%' and 'x' keys and enter subroutine number.



Subroutine number entered appears on input line.

Press 'Transfer' key.



Subroutine call appears in block.

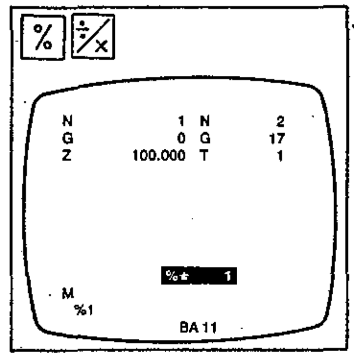
Entering a local subroutine

To generate a local subroutine, it has first to be opened (page 7-7) and transferred to the main memory of the control by pressing the 'Acknowledgment' key.

As a rule, however local subroutines are entered while generating the associated main program in mode 11.

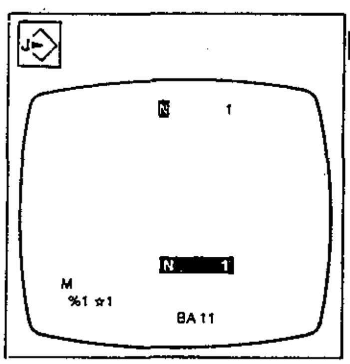
Proceed as follows:

In mode 11, press '%' and 'x' keys and enter subroutine number.



Subroutine number appears on input line.

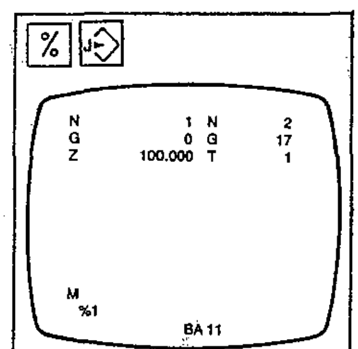
Press 'Transfer' key.



First block number of local subroutine appears on input line.

Subroutines are generated in the same manner as pro grams.

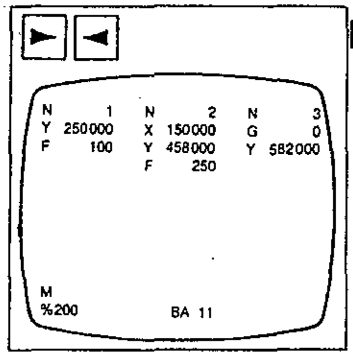
To return to main program, press '%' and 'Acknowledgment' keys.



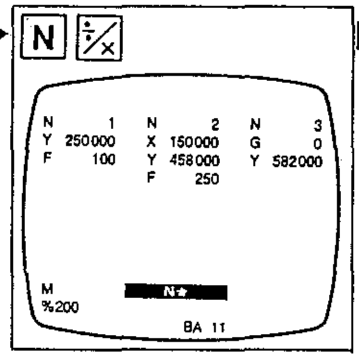
Display screen shows main program block last previously worked on.

Entering a subroutine call

Use 'Cursor right/left' keys to find the block where a subroutine call is to be programmed.

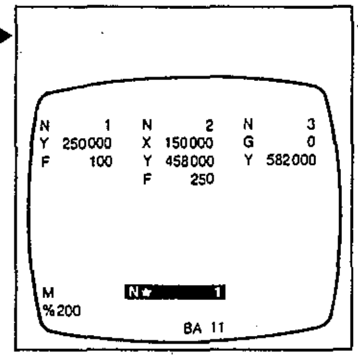


Press 'N' and 'x' keys.



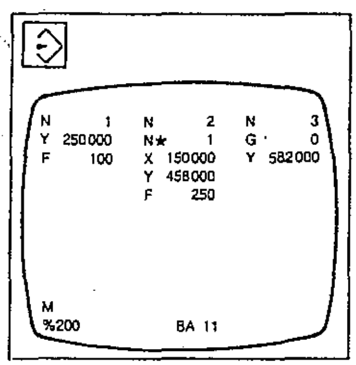
'N *' appears on input line.

Enter number of subroutine to be called.



Number appears in input line.

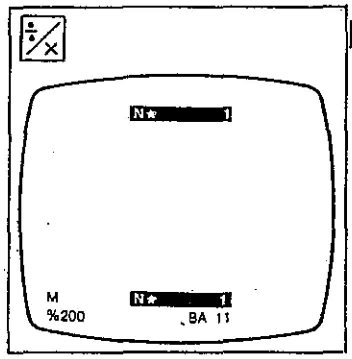
Press 'Transfer' key.



Subroutine call is inserted in correct position within block.

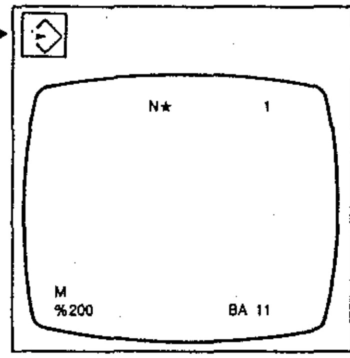
Entering a subroutine block

Upon entering the last command in the last block (usually M30 or M2), press 'x' key.



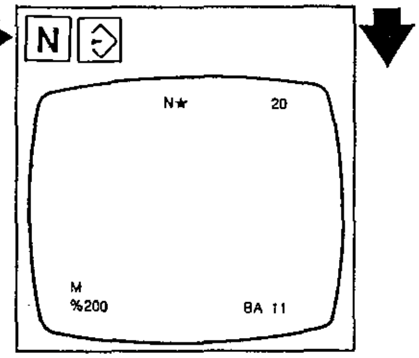
First subroutine block number appears on input line.

Either:
press 'Transfer' key.



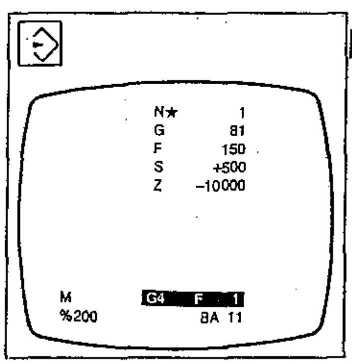
Subroutine block number is transferred to main memory.

Or:
overwrite block number by entering 'N' and new block number and press 'Transfer' key.

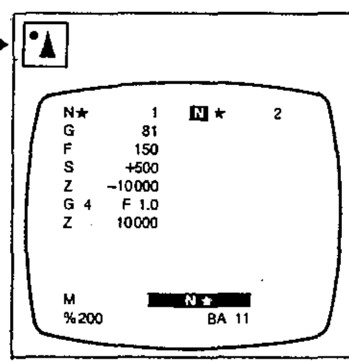


New subroutine block number is transferred to main memory.

Enter subroutine block word by word.

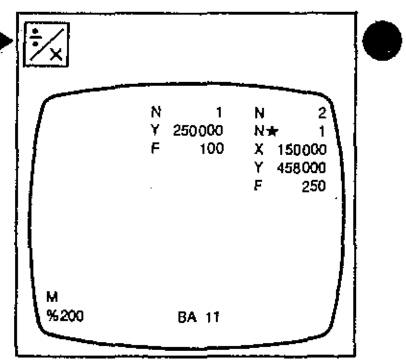


To enter next subroutine block, press 'Cursor up' key.



Subroutine block just entered moves to the left; next higher subroutine block number appears on input line.

To return to main program, press 'x' key.

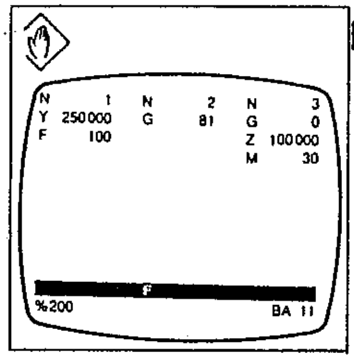


CRT shows first block of main program.

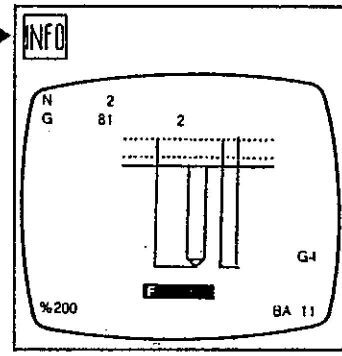
Entering a cycle

The CNC DIALOG 4 control features dialog-assisted input of drilling and milling cycles. As an additional aid for program input on the keyboard you may call a graphics display showing the cycle parameters to be entered (page 6-87).

In mode 11, enter the desired cycle call using the address keys.

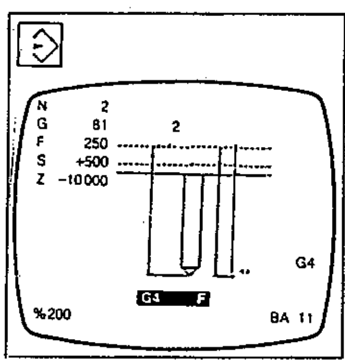


Press 'INFO' key.



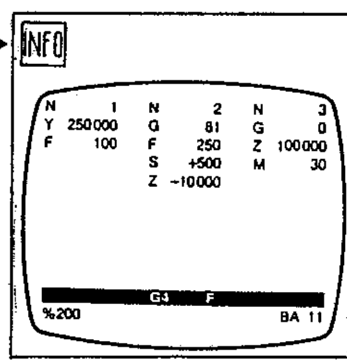
Screen shows display of cycle with the individual parameters requested by the control in the input dialog.

Continue by entering the cycle parameters as requested.



The parameter requested by the control is shown highlighted on the display screen.

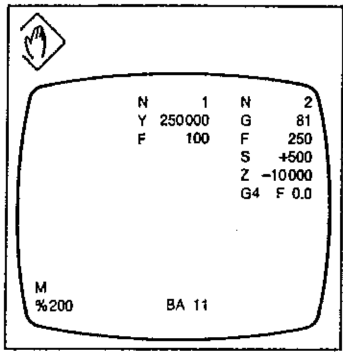
To clear graphics display, press 'INFO' key again.



Graphics disappears automatically when you proceed to next block.

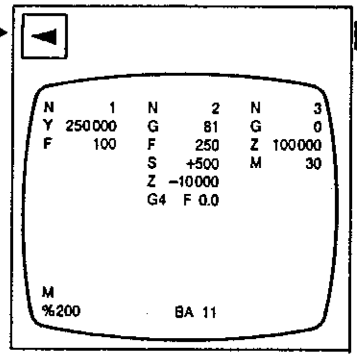
Checking the program contents

Transfer desired program or macro to main memory of control and select mode 11.



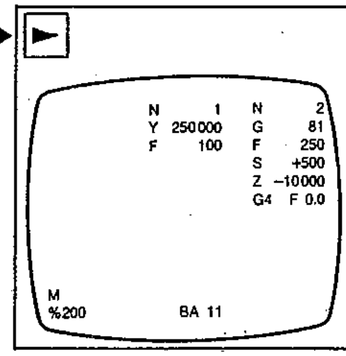
Display screen shows first two blocks of program.

Press 'Cursor left'.



Program display moves one step to the left.

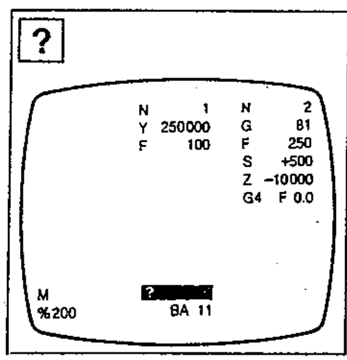
Press 'Cursor right'.



Program display moves one step to the right.

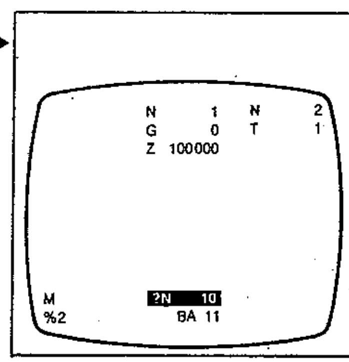
Searching a program block

Press '?' key.



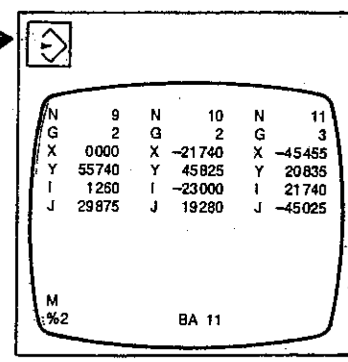
Control requests desired block number on input line.

Enter block number, for example 10.



Number appears on input line.

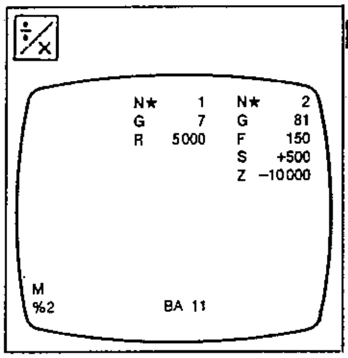
Press 'Transfer' key.



Block appears in the middle of screen, with neighbouring blocks on the right and left.

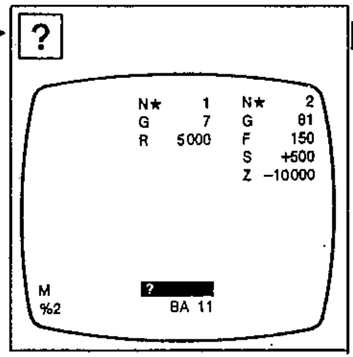
Searching a subroutine block

Press 'x' key.



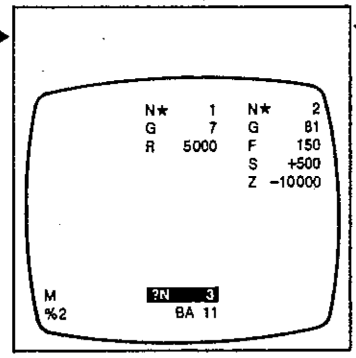
Display screen shows the first two subroutine blocks.

Press '?' key.



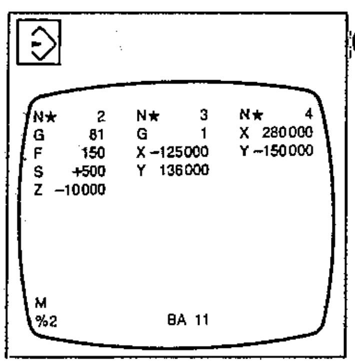
Control requests desired subroutine block number on input line.

Enter subroutine block number, for example 3.



Number appears on input line.

Press 'Transfer' key.

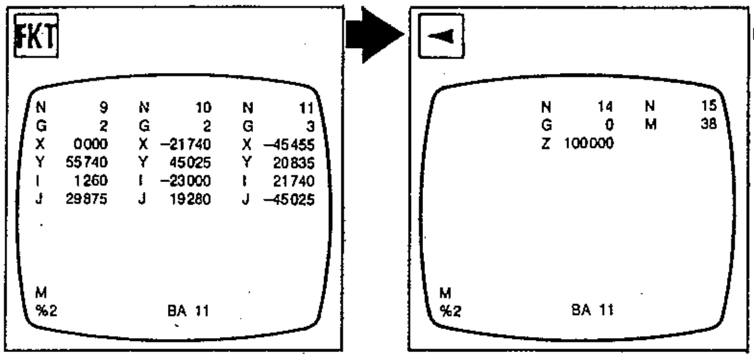


Subroutine block appears in the middle of screen, with neighbouring blocks on the right and left.

Searching the last block in a program

In mode 11, press 'FKT' key.

Press 'Cursor left'.



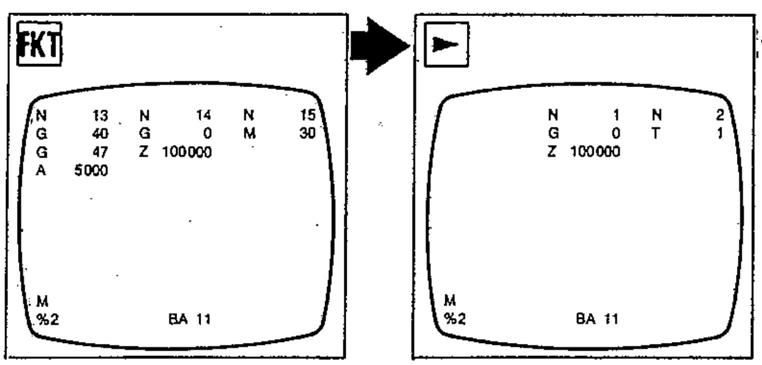
Last block of program selected appears on display screen.

● To search last subroutine block, press 'x' key and proceed in the same manner.

Searching the first block in a program

In mode 11, press 'FKT' key.

Press 'Cursor right'.

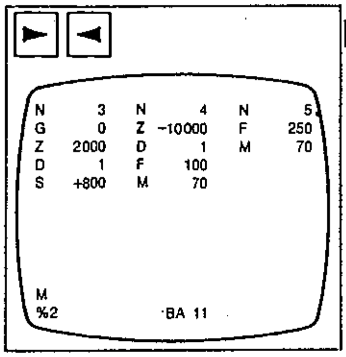


First block of program selected appears on display screen.

● To search first subroutine block, press 'x' key and proceed in the same manner.

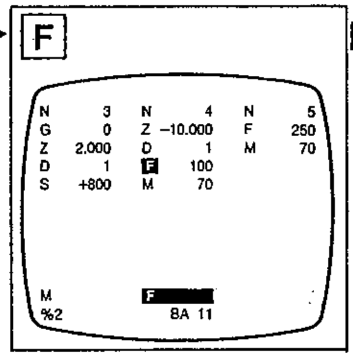
Program edit
Changing a word

In mode 11, find the desired block (page 7-26).



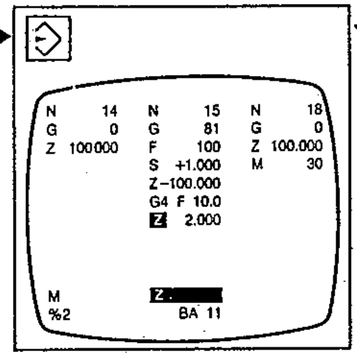
Block with word to be changed is displayed in the middle of screen.

Press address key of word to be changed, for example F.



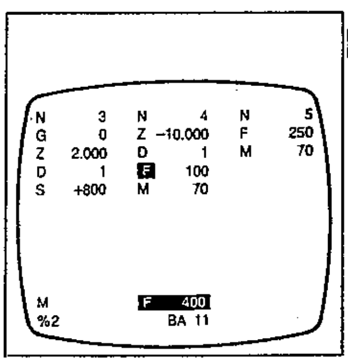
Address letter is shown highlighted on input line. Cursor is positioned on word to be changed.

If an address is contained several times in a block, use 'Transfer' key to move cursor to desired position.



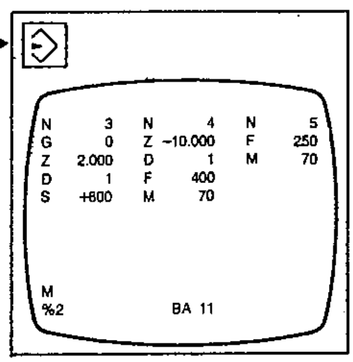
Each time key is pressed, cursor jumps to next word with the same address.

Enter new numerical value, for example 400.



New value appears in input line.

Press 'Transfer' key.

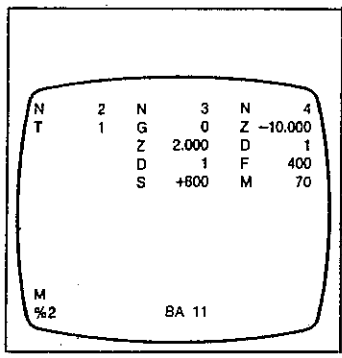


New word replaces old word; old word is deleted.

- The address of a word cannot be changed in this way.
  Proceed as follows: delete word with wrong address (page 7-32), then enter correct word (page 7-31).

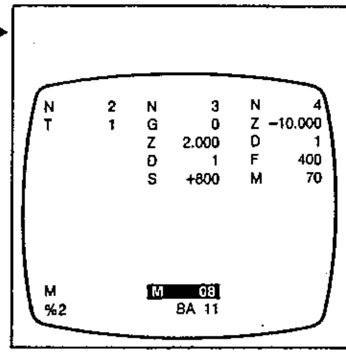
Inserting a word

In mode 11, find the desired block (page 7-27).



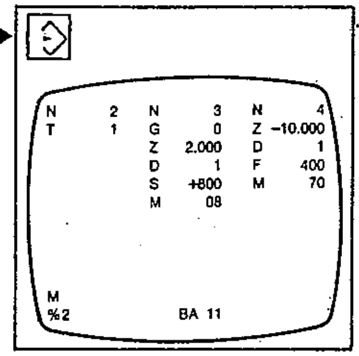
Block with word to be inserted is displayed in the middle of screen.

Enter word to be inserted, for example M08.



Word appears on input line.

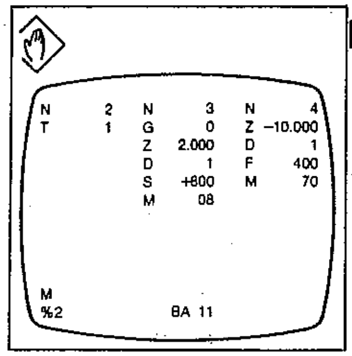
Press 'Transfer' key.



New word is transferred to memory in correct position within program block.

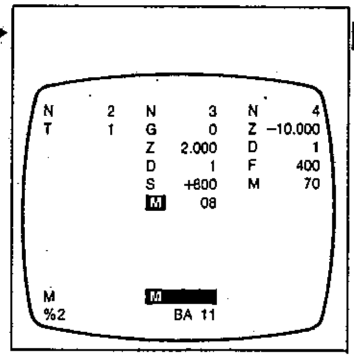
Deleting a word

In mode 11, find the desired block (page 7-27).



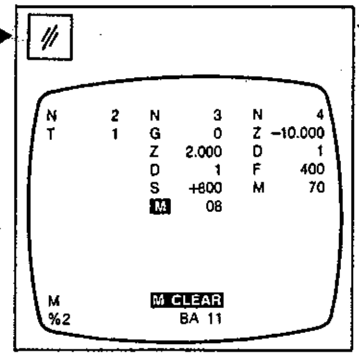
Block with word to be deleted is displayed in the middle of screen.

Press address key of word to be deleted, for example M.



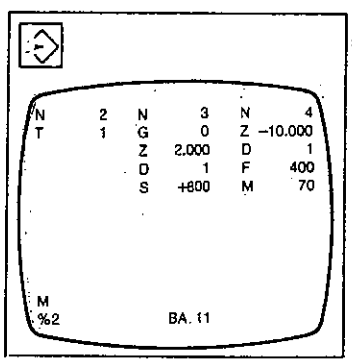
Address letter is shown highlighted on input line.

Press red 'Delete' key.



'CLEAR' appears adjacent to word address in input line.

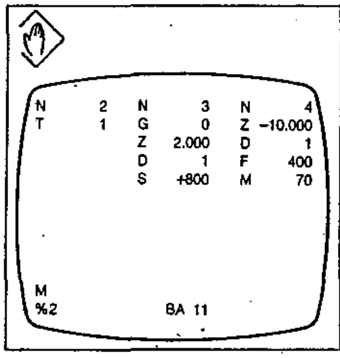
Press 'Transfer' key.



Word to be deleted disappears from input line and from block displayed and is erased from memory.

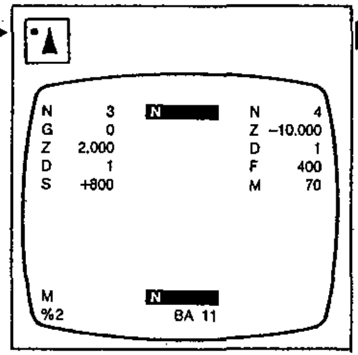
Inserting a block

In mode 11, find the block where the additional block is to be inserted. (page 7-27).



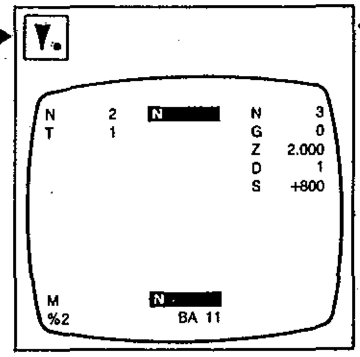
Desired block is shown in the middle of screen.

Either:
press 'Cursor down' (behind block shown).



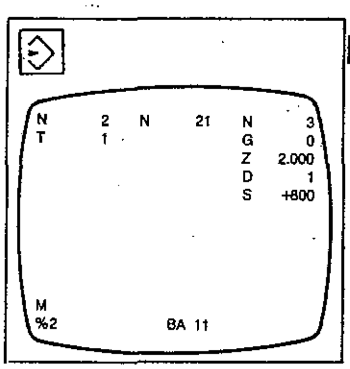
Block selected moves to the left; N address appears highlighted on input line.

Or:
press 'Cursor up' (ahead of block shown).



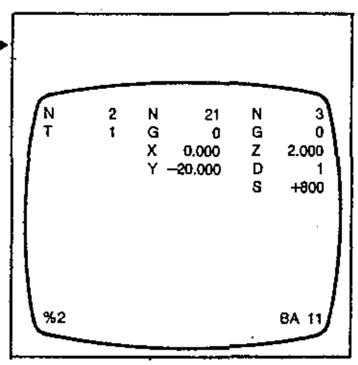
Block selected moves to the right; N address appears highlighted on input line.

Enter new block number, press 'Transfer' key.



Block number disappears from input line and is transferred to top of screen.

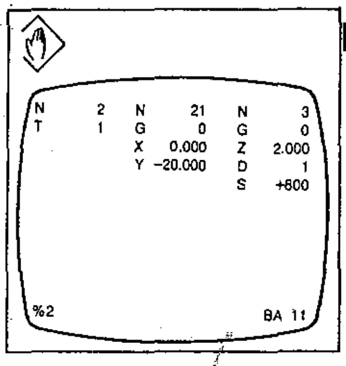
Enter new block word by word.



● The blocks in a program are always machined in the sequence of input, regardless of block numbers.
Always select a number not yet contained in the program for a new block inserted. Preferably use the adjacent block number with a different numeral in the hundreds digit.

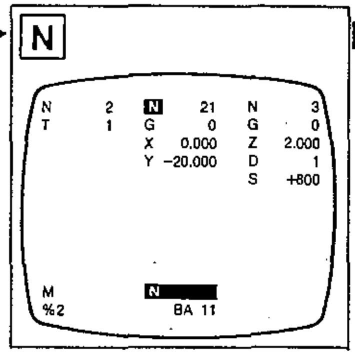
Deleting a block

In mode 11, find block to be deleted (page 7-27).



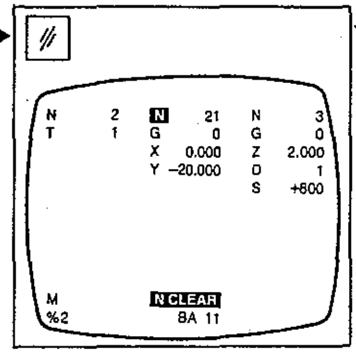
Block to be deleted is displayed in the middle of screen.

Press address address key 'N'.



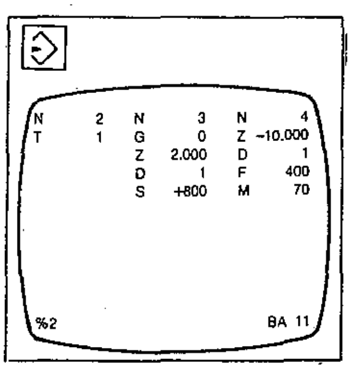
Address N appears highlighted on input line.

Press red 'Delete' key.



'CLEAR' appears adjacent to N on input line.

Press 'Transfer' key.



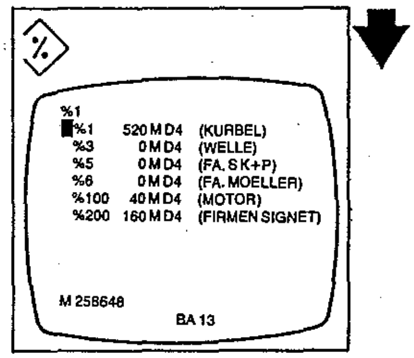
Block to be deleted disappears from screen and is erased from memory.

Tool store T
Entering a tool number

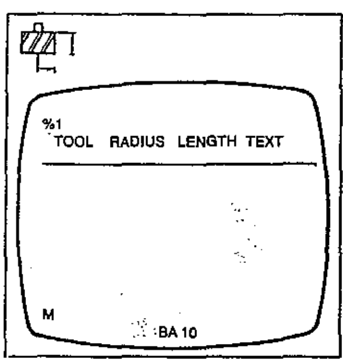
Tool store T always belongs to a certain program. Tool length and radius compensations are activated by calling a tool (T address). They are self-retaining and thus valid from the time the associated tool is changed into the spindle to the call of a new tool.
(Tool numbers may have up to six digits.)

If you decide to use tool compensations from tool store T, you should press the 'Transfer' key whenever a compensation value D is requested by the control in the input dialog.

Load program into main memory of control (page 7-6).

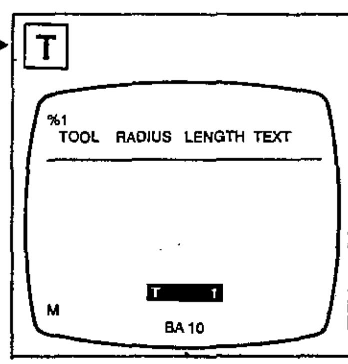


Select mode 10.



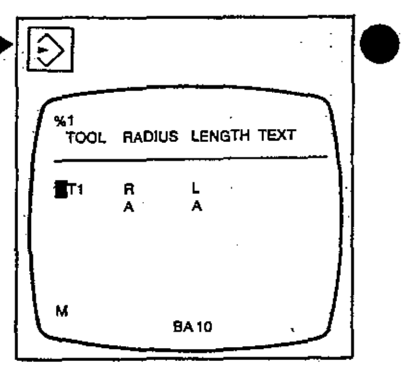
Display screen shows head of tool table for tool store T.

Press 'T' key and enter tool number.



Tool number appears on input line.

Press 'Transfer' key.

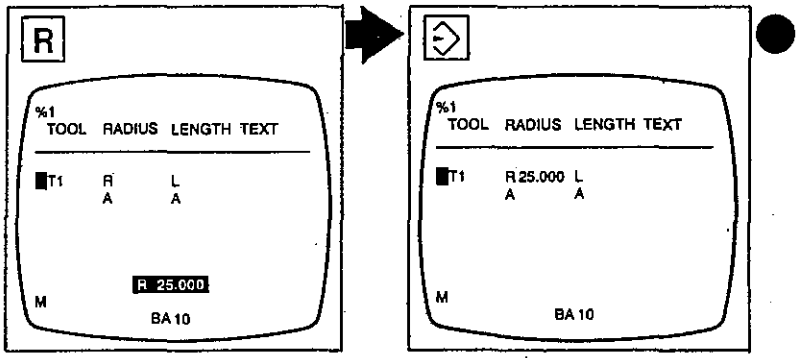


Tool number is now in tool store.

Tool store T
Entering a radius compensation value

Press 'R' key and enter radius compensation value.

Press 'Transfer' key.



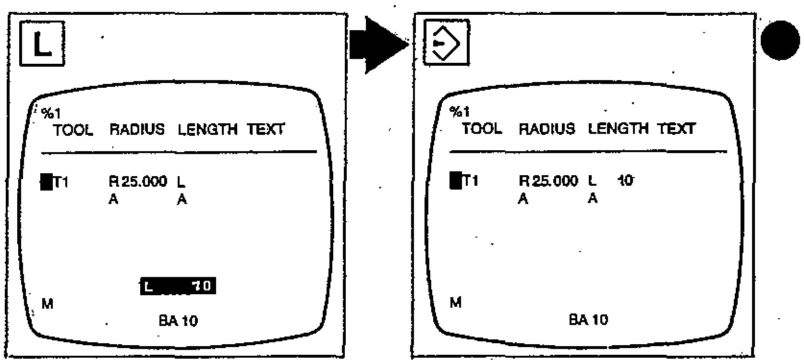
'R' and compensation value appear on input line.

Radius compensation is now in tool store.

Tool store T
Entering a length compensation value

Press 'L' key and enter length compensation value.

Press 'Transfer' key.

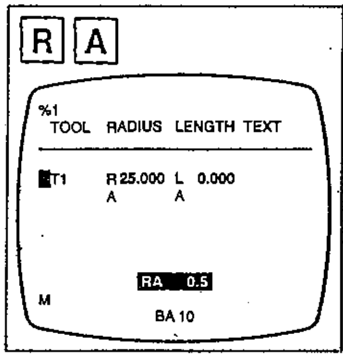


'L' and compensation value appear on input line.

Length compensation is now in tool store.

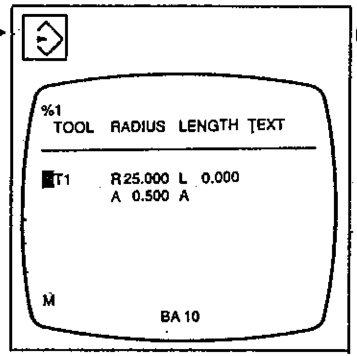
Tool store T
Entering a radius allowance on compensation value

Press 'R'and 'A' keys and enter allowance value.



'RA' and allowance value appear on input line.

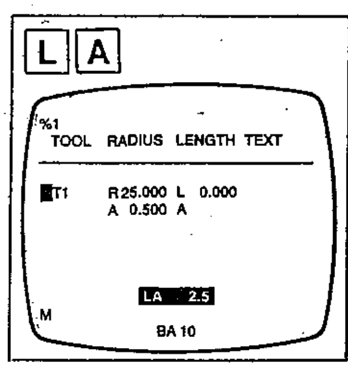
Press 'Transfer' key.



Allowance on radius is now in tool store.

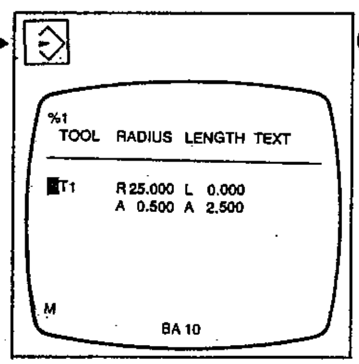
Tool store T
Entering a length allowance on compensation value

Press 'L'and 'A' keys and enter allowance value.



'LA' and allowance value appear on input line.

Press 'Transfer' key.

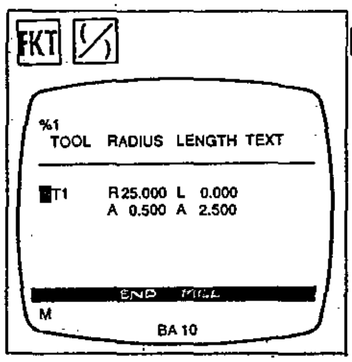


Allowance on tool length is now in tool store.

Tool store T
Entering a tool designation (comment)

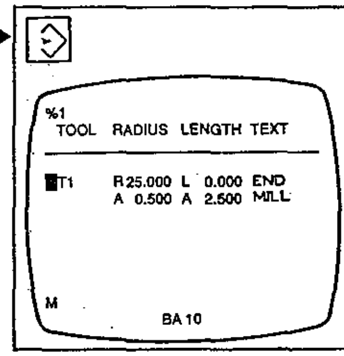
A comment of max 22 characters (for example the tool designation) can be entered for each tool. To do so, find tool number (page 7-38). Comments may be altered as required or deleted (pages 7-8 and 7-9).

Press '(', key, enter comment, then press ')' key.



Comment appears on input line.

Press 'Transfer' key.

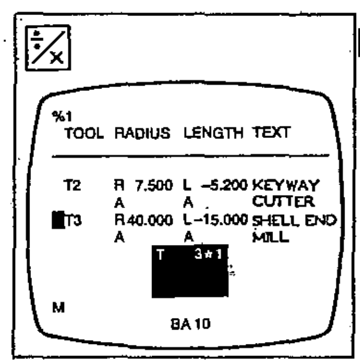


Comment now appears behind tool number in 'Text' column.

Tool store T
Entering additional compensation values

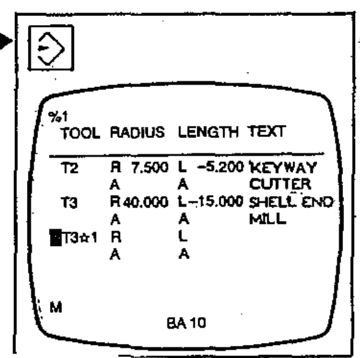
Additional tool length and radius compensations may be entered for each tool. They are marked in the tool store by * and a number. If any such * tool is called in a program, the cutting operation is continued with the same tool, but with altered compensation values.

Press 'x', key and enter number of additional compensation.



Number of tool for which a * tool is to be defined, as well as * and previously entered number of additional compensation appear on input line.

Press 'Transfer' key.



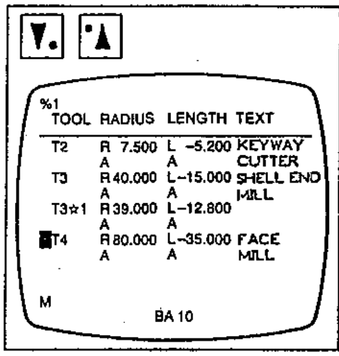
Number of * tool is now in tool store.
Additional compensations can now be entered (pages 7-35 to 7-37).

Tool store T
Searching a tool number

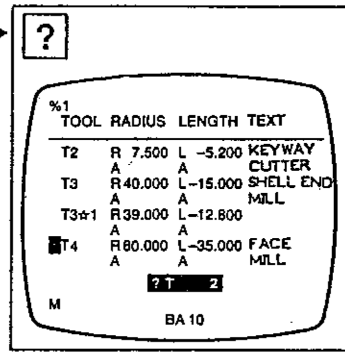
Select mode 10.
Either: use 'Cursor up/down' keys to set cursor to desired tool number.

Or:
Press '?' key and enter desired tool number.

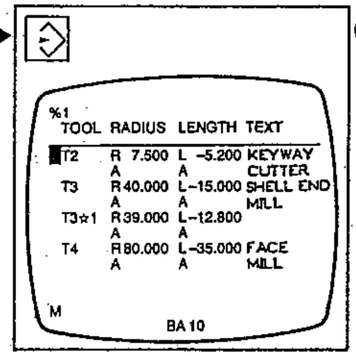
Press 'Transfer' key.



Cursor is positioned at desired tool number.



Desired tool number appears on input line.

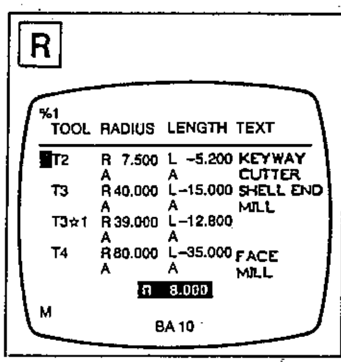


Cursor is positioned at desired tool number.

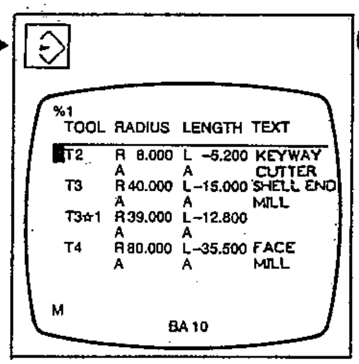
Tool store T
Changing a compensation value

Find tool number (see above). Press adress key for compensation value to be altered, for example 'R' for radius compensation, and enter new value.

Press 'Transfer' key.



'R' and new value appear on input line.

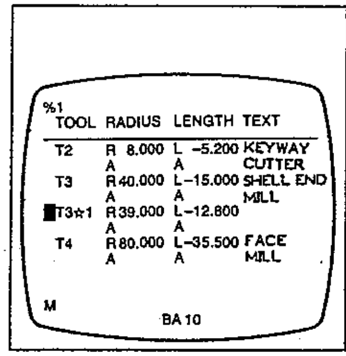


New compensation value is now in tool store.

For length compensation or allowance proceed in the same manner.

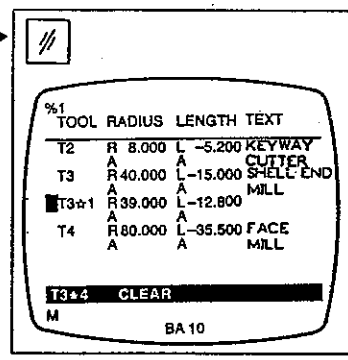
Tool store T
Deleting a tool

Select mode 10 and find desired tool (page 7-38).



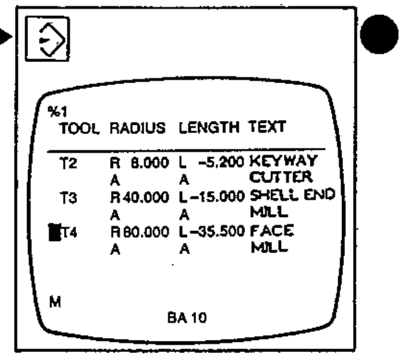
Cursor is positioned at desired tool number.

Press 'Delete' key.



Number of tool to be deleted and 'CLEAR' appear on input line.

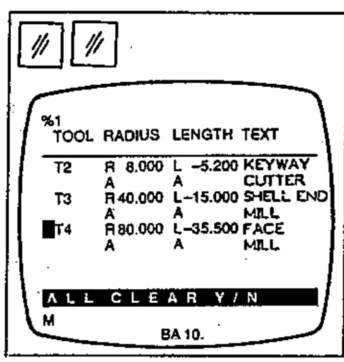
Press 'Transfer' key.



Tool data are erased in tool store.

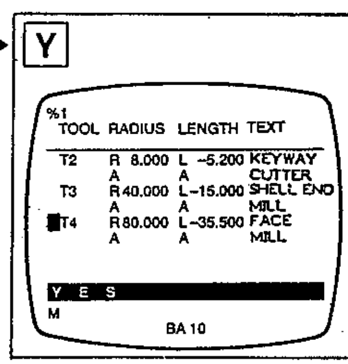
Tool store T
Deleting all tools

Select mode 10 and press 'Delete' key 2x.



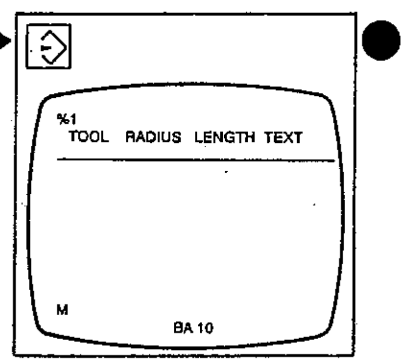
'ALL CLEAR Y/N' appears on input line.

If all tools are to be deleted, press 'Y' for YES (otherwise press 'N' for NO).



'YES' appears on input line.

Press 'Transfer' key.



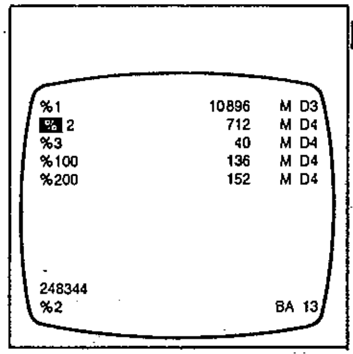
Tool store is now clear.

Tool compensation store D

Searching a tool compensation number

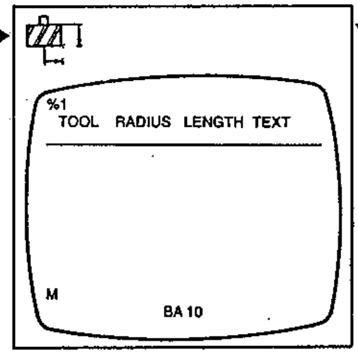
Tool compensation store D should only be used in connection with DIALOG 3 programs.
In this case, tool store T must not contain any compensation values, since otherwise the D and T compensation value may be added.
Tool compensation store D also forms part of the program. Up to 100 compensation values can be stored in compensation store D for each program.
Tool length compensations are effective in only one block.
Tool radius compensations must be entered in reply to the input dialog each time a compensation on the contour is called.

Select mode 13 (page 7-3) or open a program (page 7-6).



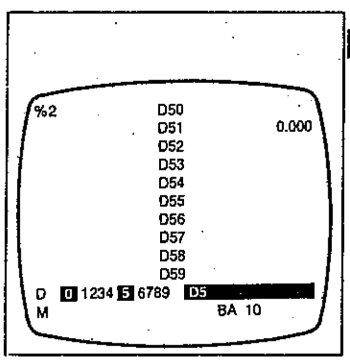
Since the tool compensation values belong to the program, there will be no access to the compensation store without a program in the main memory.

Select mode 10.



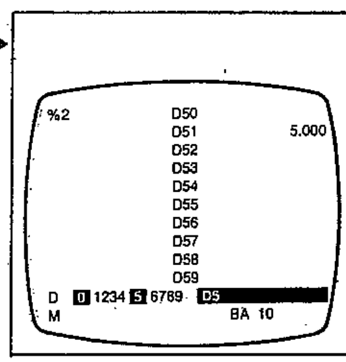
Display screen shows contents of tool store T.

To get into tool compensation store D, press address key 'D'.



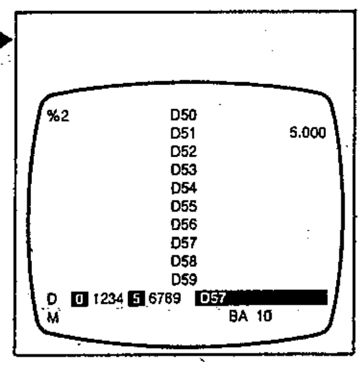
Tool compensation store D stores the program-related compensation values in decade blocks (groups of ten). Display screen shows first decade in which a compensation is stored.

Enter number of desired decade, for example 5 for 50..



Desired decade appears on display screen. Decades containing compensation values are shown highlighted in lower left of screen.

Enter number of desired compensation in such decade, for example 7.

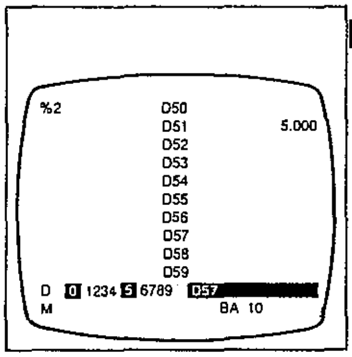


Cursor jumps to desired compensation number.

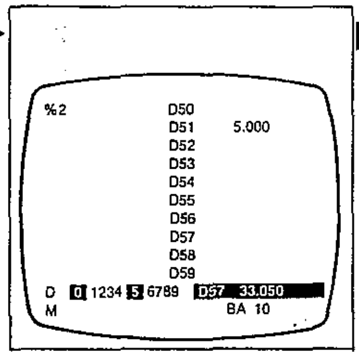
Tool compensation store D

Entering a tool compensation value

Find desired tool compensation number, for example 57.

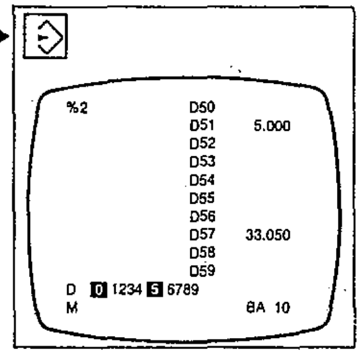


Enter compensation value with sign, for example 33.050 (= 33.05 mm).



Compensation value appears adjacent to compensation number on input line.

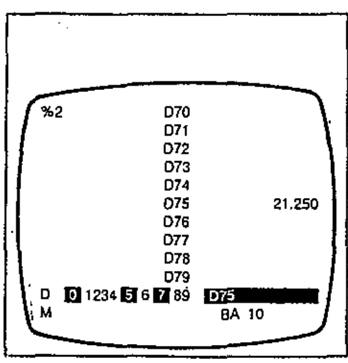
Press 'Transfer' key.



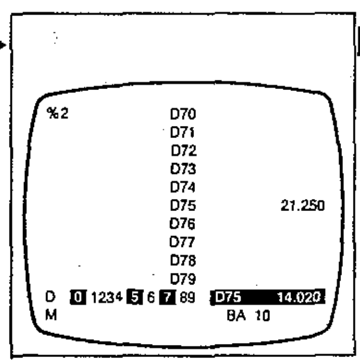
Compensation value is transferred to compensation store.

Tool compensation store D
Altering a tool compensation

Find desired tool compensation number, for example D75.

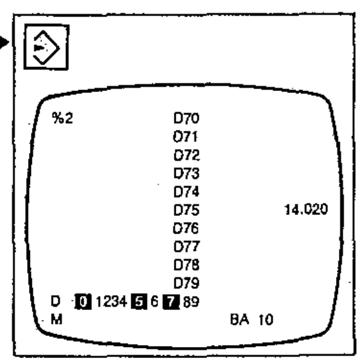


Enter new compensation value, for example +14.020.



New compensation value appears adjacent to compensation number on input line.

Press 'Transfer' key.



New compensation value is transferred to compensation store. Previous compensation value is deleted.

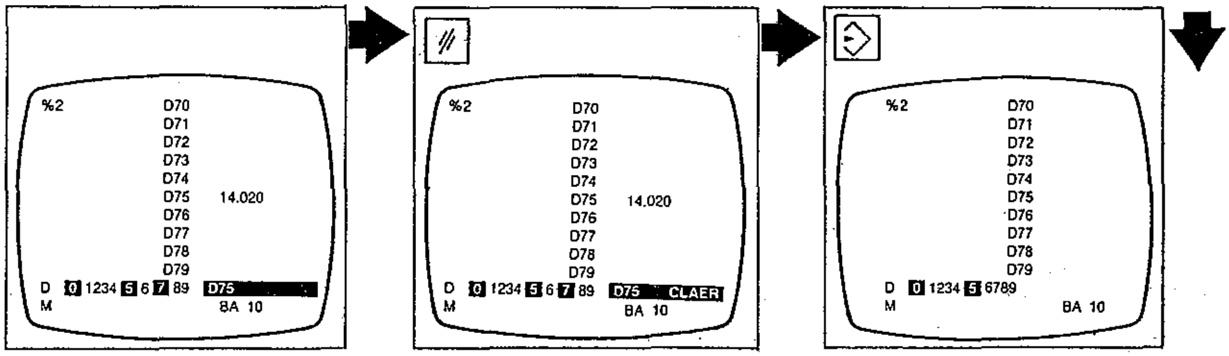
Tool compensation store D

Deleting an individual tool compensation value

Find desired tool compensation number, for example D75.

Press 'Delete' key.

Press 'Transfer' key.



'CLEAR' appears adjacent to compensation number on input line.

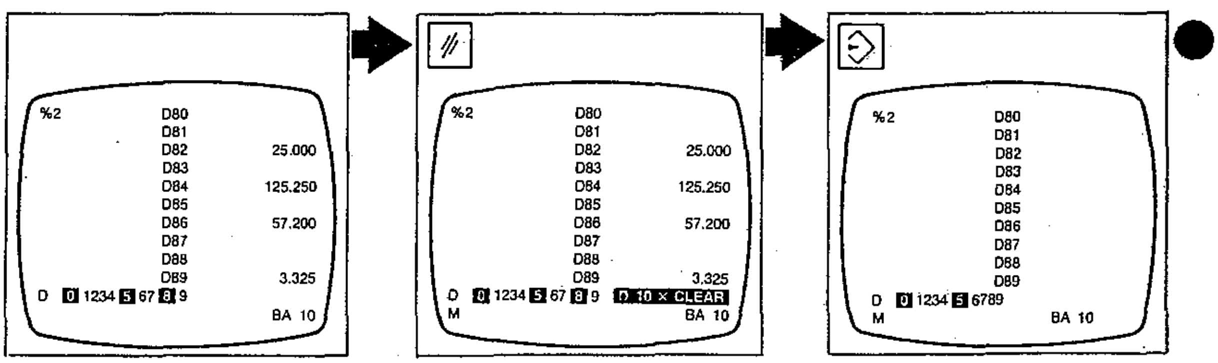
Compensation value is deleted.

Tool compensation store D
Deleting a tool compensation decade

In mode 10, enter number of decade to be deleted, for example D8.

Press 'Delete' key 2x.

Press 'Transfer' key.



Desired decade appears on display screen.

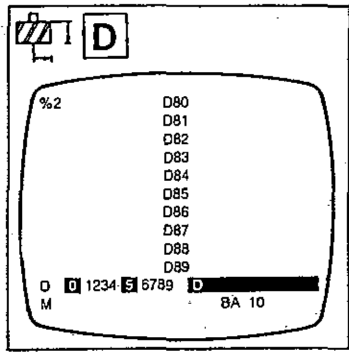
'10 x CLEAR' appears adjacent to decade number on input line.

Complete decade is deleted. Highlighted decade number 8 disappears.

Tool compensation store D

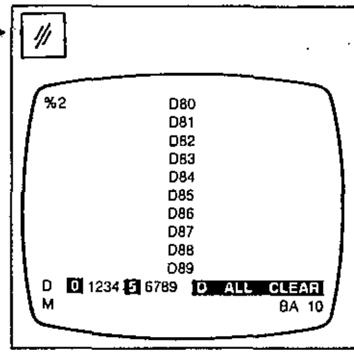
Deleting all tool compensation values

Select mode 10 and press 'D' key.



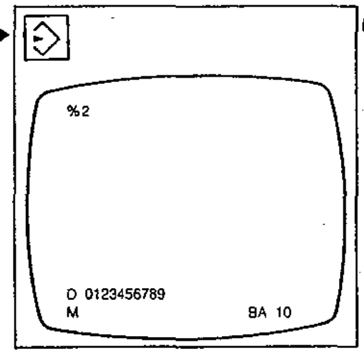
First decade containing a compensation value appears on display screen.

Press 'Delete' key 2x.



'ALL CLEAR' appears on input line.

Press 'Transfer' key.

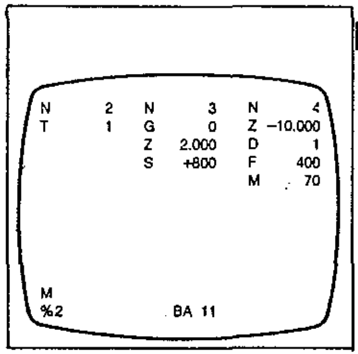


All tool compensations of the selected program are deleted.

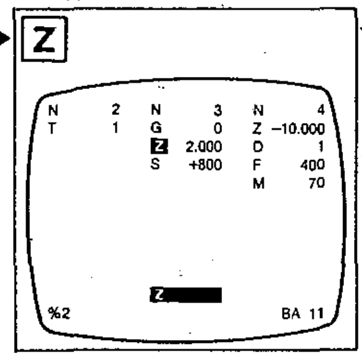
Tool compensation store D
Entering a tool length compensation call

In order for the control to know at what point in a program a tool length compensation is to be used, the tool compensation with the compensation number have to be programmed accordingly.

Load program into main memory, if required.
In mode 11, find block where tool length compensation is to be activated.

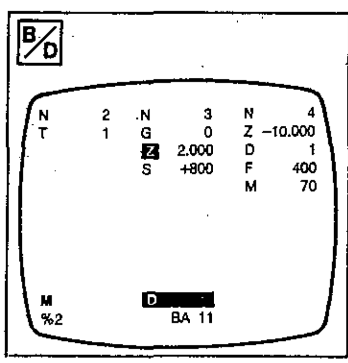


Press address key of axis where tool length compensation is to be applied (in most cases Y or Z).



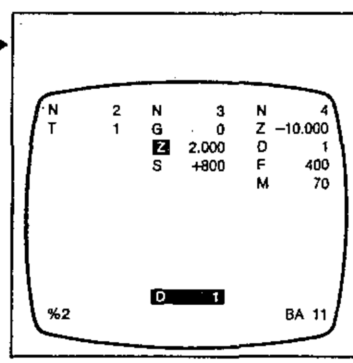
Axis address appears on input line.

Press 'D' key.



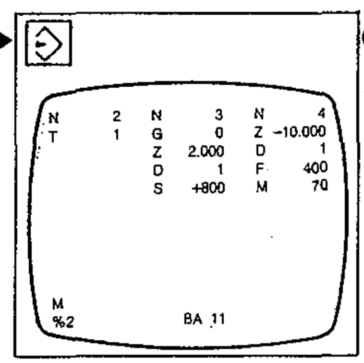
'D' appears on input line.

Enter compensation number under which tool compensation is stored in compensation store, for example 1 (always positive).



Number appears on input line.

Press 'Transfer' key.



Tool length compensation call is inserted in correct position into program block.

Cutting data calculator

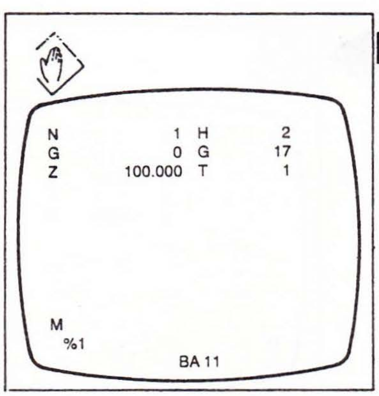
The new DIALOG 11 is equipped with a cutting data calculator relieving you of the tedious calculation of the feed rate and spindle speed in manual data input (mode 11).

The cutting data calculator can also be used for optimizing the cutting data.
First enter your parameters (tool diameter, cutting speed, number of teeth and feed per tooth or per revolution).
The calculator will produce 'odd' values for feed rate and spindle speed.

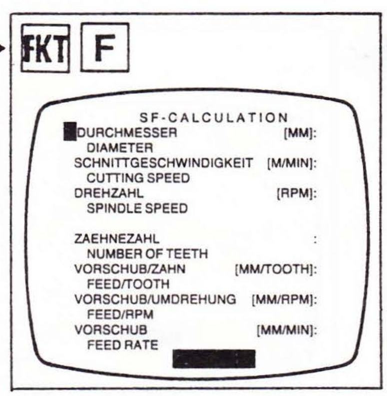
Now you enter the nearest 'straight' feed rate and spindle speed: the cutting data calculator will indicate the actual cutting speed and the corresponding feed per cutter tooth or revolution.

Activating the cutting data calculator

Select mode 11.



Press 'FKT' and 'F' or 'S' keys.



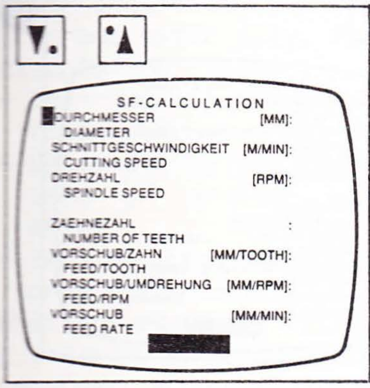
Display screen shows a table used for entering your cutting parameters.

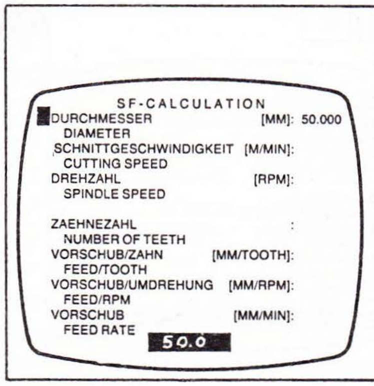
Entering a value

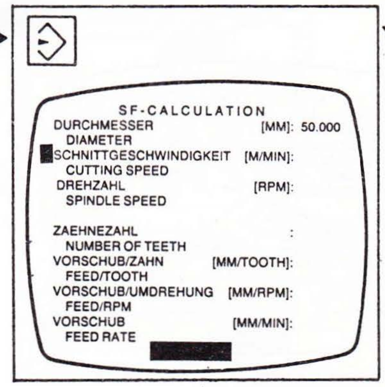
Use 'Cursor up/down' keys to move cursor to desired value.

Enter value.

Press 'Transfer' key.







Cursor is positioned at parameter for which a value is to be entered.

Value entered appears on input line.

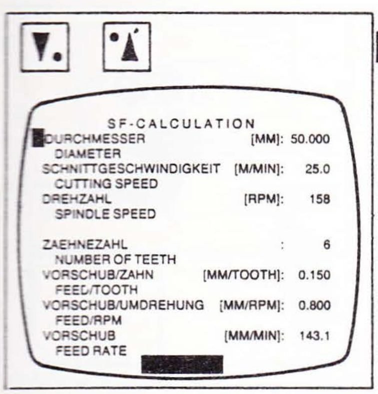
Value is transferred to cutting data table. Input line is cleared. Cursor has moved to next parameter.

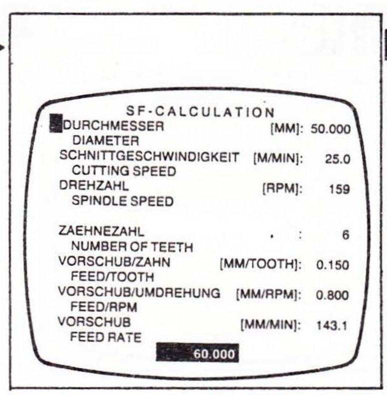
Changing a value

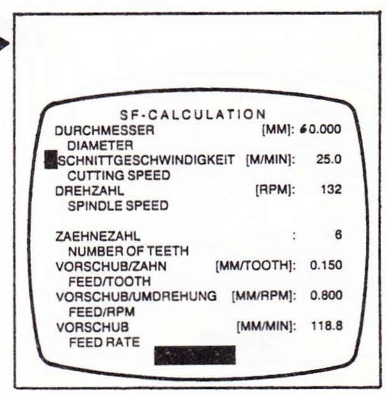
Use 'Cursor up/down' keys to move cursor to desired value.

Enter new value.

Press 'Transfer' key.







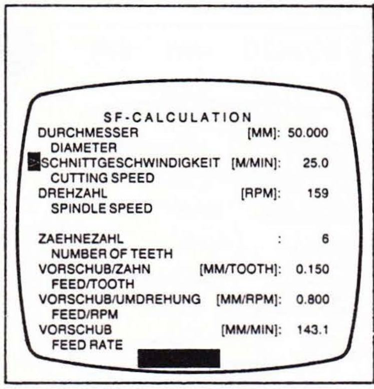
Cursor is positioned at value to be changed.

New value appears on input line.

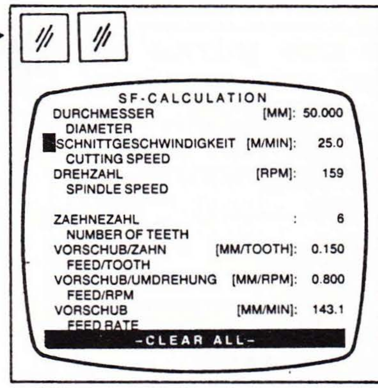
New value is shown in cutting data table. Input line is cleared.

Calculating the values for a new operation

For a completely new calculation of the cutting data:

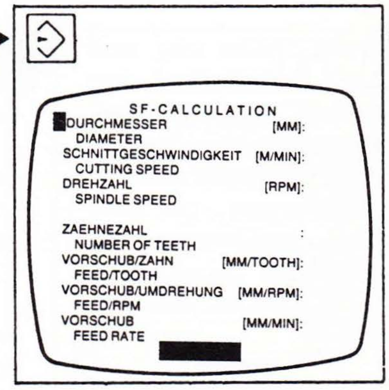


Press 'Delete' key 2x.



'CLEAR ALL' appears on input line.

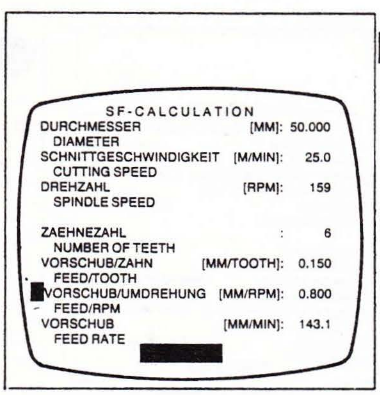
Press 'Transfer' key.



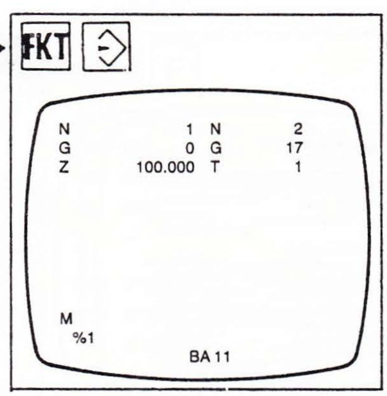
Cutting data table is cleared. New values can now be entered.

Switching off the cutting data calculator

Remember or jot down cutting data to be used.



Press 'FKT' and 'Transfer' keys.



Block last worked on appears on display screen. Program input can be continued.

Data transfer to cassette recorder

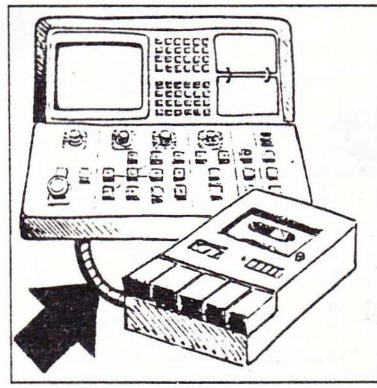
The storage capacity of the control is sufficient for storing almost any number of programs (six-digit program numbers).
On the other hand, it is often desirable to store programs externally, for example
- for the purpose of filing them;
- because they are to be used on another machine;
- because they are used only occasionally.

What you need for filing a program is a cassette recorder with counter, automatic level control and a listening-in facility during recording (we recommend a GRUNDIG recorder).
For filing we recommend an SK 2333 'Compusette' data cassette manufactured by BASF.

Commercial cassettes accommodate one or several programs, depending on their length. You may also use miniature cassettes (recording time 8 minutes) to file each program separately.

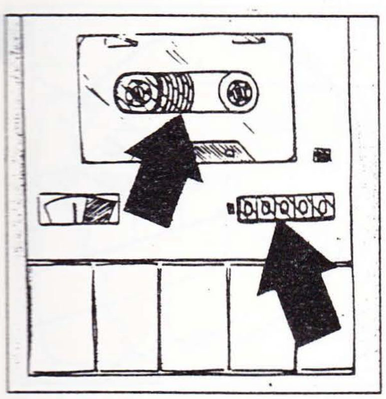
NOTE
It is possible to generate DIALOG 3 programs on the DIALOG 4 control and to rename DIALOG 4 programs into DIALOG 3 programs.
However, a DIALOG 3 program must not contain any DIALOG 4 functions, otherwise an error code will be indicated when reading out the program from the DIALOG 4, and reading out is terminated.

Connect control and cassette recorder by means of connecting cable (Stock No. 2280 000340).



Receptacle on control: at bottom of cubicle; on cassette recorder use recording input.

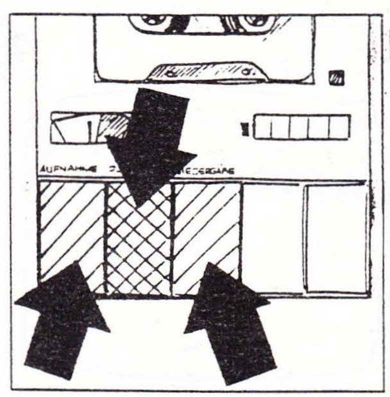
Rewind cassette and set counter on recorder to zero.



Find blank section on cassette: listen in and wait until neither uniform nor interrupted audio signal can be heard.

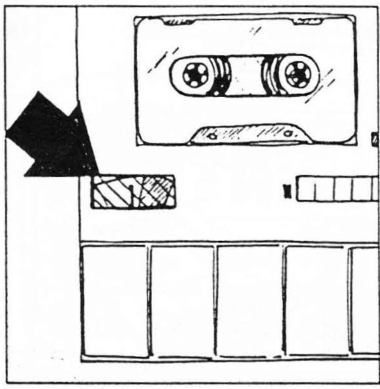
Jot down counter reading.

Press pause button on recorder, then simultaneously press recording and replay buttons.

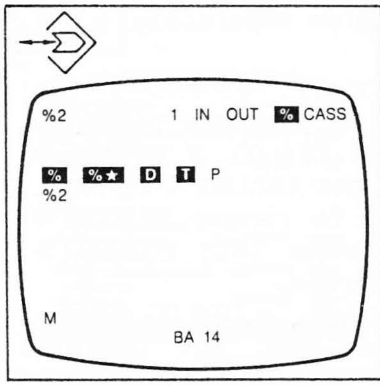


Audio signal indicates that control is ready for data output.

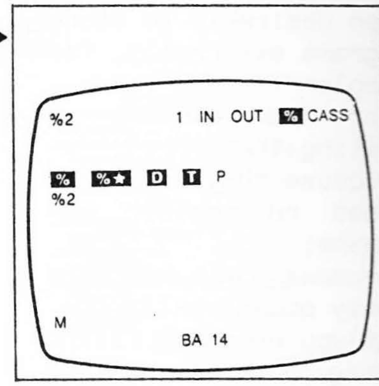
Set recorder for automatic level control, if required.



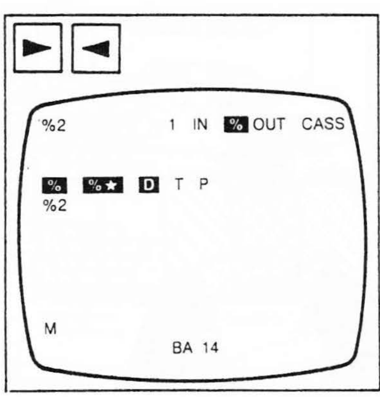
Select mode 14.



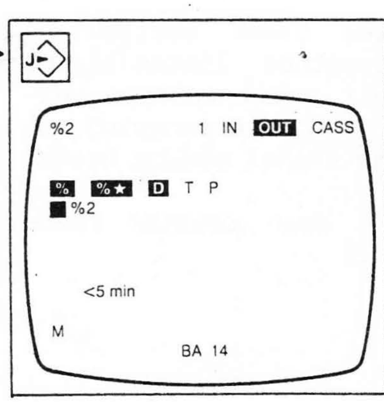
Select 'CASS' interface (see page 7-88).



To set control for data output: use 'Cursor right/left' keys and move cursor to 'OUT'.

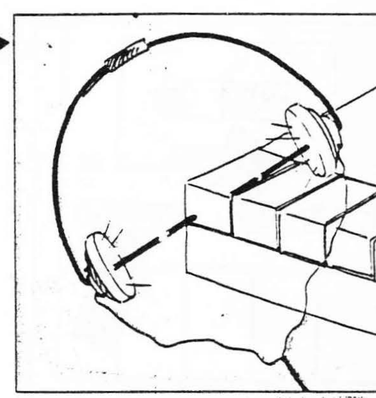


Press 'Acknowledgment' key.

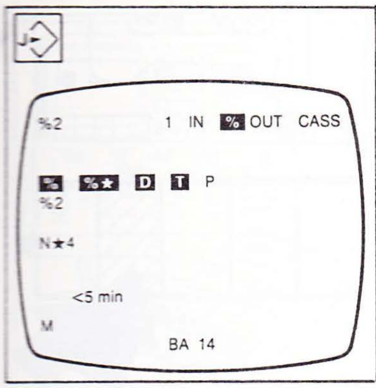


Current program is shown on the left side of display screen. Highlighted area about 'OUT' increases in size.

Check whether uniform audio signal can be heard in earphones.

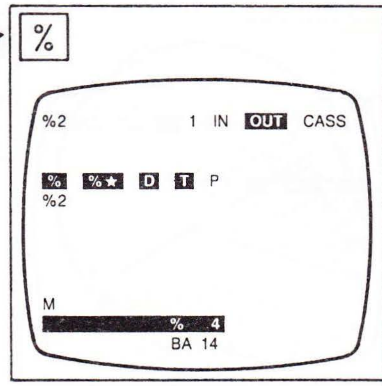


If the program shown on the screen is to be recorded:
release pause button on recorder and again press 'Acknowledgment' key.



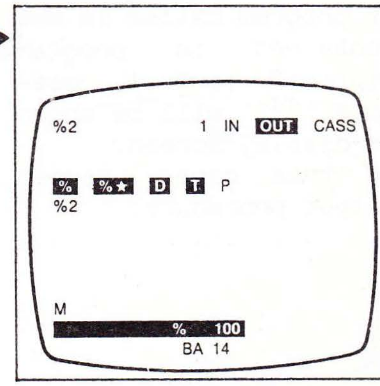
Control reads out program. Blocks being read out are shown on display screen.

If a different program is to be read out:
press '%' key.



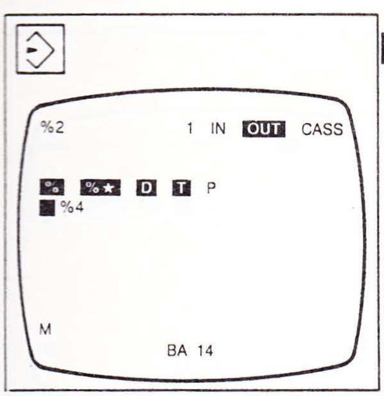
Input line appears on display screen.

Use alphanumeric keyboard to enter program number.



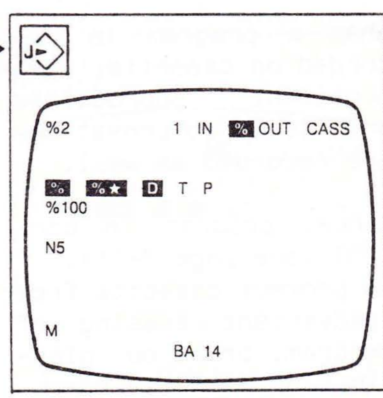
Program number appears on input line.

Press 'Transfer' key.



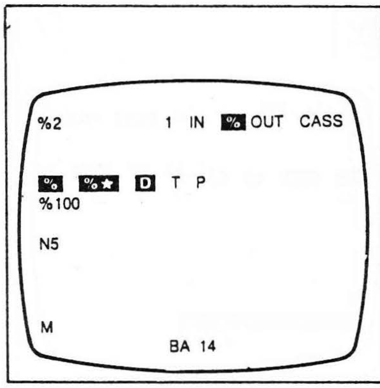
New program number appears in upper left.

Release pause button on recorder and press 'Acknowledgment' key.



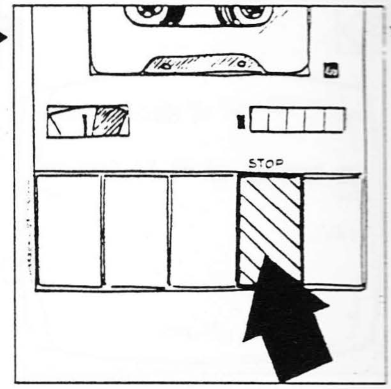
Control reads out program. Blocks and numbers tool compensations are shown on display screen as they are read out.

If program called is not contained in program store of control, message '???' will be shown on display screen.
In this case, repeat output procedure.



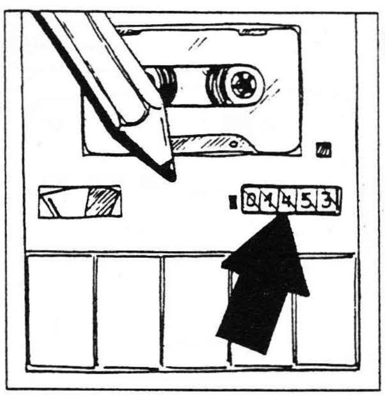
When recording is completed, highlighted area underneath 'OUT' disappears. Audio signal in earphones turns to uniform sound.

Switch off recorder.



Program is now on cassette, but still also contained in program store of control.

Jot down counter reading and write program number etc on cassette label.

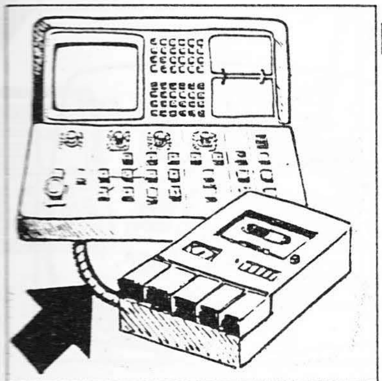


- When a program is recorded on cassette, the pertinent subroutines and tool compensations are recorded as well.

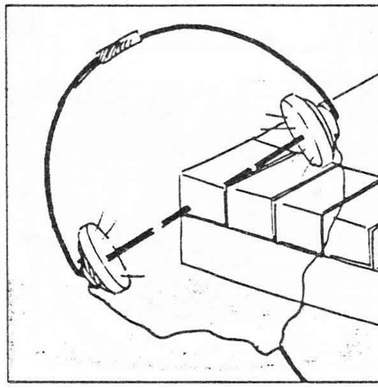
Cancel program in control (see page 7-11).
To protect cassette from inadvertent erasing of program, brake out plastic flap on outer rim.

Data transfer from cassette recorder to control

Connect control and cassette recorder by means of connecting cable.

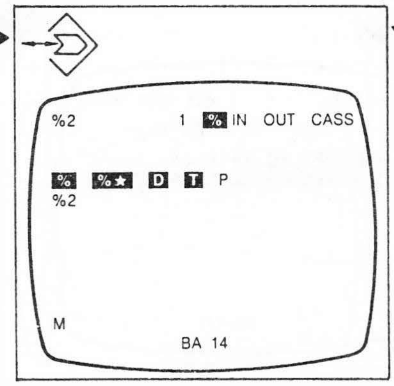


Find program start on cassette tape (counter reading on cassette label).

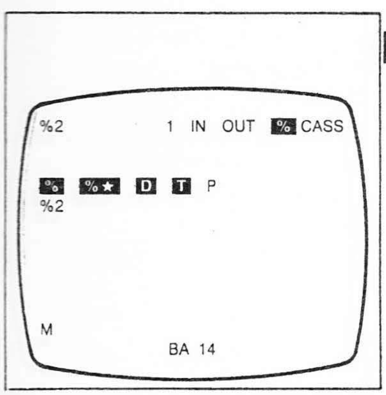


A uniform audio signal should be heard at that point.

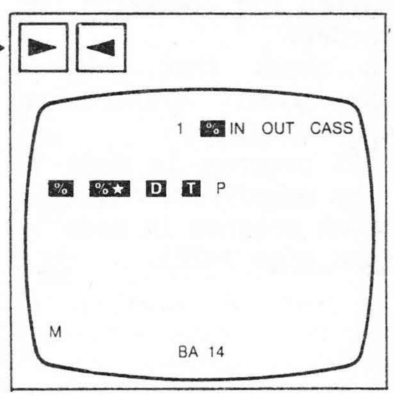
Select mode 14.



If required, select 'CASS' interface (see page 7-88).

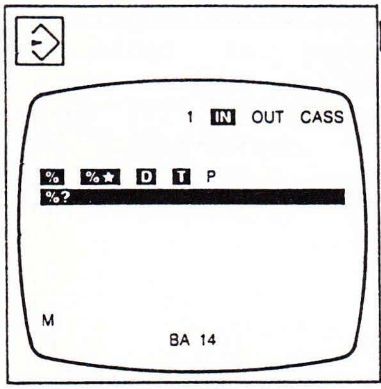


Use 'Cursor right/left' keys to move cursor to 'IN'.



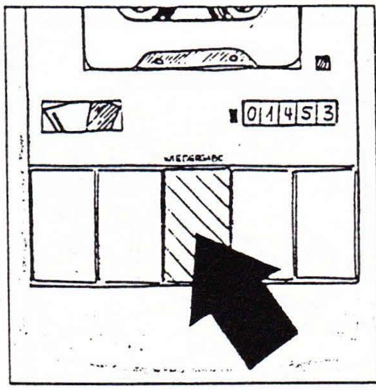
If control already contains a program under the number of the program you want to read in:
cancel program in control (see page 7-11).

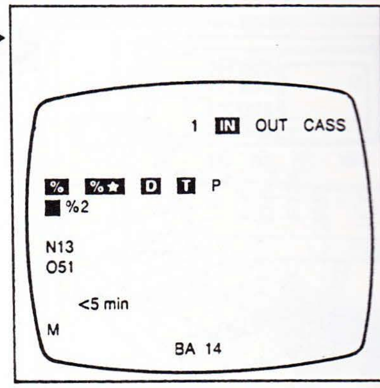
Press 'Acknowledgment' and 'Transfer' keys.



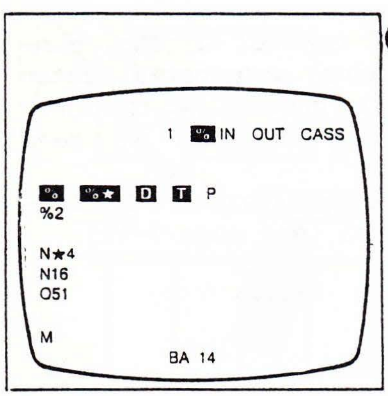
Control is now ready for program input. '%?' appears highlighted on display screen.

Press replay button on recorder.





Program number, block numbers and tool compensations are shown on display screen as they are read into control. The time it will probably take to transfer the program is indicated in minutes.

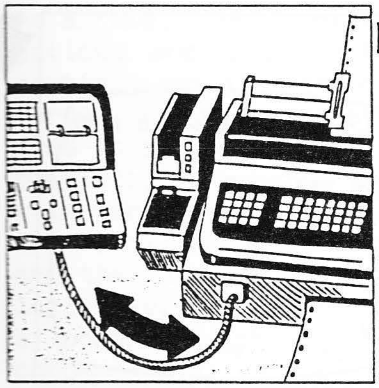


When input is completed, highlighted 'IN' returns to normal.

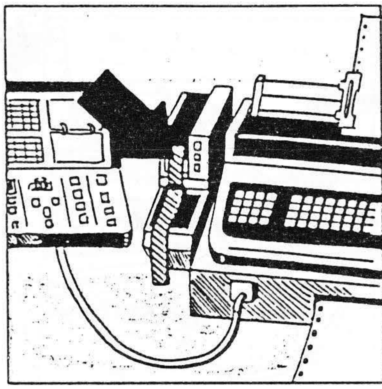
Switch off cassette recorder.
To check that program has been transferred correctly:
call program in mode 13 (see page 7-3);
check program in mode 11 (see page 7-27).

Data transfer to tape reader

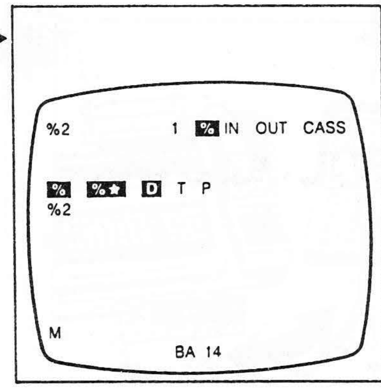
Connect control and tape reader by means of connecting cable.



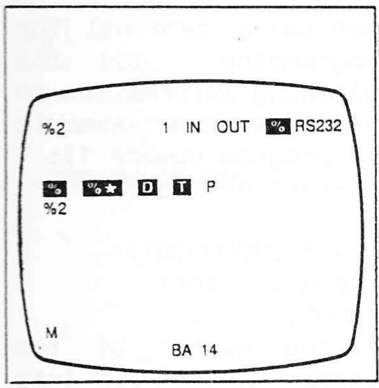
Insert blank tape.



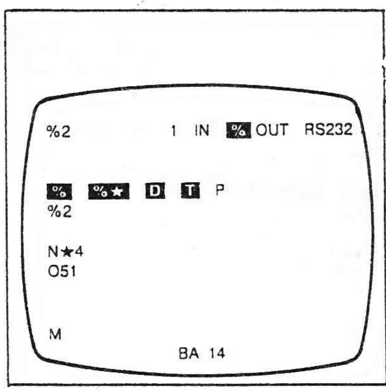
Select mode 14.



Select interface (see page 7-88).



Select program and read out data, proceeding as described for cassette recorder (page 7-49).



When output is completed, highlighted 'OUT' returns to normal.

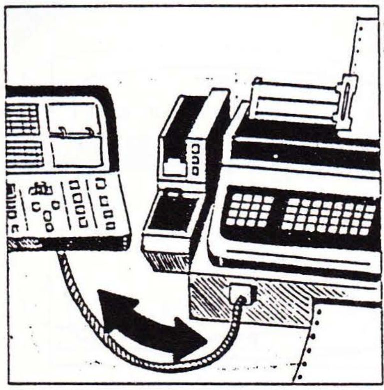
When using terminal for programming, add the following information on the tape (for example for program number 1):

at start of tape:
%
(& % 1/00000000);

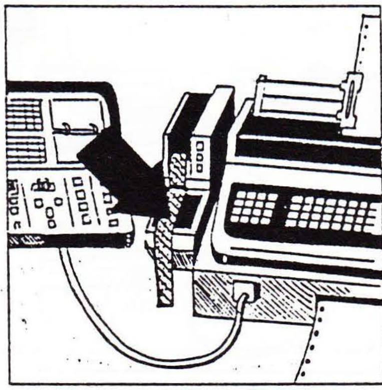
at end of tape:
? 0000.

Data transfer from tape reader to control

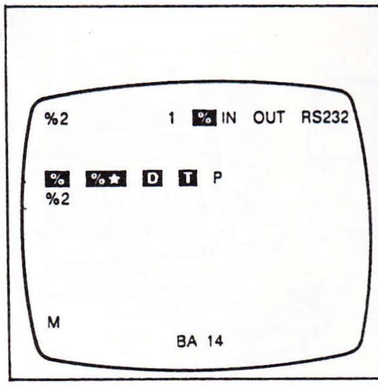
Connect control and tape reader by means of connecting cable.



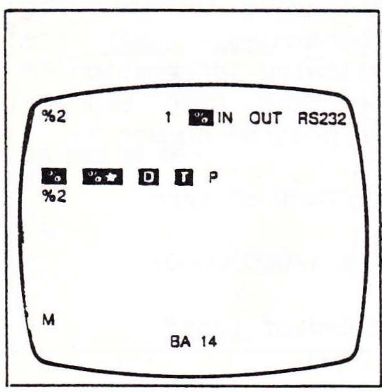
Insert tape with desired program into tape reader unit.



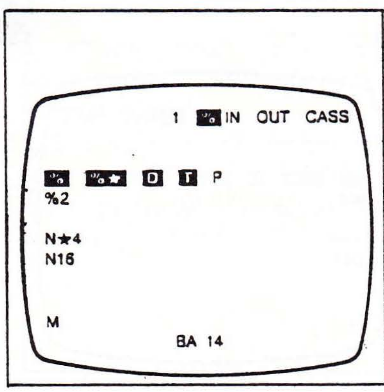
Select mode 14.



Select interface (see page 7-88).



Transfer data, proceeding as described for cassette recorder (page 7-53).



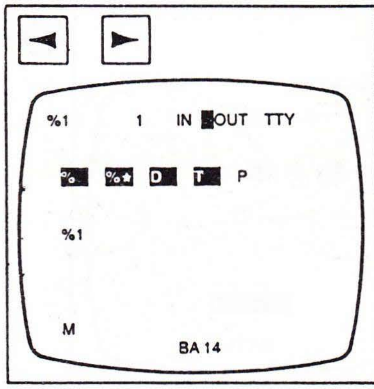
When intput is completed, highlighted 'IN' returns to normal.

● When using terminal for programming, add the following information on the tape (for example for program number 1):
at start of tape:
%
(& % 1/00000000);
at end of tape:
? 0000.
If the number of the program to be read into the control is already contained in the program store, an error message will appear.
Remedy: rename program.

Selective data transfer

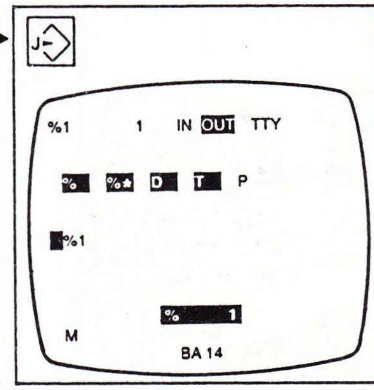
As a rule, tool compensations and local subroutines are also transferred when programs are read in and out. It is possible, however, to transfer the program, the tool compensations and the subroutines selectively. To read out a program without the compensation values in tool compensation store D, for example, proceed as follows:

Use 'Cursor right/left' keys to move cursor to 'OUT'.



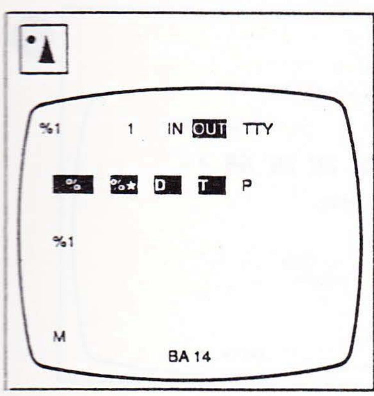
Cursor is positioned at 'OUT'.

Press 'Transfer' key.



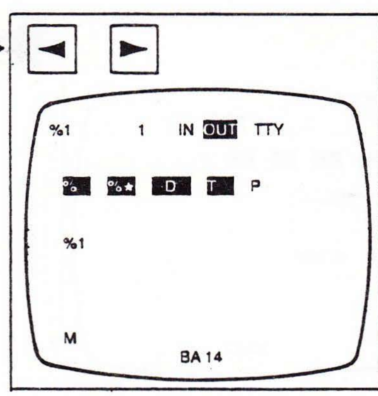
'OUT' is highlighted. Cursor is positioned at program number.
The data not to be read out can now be selected.

Use 'Cursor up' to move cursor one line up.



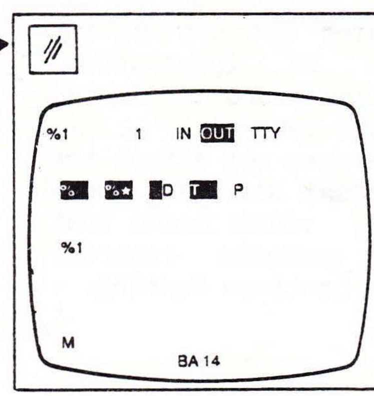
Cursor is now positioned at highlighted '%'.

Use 'Cursor right/left' to move cursor to data block not to be read out (in this case tool compensation store D).



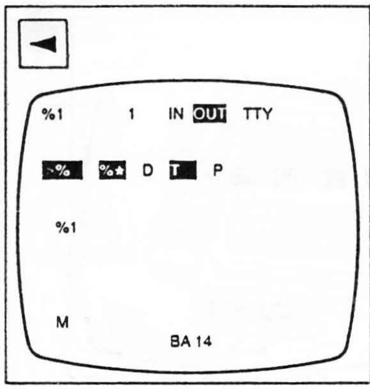
Cursor is positioned at highlighted 'D'.

Press 'Delete' key.



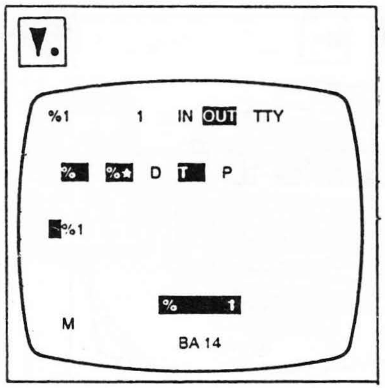
Highlighted 'D' returns to normal.

Use 'Cursor left' to move cursor fully to the left.



Cursor is positioned at '%'.

Use 'Cursor down' to move cursor to program number.



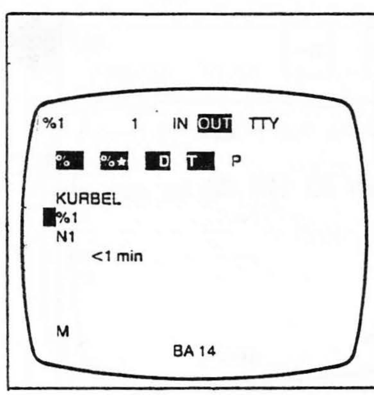
Cursor is positioned at program number.

Now start reading out the program: the data of tool compensation store D will not be read out. To exempt data blocks from being read in, move cursor to 'IN', press 'Transfer' key and proceed in the same manner.

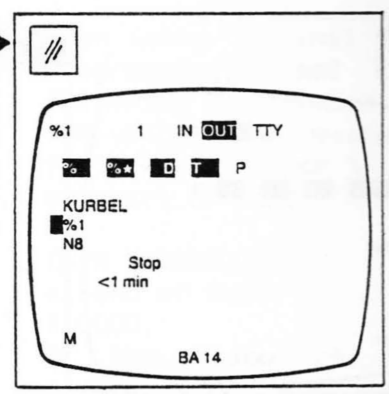
Interrupting data transfer from control

Reading out can at any time be interrupted, continued and terminated on the control. This does not affect the external storage, however, which means that the cassette recorder will continue running.

The data being read out are always displayed underneath the program number on the display screen.



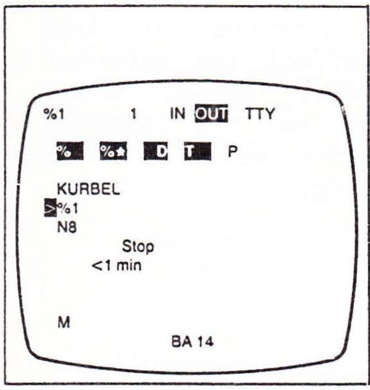
To interrupt data output, press 'Delete' key.



'Stop' appears on display screen. Data output is interrupted.

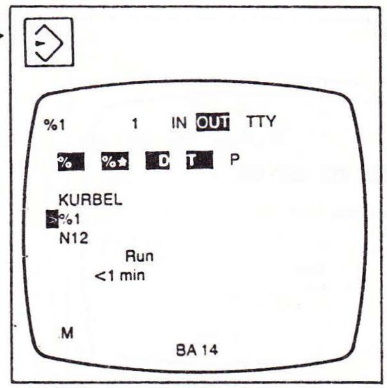
Continuing data transfer from control

Data transfer was interrupted.



Display screen shows 'Stop'.

To continue data output, press 'Transfer' key.

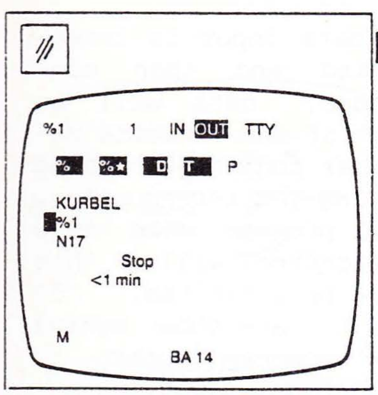


'Run' appears on display screen.
Control continues reading out the program.

● Data output can be interrupted any number of times: no data will be lost.

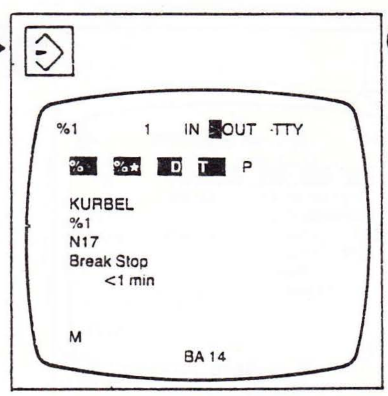
Discontinuing data transfer from control

To terminate data output, press 'Delete' key.



'Stop' appears on display screen.

Then press 'Acknowledgment' key.

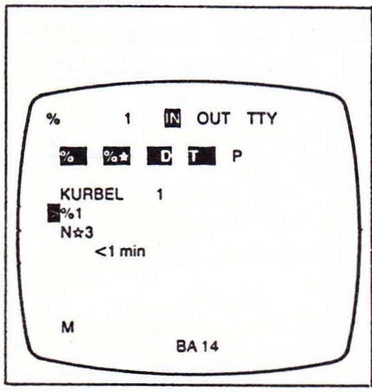


'Break Stop' appears on display screen.

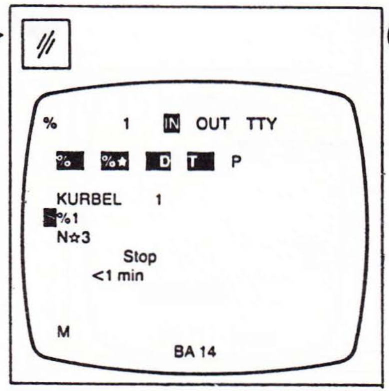
● Program output has been terminated. If you then wish to read out the data, start the procedure all over again.

Interrupting data transfer to control

The data being read in are always displayed underneath the program number on the display screen.



To interrupt data input, press 'Delete' key.

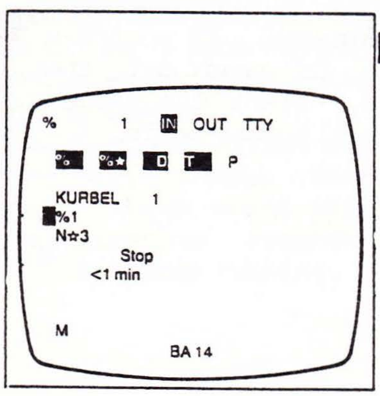


'Stop' appears on display screen.
Data input is interrupted.

● The interruption does not affect the external storage, however, which means that the cassette recorder will continue running.

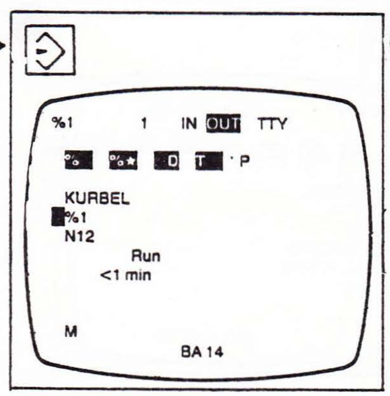
Continuing data transfer to control

Data transfer was interrupted.



Display screen shows 'Stop'.

To continue data input, press 'Transfer' key.

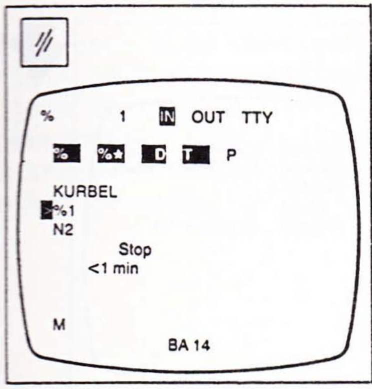


'Run' appears on display screen.
Control continues reading in the program.

● If data input is interrupted and then continued, data will be lost if the cassette recorder continues running during the interruption. The program read into the control will in this case be inhibited. 'S' and 'E' are shown behind the program number in the program list.

Discontinuing data transfer to control

To terminate data input, press 'Delete' key.



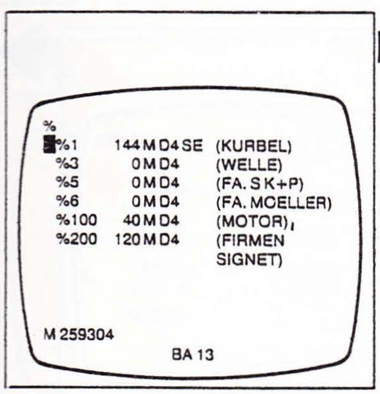
'Stop' appears on display screen.

Then press 'Acknowledgment' key.

'Break Stop' appears on display screen.

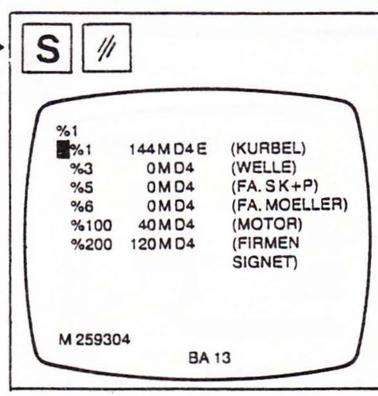
Program input has been terminated. If you then wish to read in the data, start the procedure all over again.
'S' and 'E' are shown behind the program number in the program list.
('S' means 'Program inhibited';
'E' means 'Reading-in error')

Select mode 13.



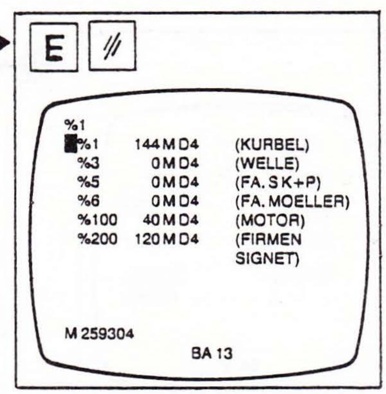
'S' and 'E' are shown behind the program number in the program list.

To work on a program not correctly transferred, press 'S' and 'Delete' keys.



'S' behind program number disappears.

Press 'E' and 'Delete' keys.

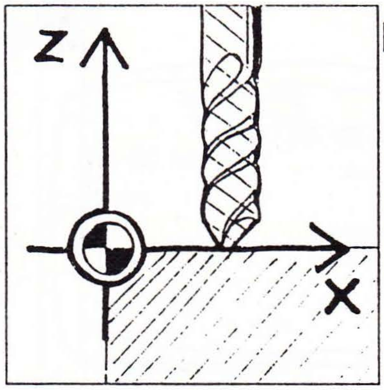


'E' behind program number disappears. Program can now be completed or corrected in mode 11.

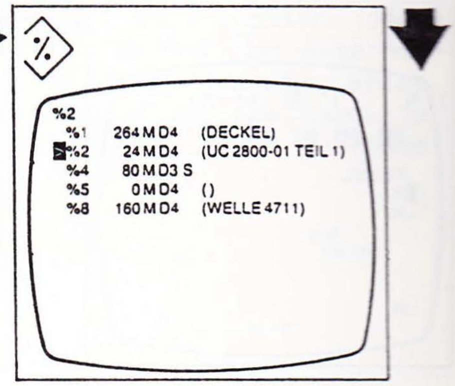
Program-controlled machine operation
Trial run block by block

A new program should first be tried out block by block before a workpiece is machined. In such a trial run, the operation is stopped upon the completion of each block. In rapid traverse blocks the motion can be slowed down by means of feed rate override control (6).

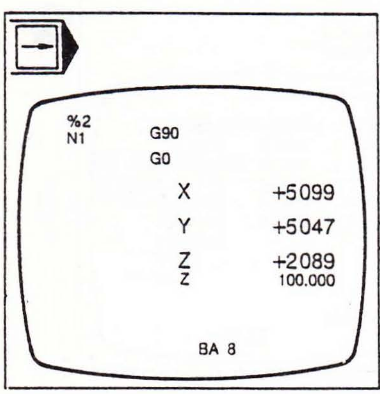
Set up the machine and move tool to starting position.



Select mode 13 and call desired program (page 7-3).

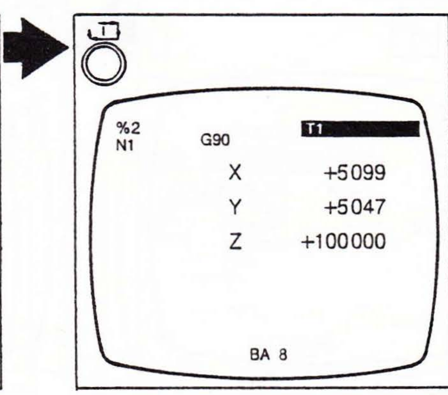


Select mode 8.



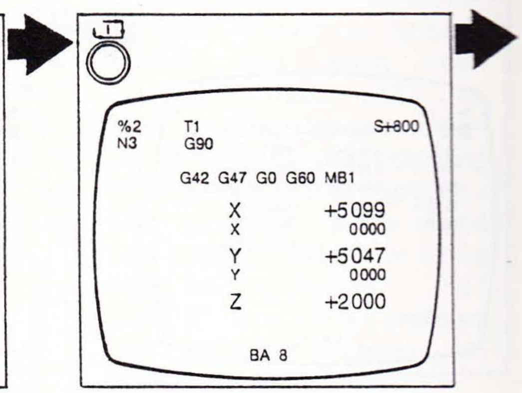
Display screen shows first program block with setpoint and actual values.

Press 'Cycle start' button (20).



Machine executes first block (setpoint display disappears) and stops in the new position. Display screen shows number of first program block with setpoint and actual positions.

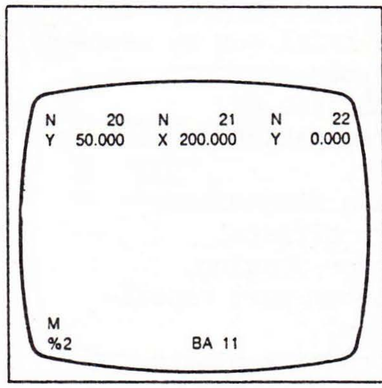
Press 'Cycle start' button (20) again.



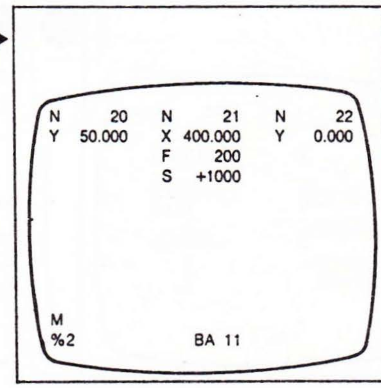
Machine executes next block and stops again. Display screen shows number of next program block and new setpoint position.

After each step continue trial run by pressing 'Cycle start'.
Check cutter path.
In the event of an error in the program:

Select mode 11, find faulty block (page 7-27) and correct as required (page 7-30). Remember block number.

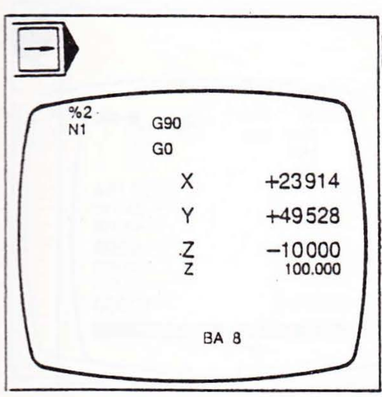


Complete corrected block to main block by entering preparatory function, spindle speed and feed rate.



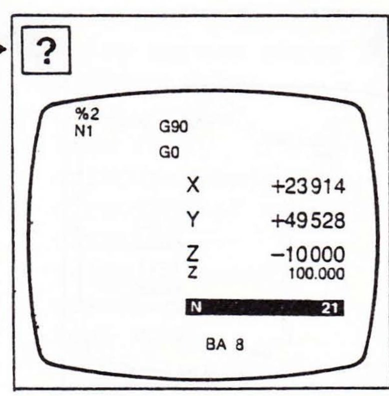
The control needs these data, otherwise cycle cannot be resumed at the point of interruption.

Select mode 8.



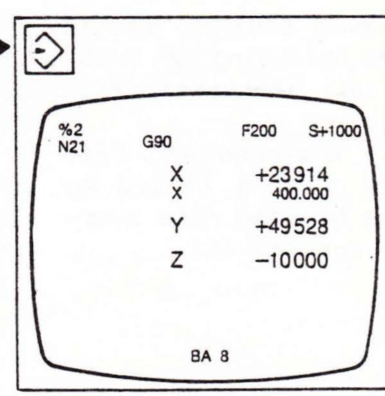
Display screen shows first program block. Machine slides are still in target positions of block where program was interrupted. Target positions of first block

Press '?' key and enter number of corrected block (for example 21).



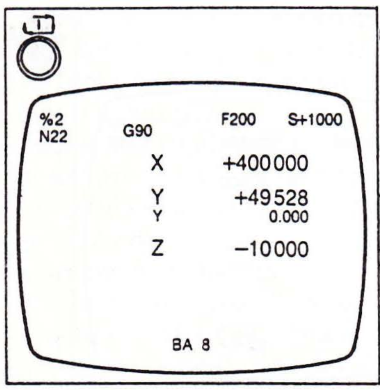
Desired number is shown on input line.
If that is a main block, display screen shows feed rate and spindle speed.

Press 'Transfer' key.



The trial run can now be continued.
Block number moves from input line to top of display screen.

Press 'Cycle start' button (20).



Machine executes corrected block and stops. Display screen shows following block.

After each block, continue trial run by means of 'Cycle start'.
In the case of
- compensation on contour,
- chain dimensions,
- zero offsets,
- mirror imaging,
- program part repetitions,
- subroutine blocks,
- complex program structures (for example, jumps),

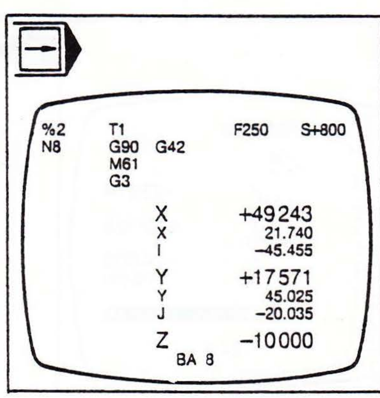
the machining operation cannot, as a rule, be continued from the corrected block because the control lacks additional information required.

In this case continue cycle from the point of last preceding tool change, moving the machine slides to their starting positions after such tool change by manual control.

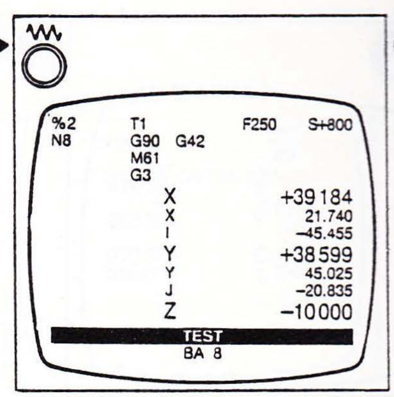
Trial run block by block at increased feed rate

You can save a little time by using a trial run at increased feed rate.
Rapid traverse and feed rates can be varied by means of feed rate override control (6).

Use mode 8 and proceed as in trial run block by block (see page 7-62).



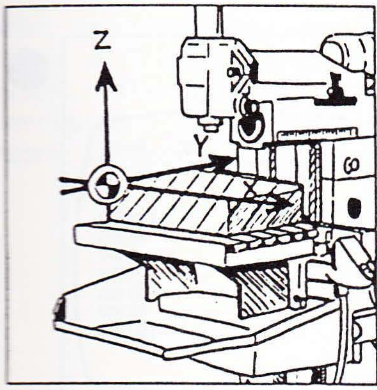
Press 'Rapid traverse' button (8) simultaneously each time you press 'Cycle start'.



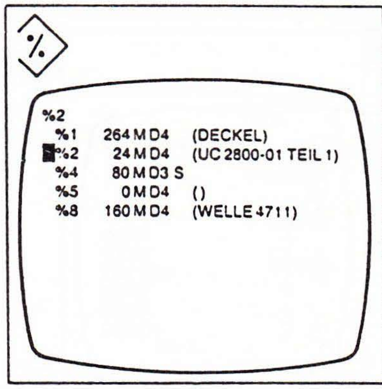
Blocks at feed rate are executed at rapid rate block by block. Display screen shows 'TEST' in inverse display.

Program controlled operation (automatic work cycle)

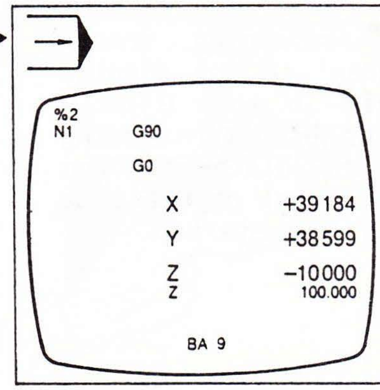
Set up machine and move tool to starting position if required.



Select mode 13 and call desired program (page 7-3).

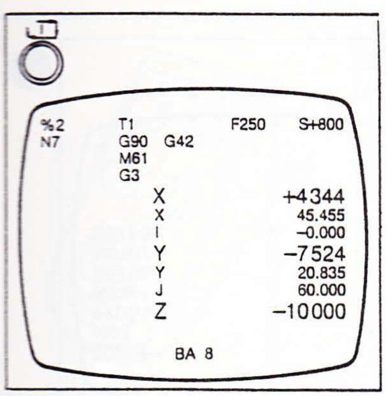


Select mode 9.



Display screen shows first program block with setpoint and actual values.

Press 'Cycle start' button (20).



The machine starts operating in automatic, program-controlled operation.

During the program run, display screen shows the following data:

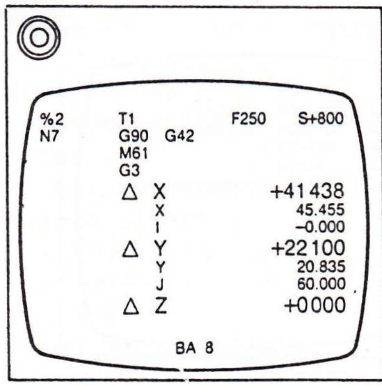
- block number,
- feed rate,
- spindle speed,
- direction of spindle rotation,
- setpoint positions,
- actual positions,
- tool number,
- G functions,
- M functions,
- compensation numbers,
- zero offsets.

For interruptions of automatic work cycle see page 7-68.

Remaining travel display

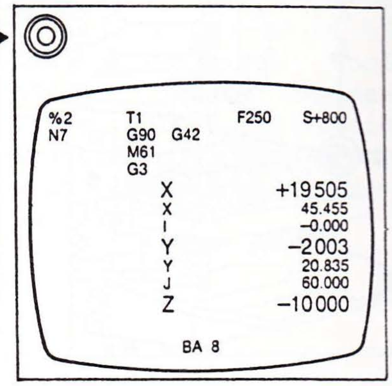
The 'remaining travel display' on the display screen in mode 8 or 9 can be used for reading off the distances from the target points in a block just started.

Press red 'Stop' button (7) during program run (in mode 8 or 9).



Display screen shows triangle ahead of axis addresses with remaining travel in each axis. Target positions are shown underneath.

Press 'Stop' again.



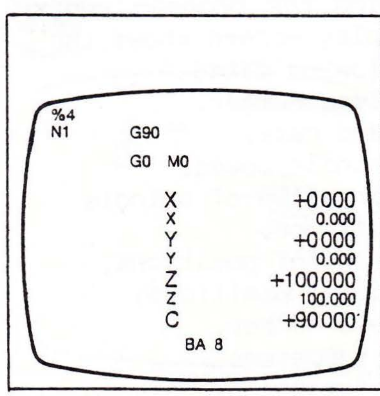
Screen returns to normal display in mode 8 or 9.

Rotating the universal table in program controlled operation

Important: command M0 (program stop) has to be programmed before universal table can be rotated in program-controlled operation.

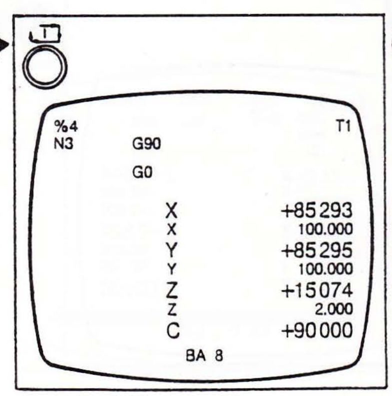
When M0 is reached, the control interrupts the automatic work cycle.

Move universal table to desired position by manual control.



New table position is shown on display screen.

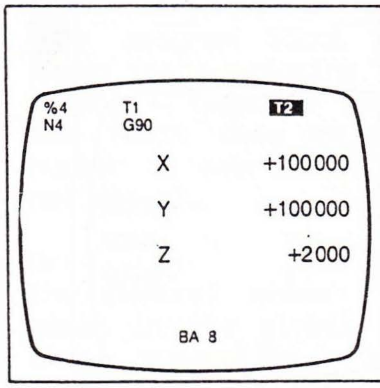
Press 'Cycle start' button (20).



Work cycle continues.

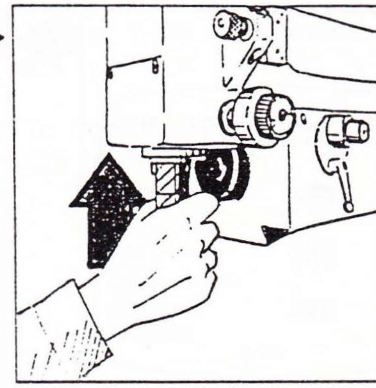
Tool change in program controlled operation

When the program contains a T address in a block, the cycle is interrupted after that block in modes 8 and 9.

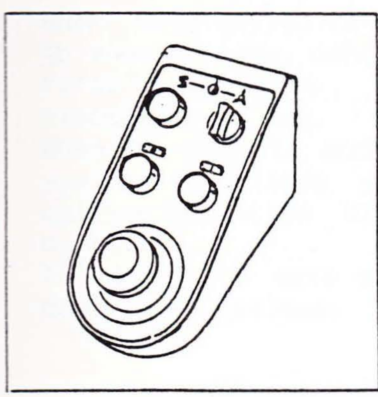


Number of next tool is shown highlighted underneath number of current tool. Tool change indicating lamps (21 and 26a) on control console and remote control start flashing.

Release tool, mount new tool and reclamp (see Chapter 3).

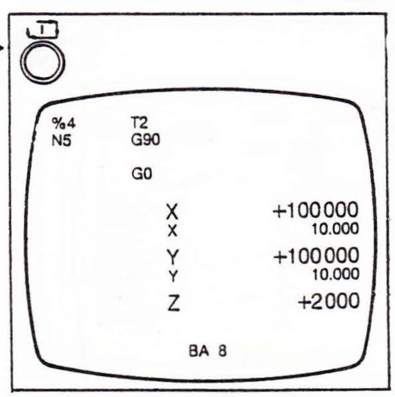


Press 'Tool change' button (21).



Tool change indicating lamps (21 and 26a) go out.

Press 'Cycle start' button (20).



Number of new tool appears in place of number of previous tool.
Program continues with new cutting tool.

- The tool shanks to be mounted should be perfectly clean. Lubricate shanks frequently, but only lightly to avoid contamination.

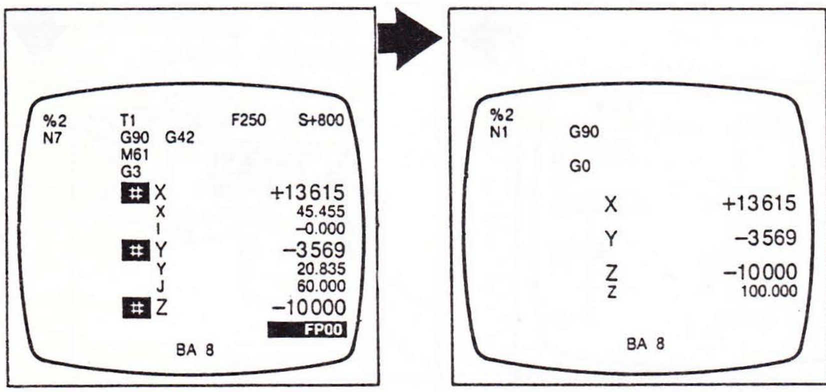
Always fit protective plug to the spindle not in use.

Only use tools of appropriate dimensions, which are suitable for the maximum spindle speeds programmed for your cutting operation.

Interrupting the program in the event of tool breakage

Press 'Emergency stop' button (5).

Jot down the block number and restart machine (see Chapter 3).



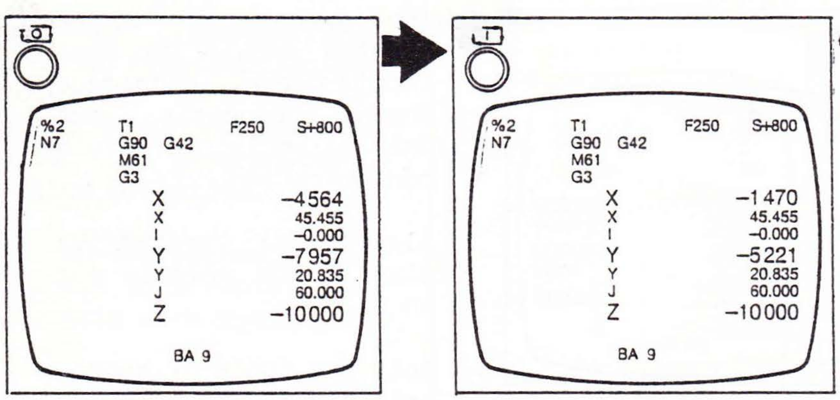
Mount new tool. Change tool compensation if necessary.
Continue program (see page 7-70).

Machine stops immediately. All machine slides are in clamped condition. Display screen shows error code FP0.

Interrupting the program by means of 'Cycle stop' button

Press 'Cycle stop' button (19).

Press 'Cycle start' button (20).

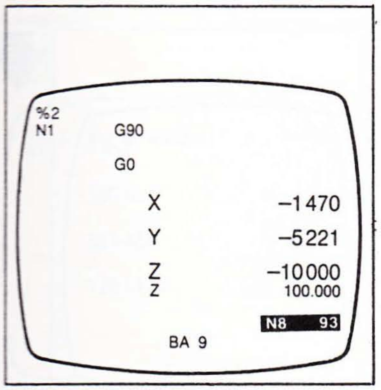


Machine slides and spindle stop; control remains energized.

Work cycle continues.

Interrupting the program in the event of program errors

If the information contained in the program is incomplete or false, the control will interrupt the cycle a few blocks in advance.



Push button 'Cycle stop' (19) starts flashing. Display screen shows error code and number of block likely to be faulty.

If fault is found in a global (%0...) or local (%*...) subroutine, the control may display two different fault codes.

Either:
the control shows the main program block in which the subroutine is called, together with the fault code (block number in subroutine is not shown).

Or:
the control shows the block in the global or local subroutine, together with the fault code (subroutine number remains unknown).

To find faulty block, proceed as follows: switch to graphics mode and run program. Cycle will continue up to last correct block.

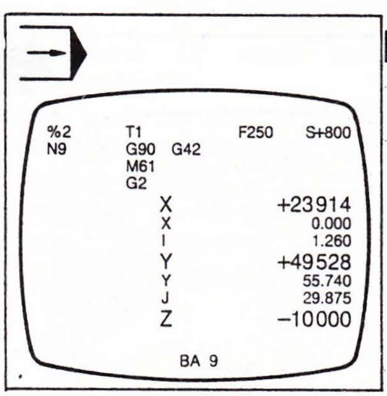
Correct error in program (page 7-30 and Chapter 8).

To continue, press 'Cycle start' button (20).

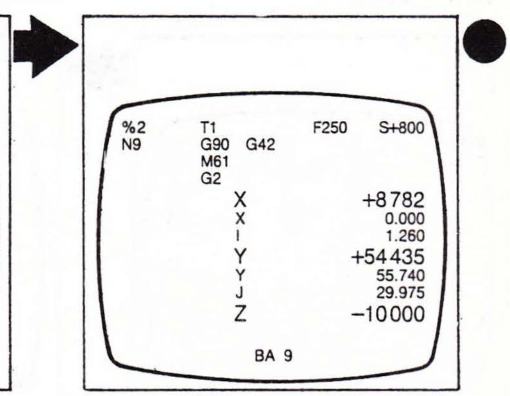
Interrupting the program by changing the mode of operation

When mode switch is set to another mode, control first completes the block in progress.
When changing to mode 8 you can continue your cutting operation block by block.
To return to auto mode proceed as follows:

Select mode 9.



Press 'Cycle start' button (20).



Work cycle continues.

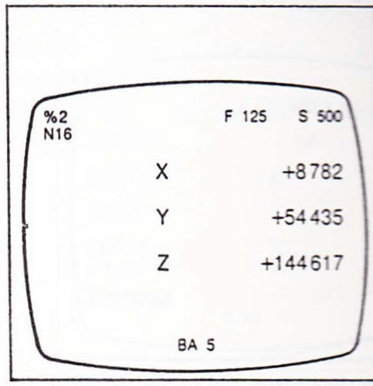
Continuation of program after interruptions

In the case of
- compensation on contour,
- chain dimensions,
- zero offsets,
- mirror imaging,
- program part repetitions,
- complex program structures (for example, jumps),

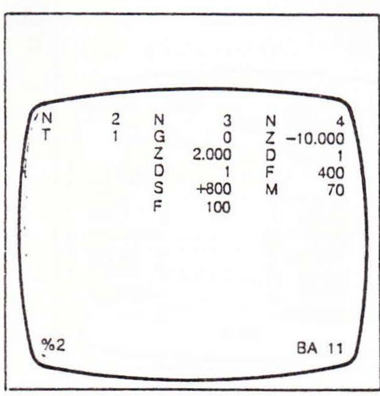
the machining operation cannot be continued from the point of interruption because the control lacks additional information required.

Preferably continue from first block after last preceding tool change or a similarly marked point in the program.

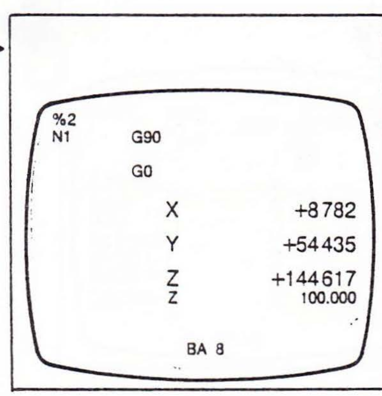
Use mode 5, 6 or 7 to move machine slides to a position where program can be continued.



If necessary use mode 11 to complete such block to a main block by entering spindle speed and feed rate.

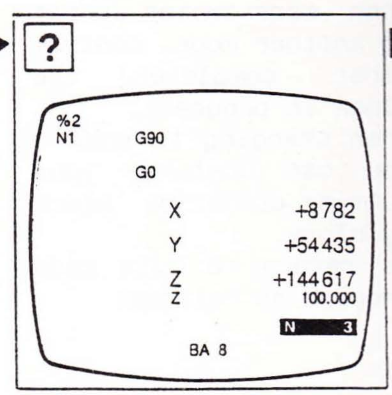


Select mode 8 or 9.



Display screen shows first block of interrupted program with setpoint and actual values.

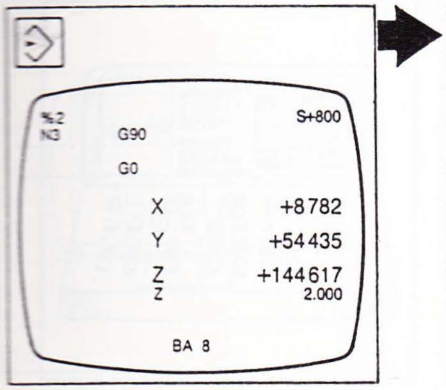
Press '?' key, enter desired block number.

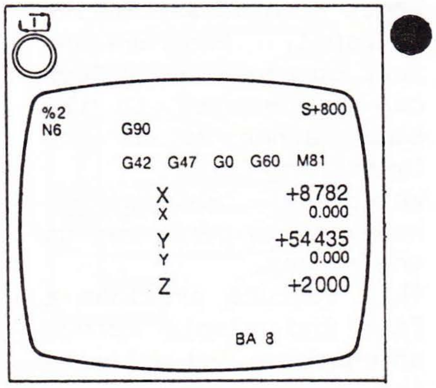


Desired number appears on input line.

Press 'Transfer' key.

Press 'Cycle start' button (20).





Block number moves to top of display screen.

Work cycle continues.

Continuation of program after interruptions within cycles

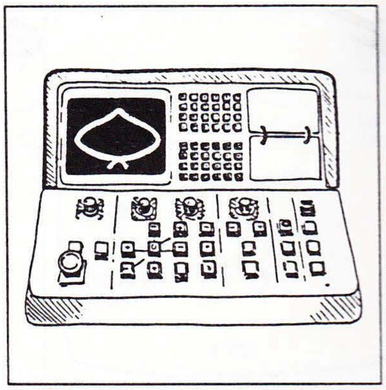
If a program is interrupted by means of 'Cycle stop' within a milling or drilling cycle, the cycle can be resumed by pressing the 'Cycle start' button, provided the mode has not been changed (important exception: tapping cycle G84, see Chapter 6).

Procedure if mode has been changed:
move tool to position in which it was before cycle was called.
Continue program from block containing cycle call (see page 7-70).

Screen graphics (option)

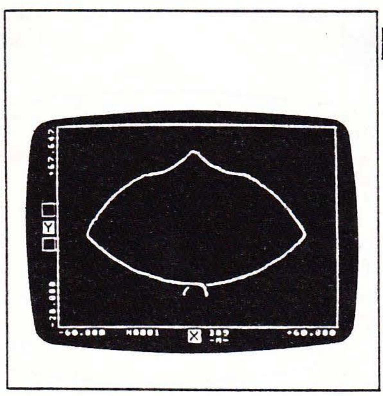
The machining of a workpiece can be simulated on the display screen in the 'graphics' operating mode. The machine slide and spindle drives are disabled while in this mode.

Modes 8 to 15 are active as usual. Programs and tool compensation values can be entered in the same manner as in the 'machining' mode.
Workpiece 'machining', however, is performed on the screen.
This feature provides a fast and simple method of checking and optimizing a part program.



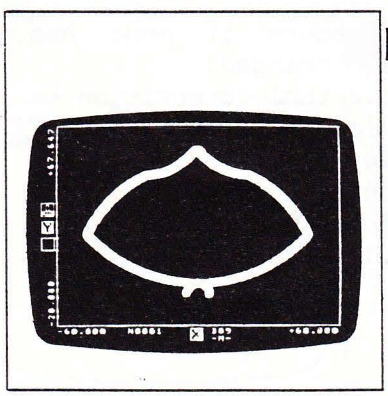
Simulation options

Simulation of cutter axis path



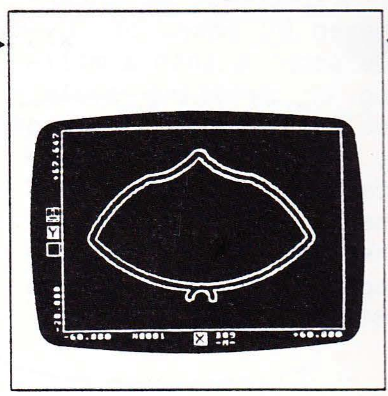
The path of the cutter axis is shown in the form of a line. Points where infeed motions are performed are marked by a small circle.

Simulation of tool path

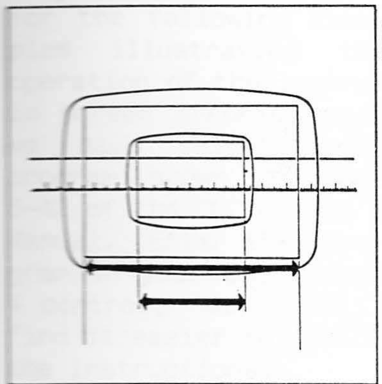


The path of the cutting tool is shown as a solid surface. The control takes the tool radius entered in the tool compensation store into account.

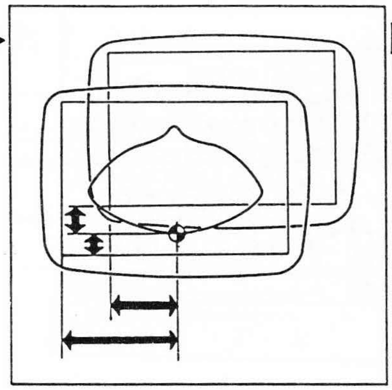
Simulation of workpiece contour



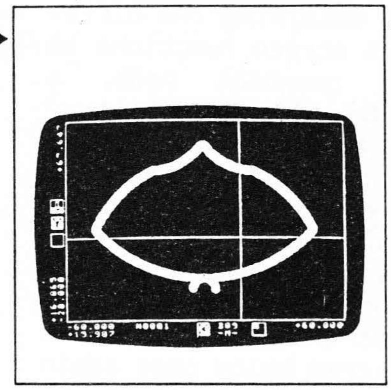
Upon 'draining' of the solid surface of the cutter path, the screen shows the workpiece contour.



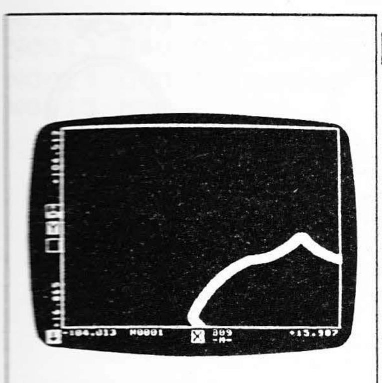
Any simulation scale can be selected in mode 16: you simply specify to what length on the actual workpiece the screen width should correspond.



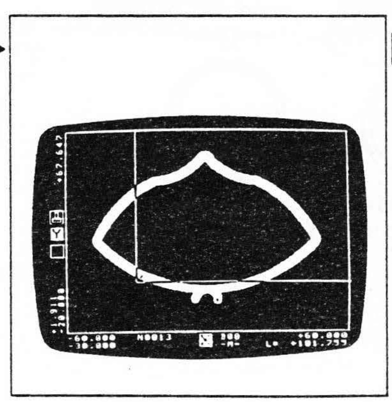
In addition, you may select any location for the zero point (based on lower left corner of screen) of the program to be simulated on the screen.



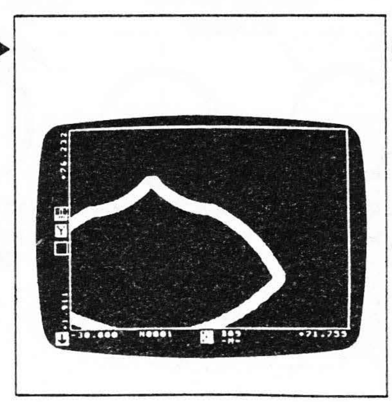
The cross wires can be used to shift the screen display over the workpiece, while maintaining the simulation scale.



The desired quadrant is then transferred to the screen in original size.



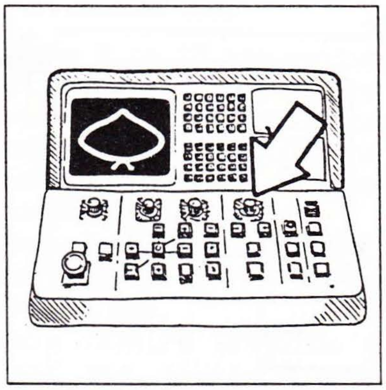
With the aid of a window you can select sections of the screen display for enlargement.

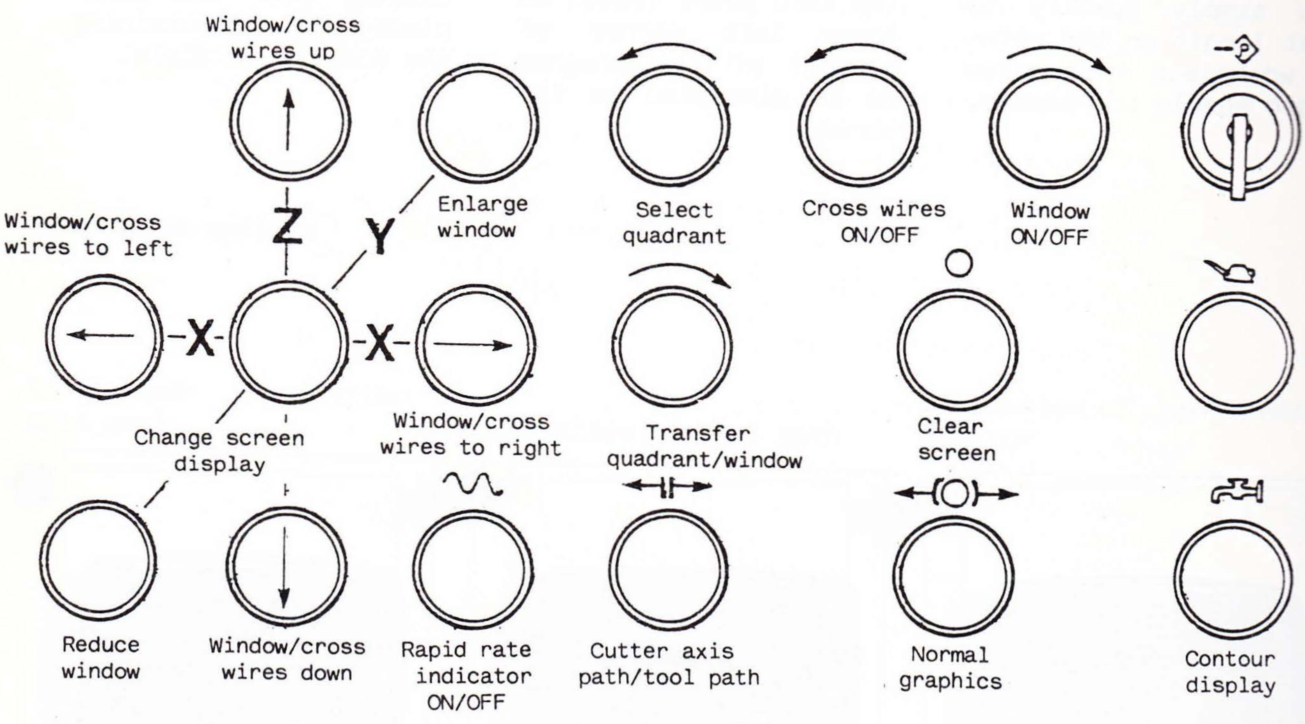


The window is then enlarged to fill the entire screen, which means that the simulation scale is altered.

Push button assignment in graphics mode

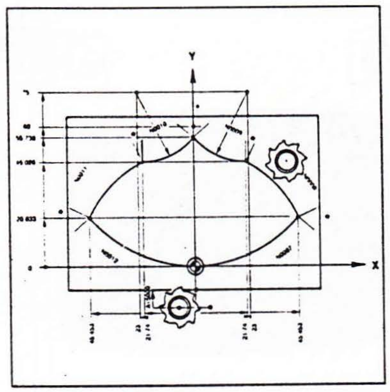
For obtaining the different screen functions in the graphics mode, a number of push buttons on the control console of the machine has been given alternate functions which are effective in the graphics mode. For quick reference you will find the diagram below once again on the rear cover of this manual.





Operating the screen graphics

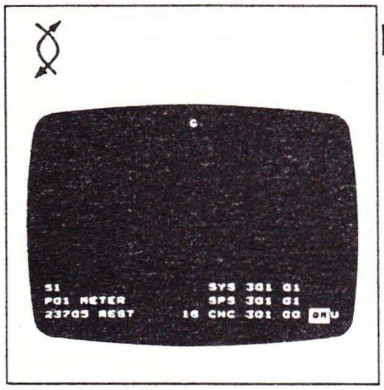
For the following examples illustrating the operation of the control in screen graphics mode we have selected the program shown on page 6-42 of the CNC DIALOG 4 Manual. Enter this program in your CNC DIALOG 4 control, so you will find it easier to follow the instructions.




```
N0001 G00 Z+100000
N0002 T01
N0003 G00 Z+2000 D+01 S+800
N0004 Z-10000 D+01 F100 M70
N0005 F250 M70
N0006 G42 D+51 G47 A+5000 X+0 Y+0
      G00 G60 M61
N0007 G03 X+45455 Y+20835 I+0
      J+60000
N0008 G03 X+21740 Y+45025 I-45455
      J-20835
N0009 G02 X+0 Y+55740 I+1260
      J+29975
N0010 G02 X-21740 Y+45025 I-23000
      J+19260
N0011 G03 X-45455 Y+20835 I+21740
      J-45025
N0012 G03 X+0 Y+0 I+45455 J+39165
N0013 G40 G47 A+5000
N0014 G00 Z+100000
N0015 M30
```

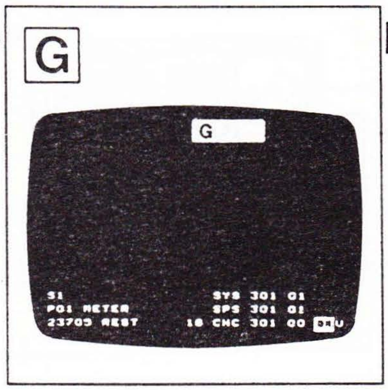
Switching to screen graphics mode

Select mode 16.



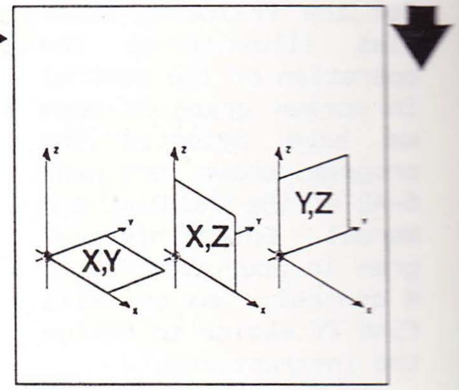
Letter G appears on input line.

Press 'G' key.

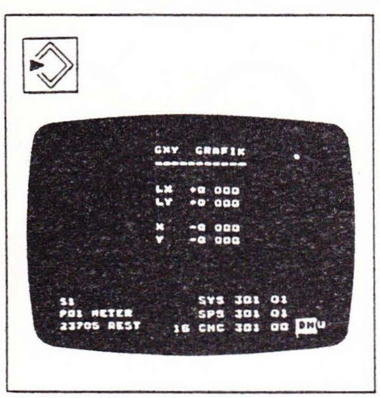


'G' appears highlighted on screen.

The highlighted area is used to enter the coordinate axes of the plane to be displayed on the screen.



Press address keys of the desired axes, then 'Transfer' key.



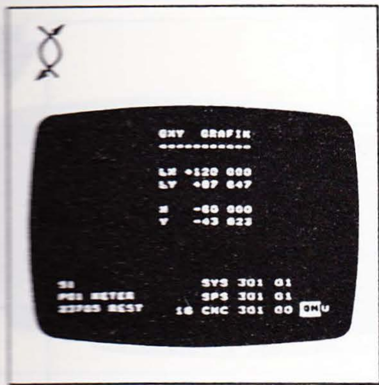
The machine is now in graphics mode. Drives are disabled as soon as next mode change is entered.

Functions obtainable in graphics mode:

- programs can be selected (mode 13) and altered (mode 11);
- tool compensation values can be entered (mode 10);
- programs can be 'machined' (modes 8 and 9). Screen display shows machining operation in the plane selected;
- programs can be read in and out (mode 14).

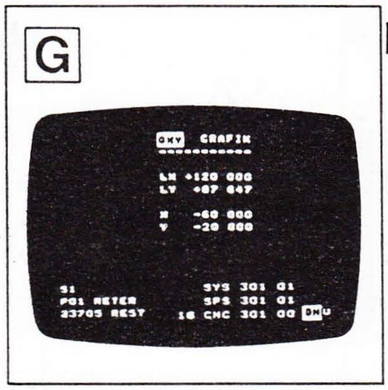
Switching to machining mode

Select mode 16.



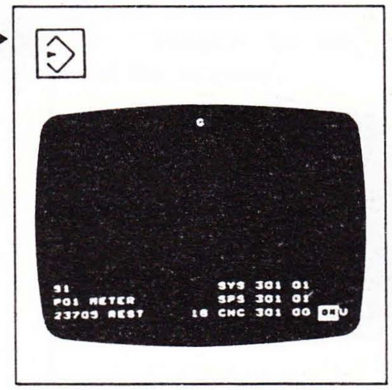
Plane, dimensional and positioning data are displayed on screen.

Press 'G' and 'Delete' keys.



'G appears highlighted on input line.

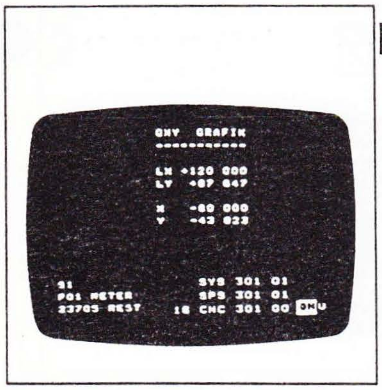
Press transfer key.



The machine is now in 'machining' mode.
Drive motors are operative again as soon as next mode change is entered.

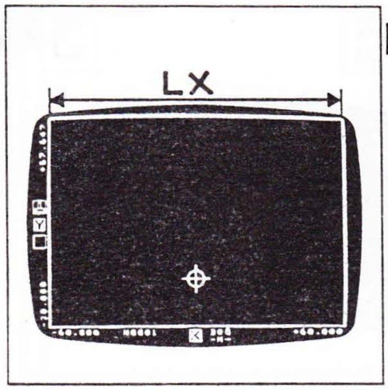
Simulation scale and position of image

Switch control to graphics mode.

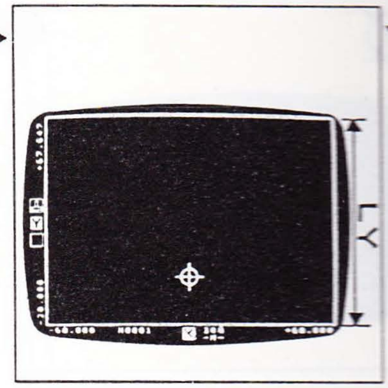


Plane, dimensional and positioning data for size and position of image are displayed on screen.

Simulation scale:



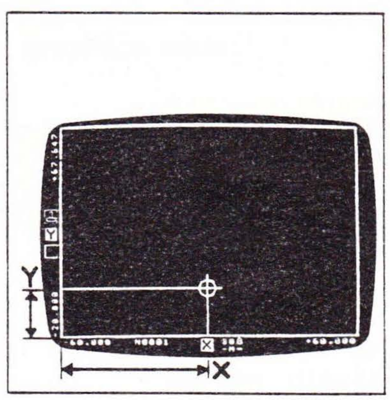
Line 1 refers to screen width.
The letter indicates the axis direction corresponding to the transverse direction on the screen.
The number shown indicates the actual length on the workpiece corresponding to the screen width.



Line 2 refers to screen height.
The letter indicates the axis direction corresponding to the vertical direction on the screen.
The number shown indicates the actual length on the workpiece corresponding to the screen height.

Regardless of whether you enter the numerical value for line 1 or 2, the control will in each case calculate the other numerical value in accordance with the screen format.

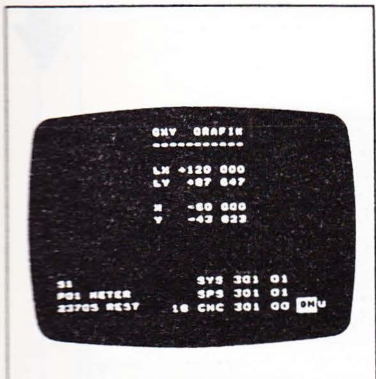
Position of image:



Line 3 and 4 indicate the location of the lower left corner point (based on the current program zero point).

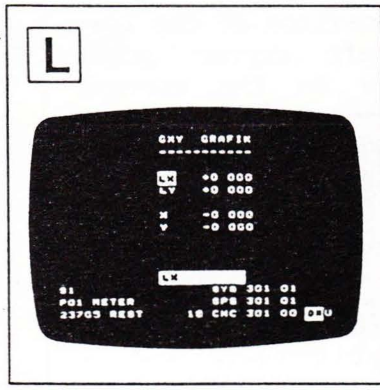
Selecting the simulation scale

Switch control to graphics mode.



Plane and dimensional data are displayed on screen.

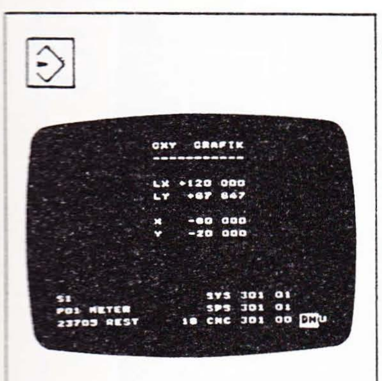
Press 'L' and 'X' or 'Y' keys.



'L' and selected coordinate appear highlighted on input line.

Highlighted area is used to enter length to be simulated on screen.

Enter desired length of image and press 'Transfer' key.



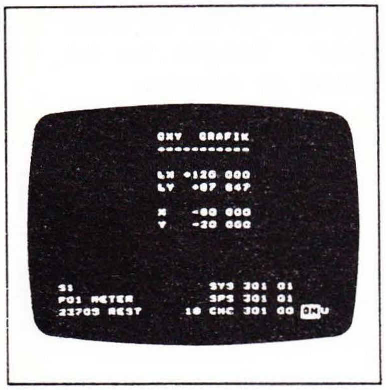
Desired length of image appears on line 1.

The corresponding simulation length in the second axis is calculated by the control.

Example X/Y plane:
you enter LX, the control calculates LY.
Or else, you enter LY, the control calculates LX.

Entering/shifting the image position

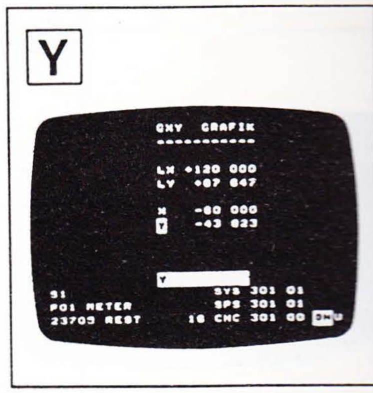
Switch control to graphics mode.



Plane, dimensional and positioning data for size and position of image are displayed on screen.

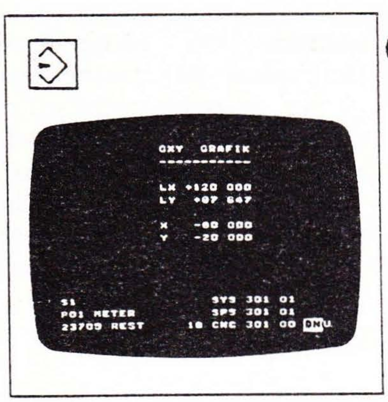
The position of the lower left corner point (based on the current program zero point) can be shifted in both axis directions.

Press address key of axis in which position is to be shifted.



Coordinate appears highlighted on screen.

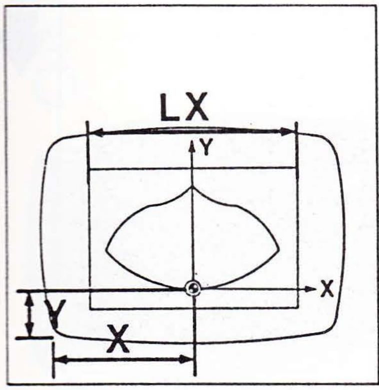
Enter new coordinate (based on program zero) and press 'Transfer' key.



Desired coordinate appears on screen.

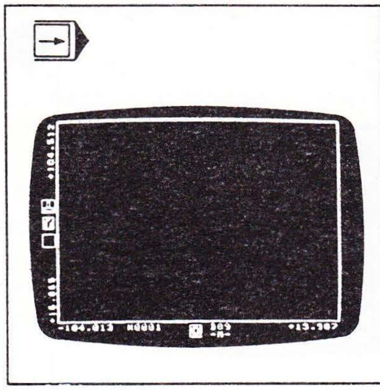
Displaying the tool motions (running the program)

Select program (in mode 13), then select suitable simulation scale and image position (in mode 16).



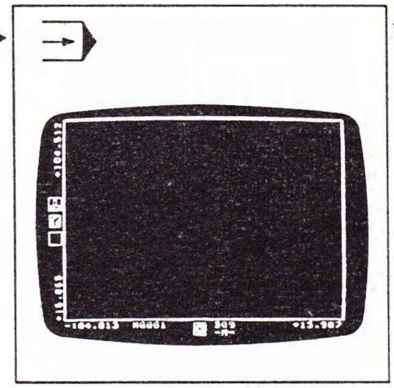
If you want to run the program block by block:

Select mode 8.



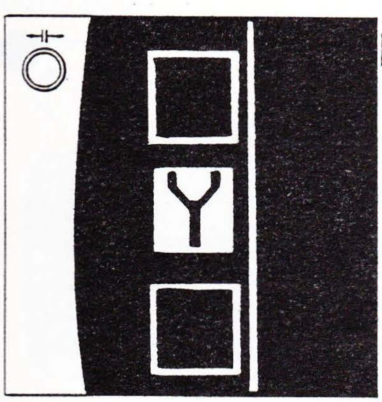
If you want to run the program without interruption:

select mode 9.



If you want to simulate the tool path:

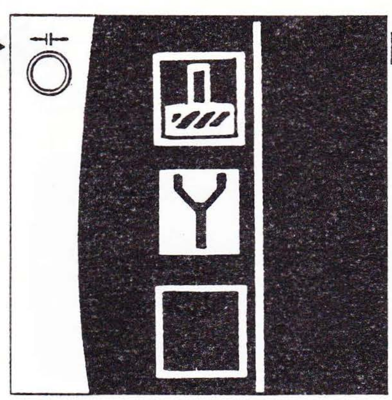
press 'Cutter axis path/ tool path' button.



Tool indicator disappears from left edge of screen.

If you want to simulate the cutter axis path:

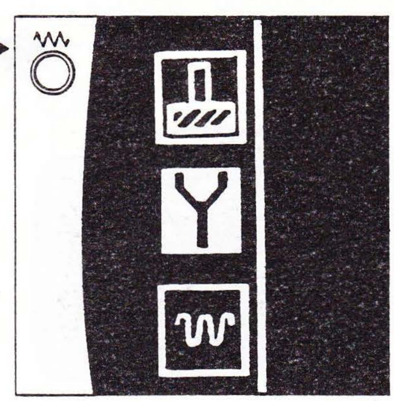
press 'Cutter axis path/ tool path' again.



Tool indicator appears at left edge of screen.

If you want to display rapid traverse motions:

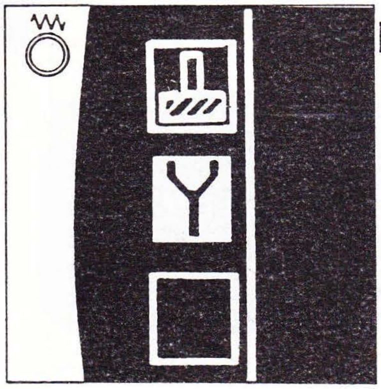
press 'Rapid ON/OFF' button.



Rapid rate indicator appears at left edge of screen.

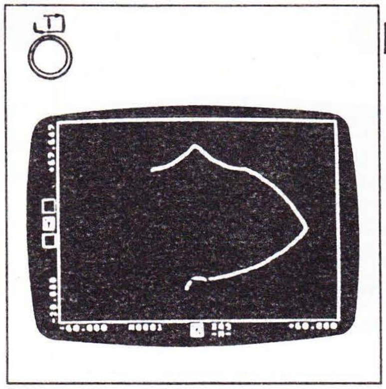
If you do not want to display rapid traverse motions:

press 'Rapid ON/OFF' button again.



Rapid rate indicator disappears from left edge of screen.

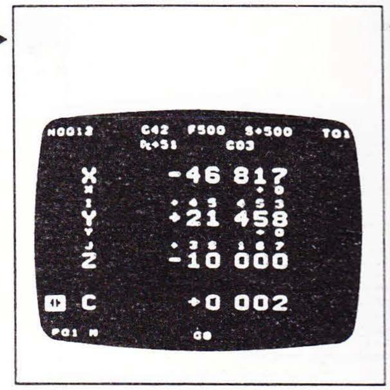
Press 'Cycle start' button (20).



Programmed cutter paths are displayed on screen.

If you want to display currently machined program block on screen:

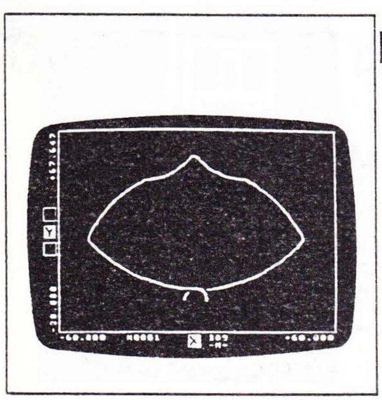
press 'Change screen display' button.



Screen display in effect in current operating mode is shown on screen.

If you want to return to graphics simulation:

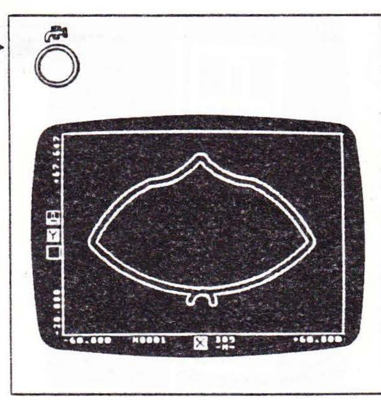
press 'Change screen display' button again.



Graphics simulation appears on screen.

After 'machining', the contour can be displayed:

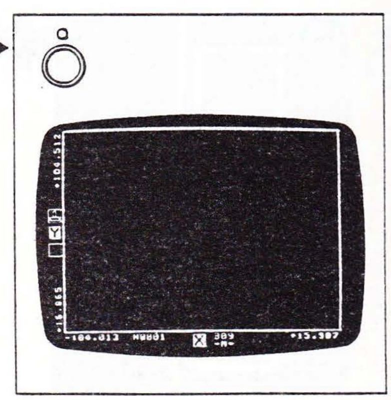
press 'Display contour' button.



Milled contour appears as frame on screen.

To clear screen display:

press 'Clear screen' button.

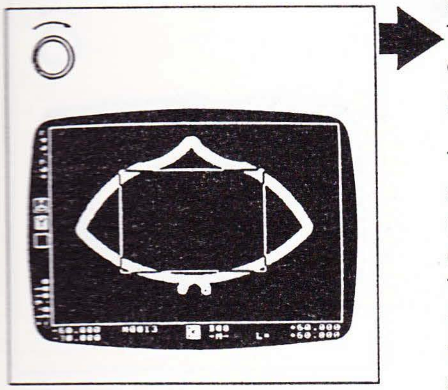


Screen is blank again.

Shifting/enlarging the image (window)

When 'machining' is completed:

press 'Window ON' button.



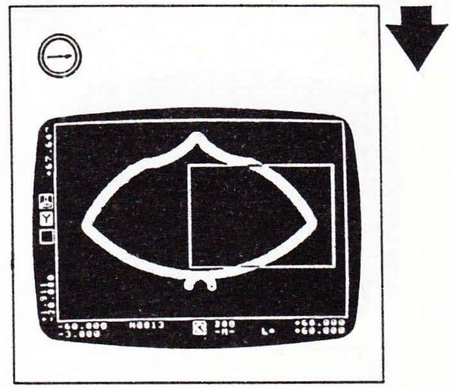
Window is shown on screen.

The coordinates of the lower left corner point appears at lower left of screen.
They are based on the current program zero point.

The window may be shifted up and down, to the right and left, enlarged or reduced in size.
This makes it possible to accurately determine the window section to be simulated if the program is run again.

To shift window:

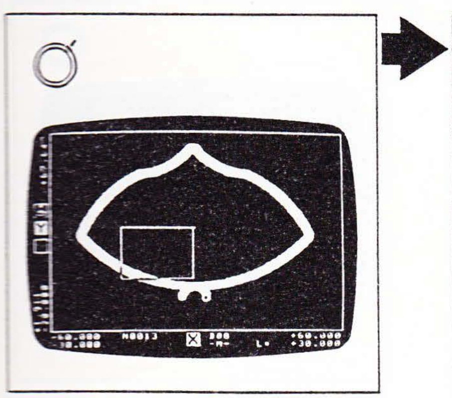
press direction button (7) for the desired direction.



Window moves across screen until button is released.

To reduce window size:

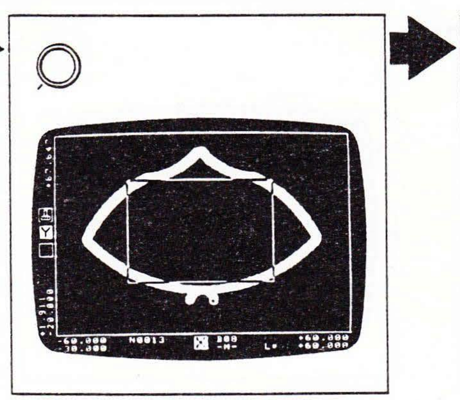
press 'Reduce window' button.



Window continues decreasing in size until button is released.
Lower left corner point keeps its position.

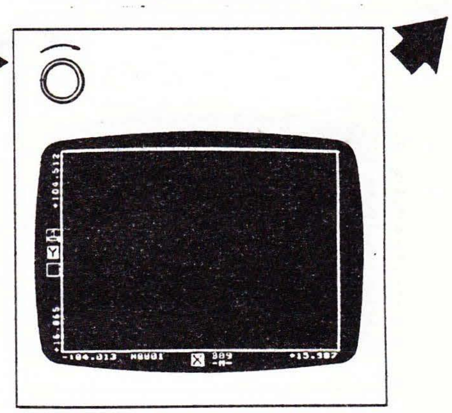
To increase window size:

press 'Enlarge window' button.



Window continues increasing in size until button is released.
Lower left corner point keeps its position.

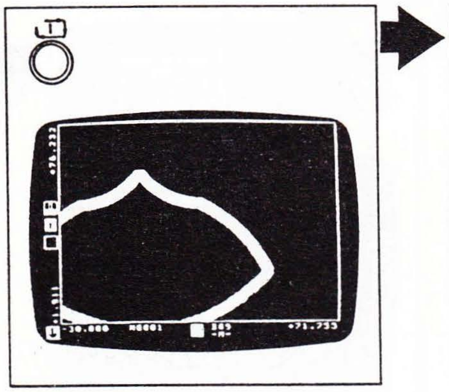
Press 'Transfer quadrant/window' button.



Screen is cleared.

To rerun the program:

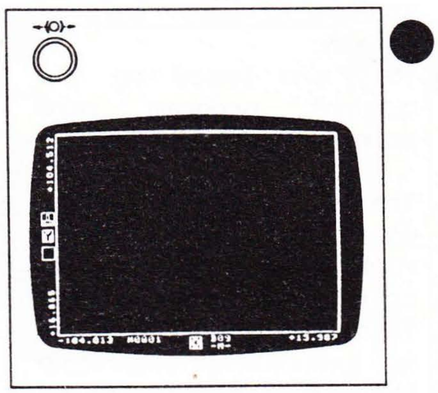
press 'Cycle start' button (20).



Contents of window appear in full format on screen.

To return screen to original display:

press 'Normal graphics' button.

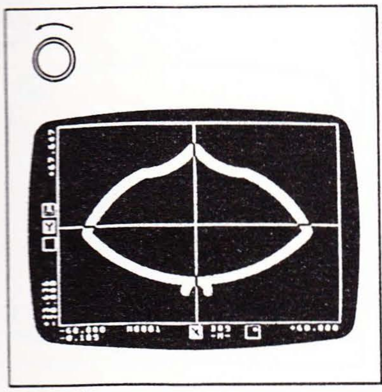


Screen display returns to normal; screen is cleared.

Shifting the image (cross wires)

When 'machining' is completed:

press 'Cross wires ON/OFF' button.



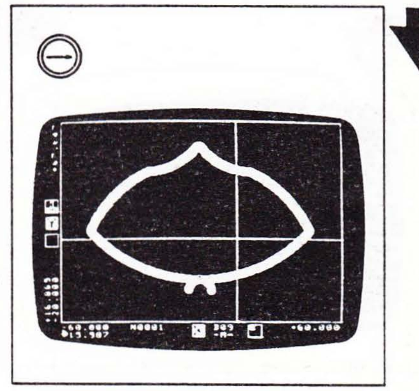
Cross wires appear on screen.

Coordinates of point of intersection appear at lower left of screen. They are based on the current program zero point.

The cross wires can be shifted up and down and to the right and left, and then one of the quadrants can be enlarged to full screen format.

To shift cross wires:

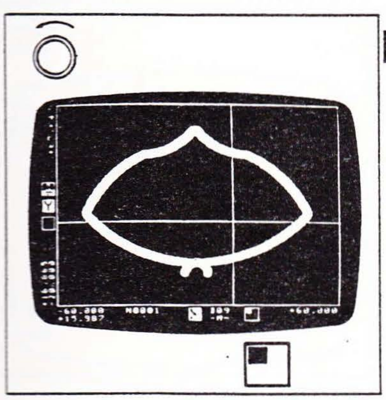
press direction button (7) for the desired direction.



Cross wires move across screen until button is released.
Coordinates of the point of intersection are displayed at lower left edge of screen.

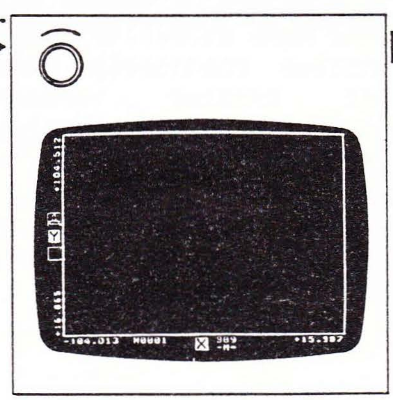
To select quadrant:

press 'Select quadrant' button repeatedly until desired quadrant appears.



Quadrant selected is shown at lower edge of screen.

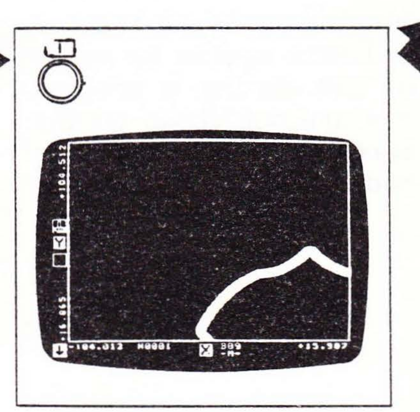
Press 'Transfer quadrant/window' button.



Screen is cleared.

To rerun the program:

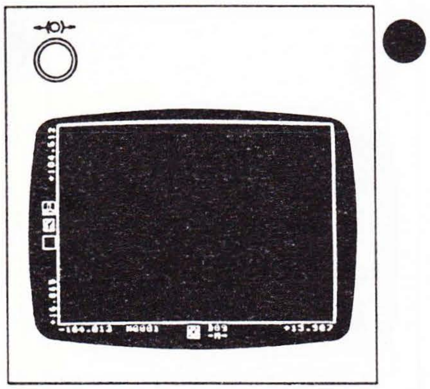
press 'Cycle start' button (20).



Contents of quadrant appear on screen in full format. The scale remains the same.

To return to normal display:

press 'Normal graphics' button:



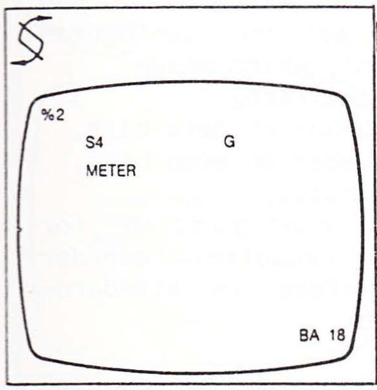
Screen display returns to normal.

Set actual value (G54)

Displacements of the coordinate system by means of G54 during a program run are not shown on the screen in the graphics mode.

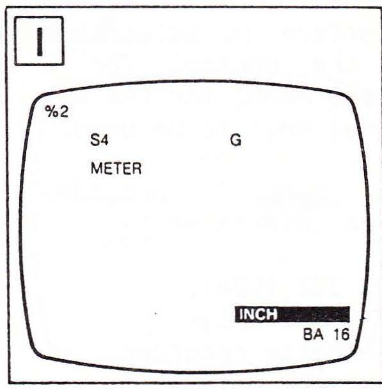
Switching the measuring system from metric to inch display

Select mode 16.



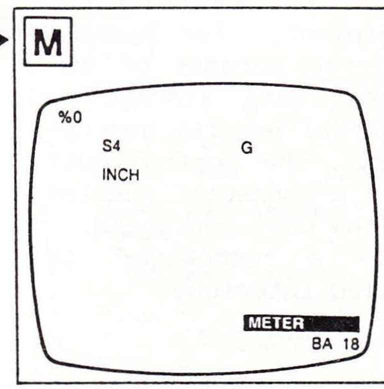
Active measuring system is shown in clear text on screen.

To switch to inch read-out:
press 'I' key.



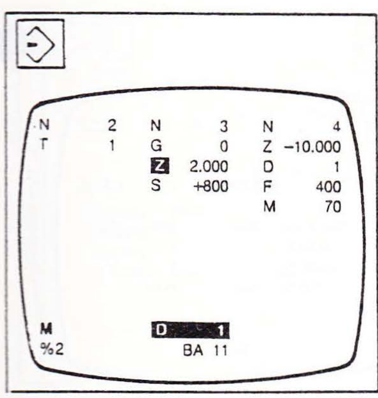
INCH is shown on input line.

To switch to metric (mm) display:
press 'M' key.



METER is shown on input line.

Press 'Transfer' key.



METER or INCH moves to top of screen; system displayed is activated.

The measuring system remains activated when the main disconnect switch (master switch) is switched off.
If a metric program is called in inch mode, the control switches to metric mode automatically.

Interfaces

For interfacing external equipment, for example for the purpose of external data storage or external program generation, the control must have a connector adapted to the unit concerned. Such a connector is called interface.

On the CNC DIALOG 4, the interface is selectable on the control (on a screen menu) for the external unit to be used.

The control provides three interfaces:

- RS 232 (V24),
- TTY (20 mA),
- cassette recorder (Kansas City).

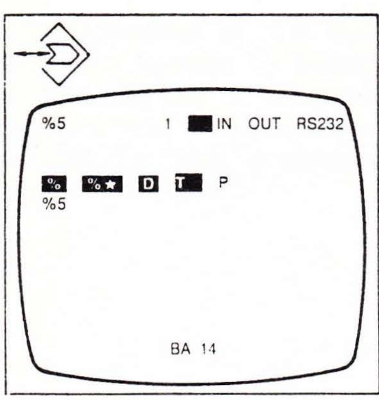
For RS 232 and TTY you can set the 'configuration', which means
- baud rate,
- number of data bits,
- number of stop bits,
- parity.

The configuration for the cassette recorder interface is standardized.

Information on the interface and configuration you need for your external unit will be found in the operator's manual for the external unit concerned.

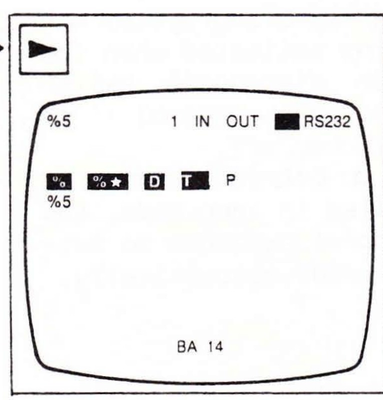
Selecting an interface

Select mode 14.

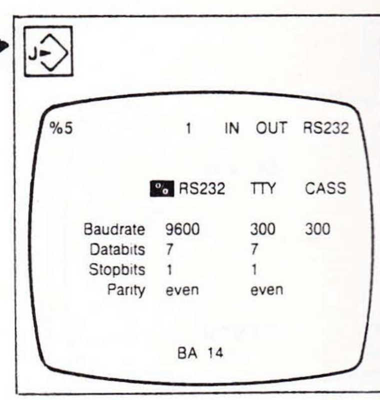


Currently active interface is shown in upper right of screen.

Use 'Cursor right/left' to move cursor to interface shown.

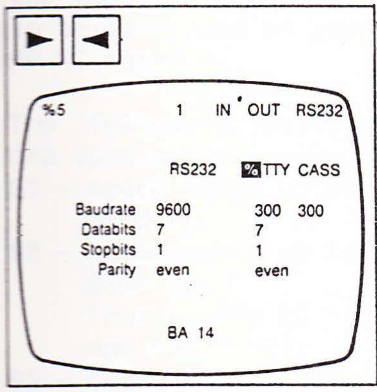


Press 'Acknowledgment' key.

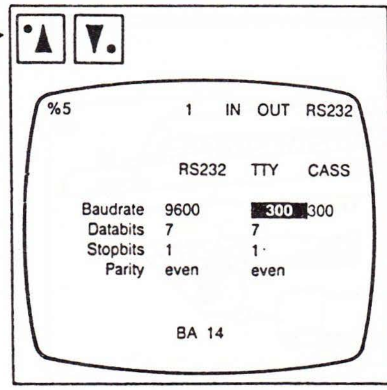


Menu for selecting the configuration appears on screen.

Use 'Cursor right/left' to move cursor to desired type of interface.

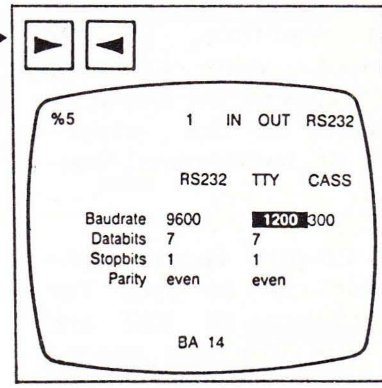


Set interface configuration, if required, using 'Cursor up/down' to proceed from line to line.



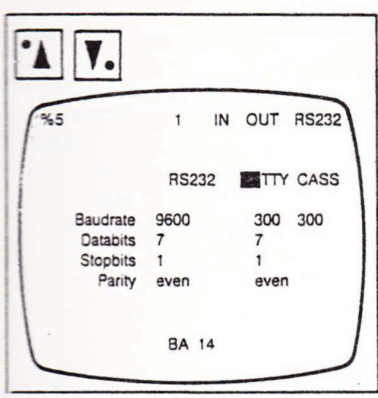
Cursor moves one line down each time key is pressed.

Use 'Cursor right/left' to set desired value.

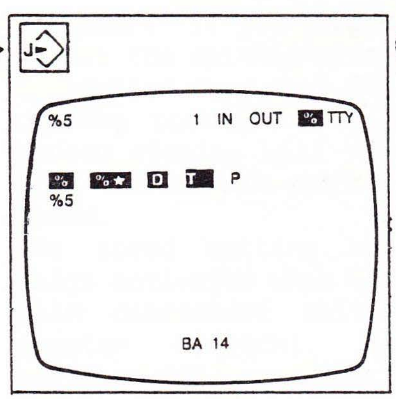


The interface configuration values are standardized, so you cannot select any intermediate values.

Upon selecting configuration values, use 'Cursor up/down' to move to desired interface.



Press 'Acknowledgment' key.

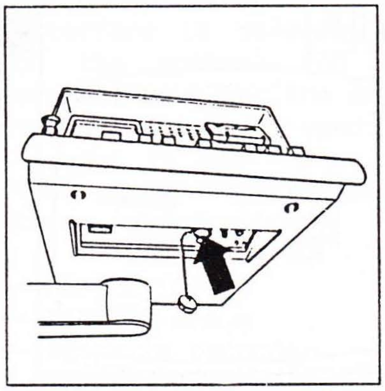


The new interface is now active. Type and configuration are shown at bottom of screen.

- The interface selected remains activated when the main disconnect switch (master switch) is switched off.

Connecting external equipment

Having selected the correct interface, you can connect your external unit to one of the connectors on the underside of the control console.



The 25-pin female connector can be used for interfaces RS 232 and TTY and for DNC operation. To connect a cassette recorder, use the adjacent jack.

Setting the spindle speed indication

The spindle speed indication on your CNC DIALOG 4 has to be set for the machining operation to be performed on your FP-NC machine.

The following settings are obtainable:

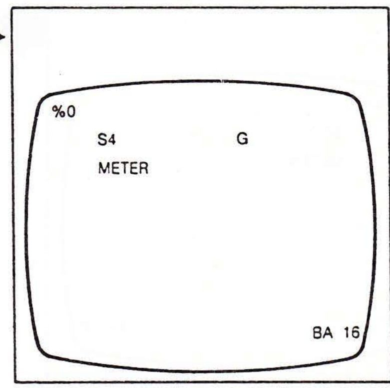
S1 - speed range up to 3150 rpm;

S2 - speed range up to 6300 rpm (not on FP4 TC and FP6/7 NC);

S3 - for use of three-spindle milling head;

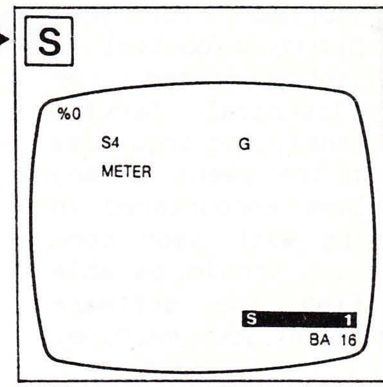
S4 - for other special spindle heads.

Select mode 16.

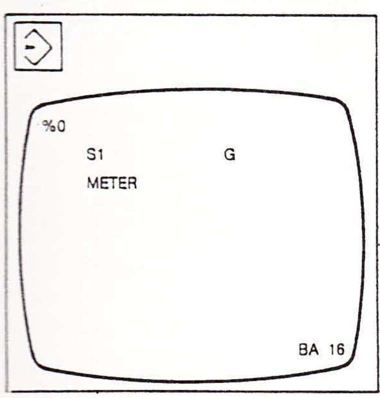


Spindle speed range is shown on screen.

Press 'S' key and enter number of desired range.



Press 'Transfer' key.



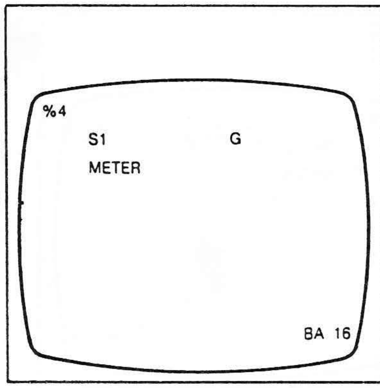
The speed indication is now correct for your machining operation.

Remember: if you forget to set the spindle speed indication for your machining operation, the screen display will not show the actual spindle speed.
The speed setting remains activated when the main disconnect switch (master switch) is switched off.

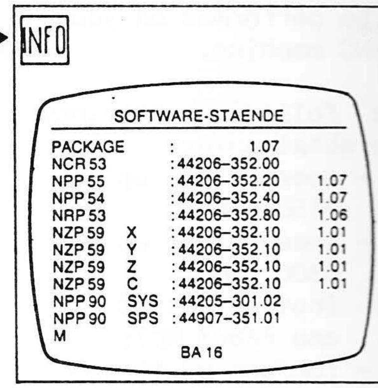
Checking the software version

The software for your CNC DIALOG 4 control is constantly updated. For our Technical Service personnel, for inquiries or in the event of any problems encountered in working with your control you should be able to find the software version of your machine.

Select mode 16.



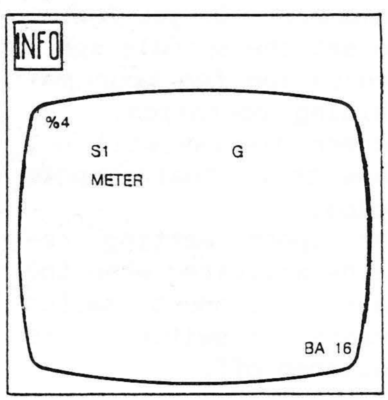
Press 'INFO' key.



Screen shows a table.

The software version of your machine will be found in the first line, opposite the word 'PACKAGE'. The table underneath informs our service technicians of the individual modules used in your CNC DIALOG 4 control.

To return to normal display: press 'INFO' key again.



Screen returns to normal display of the mode selected.