Operating Manual

HEIDENHAIN TNC 150 B/TNC 150 Q

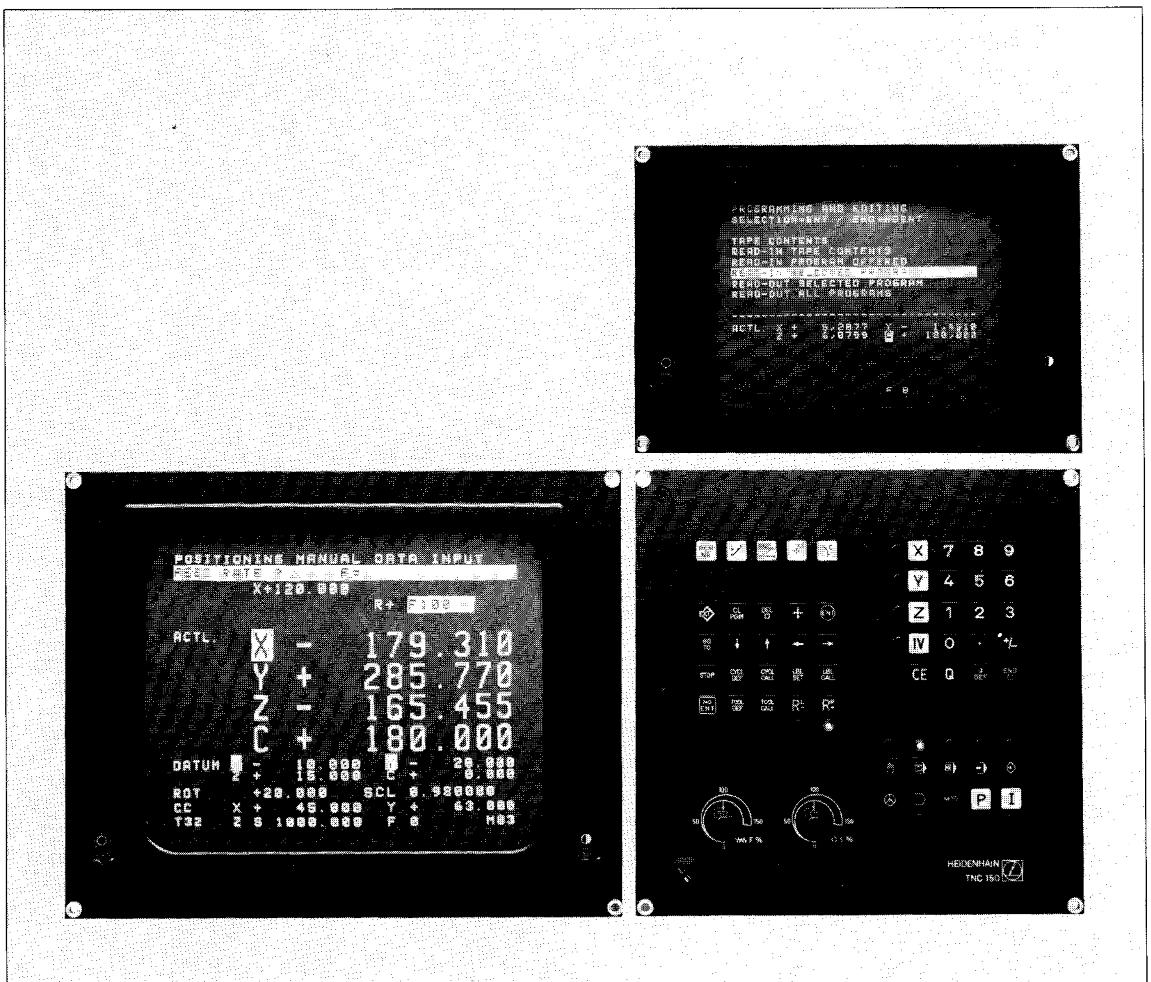
Contouring Control

DR. JOHANNES HEIDENHAIN

Precision Mechanics, Optics and Electronics · Precision Graduations

P.O. Box 1260 · D-8225 Traunreut · Telephone (08669) 31-0

Telex: 56831 · Telegramme: DIADUR Traunreut



Issue 9/85

Dialogue initiation

Dialogue initiation with key	Mode of operation: Manual mode	Single block MDI	Programming and editing	Automatic mode	See section	Page
PGM NR			Program management		M 1	41
X Y Z IV	Setting datum point	Single axis positioning (positioning block)	Programming for pure single axis positioning		K 2 N O	37 100 103
L			Programming: . Single axis machining . 2D-straight line . 3D-straight line		M 3.2.3.1 M 3.2.3.2	52 54
RND			Programming: . Rounding of corners . Contour tangential approach and departure		M 3.2.3.6 M 3.2.6.2	58 63
CC			Programming: Circle centre or Pole		M 3.2.2 M 3.2.3.3	50 55
C			Programming: 2D-circular arc 3D-helix		M 3.2.3.4 M 3.2.3.5	55 57
STOP	Auxiliary function	Discontinuation of program run: Acknowledge ext. stop	Programmed halt	Discontinuation of program run: Acknowledge ext. stop	K 3 M 4 P 2	39 64 106
TOOL DEF			Tool data: Tool definition		M 2.1	42
TOOL CALL	Spindle rpm	Tool call-up	Tool data: Tool call-up		M 2.2	44
LBL SET			Label: Setting label		M 6.1	73
LBL CALL			Label: Label call-up		M 6.2	73
CYCL DEF			Canned cycles: Cycle definition		M 7	81
CYCL CALL			Canned cycles: Cycle call-up		M 7.3	94
CL PGM			Clear program		M 8.7	98
Q DEF			Definition of parameter functions		M 5	64
MOD	Mode (supplementary operating modes): Vacant blocks, mm/inch conversion, Position data display: Actual value/Nominal value/ Target distance/Lag/Position display large/ small, Baud rate, Working range, NC-software Number, PLC-software Number, Code No.				J	30
EXT			Entry of programs via data interface	Output of programs via data interface	Q	109

Basic-Symbols	Meaning
	Machine traverse "automatic"
	Program test
□	Block*
◇	Memory for machining program (store)

* The machining program consists of individual "program blocks".

Keys for programming of contours and entry of program number

Key symbol	Abbreviation for	Meaning	See section	Page
PGM NR	PROGRAM NUMBER	Designation of a new number for program or machining program. Selection of a program.	M1.1 M1.2	41 42
L	LINE	Straight cut traverse (simultaneously in 3 axes, 2 axes or only in 1 axis)	M3.2.3.1 M3.2.3.2 N	52 54 100
RND	ROUND	."Rounding off" of corners (programming of arcs with tangential transitions) .Tangential contour approach (run-on) and departure (run-off)	M3.2.3.6 M3.2.6.2	58 63
CC	CIRCLE CENTRE	.Circle centrepoint for circular path .Pole for nominal value entry in polar co-ordinates	M3.2.3.3 M3.2.2	55 50
C	CIRCLE	.Circular arc .Helix	M3.2.3.4 M3.2.3.5	55 57

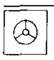
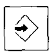
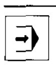
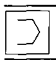
Keys for entry values and axis selection

Key symbol	Abbreviation for		Meaning	See section	Page
0 ... 9	---	Entry value/ axis selection keys	Decimal keyboard for numerical values	G 3	26
·	---		Decimal point	G 3	26
+/-	---		Sign change	G 3	26
X Y Z	Axes: X, Y, Z		Axis selection for datum set and programming of position values	K 2 M 3.2.3 N	37 52 100
IV	Fourth axis				
Q	PARAMETER		Parameter entry	M 5.1	65
Q DEF	PARAMETER DEFINITION		Definition of parameter	M 5.2	65
CE	CLEAR ENTRY		For delection of entry values or cancellation of fault/error display	H 2 G 3	27 26
END □	END BLOCK		Complete block	M 3.2.4	59

If, in the selected operating mode, a button which has no function is inadvertently pressed, the error "BUTTON NON-FUNCTIONAL" is indicated.
This error code can be cancelled by pressing CE.

TNC 150 Keyboard

Operating mode-keys

Key symbol	Meaning	See section	Page
	**Manual** mode of operation **1.** The control operates as a conventional digital readout. The machine can be traversed via the axis-direction buttons.	K	36
	2. Datum set		
	Traversing of machine axes via the **electronic handweel**	L	40
	**Positioning with MDI** (manual data input) Single axis automatic traversing. **One single block** can be entered only, but not stored (single axis positioning block or tool call). The stored machining program is not influenced. Contouring operation, with canned cycles, subprograms or program part repeats is not possible in this operating mode.	O	103
	**Program entry and editing** Programming is dialogue-guided, i.e. all necessary data for programming is asked for by the control in plain language dialogue and in the correct sequence. A machining program can comprise the following types of program blocks: .Straight cut ("single axis" programming, linear interpolation (2 axes) or 3D-linear path) .Circle centre .Circular path .Helix .Tool definition .Tool call .Cycle definition .Cycle call .Label set .Label call: subprogram or program part repeat .Parameter programming (mathematical and logical functions) .Programmed stop	M	41
	**Single block program run** A press of the start-button is required to execute each individual program block.	P	104
	**Automatic** (complete program sequence) With single press of start-button, the stored program sequence is run to a programmed stop or to the end.	P	104
	Program test without machine movement	M 9	99
MOD	**MODE** (supplementary operating modes) .mm/inch conversion .Position data display: Actual position Distance to reference marks Lag Nominal positions Distance to nominal position .Position display large/small .Baud rate .Limiation of working range .Vacant blocks .NC: Software No. .PLC: Software No. .Code No. .4th Axis on/off	J	30
I	**Incremental dimension (chain dimensions);** When off: absolute dimensions	M 3.2	49
P	Entry of nominal values in **polar co-ordinates;** when off: right-angled (Cartesian) co-ordinates	M 3.2.2	50

Programming and editing keys

Key symbol	Abbreviation for		Meaning	See section	Page
EXT	---		External data input or output	O	110
⌖	---		Actual position value: Transfer of actual machine position data as entry value for programming (Playback of Position data or programming of tool length)	N 2	102
CL PGM	CLEAR PROGRAM	Editing keys	Clear complete program content	M 8.7	98
DEL □	DELETE BLOCK	Editing keys	Delete previously entered block	M 8.3	96
ENT	ENTER		Enter into memory	G 2 G 3	25 26
GO TO	GO TO BLOCK	Editing keys	Block search key	M 8.1	95
↓ ↑	---	Editing keys	"Page" program blockwise forwards or reverse	M 8.2 M 8.6	95 98
← →	---	Editing keys	Cursor setting for program word selection	M 8.5 M 8.6	97 98
STOP	STOP		Programmed stop or discontinuation of positioning	M 4 P 2 P 3	64 106 107
CYCL DEF	CYCLE DEFINITION	Cycle keys	Definition of canned cycle	M 7	81
CYCL CALL	CYCLE CALL	Cycle keys	Call-up of canned cycle	M 7.3	94
LBL SET	LABEL SET	Label keys	Allocation of program label for subprogram or program part repeat	M 6.1	73
LBL CALL	LABEL CALL	Label keys	Call-up of program label (Jump to label No.)	M 6.2	73
NO ENT	NO ENTER		No enter: The data (entry) requested by the dialogue is not required.	G 2	25
TOOL DEF	TOOL DEFINITION	Tool keys	Tool definition (Tool No., length, radius)	M 2.1	42
TOOL CALL	TOOL CALL	Tool keys	Call-up of required tool (Tool No., axis, rpm)	M 2.2	44
R R+	---	Radius compensation keys	**.In contouring operation:** The milling cutter is located to the **right** of the contour **in the feed direction.** **.In single axis positioning operation:** Radius compensation "plus": the tool offset **extends the traverse.**	M 3.1 N 1	46 100
R L-	---	Radius compensation keys	**.In contouring operation:** The milling cutter is located to the **left** of the contour **in the feed direction.** **.In single axis positioning operation:** Radius compensation "minus": the tool offset **shortens the traverse.**	M 3.1 N 1	46 100

Contents

This operating manual is valid for the following controls

TNC 150-versions with interface for an external machine PLC:

Transducer inputs: sinusoidal signals	Transducer inputs: square wave signals
TNC 150 B	TNC 150 BR
TNC 150 F (without 3D-movement)	TNC 150 FR (without 3D-movement)

TNC 150-versions with PLC-board(s):

Transducer inputs: sinusoidal signals	**Transducer inputs: square wave signals**
TNC 150 Q	TNC 150 QR
TNC 150 W (without 3D-movement)	TNC 150 WR (without 3D-movement)

HEIDENHAIN is constantly working on further developments of its TNC-controls. It is therefore possible that details of a certain control may differ slightly to the control version which is being described herein. Due to the operator being "guided" by the plain language dialogue, such differences will prove insignificant.

A) Brief description of TNC 150

The HEIDENHAIN TNC 150 is a **4-axis contouring control.** Axes X, Y and Z are primarily intended for linear traversing and the fourth axis is normally used for connection of a rotary table. With each control switch-on, the fourth axis may be made active or inactive.

The following is possible with TNC 150:
.circular interpolation in 2 out of 4 axes,
.linear interpolation in 3 out of 4 axes
and
.helical interpolation in 3 out of 4 axes.

Circular and helical interpolation is only possible with the fourth axis if it is being used as a linear axis.

Programming is dialogue-guided; i. e. after "dialogue initiation" by the operator, all necessary data required for program entry is asked for by the TNC 150 in plain language.

Dialogue texts, machining programs, entry values, fault/error indication and position data are clearly indicated on the screen of the visual display unit (VDU)

The resolution for position display is

0.001 mm or 0.005 mm
or 0.0001 inch or 0.0002 inch in imperial mode,
angle resolutions 0.001° or 0.005°

The resolution is determined by the machine tool manufacturer.

Position values may be entered in steps of

0.001 mm or
0.0001 inch and 0.001° for angles.

Program management
The TNC 150 has a program management facility for 24 different programs with a total of 1200 blocks.

Program entry
with **linear or circular interpolation:**
manually through key-in

.to program sheet or workpiece drawing – also during machining (background programming)

or externally

via the V.24-compatible data transfer interface (e. g. with HEIDENHAIN magnetic tape cassette units ME 101/ME 102 or with other commercially available peripheral units).

with pure **single axis operation:**
manually with key-in.

.to program sheet or workpiece drawing – also during machining (background programming)

.**or** during conventional machining operation in the manual mode by entering actual position data from position display as nominal values **(Playback)**

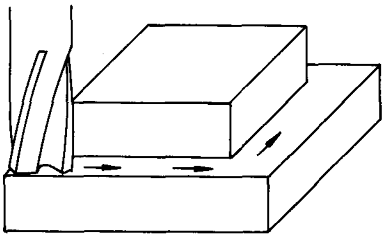
or externally

.via the V.24-compatible data transfer interfaces as explained above.

The HEIDENHAIN ME 101/ME 102 magnetic tape cassette units have been especially designed for external storage of TNC-programs on magnetic tape cassettes. On the rear of these units, connections are provided for data input and output (V.24 or RS-232-C compatible) so that a TNC 150 and e. g. a printer unit, may be simultaneously connected.
Programs which have been entered externally can be edited or optimised if required.

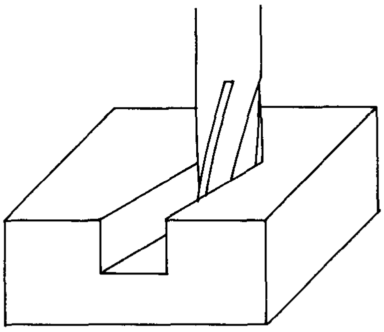
Programs which have been compiled on the TNC 145 can be used on the TNC 150. When being read into the TNC 150, such programs are automatically adapted to the TNC 150; e. g. the diagonal path cycle of the TNC 145 is transformed into 3D-Linear interpolation by the TNC 150.
An existing TNC 145 program library can therefore be further utilised in the TNC 150.

Single axis machining



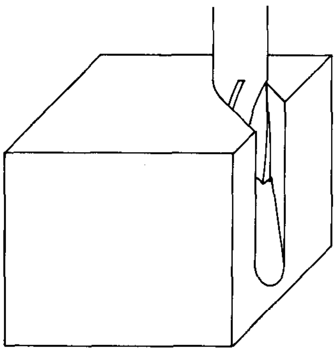
Many workpieces require only the simplest of machining operations: single axis machining, i. e. only one axis is traversed at a time.

2-axis linear interpolation



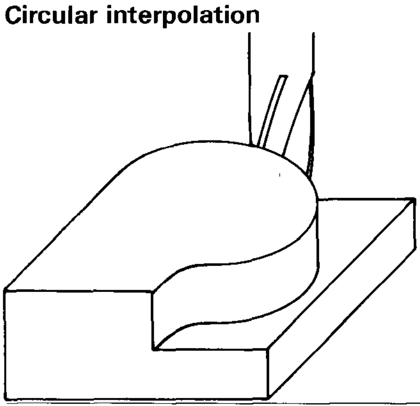
Two axes are moved simultaneously so that the tool follows a straight path.

3-axis linear interpolation



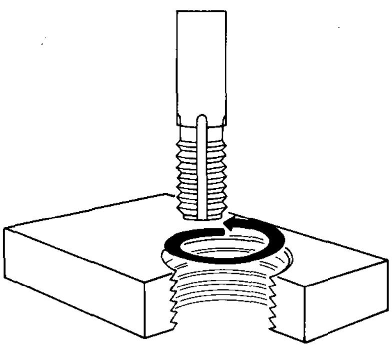
Three axes are moved simultaneously so that the tool follows a 3D-straight path.

Circular interpolation



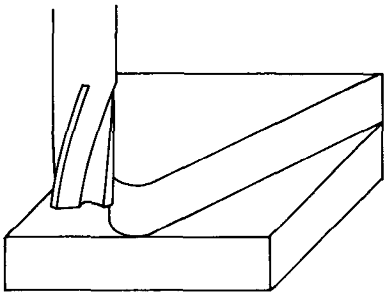
Two axes are moved simultaneously so that the tool describes a circular (arc) contour.

Helical interpolation



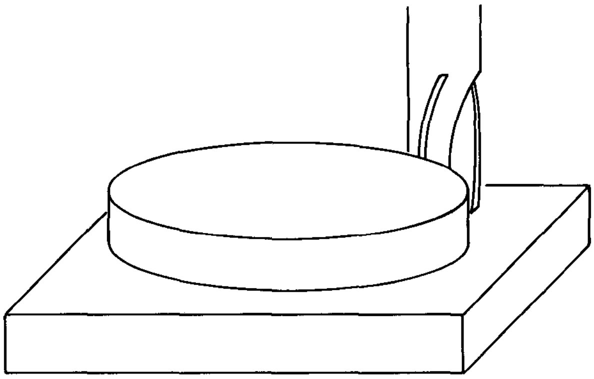
Circular movement is performed in the working plane with simultaneous linear motion in the tool axis. Helical interpolation is used mostly for the manufacture of large diameter internal and external threads.

Applying corner radii



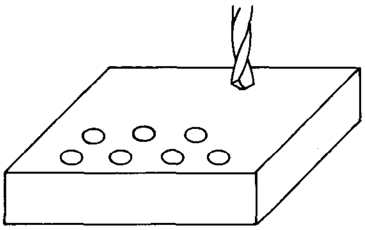
Rounding of corners is especially easy to program. The entry of the rounding-off radius is sufficient. During machining, the corner radius is inserted with a tangential transition to the remaining contour.

Contours derived from mathematical formulae



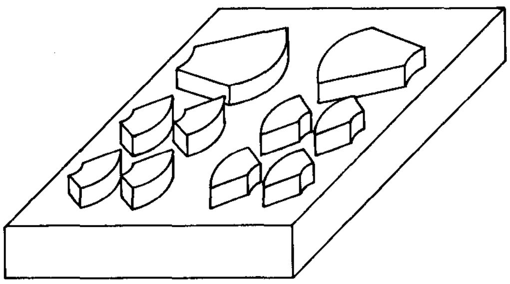
With the aid of parameter programming, contours can be machined which have been calculated using mathematical formulae (e. g. ellipses).

Hole patterns



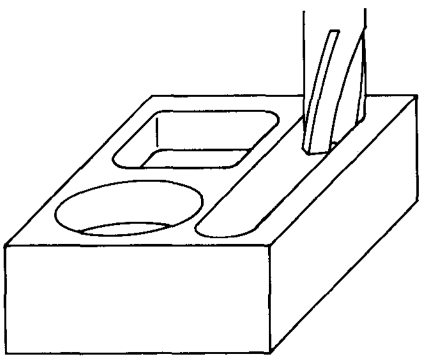
Holes and threads can be programmed easily and fast with the aid of subprograms and canned cycles.

Repetitive contours



"Datum shift" and "mirror image" cycles in conjunction with subprogramming and parameter programming simplify and shorten programming effort for repetitive contours and shapes.

Simple pockets and slots



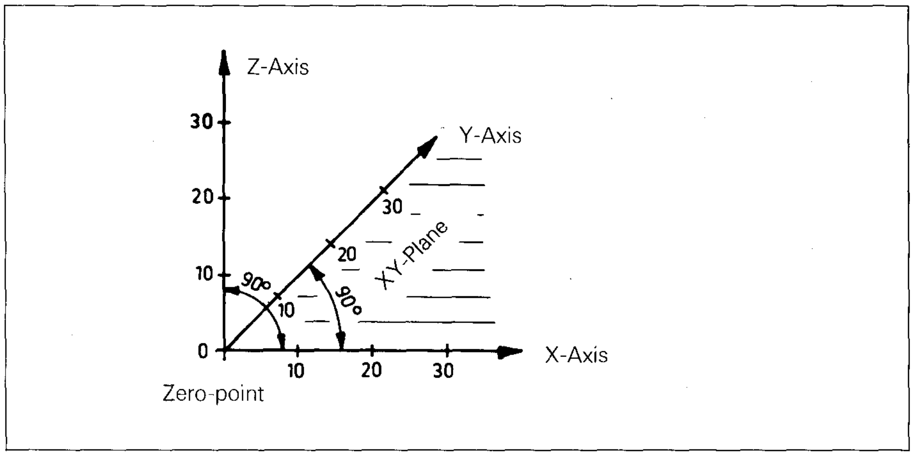
TNC 150 has pre-programmed canned cycles for rectangular pockets, circular pockets and slots.

C) Dimensions/Co-ordinates

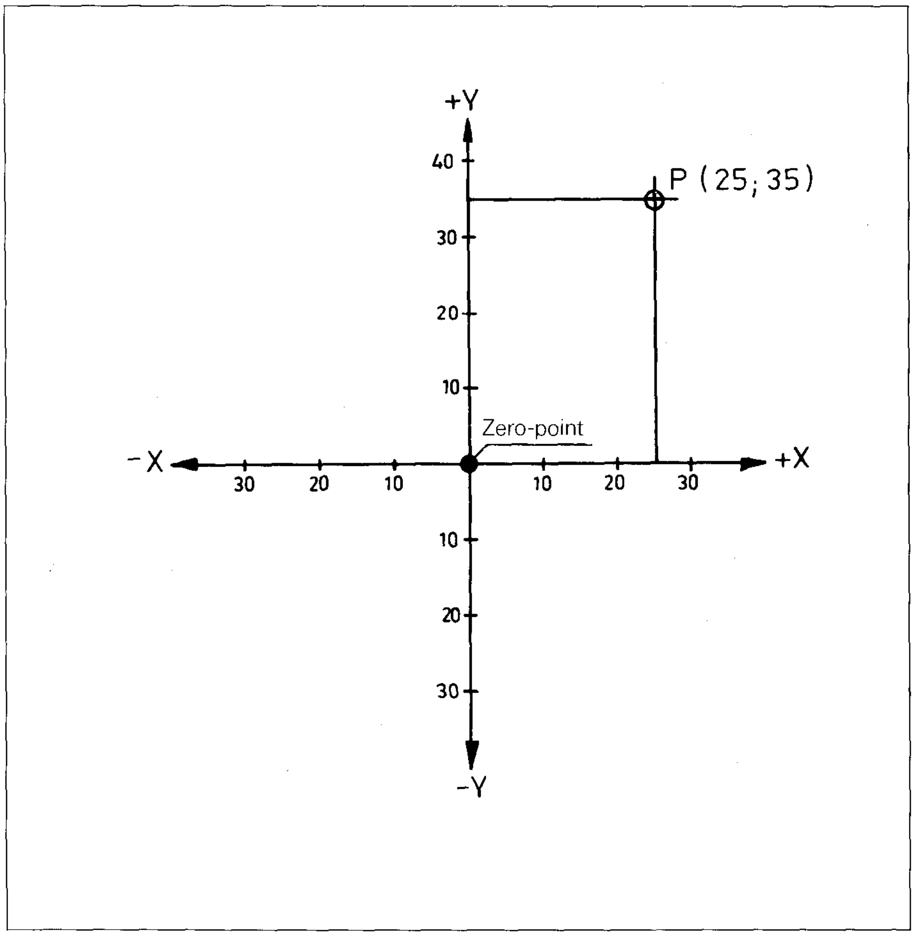
C 1) Cartesian co-ordinates

One must differentiate between the **"actual position"** of machine and workpiece, i.e. the momentary position, and the **"nominal position"**, as per machining program.
As an aid for locating positions within a plane or in space, so-called **"co-ordinates"** or a "co-ordinate system" are used.

The TNC 150 displays **actual positions in right-angled co-ordinates** – also referred to as "Cartesian co-ordinates". **Nominal positions** for machining can be programmed either in **"Cartesian co-ordinates" or in "Polar co-ordinates"** (refer to section M.3.2).



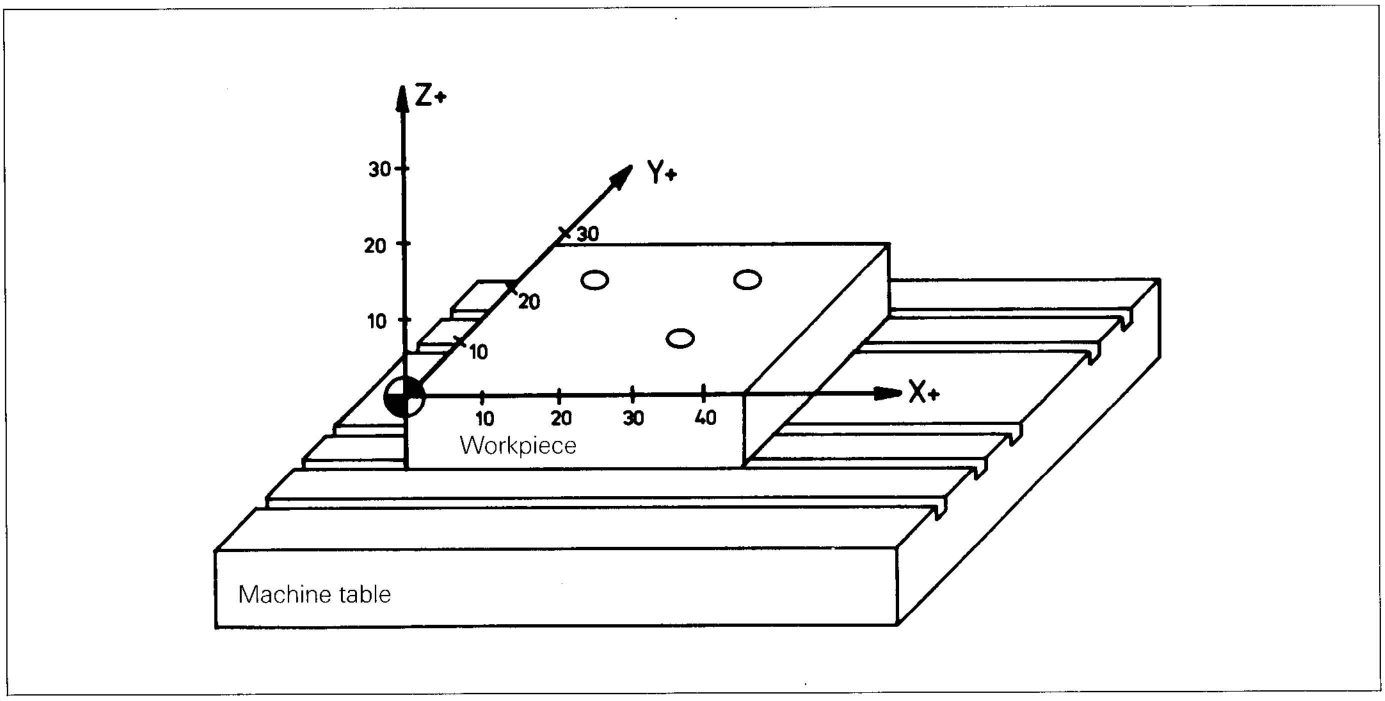
A **right-angled co-ordinate system** is formed by three co-ordinate axes X, Y and Z which are perpendicular to each other. The two axes X and Y constitute the XY-plane. All three axes have a common point of intersection the so-called zero-point (or "origin").



Every position or every point of the XY-plane is determined by two co-ordinates, i.e. by its X-value and Y-value. The illustrated point "P" has the co-ordinates X = 25 mm and Y = 35 mm. In the same manner, a point in space is determined by its three co-ordinates X, Y and Z.

C 2) Workpiece datum

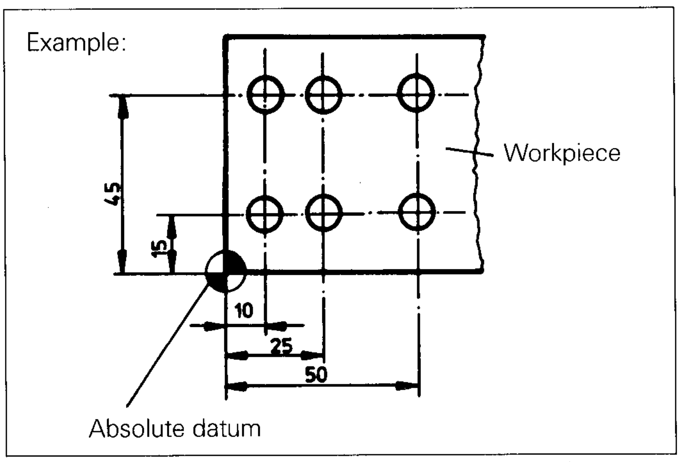
To determine positions in a machining program, the co-ordinate system is established such, that program preparation is easy and convenient. E.g. the co-ordinate axes can coincide with the workpiece edges (the workpiece is clamped to the machine table so that its co-ordinate axes are parallel to the machine axes). The co-ordinate zero-point is the reference point (or datum) for all absolute dimensions of the machining program. This point is designated by the symbol .



C 3) Absolute/Incremental dimensions

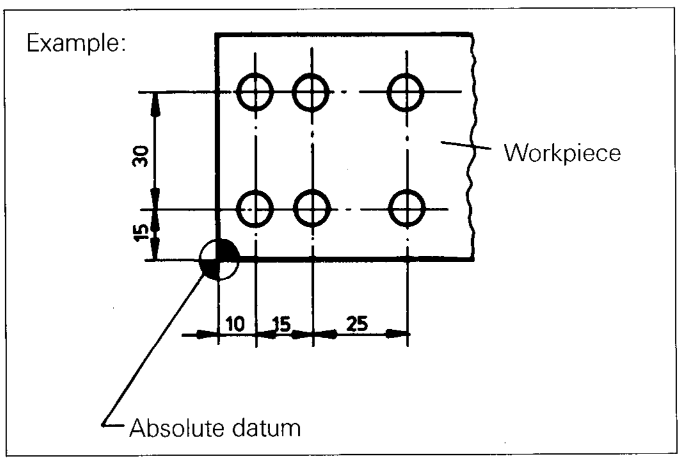
Workpiece dimensions are either absolute or incremental.

Absolute dimensioning



The lower left-hand corner of the workpiece is the "absolute datum" for dimensioning.
The machine is to be traversed **to** the entered dimension. It traverses **to** the entered **nominal position value.**

Incremental dimensioning



Dimensioning commences from the lower left-hand corner of the workpiece as a chain of values.
The machine is to be traversed **by** the entered **nominal position value starting from the actual position previously reached.**

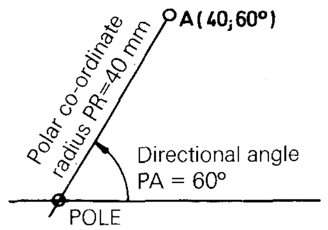
Programming in absolute dimensions offers the advantage of making geometric amendments of single positions without affecting other positions. Re-entry into an interrupted program after power failure or any other defect is also more simple with absolute programming. Furthermore, a suitable location of the zero-point may dispense with negative values.

On the other hand, **incremental programming** may reduce calculation work.

C 4) Polar co-ordinates

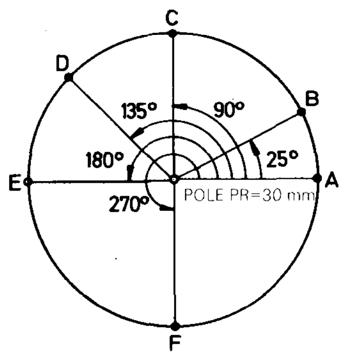
TNC 150 also offers the possibility of entering nominal position values by in using polar co-ordinates.

With polar co-ordinates, points **in one plane** are referenced to a polar co-ordinate datum – **the "pole"** – and are defined by the radius from the pole to the required position and the angle of direction (polar angle).



a) Radius and directional angle programmed in absolute dimensions

Example:

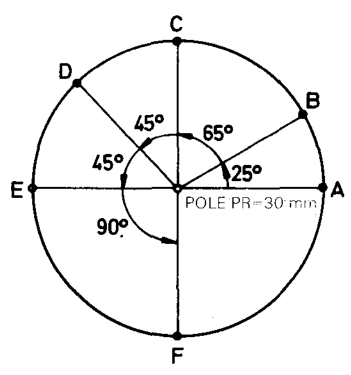


Polar co-ordinates of points A, B, ..., F;
PR and PA absolute

A (30_A; 0°_A)
B (30_A; 25°_A)
C (30_A; 90°_A)
D (30_A; 135°_A)
E (30_A; 180°_A)
F (30_A; 270°_A)

b) Radius programmed in absolute dimensions and directional angle programmed in incremental dimensions

Example:



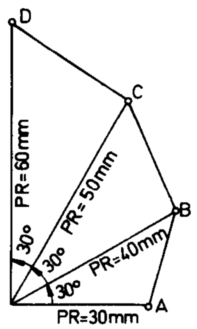
The first point A must be defined in absolute: A (30_A; 0_A°)

Polar co-ordinates of points B, C, ..., F;
PR absolute, PA incremental:

B (30_A; 25°_I)
C (30_A; 65°_I)
D (30_A; 45°_I)
E (30_A; 45°_I)
F (30_A; 90°_I)

c) Radius and directional angle programmed in incremental dimensions

Example:

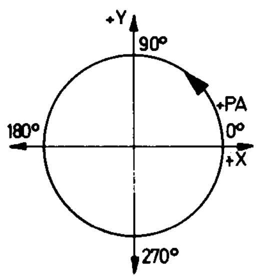


The first point A must be defined in absolute: A (30_A; $0°_A$)

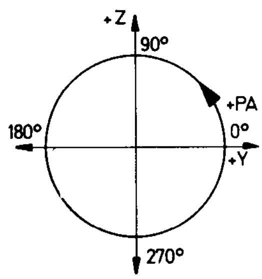
Polar co-ordinates of points B, C, D
PR and PA incremental:

B (10_I; $30°_I$)
C (10_I; $30°_I$)
D (10_I; $30°_I$)

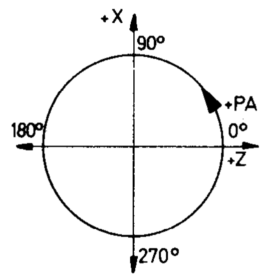
Definition of planes and 0°-axes



In the **X/Y**-plane the 0°-axis lies on X+



In the **Y/Z**-plane the 0°-axis lies on Y+



In the **Z/X**-plane the 0°-axis lies on Z+

The **positive direction** of the angle **"PA"** corresponds to an **anti-clockwise** (ccw) direction (rotation to the left).

C 5) NC-Dimensioning of workpieces

With machines fitted with TNC-Controls, geometric and technical data necessary for workpiece machining can be entered via the keyboards. In order to make shop-floor programming economical and less time-consuming, it is advisable to use either drawings which have been dimensioned for direct TNC – entry or pre-prepared program lists.

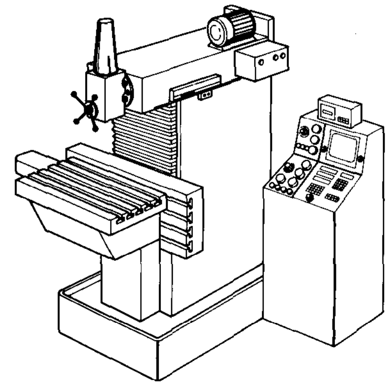
C 6) Axis designation for NC machines

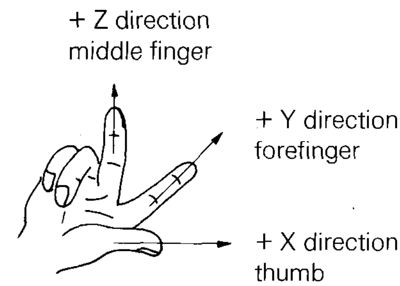
The allocation of co-ordinate planes to the traversing direction of numerical controlled machines are explained in the appropriate NC-standards of ISO.

C 6.1) The three main axes

The three main axes are defined by NC-standards. Traversing directions can be determined by the "right hand rule". In addition, the traversing direction of the tool-axis **towards the workpiece** corresponds to **the negative traversing direction.**

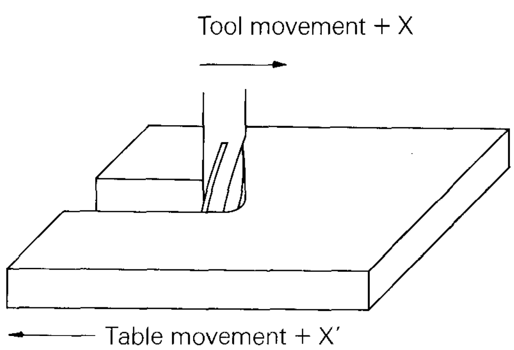
Example:
Universal milling machine





"Right hand rule":
Coordinates are correlated to the fingers.

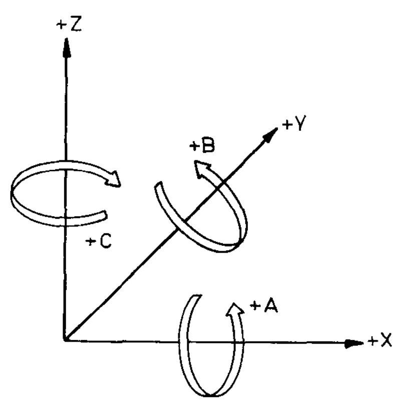
When programming only **tool movement** is considered (relative movement of tool) i. e. whilst programming the operator always assumes that the tool is moving.

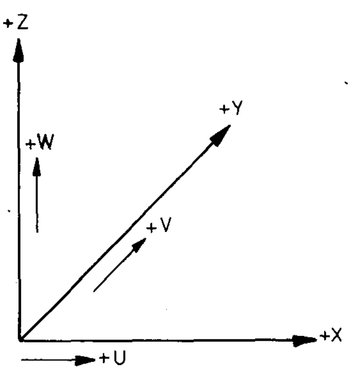


With the universal milling machine as illustrated above, the milling tool should, for example, traverse in a positive direction. However, due to the table moving in this axis and not the tool, the table must move in the left-hand direction. The relative movement of the tool is therefore in the right-hand direction, i. e. in the positive X direction. In this case, the traversing direction of the table is designated X'.

C 6.2) The fourth axis

The machine tool manufacturer decides whether the fourth axis is to be used for a rotary table or as an additional linear axis and also which designation this axis will receive on the display screen:

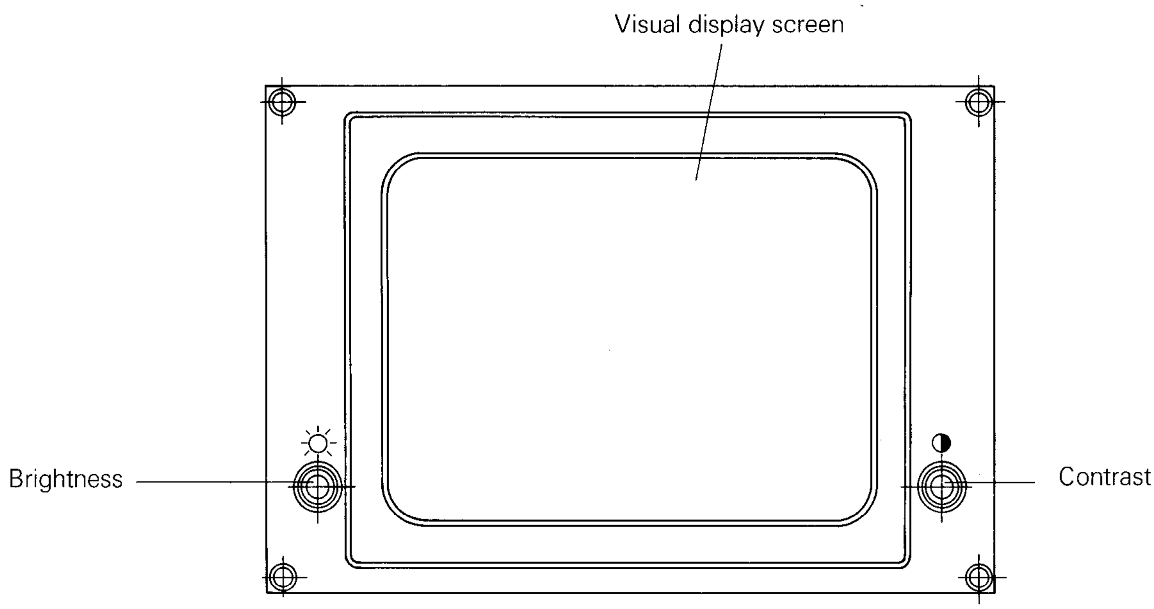


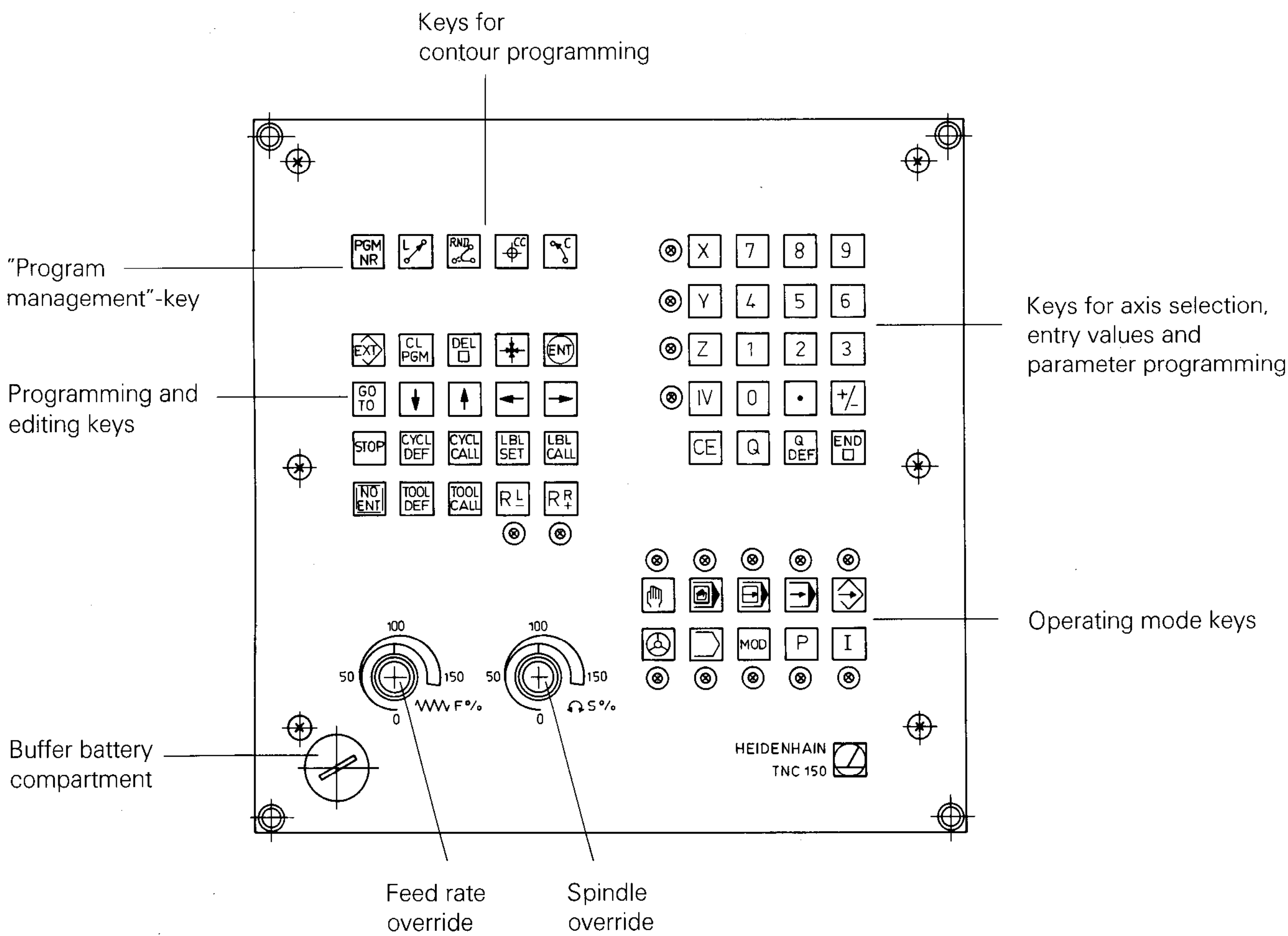


Rotary axis
The rotary axis is designated with the letters **A, B** or **C;** the correlation to the main axes and the rotating direction is shown in the above illustration.

Linear axis
If the fourth axis is to be used as a linear axis, the designation of this axis is **U, V** or **W.**
The correlation to the main axes is shown above.

D) Keyboards and displays of TNC 150





E) Main functions of TNC 150

The TNC 150 controls the automatic machining of a workpiece in accordance with a program which is entered into and stored within the TNC 150. The program contains all the required data for tools, spindle speeds, axis movements and switching procedures (coolant on/off, rotating direction or spindle stop etc.). Up to 24 machining programs can be stored simultaneously ([PGM NR] -key).

Machining programs comprises individual "blocks".
For execution of a stored program, the operating mode **"automatic"** ([→] -key) or **"program run single block"** ([→] -key – each block is started individually) – may be selected.

For machining operations with single axis positioning only, entry and execution can be made block by block: **operating mode "single block positioning with MDI"** ([→] -key).

Machine set-up operation can be carried out with the **electronic handwheel** ([handwheel] -key).

Datum-set and reference mark approach are performed in the **"manual"** mode ([hand] -key). This mode is otherwise only for conventional digital readout operation.

From the range of **tools** entered with the tool definition blocks ([TOOL DEF] -key), the required tool is selected with a tool call block before commencement of machining ([TOOL CALL] -key).

The **"program entry"** mode is initiated with the [◇] -key (the respective lamp is then on).

For programming of the **tool path,** only the workpiece contour and drawing dimensions have to be entered: length and radius of the tool are automatically taken into account by the TNC 150. In order to describe the contour, the type of path (linear [L] or arc [CC] [C]) and nominal target position is entered. Only the [RND] -key has to be pressed for automatic insertion of tangential transition radii or automatic rounding of corners.
Nominal position programming is not only in right-angled (Cartesian) co-ordinates – as with most controls –, but also in polar co-ordinates in either absolute or incremental dimensions as well as in mm or in inch.

For pure single axis machining, programming can also be carried out via the axis-keys [X], [Y], [Z] and [IV] as with HEIDENHAIN-controls TNC 131/135 i.e. greater simplicity.

Furthermore, with single axis positioning, transfer of actual position data (display values) as nominal values is also possible (Playback).
The **tool path feed rate** is programmed in mm/min. or 0.1 inch/min. or in °/min. (with rotary tables).
A substantial reduction of programming is made possible by **canned machining cycles:**
.Pecking (Deep hole drilling)
.Tapping
.Slot milling
.Pocket milling
.Circular pocket milling
The TNC 150 also offers cycles for:
.Datum shift
.Mirror image
.Co-ordinate system rotation and
.Scaling

Required values are entered with the cycle definition ([CYCL DEF] -key) and the cycle is retrieved with the [CYCL CALL] -key.

An important aid for programming is offered by the TNC 150 through **subprograms and program part repeats:**

Program sections can be "labelled" via the [LBL SET] -key and then be retrieved as often as required via the [LBL CALL] -key.

The [Q] and [Q DEF] -keys permit input of **parameters** in place of co-ordinate or feed rate values. This parameter programming feature enables contours to be increased or decreased in size or the machining of special contours calculated via mathematical formulae.
A stored machining program may be checked by using the **"program test"** mode [→] which is performed without machine movement.
Program editing i.e. corrections or optimisation of programs by amending block-words, complete blocks or insertion or deletion of blocks is performed with the [GO TO], [↓], [↑], [←], [→] -keys.

Program entry and output via an **external data medium** is initiated with the [EXT] -key.

After control switch-off or power failure, a buffer battery ensures that the memory content i.e. machining program and machine parameters (control functions adapted to the machine characteristics) is not erased. In order to prevent loss of this data, a **BATTERY CHANGE MUST BE MADE WITH THE CONTROL SWITCHED ON!** If loss of power occurs when the battery is discharged or missing, **THE MACHINING PROGRAM AND MACHINE PARAMETERS** must be re-entered (see section R).

F) Machine-specific data

F 1) Feed rate F

The required contouring speed (tool path feed rate) is programmed in mm/min. (or 0.1 inch/min.) and in °/min. with rotary tables. For maximum feed (rapid traverse).

the value 15999 for mm-programming
and 6299 for inch-programming
is to be entered in accordance with the input range.

Max. feed rates and traverses of individual machine axes are determined by the machine tool manufacturer and specified in the machine tool operating manual.

F 2) Auxiliary functions M

M-functions are programmed for control of miscellaneous machine functions (e.g. "spindle switch-on" etc.). M-function entry is requested by the dialogue.

Special M-functions which affect program run

M 00	Interrupts program run after completion of the appropriate block and provides the command "spindle HALT" and "coolant OFF".
M 02	Interrupts program run after completion of the appropriate block and selects block 1; furthermore, "spindle HALT" and "coolant OFF" are also commanded. Depending on the entered machine parameters, the status display is cancelled from the VDU-screen.
M 03	"Spindle clockwise" – at beginning of block.
M 04	"Spindle counter-clockwise" – at beginning of block.
M 05	"Spindle HALT" – at end of block.
M 06	"Tool change" – further functions as per M 00.
M 08	"Coolant ON" – at beginning of block.
M 09	"Coolant OFF" – at end of block.
M 13	"Spindle clockwise" and "coolant ON".
M 14	"Spindle counter-clockwise" and "coolant ON".
M 30	Functions as per M 02.
M 90	Constant contouring speed at corners. **The function of M 90 depends on the machine parameters entered with the initial commissioning procedure.** Detailed information may be obtained from the machine tool manufacturer (see section M 3.2.5)
M 91	If M 91 is programmed within a positioning block, the programmed nominal position value is not referenced to the original workpiece datum (see section K 2), but to the transducer reference point.
M 92	If M 92 is programmed within a positioning block, the programmed nominal position value is not referenced to the original workpiece datum (see section K 2), but to a position which has been defined by the machine tool manufacturer via a machine parameter (e. g. a tool change position). Tool compensation is ineffective with this block.
M 94	If M 94 is programmed within a position block, for axis IV (with rotary axis application), the position display is automatically reduced to the corresponding position value below 360° before commencement of positioning.
M 95 **M 96**	Change of approach behaviour at beginning of contour (see section M 3.2.6.1)
M 97	Correction of tool path intersection for external corners (see section M 3.1)
M 98	Contour offset completed (see section 3.2.6.1)
M 99	Same functions as "CYCL CALL" (see section M 7.3)

Unassigned M-functions are utilized by the machine tool manufacturer. These are explained in the machine operating manual.

The following M-functions are programmable:

M-function (M-Functions which affect program run are indicated)	Output at block beginning	end
M 00		X
M 01		X
M 02		X
M 03	X	
M 04	X	
M 05		X
M 06		X
M 07	X	
M 08	X	
M 09		X
M 10		X
M 11	X	
M 12		X
M 13	X	
M 14	X	
M 15	X	
M 16	X	
M 17	X	
M 18	X	
M 19		X
M 20	X	
M 21	X	
M 22	X	
M 23	X	
M 24	X	
M 25	X	
M 26	X	
M 27	X	
M 28	X	
M 29	X	
M 30		X
M 31	X	
M 32		X
M 33		X
M 34		X
M 35		X

M-Function	Output at block beginning	end
M 36	X	
M 37	X	
M 38	X	
M 39	X	
M 40	X	
M 41	X	
M 42	X	
M 43	X	
M 44	X	
M 45	X	
M 46	X	
M 47	X	
M 48	X	
M 49	X	
M 50	X	
M 51	X	
M 52		X
M 53		X
M 54		X
M 55	X	
M 56	X	
M 57	X	
M 58	X	
M 59	X	
M 60		X
M 61	X	
M 62	X	
M 63		X
M 64		X
M 65		X
M 66		X
M 67		X
M 68		X
M 69		X
M 70		X

M-Function	Output at block beginning	end
M 71	X	
M 72	X	
M 73	X	
M 74	X	
M 75	X	
M 76	X	
M 77	X	
M 78	X	
M 79	X	
M 80	X	
M 81	X	
M 82	X	
M 83	X	
M 84	X	
M 85	X	
M 86	X	
M 87	X	
M 88	X	
M 89	X	
M 90	X	
M 91	X	
M 92	X	
M 93	X	
M 94	X	
M 95		X
M 96		X
M 97		X
M 98		X
M 99		X

F 3) Spindle speeds S

Tool spindle speeds are programmed with a tool call (see section M 2.2).
The following spindle speeds may be programmed:

rpm	rpm	rpm	rpm	rpm
0	1	10	100	1000
0,112	1,12	11,2	112	1120
0,125	1,25	12,5	125	1250
0,14	1,4	14	140	1400
0,16	1,6	16	160	1600
0,18	1,8	18	180	1800
0,2	2	20	200	2000
0,224	2,24	22,4	224	2240
0,25	2,5	25	250	2500
0,28	2,8	28	280	2800
0,315	3,15	31,5	315	3150
0,355	3,55	35,5	355	3550
0,4	4	40	400	4000
0,45	4,5	45	450	4500
0,5	5	50	500	5000
0,56	5,6	56	560	5600
0,63	6,3	63	630	6300
0,71	7,1	71	710	7100
0,8	8	80	800	8000
0,9	9	90	900	9000

When entering the machine data, the machine tool manufacturer lays down a series of "permissible" spindle speeds. If an rpm is programmed which is outside of this range, the error **WRONG RPM** is indicated during program run.

Spindle speeds are output either

.BCD-coded
or
.analogue.

With analogue output of the spindle speed, the programmed spindle speeds do not have to correspond to the values in the table. Any required speed may be entered, provided that the max. spindle speed is not exceeded and the lowest spindle speed is not below the min. speed.

Moreover, with analogue output of the spindle speed, the programmed spindle speed is superimposed by the %-factor which is set on the **"spindle override"** potentiometer.

As of software version 0……09.

The max. entry value with analogue output of the spindle speed has been increased to 30000 rpm.

F 4) Tool numbers T

The tool number is programmed via the tool call (see section M 2.2).

Tool numbers 0…254 are available for programming.

When using an automatic tool changer, only tool numbers 0…99 may be programmed as the control output is unable to provide three-digit numbers.

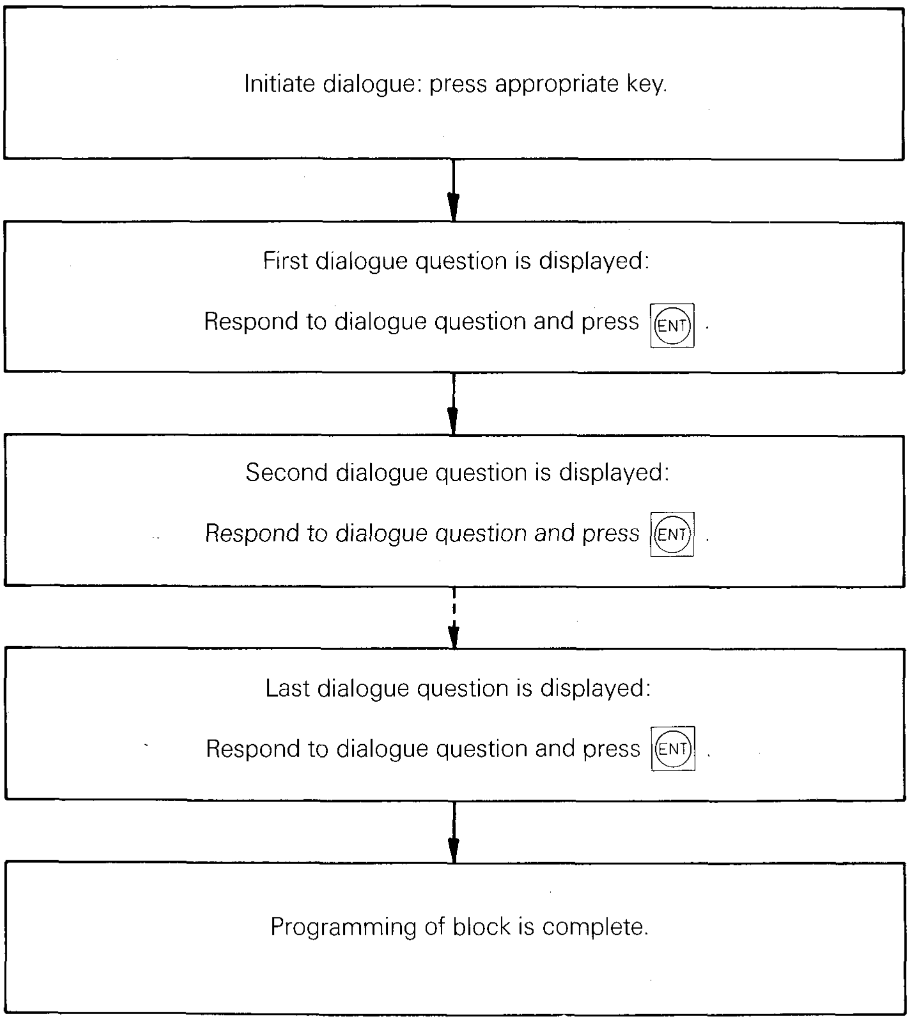
G) Dialogues of TNC 150

Operation and programming of the HEIDENHAIN TNC 150-Control is characterised by the plain language dialogue. After the operator has initiated a dialogue, the control takes over the full guidance with respect to program entry by means of direct questions in plain language.

G 1) Dialogue initiation

Keys for dialogue initiation are explained on fold-out page 2.

G 2) Rules for responding to dialogue questions in program blocks



[NO ENT]-key

Certain dialogue questions can be responded to – without entry of a numerical value – by pressing the [NO ENT] -key:

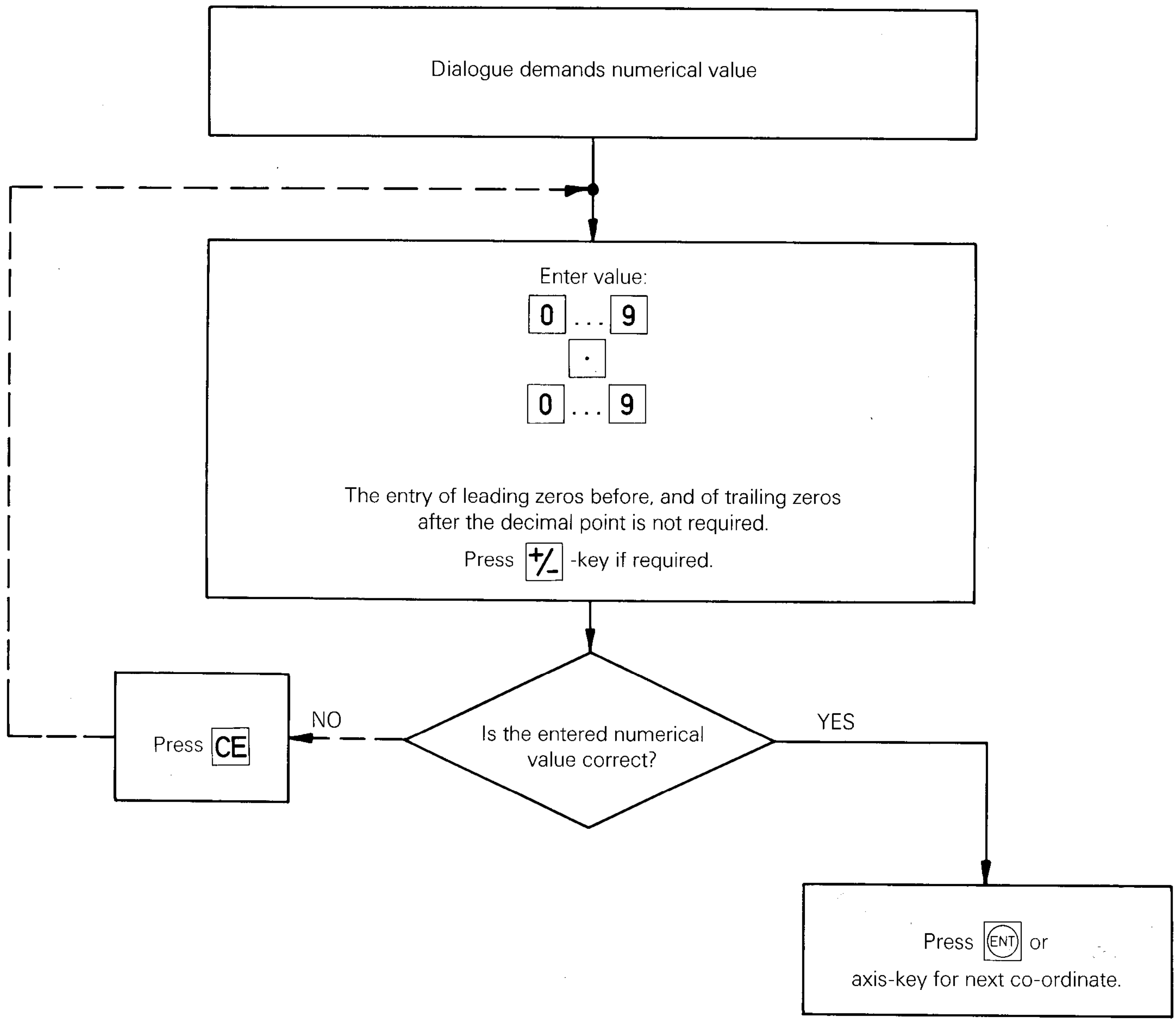
When executing the program, the data last programmed is valid. These types of dialogue questions are especially dealt with in the individual sections of this manual.

[END □]-key

With positioning blocks and tool calls, block entry can be terminated in advance by pressing [END □] .

With program execution, the last values programmed are also valid (see also section M 3.2.4).

G 3) Entry of numerical values



Entry step of dimensions and co-ordinates:

.Lengths down to 0.001 mm or 0.0001 inch
.Angles down to 0.001°

Entry range:
.for lengths ± 30000.000 mm or 1181.1024 inches
.for polar co-ordinate angles ± 14400°
.for fourth axis as rotary axis ± 30000.000°

H) Fault/Error prevention and diagnosis

H 1) Fault/error indication

The TNC 150 possesses an extensive monitoring system for entry and operating errors and for diagnosis of technical defects within the control/machine-system.

The following is under supervision:
.Programming and operating errors
e. g. error indication
KEY NON-FUNCTIONAL
CIRCLE END POS. INCORRECT
ENTRY VALUE INCORRECT

.Internal control electronics
e. g. fault indication
TNC-OPERATING TEMP. EXCEEDED
EXCHANGE BUFFER BATTERY
TNC-ELECTRONICS DEFECTIVE

.Transducers and certain machine functions
e. g. fault indication
X-MEASURING SYSTEM DEFECTIVE
GROSS POSITIONING ERROR
RELAY EXT. DC VOLTAGE MISSING

The control differentiates between minor and major faults. **Major faults** are indicated by a **flashing** signal (e. g. malfunctioning of measuring systems, drives and control electronics). This simultaneously activates an automatic machine switch-off via the **EMERGENCY STOP** contact of the control.

H 2) Cancellation of fault/error indication

.minor faults/errors
e. g. **KEY NON-FUNCTIONAL**
These errors can be cancelled by pressing CE .
.major faults/errors
e. g. **GROSS POSITIONING ERROR**
These faults/errors (indicated by a **flashing** signal) can only be cancelled by **switching off the mains power.**

H 3) Fault indication "Exchange buffer battery"

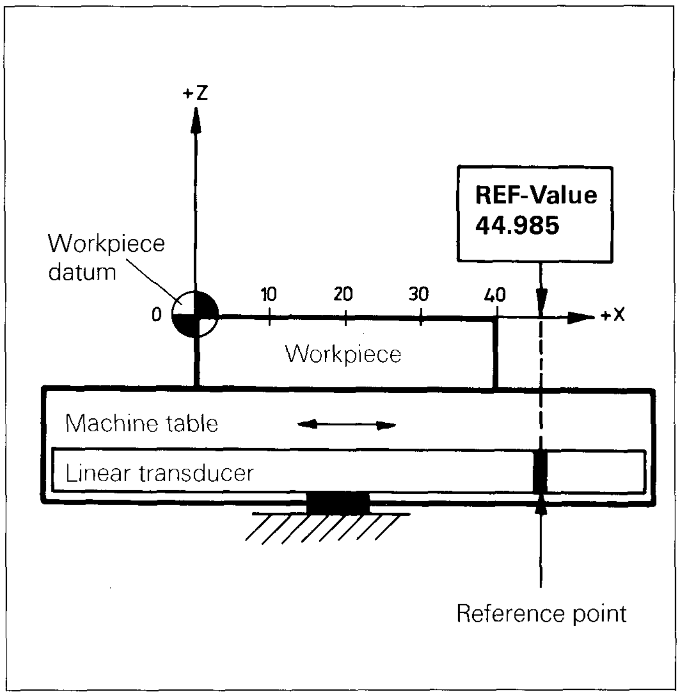
If the dialogue display indicates **EXCHANGE BUFFER BATTERY,** new batteries must be inserted ("empty" batteries retain the program content for at least 1 week). The buffer battery compartment is located beneath the screw-cover in the lower left-hand corner of the operating panel (see section D 1). When exchanging the batteries, special care should be taken that the polarity is correct (plus-pole of battery outwards).

The batteries to be used have IEC-designation "LR 6" and must be of the leak-proof type. We especially recommend the use of Varta batteries type "4006".

With discharged (or missing) buffer batteries, the program memory is supplied by the mains power supply. Continuation of operation is therefore possible – however, the memory content will become erased in the event of a mains power failure. It must be remembered that **the TNC must be switched on during a battery change: If a mains power failure occurs during a battery change (discharged or missing batteries), a new entry of the machine parameters and the machining program is necessary (see section R)!**

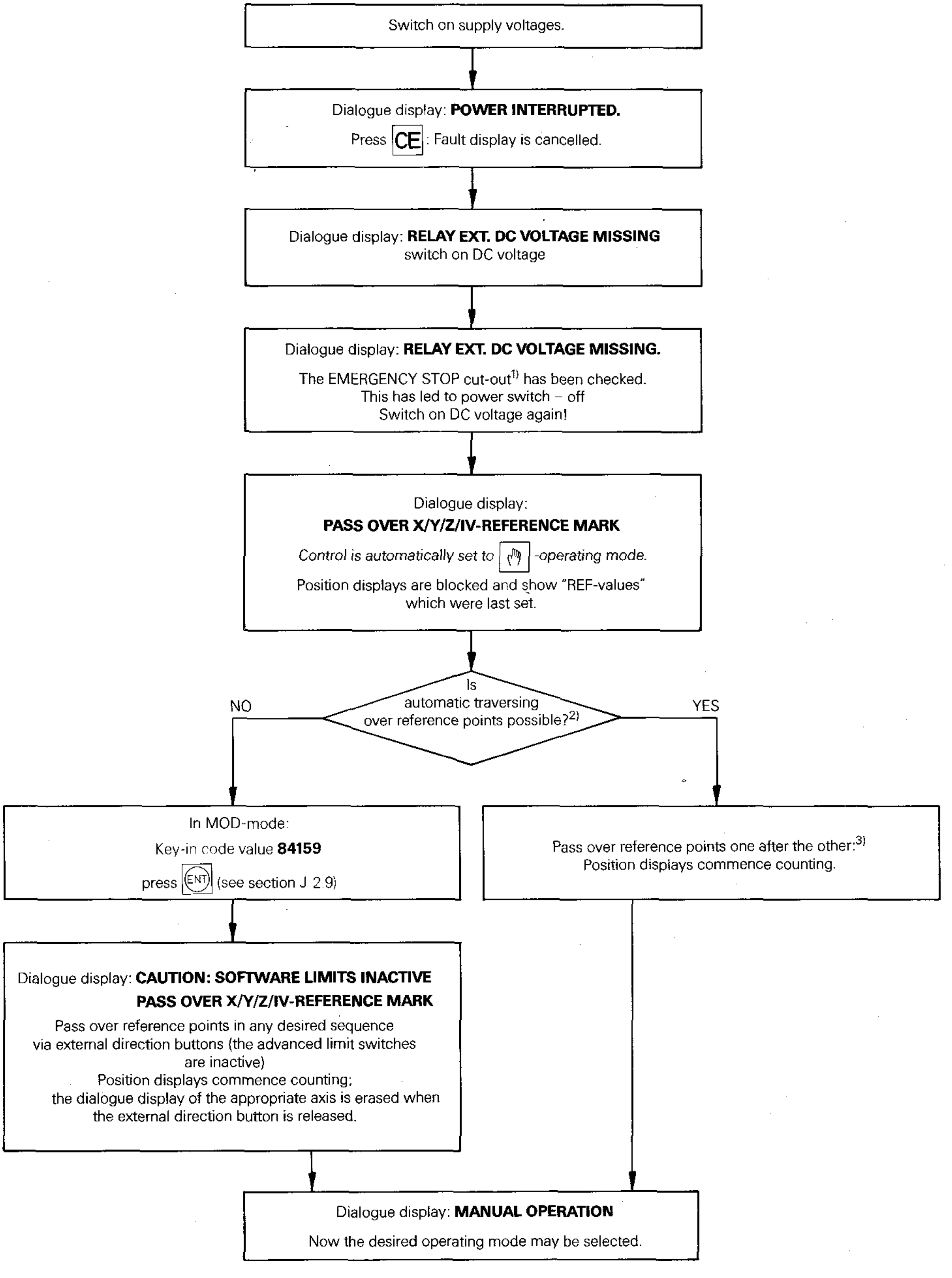
I) TNC 150 switch-on and reference mark routine

The transducers of all machine axes possess reference marks. These marks, when passed over, produce a reference mark signal, which is then processed into a square-wave pulse within the control. The pulse determines a definite correlation between **positions** of the particular machine axis and the **position value.**
The position of the reference mark on the machine axis is referred to as the "reference point".
The reference points must be traversed over after every interruption of power (due to the TNC 150 being equipped with software-limit switches) otherwise all possibilities of further operation are inhibited! Moreover, by traversing over all reference marks, **the workpiece datum which was last set before interruption of power, is reproduced** (see section K 2).



When setting a datum, certain numerical values are allocated to the reference points, the so-called **"REF-values".** These values are automatically stored within the control so that if, after interruption of power, the last datum which was previously set can be easily reproduced by traversing over the axis-reference marks.

TNC 150 switch-on and reference mark routine is performed as follows:



1) The EMERGENCY STOP-check is carried out with control switch-on. The EMERGENCY STOP-circuit is extremely important for operational safety of both machine and control.
2) The speed, axis sequence and traversing direction for automatic traversing over the reference points have already been programmed with the machine parameters (see section R). Before every reference mark routine, check that no obstructions e. g. jigs, are present.
3) *Automatic traversing over the reference points is activated via the external start-button. For reasons of safety, each axis must be individually started.* The position displays only commence counting when the reference points have been passed over; the dialogue display of each axis is then erased.

J) Supplementary operating modes [MOD]

The following supplementary operating modes may be selected via the [MOD]-key:

.Vacant blocks
.Changeover mm/inch
.Fouth axis on/off
.Position data display: Actual position
"Distance to go" to reference points
Trailing error (lag)
Nominal position
"Distance to go" to nominal position
.Position display enlarged/small
.Baud rate
.Traversing range limits
.NC-Software number
.PLC-Software number
.Code number

If program run has been started in [→] mode, only the following supplementary modes may be selected:
.Position display enlarged/small
.Vacant blocks

If the error **POWER INTERRUPTED** is displayed on the screen, only the following supplementary modes can be selected:
.Code number
.Fourth axis on/off
.NC-software number
.PLC-software number

After cancellation of the error **POWER INTERRUPTED** the mode "Fourth axis on/off" can no longer be selected.

J 1) Selection and cancellation of supplementary operating modes

[MOD] may be selected in any other existing operating mode:

Press [MOD].
↓
Select desired MOD-function with [↓], [↑] -keys or repetitive pressing of [MOD].

Cancellation of MOD-routine is by pressing [DEL □].

If a numerical value was amended prior to cancellation, this value must be stored by pressing [ENT].

J 2) Explanation of MOD-functions

J 2.1) Vacant blocks

The MOD-function **VACANT BLOCKS** indicates the number of program blocks which are still available in the memory.

Example of display:

VACANT BLOCKS = 1179

J 2.2) Changeover mm/inch

The control can operate in either metric or imperial mode. Changeover of position displays from "mm to inch" is as follows:

After selection of MOD function:
Dialogue question: **CHANGE MM/INCH**

↓

Press ENT.
Control changes from mm-to INCH operation or vice-versa.

J 2.3) Position data display

TNC 150 can be switched over for display of the following position data:

Display type	**VDU screen abbreviation**	**Remarks**
Actual position	**ACTL.**	Display of actual momentary position
Distance to go to reference points	**REF**	Display of remaining distance to reference points (marks) of transducer
Trailing error (lag)	**LAG**	Display of deviations between nominal and actual positions: Nominal value-actual value
Nominal position	**NOML.**	Display of momentary nominal position calculated by the control
Distance to go	**DIST.**	Display of "distances to go" to nominal target position (differences between programmed nominal position and momentary actual position)

After selection of MOD-function:
Dialogue display: **POS. DATA DISPLAY LARGE/SMALL**

↓

Press ENT.
Display is switched over from small to enlarged characters or vice-versa.

J 2.4) Position display enlarged/small

In operating modes **program run-single block** and **automatic program run,** the position data on the TNC 150 screen can be switched over from small screen characters to enlarged screen characters.

Small display:
Four program blocks and position (in small characters).

Enlarged display:
The current program block and position (in large characters).

After selection of MOD-function:
Dialogue display: **POSITION DATA DISPLAY LARGE/SMALL**

↓

With repetitive pressing of [ENT] required display type is selected.

J 2.5) Switchover of Baud rate

With TNC 150 the transfer rate of the data interface V.24 (RS-232-C) is automatically set to 2400 Baud for connection to a HEIDENHAIN magnetic tape cassette unit ME 101/ME 102.
If a peripheral unit with another Baud rate is to be connected to the TNC 150 (without interconnection of an ME-unit), the corresponding Baud rate must be entered.

After selection of MOD-function:
Dialogue display: **BAUD RATE**

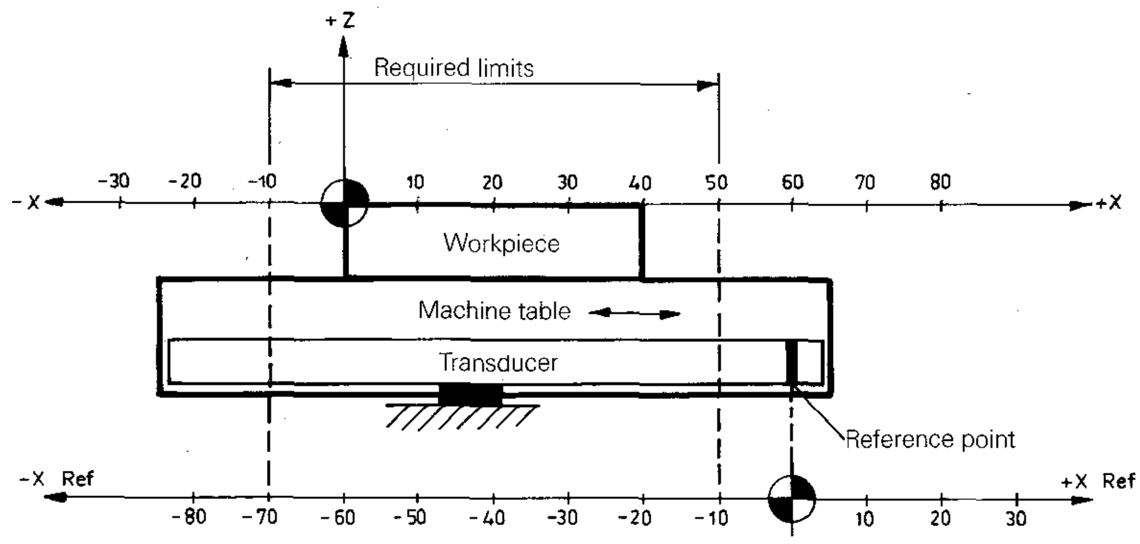
↓

If necessary, enter new Baud rate (110, 150, 300, 600, 1200 or 2400 Baud) and enter into memory with [ENT]

The Baud rate is also entered into the memory by advancing the MOD-functions via the [↑], [↓] or [MOD] -keys.

J 2.6) Traversing range limitation

The traversing ranges can be predetermined by the software safety limits, e.g. in order to prevent a collision. This limitation is determined in every axis one after the other in the + and − direction and with reference to the reference points. To determine the limit locations, the display must be switched over to "REF".



Limit X+ = −10.000
Limit X− = −70.000

Select [manual] or [handwheel] mode.

↓

Press [MOD] :
Select MOD-function **POSITION DATA**
Switch position readout to "REF"

↓

Select MOD-function **LIMIT**
for required axis and direction.

↓

Traverse to limit positions via external direction buttons
or electronic handwheel

↓

Program displayed values:
[0] ... [9] and [ENT].

.
.
.
etc.

If operation is **without traversing range limitations,** it is recommended that **+ 30000 mm** and **− 30000 mm** be entered for the appropriate axis.

J 2.7) Display of NC-software number

The appropriate NC-Software number can be displayed by means of this MOD-function

Example of display:

NC: SOFTWARE NUMBER 221 804 04

J 2.8) Display of PLC-software number

The appropriate PLC-software number can be displayed by means of this MOD-function

Example of display:

PC: SOFTWARE NUMBER 221 510 01

J 2.9) Code number

Certain operating modes can be selected by using code numbers via this MOD-function. By entering the code number **84 159**, machine axes can be traversed via the external direction buttons without prior traversing over reference marks (see section I):

Switch on control
Dialogue display: **PASS OVER REFERENCE MARK**

↓

Press [MOD].

↓

Select MOD-function **CODE NUMBER** with single press of [↑]:
Enter code number **84 159**

↓

The axes can be traversed via the external direction buttons;
positions displays commence counting upon crossing
of reference marks.

J 2.10) Fourth axis on/off

It is **only possible** to activate or inactivate the fourth axis (i.e. for optional rotary table operation) immediately after control switch-on. However, **before** cancellation of the error display **POWER INTERRUPTED.**

Switch on control
Dialogue display: **POWER INTERRUPTED**

↓

Press [MOD] and then [↓]
Dialogue display: **POWER INTERRUPTED**
AXIS 4 ON/OFF

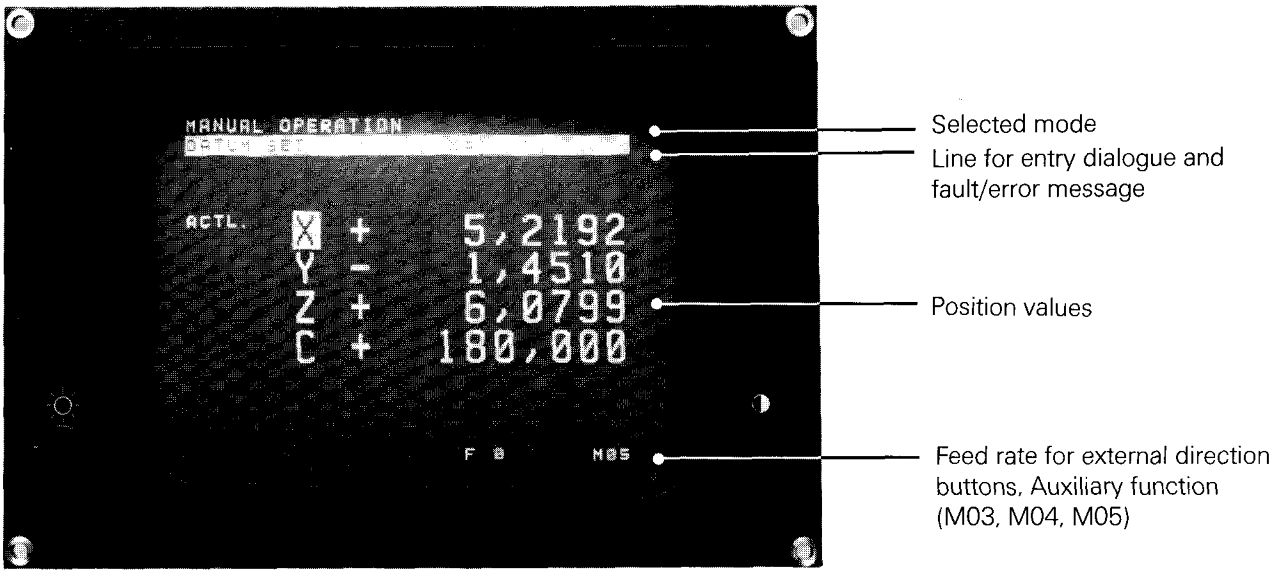
↓

Press [ENT]
The control activates or inactivates the fourth axis
depending on the previous condition

Please note:
This MOD-function can no longer be selected after cancellation of the **POWER INTERRUPTED** display.

K) Manual operation

VDU-display:



K 1) Manual traversing of machine axes

When switching on the control, the "manual" mode is automatically selected. The machine can be traversed via the axis direction buttons on the machine control panel. The traversing speed can be set either

a) via the override potentiometer of the control or

b) via an external potentiometer

depending on how the TNC 150 has been adapted to the machine.

The machine axes can be traversed in two ways:

.key-in operation
The desired axis direction button is pressed and the selected machine axis will traverse. It is stopped when the button is no longer being pressed.

.continuous operation
If, after pressing the axis direction button, the external start-button is pressed, the machine axis will continue to traverse even when the buttons are no longer being pressed. Stopping is activated by pressing the external stop-button.

K 2) Setting datum

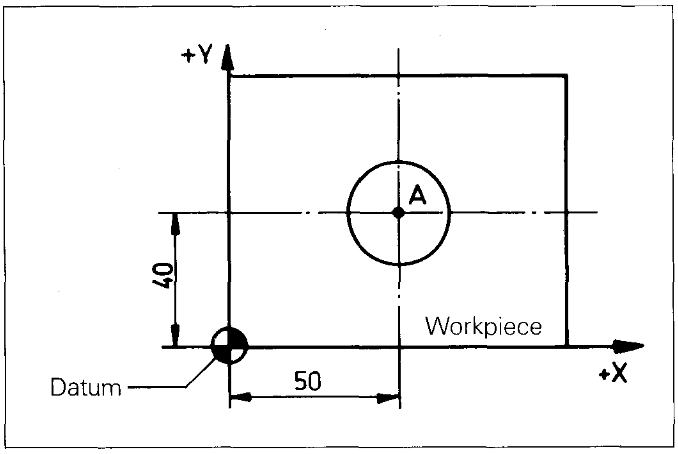
In order to machine a workpiece, the display values must correspond to the workpiece positions. When setting a datum, the three position displays are pre-set to defined values (i. e. numerical values are set into the displays as starting values whereby the machine axes already have a certain position). If, for instance, the workpiece dimensions of the sketch below are referenced to the lower left-hand corner, this corner can be declared as the "workpiece datum" and the value 0 is allocated for the X and Y-axes.

For this, either

a) the workpiece datum can be approached (e. g. with an optical edge finder) and the X and Y-displays be set to 0.

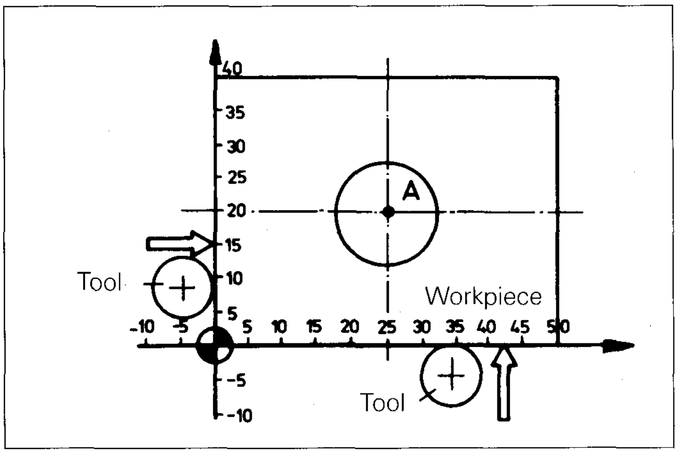
b) the known position A is approached (e. g. with a centring device for the bore) and the X-display set to 50 and the Y-display set to 40

or

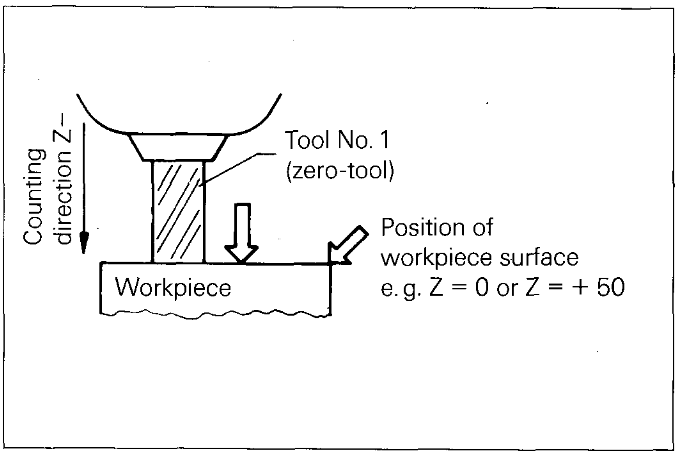


c) the workpiece datum is determined by "touching" the workpiece with the tool (or a mechanical edge finder) which has a diameter of 10 mm, the left-hand workpiece edge is approached first and when touched, the X-display is set to −5. Similarly, the lower workpiece edge is approached and touched and the Y-display is set to −5.

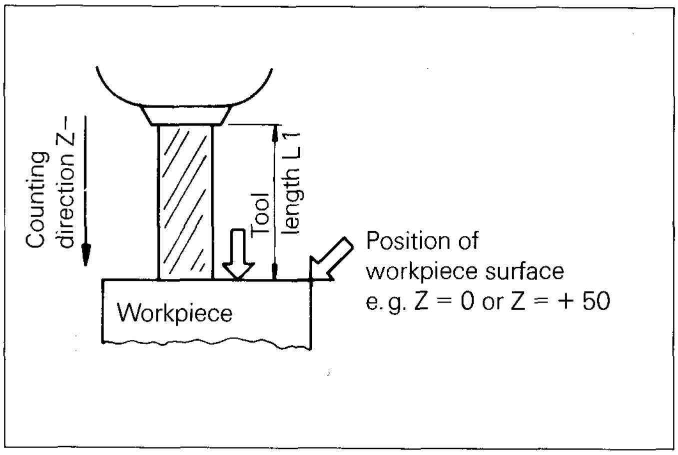
The presetting of both axes corresponds to case b) *(instead of 50 and 40, the value −5 is to be entered).*



In this example, the Z-axis corresponded to the tool axis. Determination of the workpiece datum for the Z-axis is performed in various ways depending on the type of tool being used.



a) Tools in chuck (with or without length stop)
In order to determine the workpiece datum for the tool axis, the first tool must be inserted (Tool 1 = zero-tool, see also section M 2.1 "Tool definition"). If, for example, the workpiece surface is to be referenced as 0, the tool tip must touch the workpiece surface and the Z-axis then set to 0 for this position (as per a) for axes X and Y).
If the workpiece surface is to have a value other than 0, then the tool axis must be pre-set to this value e.g. + 50.
The compensation values for remaining tools are referenced to tool 1 (zero-tool).



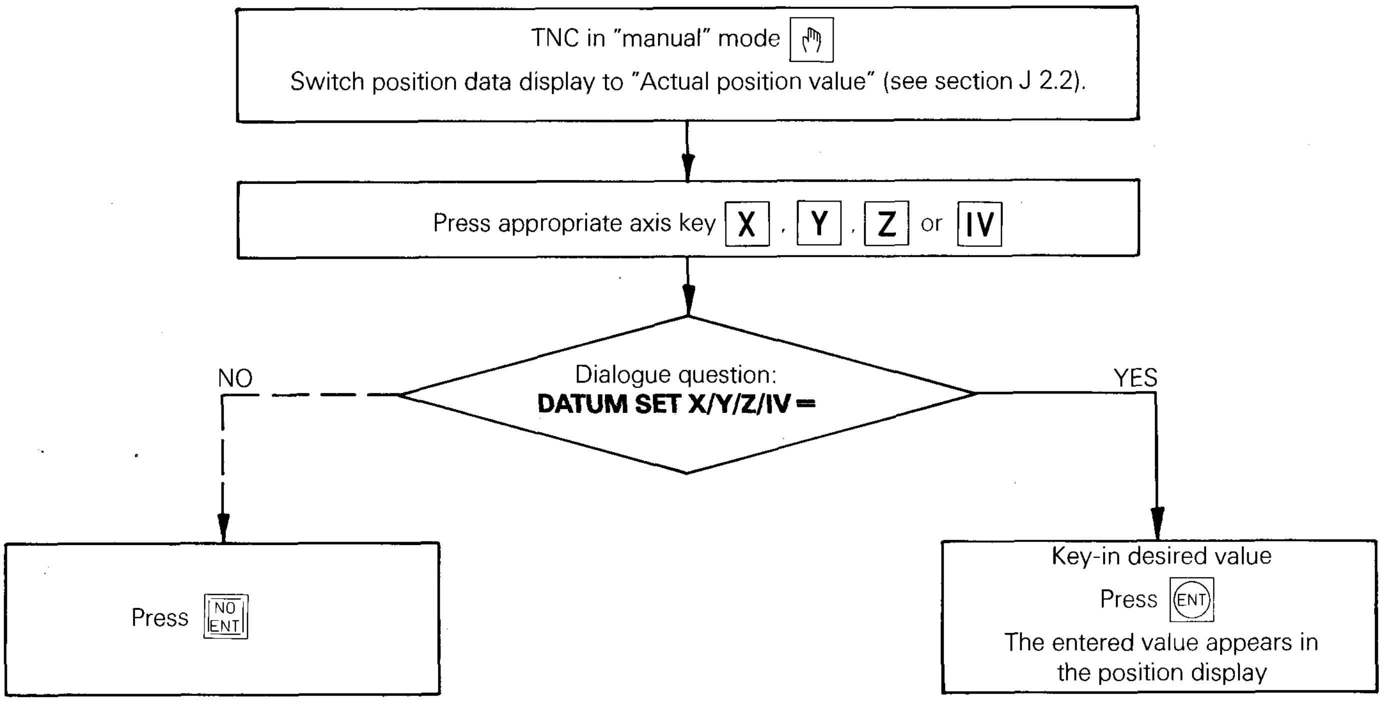
b) Pre-set tools
With pre-set tools, the tool length is already known. The workpiece surface is touched with any available tool. In order to set the workpiece surface to 0, the tool axis must be pre-set to the length + L 1 of the appropriate tool.
If the workpiece surface has a different value to 0, the tool axis must then be set to the datum value as follows:

(Datum value Z) = (tool length L 1) + (surface position)

Example:
Tool length L = 100 mm; workpiece surface position + 50 mm

(Datum value Z) = 100 mm + 50 mm = 150 mm

Presetting of position displays is performed as follows:



If position data display is switched to "Distance to go to REF-points" (see section J 2.2) the datum cannot be set.

K 3) Output of spindle speeds and auxiliary functions in "manual" mode

With TNC 150, output of spindle speeds and auxiliary functions is also possible in the [manual] operating mode.

Spindle speeds:

Press [TOOL CALL].

↓

Dialogue question: **SPINDLE SPEED RPM?**
Key-in required spindle speed.

↓

Press external (START) -button.

Auxiliary functions:

Press [STOP].

↓

Dialogue question: **AUXILIARY FUNCTION M?**
Key-in required auxiliary function.

↓

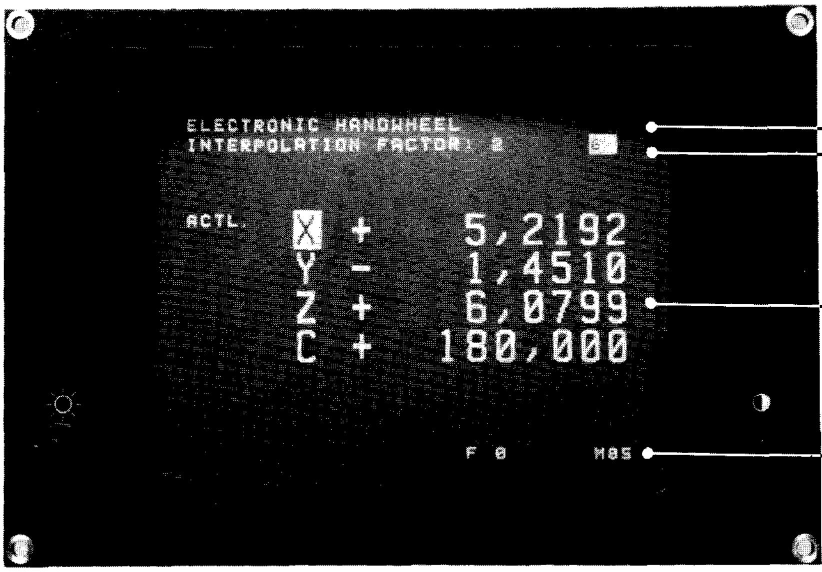
Press external (START) -button.

L) "Electronic handwheel" mode

The control can be equipped with an electronic handwheel for easy set-up operation.

The handwheel is active when the -mode is on.

VDU-display:



Selected mode

The subdivision factor determines the amount of traverse per handwheel rotation
2 = current subdivision factor
6 = newly entered subdivision factor

Position values

Feed rate for external direction buttons, Auxiliary function (M03, M04, M05)

Switch-over between axes is performed by pressing the appropriate TNC-axis key X, Y, Z or IV.
The traversing speed is determined by a subdividing factor.
The required subdividing factor is keyed-in and transferred by pressing ENT.
Available entry values: 1...10.

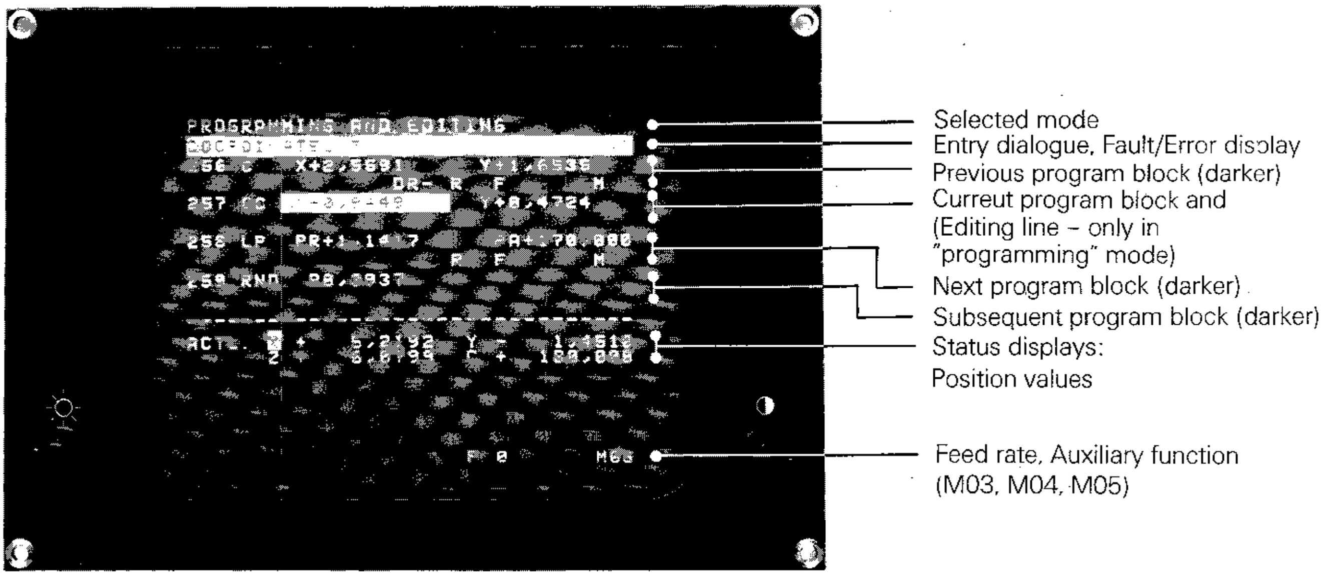
Subdividing factor	Traversing distance in mm/ rev. of handwheel
1	10 mm
2	5 mm
3	2.5 mm
4	1.25 mm
5	0.625 mm
6	0.313 mm
7	0.156 mm
8	0.078 mm
9	0.039 mm
10	0.020 mm

Depending on the rapid traversing speed of the machine, the subdividing factor is inhibited for high speeds.

The external axis direction buttons also remain active in this mode!

M) "Programming" mode [⇔]

VDU-display:



M 1) Program organisation

TNC 150 organise a library of 24 different programs with a total of up to 1200 blocks.

☞ A machining program may contain up to 999 program blocks.

M 1.1) Designation of a new program

A program can only be entered after it has been allocated with a program number. The program number may have up to 8 digits.

Programm designation is performed in the [⇔]-mode.
Dialogue initiation: press [PGM NR]

Dialogue question	Response
PROGRAM SELECTION PROGRAM NUMBER=	Key-in program number; press [ENT].
MM = ENT/INCH = NO ENT	If machining program entry is in mm; press [ENT]. If machining program is entered in INCH; press [NO ENT].

As an example, the TNC 150-screen would display the following blocks after entry of the program number 100 052 31:

0 BEGIN PGM 100 052 31 MM

1 END PGM 100 052 31 MM

If blocks are now entered, these will be inserted between the BEGIN-block and the END-block.

0 BEGIN PGM 100 052 31 MM

1 L X + 20,000 Y + 35,000

RO F100 M

2 END PGM 100 052 31 MM

For entry of a **second program,** the [PGM NR]-key must be re-pressed
The display shows:

PROGRAM SELECTION

PROGRAM NUMBER =

100 052 31

The VDU-screen display indicates that a program with the designation number 100 052 31 is already stored.

M 1.2) Selecting a program

Dialogue initiation: Press [PGM NR]

Dialogue question	Response
PROGRAM ADDRESS PROGRAM NUMBER =	Either enter program number or select program number displayed on the VDU-screen via [↓], [↑] and [←] [→] Press [ENT] The beginning of the selected program is displayed

M 2) Compensation values for tool length and radius

M 2.1) Tool definition [TOOL DEF]

The TNC 150 allows for tool compensation. Therefore, the entry of a machining program can be made directly from the drawing dimensions of the workpiece contour. For tool compensation, the length and the radius of the tool must be defined. This data is entered with the TOOL DEFINITION.

Tool definition entry may take place at any location within the machining program. A conventient search routine facility enables a certain tool definition to be easily called up for inspection or amendment (see section M 8.6).

Dialogue initiation: Press [TOOL DEF]

Dialogue question	Response
TOOL NUMBER?	Key-in tool number; press [ENT].
TOOL LENGTH L?	Enter numerical value or parameter (see section M 5) for length compensation; press [ENT].
TOOL RADIUS R?	Enter numerical value or parameter (see section M 5); press [ENT].

Dialogue question:
TOOL NUMBER?

Possible entry values:
.for machines **without** automatic tool change: 1 - 254
.for machines **with** automatic tool change: 1 - 99

No tool may be allocated with the number 0 (this tool number has already been allocated internally for "no tool" i. e. for length L = 0 and radius R = 0).

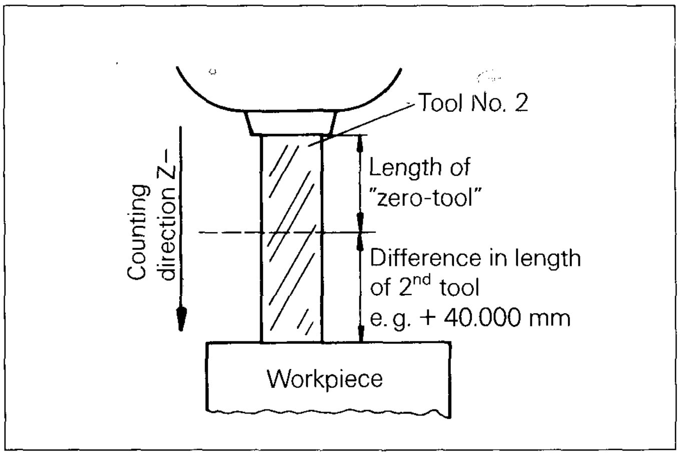
Dialogue question:
TOOL LENGTH L?

Entry range: ± 30 000.000 mm

The compensation value for the tool length L can be determined in various ways:

a) Tools in chuck without length stop
Firstly, the datum of the tool axis must be defined (see section K 2).
The surface of the workpiece is touched with the tip of the first tool and the position display of the appropriate axis (e. g. Z-axis) is pre-set. The first tool is defined as the "zero-tool", i. e. tool length L = 0 is entered into the tool definition for the first tool:

Tool length L = 0



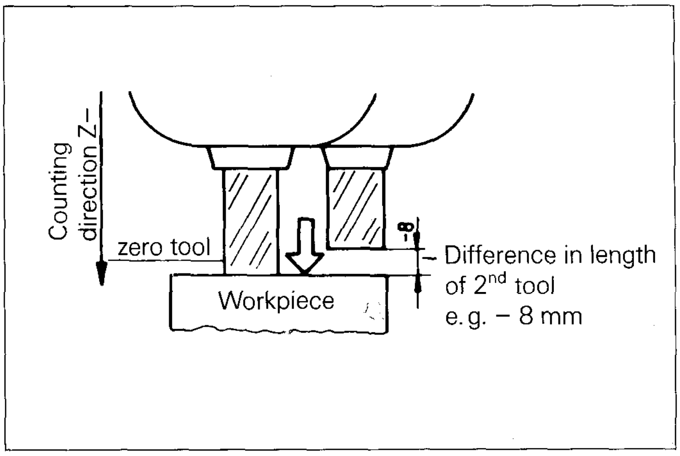
For all subsequent tools (also with a re-insertion of tool 1) the difference in length, with respect to the first tool, must be entered. If the workpiece surface has been declared with the position Z = 0, the length compensation can be determined after insertion of the new tool by touching the workpiece surface. The compensation value is indicated in the position display of the Z-axis and can be transferred as an entry value by means of the [+]-key (including sign). This value is entered in the tool definition for the appropriate tool:
e. g. Tool length L = 40.000

If the workpiece surface does not correspond to 0, the tool length must be determined after datum set as follows:
Touch workpiece surface and note down the value in the position display of the tool-axis (with sign). Now determine the compensation value L according to the following formula:

(Compensation value L) = (Actual position value Z) — (Position surface)

Example:
Position value of Z-axis = + 42, position of surface = + 50
Compensation value L = (+ 42) − (+ 50) = − 8.
This value must be entered into the appropriate tool definition:

Tool length L = − 8.



b) Tool in chuck with length stop
The compensation value for the tool length is defined as in a). A compensation value which has been defined, does not change after removal or insertion of the tool.

c) Pre-set tools
With pre-set tools, the tool length is determined on a tool setting device, i. e. all tool lengths are already known and do not have to be determined at the machine. The length definition corresponds to the tool lengths which have to be determined on the tool-presetter.

Dialogue question:
TOOL RADIUS R?

Entry range: ± 30000.000 mm

The tool radius is always entered as a positive value. Negative values for tool radius compensation can only be applied in one special case (see section N 2, Playback programming).
When using drilling tools, the tool radius can be programmed with 0.

The tool definition allocates one program block.

TOOL DEF 28 L+ 100,000
R+ 20,000

M 2.2) Tool call/Tool change [TOOL CALL]

With a tool change, the data for the new tool is called up with the [TOOL CALL]-key.

Dialogue initiation: [TOOL CALL]-key

Dialogue question	Response
TOOL NUMBER?	Key-in tool number; press [ENT].
WORKING SPINDLE AXIS X/Y/Z?	Press axis-key [X], [Y] or [Z]; **do not** press [ENT]-key.
SPINDLE SPEED S RPM?	Key-in spindle speed; press [ENT].

Block entry may be terminated in advance by pressing the [END]-key.
If dialogue questions are responded to with [NO ENT], data entry is omitted and the next dialogue question appears.
In this case, the data entered with the previous tool call block remains valid.

Dialogue question:
TOOL NUMBER?

Possible entry values:
.for machines without automatic tool changer: 0 – 254
.for machines with automatic tool changer: 0 – 99
(the control only provides tool numbers 0 – 99 in coded form).

Dialogue question:
WORKING SPINDLE AXIS X/Y/Z?

Possible entry data: X, Y, Z or if required, U, V, W by press [IV]

Definition of axis to which the spindle-axis is parallel. The tool length compensation is effective in this axis; the radius compensation is effective in the other two axes (if reqd.).

Programming of the fourth axis within a tool call is only possible if the fourth axis is linear.

Dialogue question:
SPINDLE SPEED S RPM?

Programmable spindle speeds are given in the table of S-functions (section No. F 3).

Entry is with a maximum of 4 digits in rev./min. If necessary, the control rounds-off the value to the next standard value. If, however the entered spindle speed is outside of the permissible speed range (defined by machine parameters), the error **WRONG RPM** is displayed.

The tool call only allocates one program block:

TOOL CALL 29 Z S 1000,000

Tool call with tool number 0
If, in a machining program traverses are to be made without tool compensation, the tool call is to be programmed with the tool number 0. A tool with number 0 is already pre-programmed as "no tool", i.e. length L = 0 and radius R = 0.

TOOL CALL 0 Z S 0,000

If the tool call is initiated in the [▣▶], [⇉▶] or [→] the active tool compensations are disregarded and the machine traverses to the nominal positions without compensation.

Please note:
Depending on the machine parameters which have been entered, the dialogue question

NEXT TOOL NUMBER?

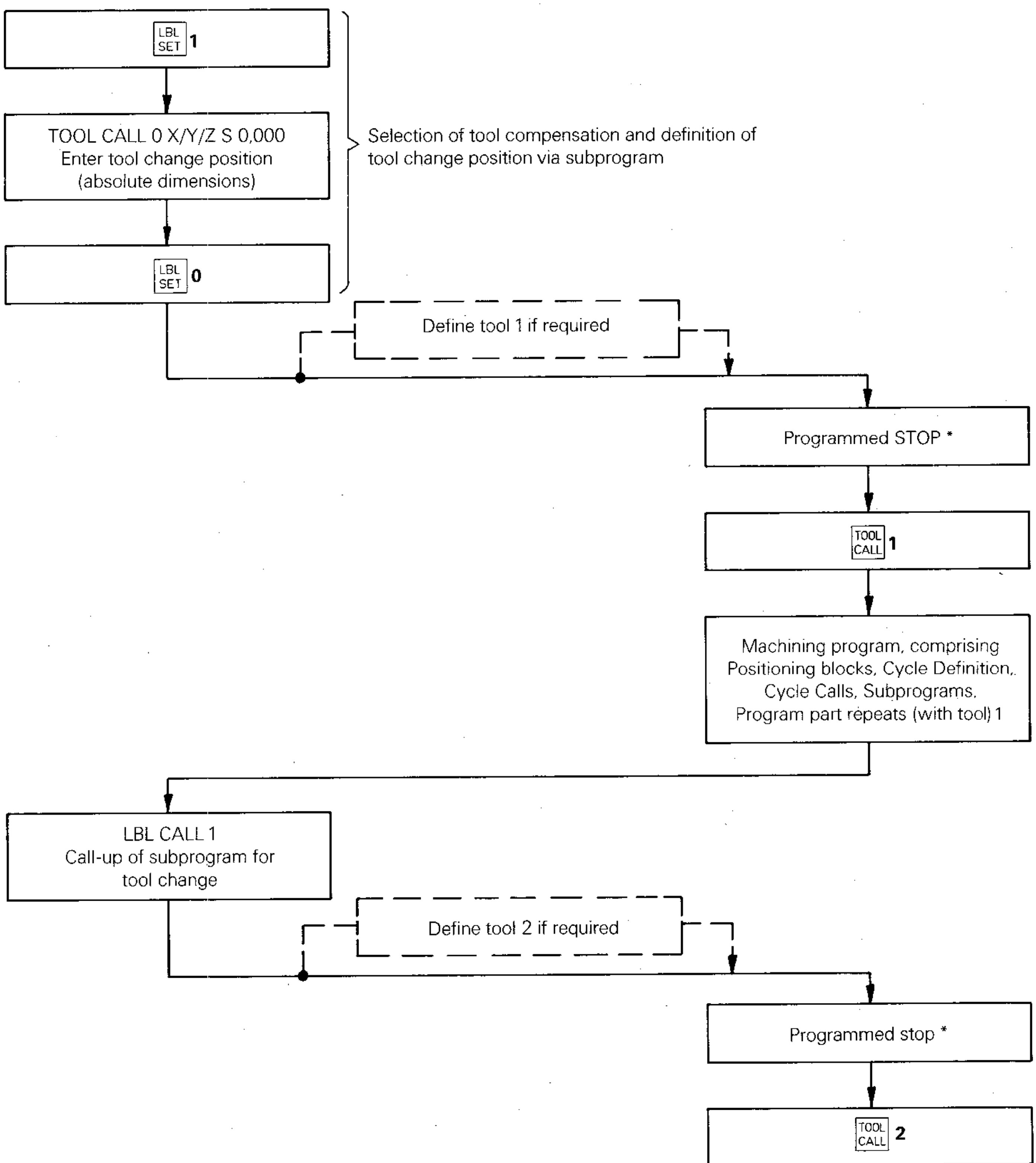
can appear after the dialogue question

TOOL NUMBER?

The output of the next tool number is only required if the machine is equipped with an automatic tool changer which searches for the next tool whilst operation is being carried out with the current tool. More detailed information can be obtained from your machine tool manufacturer.

A STOP is to be programmed before every tool change. The STOP can be neglected only when the tool call is required for an rpm-change.

Programming sequence for a tool change



etc.

* The stop can be programmed:

.via the STOP-key (see section M 4), or

.via auxiliary function M 00 or M 06 (see section F 2)

M 3) Programming for workpiece contour machining

M 3.1) Tool contouring offset R^{R}_{+} R^{L}_{-}

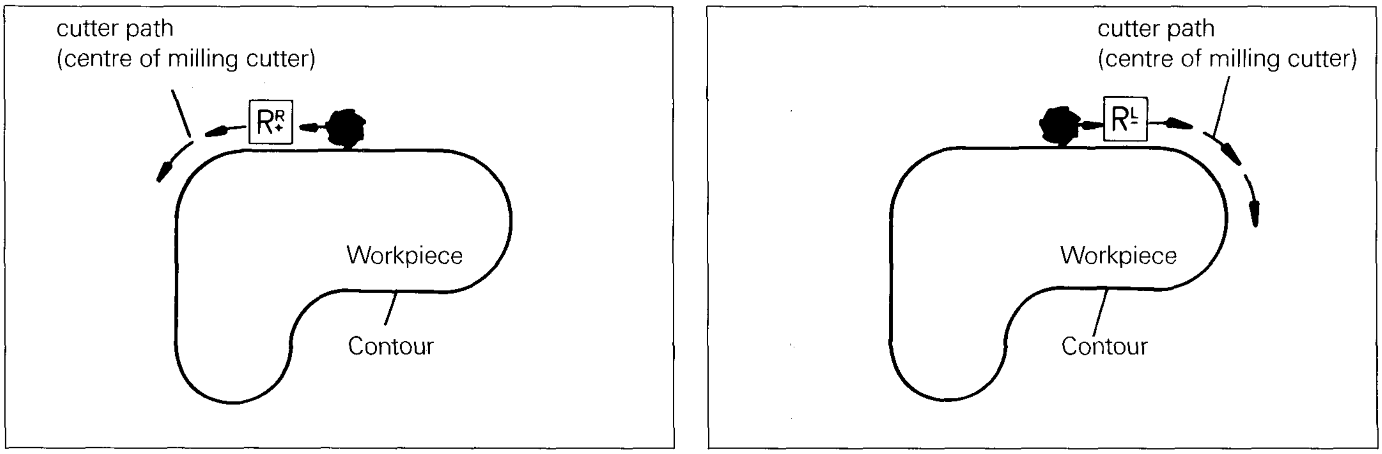
With TNC 150, the actual workpiece contour may be programmed. Tool length and radius is automatically compensated for by the control. Since the data entered for the tool length is sufficient, the radius compensation must also define whether the tool is located to the right or left of the contour in the traversing direction:

R^{R}_{+} -key: In the traversing direction, the centre of the milling cutter travels on the **right-hand side** of the contour.

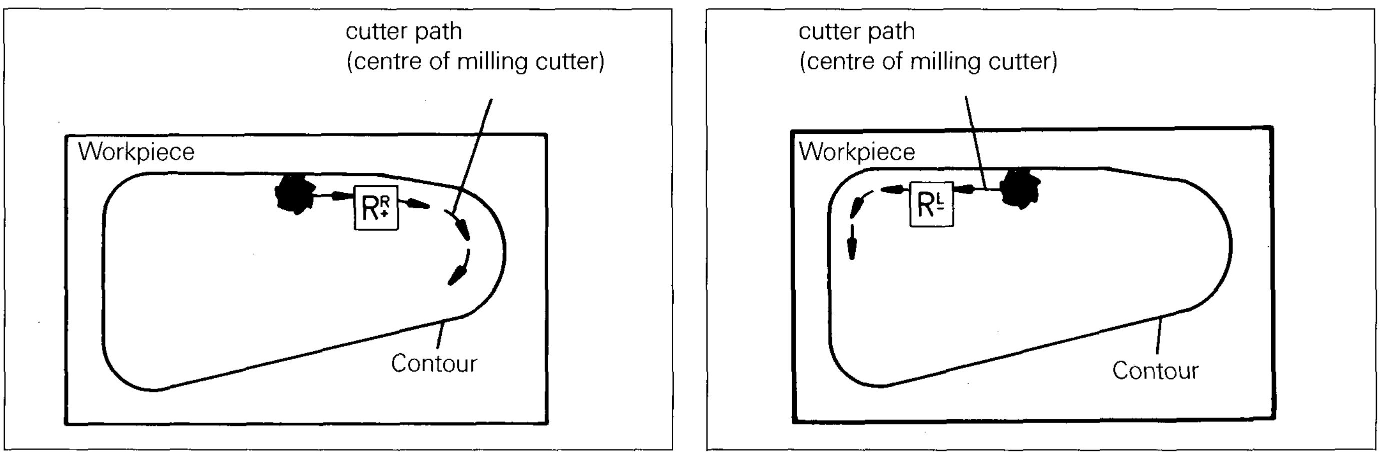
R^{L}_{-} -key: In the traversing direction, the centre of the milling cutter travels on the **left-hand side** of the contour.

The double function of both keys is explained in section N.

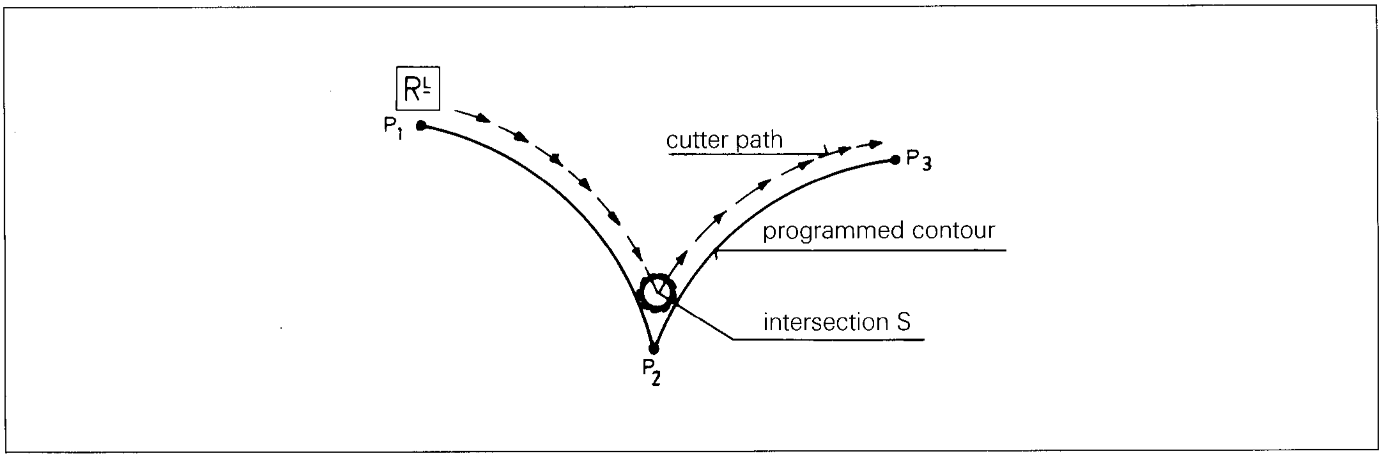
Milling an external contour



Milling an internal contour

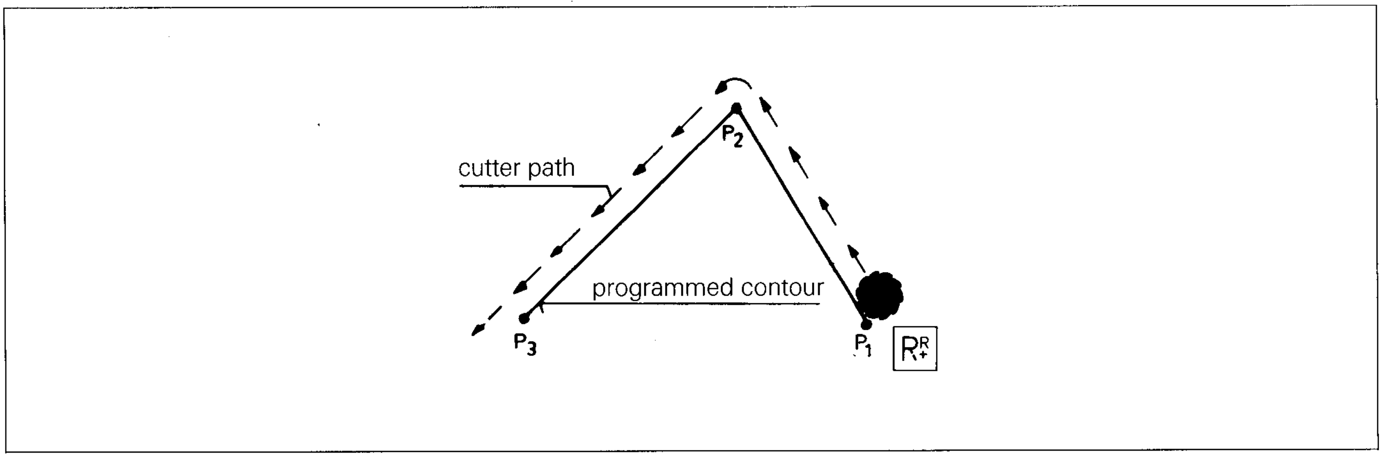


Automatic calculation of contour path intersection for internal corners



The TNC 150 automatically determines the point of intersection S for the cutter path which is parallel to the workpiece contour and also guides the cutter in its correct path. The control prevents the tool from forming a recess at point P 2 which could damage the workpiece.

Automatic insertion of transitional arcs on external corners



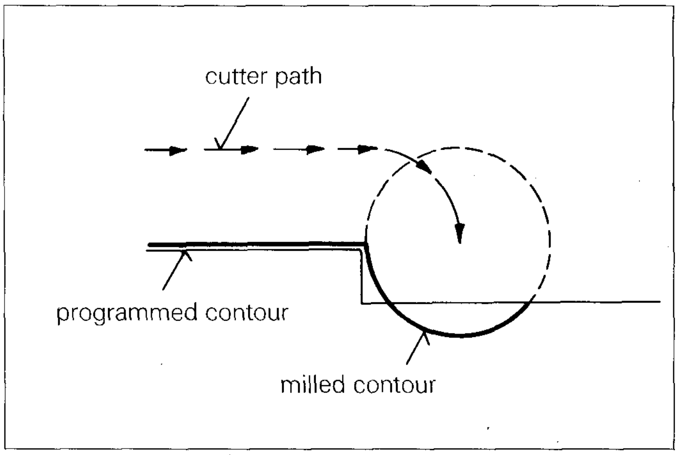
The control automatically provides a transitional arc at external corner P 2. In most cases, the cutter rolls at a constant speed around the corner. If the programmed feed rate is too high for the transitional arc, the cutter speed around corner P2 is automatically reduced to the value which is permanently programmed within the TNC.

A constant contouring speed can be impelled by programming the auxiliary function **M 90,** (see section F 2).

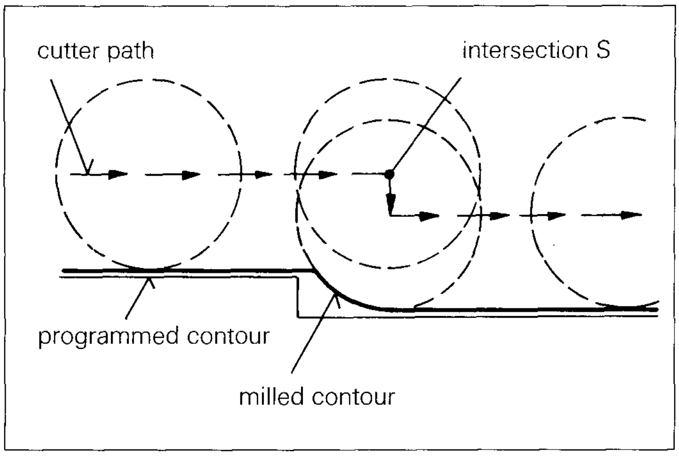
Correction of tool path intersection for external corners: M 97

If **no transitional radius** is to be inserted on an external corner, the **M 97** function is to be programmed into the appropriate block.

Examples:

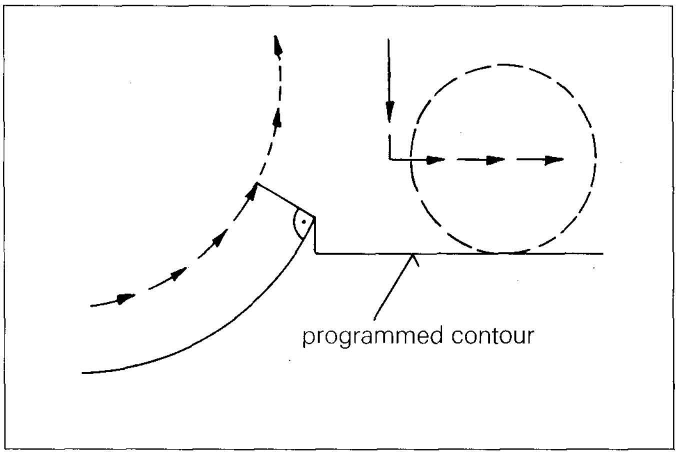


Without M 97: The transitional radius would damage the workpiece.

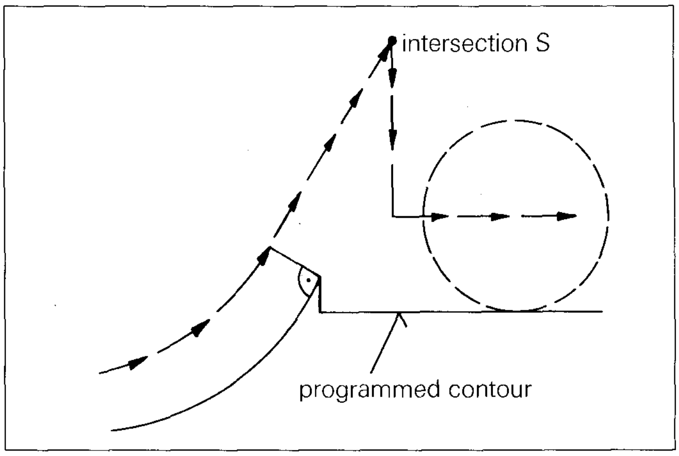


With M 97: No transitional radius is inserted; the control determines the tool path intersecting point S thus preventing damage to the workpiece.

Special case:



The control cannot determine the tool path intersection with M 97.



Remedy: A block is inserted:
L IX+0,000 I Y+0,000
RL F100 M 97
The control determines the point of intersection S and the contour can be milled.

M 3.2) Programming of workpiece contour (Geometry)

In the X, Y and Z axes, TNC 150 can control the machining of contours which comprise **straight sections** (Linear interpolation: simultaneous traversing in two or three axes or traversing in one axis only = single axis traversing) and/or **circular arcs** *(simultaneous traversing in two axes).*

Straight sections are determined by their end positions ([L] -key).
Circular arcs can be determined either by the circle centre ([CC] -key) and starting and end positions ([C] -key) or – when the circular arc describes a tangential transition into the final contour – by the radius only (rounding-off key [RND]).

Helices can be programmed with circular interpolation ([CC] and [C] -keys) in polar co-ordinates and an additional linear movement in the axis which is perpendicular.

TNC 150 also provides for **tangential approach into, and departure from a contour** by following a circular path.
The **"fourth axis"** can perform a linear interpolation routine with any one of the main axes X, Y or Z. By using the fourth axis as a rotary axis in linear interpolation with one of the main axes, a helix can be manufactured.
If the fourth axis is being used for rotary motion (on a rotary table), nominal positions are entered in degrees (°) and feed rate in degrees/min. (°/min.).
Radius compensation is not considered in the fourth axis.
Contour points (i.e. nominal positions) may be entered in "absolute" or "incremental" (chain) dimensions or in cartesian or polar co-ordinates.

With **incremental programming** the [I] -key must be pressed (the indicator lamp is then on). By re-pressing this key (indicator lamp off) the control is returned to the absolute programming mode.
The [I] -key may either be pressed prior to dialogue initiation or afterwards, but before activation of the [ENT] or [END] -key (see section M 3.2.4).

Entry step for dimensions and co-ordinates:
.Lengths down to 0.001 mm or 0.0001 inch
.Angles in degrees down to 0.001°

.**Entry range:**
.Lengths: ± 30000.000 mm or 1181.1024 inches
.Polar co-ordinate angles: absolute ± 360°, incremental ± 5400°
.Fourth axis rotary: ± 30000.000°

M 3.2.1) Entry of positions in Cartesian co-ordinates

The [P] -key must not be pressed.

If the [L], [CC] or [C] -key is pressed, the following dialogue question is displayed:

COORDINATES ?

Response:

For positioning or machining in **one axis only** (single axis traversing, programmed via [L] -key)
.press [I] if reqd.
.press axis-key and enter numerical value,
.press [ENT] or [END] (see section M 3.2.4).

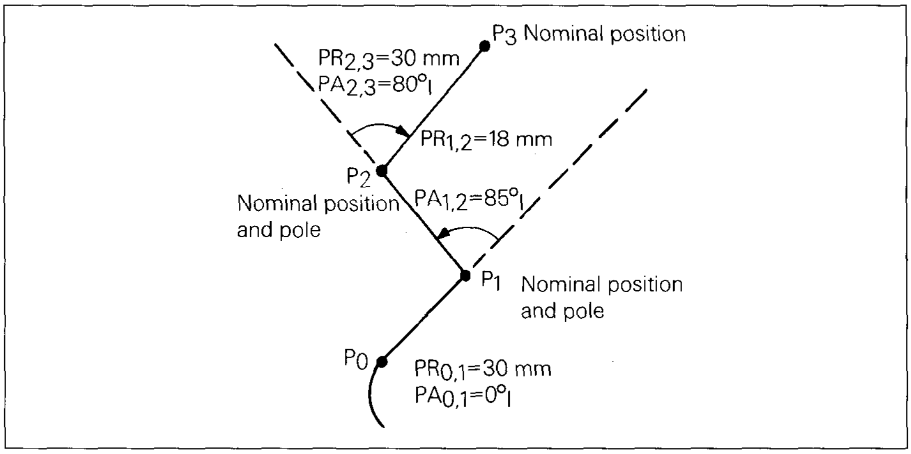
Entry of **2 co-ordinates**
.press [I] if reqd.
.press first axis key and enter numerical value then
.press second axis key and enter numerical value
.press [ENT] or [END] (see section M 3.2.4).

The entry of **3 co-ordinates** is performed similarly (3D-traverse programmed with [L])
.press [I] if reqd.
.press first axis key and enter numerical value
.press second axis key and enter numerical value
.press third axis key and enter numerical value
.press [ENT] or [END] (see section M 3.2.4).

M 3.2.2) Entry of positions in polar co-ordinates [P]

Nominal position values can also be programmed in polar co-ordinates (see section C 4). First of all, the **pole** (co-ordinate origin) must be defined. It can be defined in two ways:
.either the last nominal position value can be used as the pole
.or the pole is defined by means of Cartesian co-ordinates.

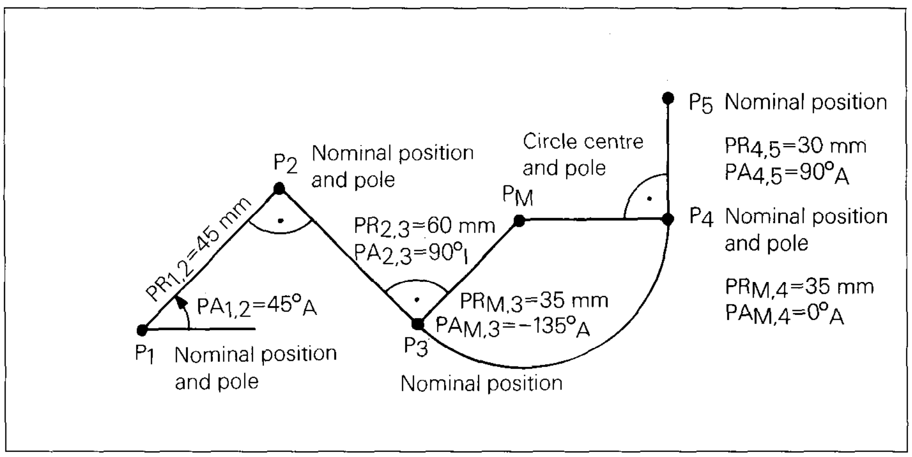
Examples:
The utilization of the last **nominal position** as a pole-value is mainly used for the programming of linear paths.



As an example, the series of straight lines as shown P_0, P_1, P_2, P_3 may be programmed by merely entering the radii and angles of direction.

With incremental programming, the polar co-ordinate angle is referenced to the direction last programmed.

A contour comprising straight sections and an arc



Press [CC] for programming of pole.

The dialogue question
COORDINATES?

is **answered** as follows:

.Press [I] for entry of Cartesian co-ordinates of pole if reqd.
.Press axis key and enter numerical value
.Press axis key and enter numerical value
.Press [ENT] or [END] -key;
.If the previous nominal position value is to be declared as a the pole, press [NO ENT].

If [NO ENT] or [END] is pressed after entry of the first co-ordinate, the second co-ordinate for the previous nominal position is used.

The programming of the fourth axis in a CC-block is only possible if the fourth axis is linear.

The pole definition allocates one program block:
either **CC X+10,000 Y+20,000** with polar co-ordinate programming
or **CC** when using the previous nominal position as the pole.

When determining a pole, the "Cartesian datum" is retained so that after entry of polar co-ordinate blocks, programming of Cartesian co-ordinates may be resumed.

When programming a positioning block in polar co-ordinates, the [P] -key must be pressed before initiating the dialogue with [L] or [C]. (The indicator lamp is then on)!

Dialogue initiation: press [L]

Dialogue question:
POLAR COORDINATES-RADIUS PR?

Response:

.press [I] if reqd.

.enter radius value "PR"

.press [ENT].

Next dialogue question:
POLAR COORDINATES-ANGLE PA?

Response:

.press [I] if reqd.

.enter polar angle "PA" in degrees

.press [ENT] or [END] (see section M 3.2.4)

Dialogue initiation with [C] :

Dialogue question:
POLAR COORDINATES-ANGLE PA?

Response as per linear interpolation.

When performing circular interpolation with polar co-ordinates, the radius of the circle end point need not be programmed. The control determines the radius automatically by using the circle starting point and centre position.

M 3.2.3) Complete positioning blocks

M 3.2.3.1) 2D-Linear interpolation and single axis traversing [L]

Dialogue initiation: press either [L] or [P] with polar co-ordinates and then [L]

Dialogue questions	Response
COORDINATES? or **POLAR COORDINATES-RADIUS PR?** and **POLAR COORDINATES-ANGLE PA?**	Enter co-ordinates as per section M 3.2.1 or M 3.2.2; press [ENT].
TOOL RADIUS COMP.: RL/RR/ NO COMP.?	Enter radius compensation if reqd. (see section M 3.1); .press [RR+] or [RL-] .press [ENT].
FEED RATE? F=	Enter feed rate (see section F 1); press [ENT].
AUXILIARY FUNCTION M?	Enter auxiliary function (see section F 2); press [ENT].

If dialogue questions are responded to by pressing [NO ENT], data entry is omitted – the next dialogue question is displayed.

If several M-functions are required in one block and have not been accomodated into previous blocks, these may be programmed as single positioning blocks containing only an M-function. The number of blocks corresponds to the required number of M-functions (press [NO ENT]-key for all preceding dialogue questions).

If **no** M-function is required within a block, press [NO ENT] in response to dialogue question for M-function.

Linear interpolation allocates one program block:

L X + 20,500 I Y + 49,800
RL F 100 M

or

LP PR + 80,000 PA + 45,000
RR F 100 M

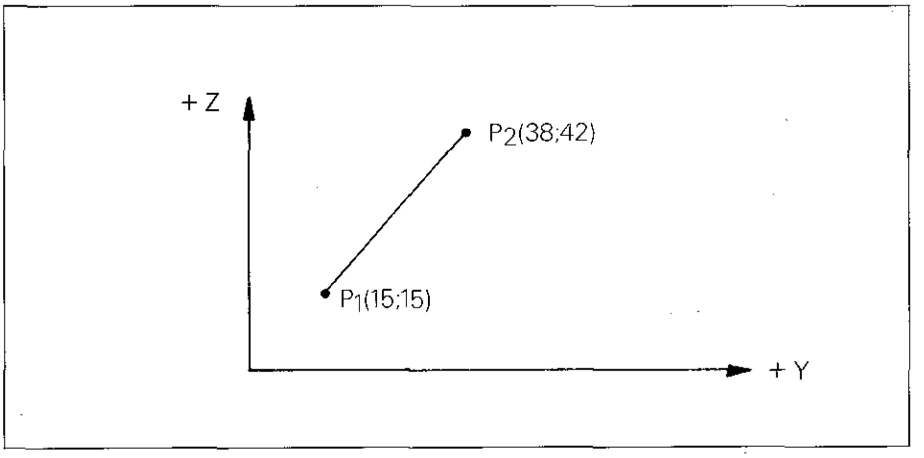
Examples:

2D-Linear interpolation in Cartesian co-ordinates

The tool is in the Position P_1.
It is to travel to the target position P_2 (co-ordinates $Y_2 = 38$ and $Z_2 = 42$) in a straight path.

Program blocks:

```
1    L Y + 15.000 Z + 15.000
               R0 F 100 M
2    L Y + 38.000 Z + 42.000
               R0 F 100 M
```

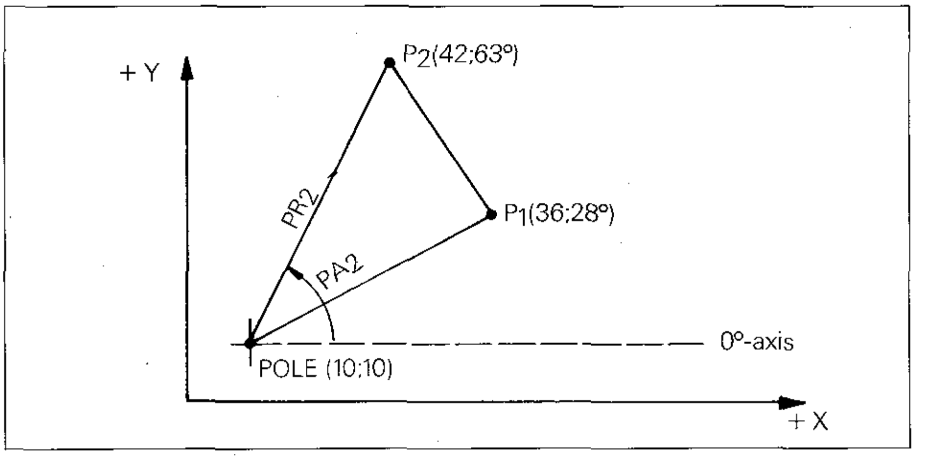


2D-Linear interpolation in polar co-ordinates

The machine is stationary at point P_1.
The nominal position P_2 *is defined by* the radius $PR_2 = 52$ mm and the polar angle $PA_2 = 63°$. The machine will traverse in a straight path from point P_1 to point P_2.

Program blocks:

```
1 CC X  + 10.000 Y  + 10.000
2 LP PR + 36.000 PA + 28.000
                 R0 F 100 M
3 LP PR + 42.000 PA + 63.000
                 R0 F 100 M
```



2D-Linear interpolation with the fourth axis

The fourth axis may be used in linear interpolation with any one of the mains axes [X], [Y], [Z].

Dialogue initiation: (polar co-ordinate entry is not possible)

When responding to dialogue questions, the following should be noted:

.When using the fourth axis as a rotary table axis:
Nominal position value entry in (°)
and
Feed rate entry in (°/min.).

.Radius compensation is only considered when the fourth axis is linear.

If the function M 94 is programmed within a positioning block for the fourth axis (fourth axis operation rotary), the position display for the fourth axis is automatically reduced to a corresponding angle value below 360°.

Linear interpolation with the fourth axis allocates one program block:

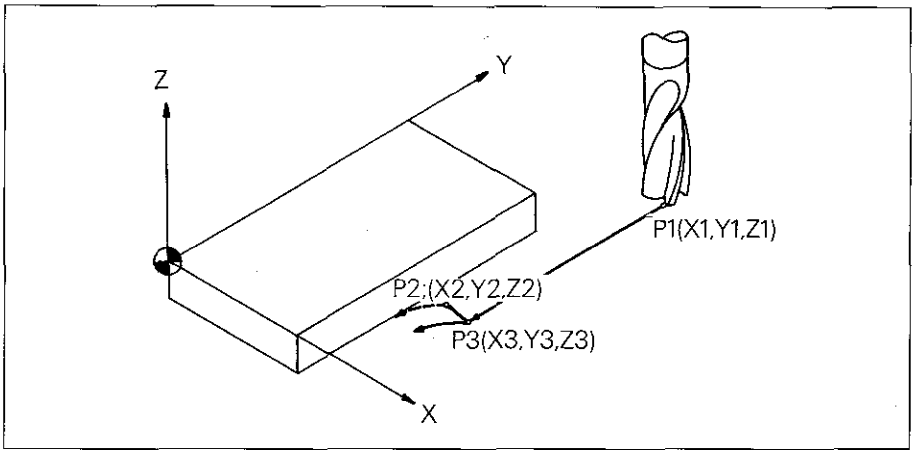
L Z + 50,000 C + 720,000
R0 F 20 M

The feed rate is given mainly in mm/min. When milling in connection with a rotary table, the feed rate must be converted to °/min (refer to "Programming examples TNC 150" which is available upon request).

M 3.2.3.2) 3D-Linear interpolation

TNC 150 enables simultaneous positioning in three axes with complete tool radius and length compensation.

Example:
The tool is located in position P1. The nominal position P2 has the co-ordinates X2, Y2, Z2. The control calculates the compensated co-ordinates X3, Y3, Z3 and traverses to point P3 in a 3D-path.

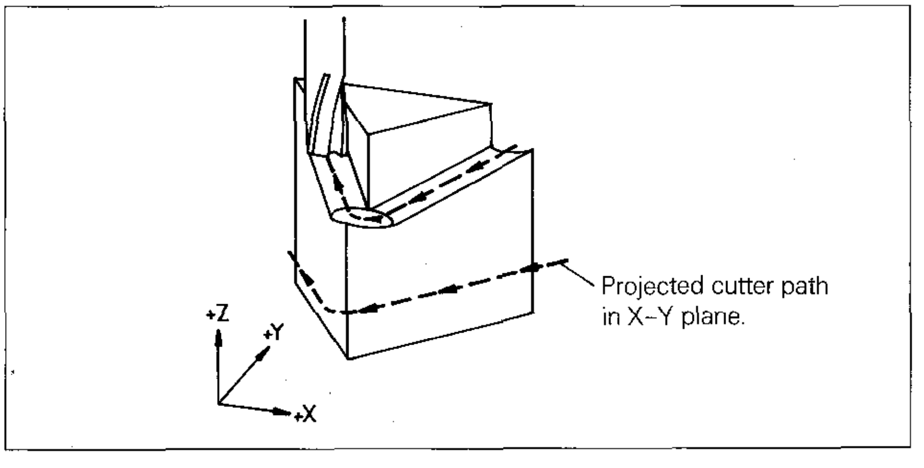


Dialogue initiation: press [L]

Dialogue question	Response
COORDINATES?	Enter first second and third co-ordinate of nominal position in Cartesian (see section M 3.2.1) and press [ENT].
TOOL RADIUS COMP.: RL/RR/ NO COMP.?	Enter radius compensation if reqd. (see section M 3.1); .press [R+] or [R-] .press [ENT].
FEED RATE? F =	Enter feed rate (see section F 1); press [ENT].
AUXILIARY FUNCTION M?	Enter auxiliary function (see section F 2); press [ENT].

If dialogue questions are answered with [NO ENT], data entry is omitted – the next dialogue question is displayed.

Insertion of radii and compensating arcs
Radii and compensating arcs are inserted such, that the projection of the cutter path is perpendicular to the tool axis in 2D.



3D-Linear interpolation allocates one program block:

L X + 63,000 Y + 49,000
Z + 39,000 RL F 100 M

M 3.2.3.3) Definition of circle centre [CC]

The [CC]-key is used for determining the circle centre point. The procedure corresponds to the pole routine for polar co-ordinates.

Dialogue initiation: press [CC]

Dialogue question	Response
COORDINATES?	Either enter co-ordinates of circle centre (see section M 3.2.1) and press [ENT], or press [NO ENT] if the previous nominal value is to be used as circle centre.

If [NO ENT] or [END] is pressed for the first co-ordinate, the previous nominal position value is re-used for the second co-ordinate.

The circle centre definition allocates one program block:

CC X + 15,000 Y + 23,000

If the previous nominal position value is used as the circle centre, the following block is displayed:

CC

Programming of the fourth axis within a CC-block is only possible if the fourth axis is linear.

M 3.2.3.4) Circular path programming [C]

Define circle centre (see section M 3.2.3.3)

Dialogue initiation: either [C] or [P], with polar co-ordinates and then [C]

Dialogue question	Response
COORDINATES? *or* **POLAR COORDINATES ANGLE PA?**	Enter co-ordinates (see section M 3.2.1 or M 3.2.2); press [ENT].
ROTATION CLOCKWISE: DR–?	By pressing [+/–]-key, enter .rotation CW (clockwise): DR– (negative direction of rotation) or .rotation CCW (anti-clockwise): DR+; (positive direction of rotation) .press [ENT].
TOOL RADIUS COMP.: RL/RR/ NO COMP.?	Enter tool radius compensation (see section M 3.1); .press [R+] or [R–] .press [ENT].
FEED RATE? F =	Enter feed rate (see section F 1); press [ENT].
AUXILIARY FUNCTION M?	Enter auxiliary function (see section F 2); press [ENT].

If dialogue questions are answered with [NO ENT], data entry is omitted; the next dialogue question is displayed.

Circular interpolation allocates one program block:

C X + 20,000 Y + 50,000
DR– RR F 80 M

or

CP PA + 180,000
DR + RL F 40 M

Programming of the fourth axis within a circular interpolation-block is only possible if the fourth axis is linear.

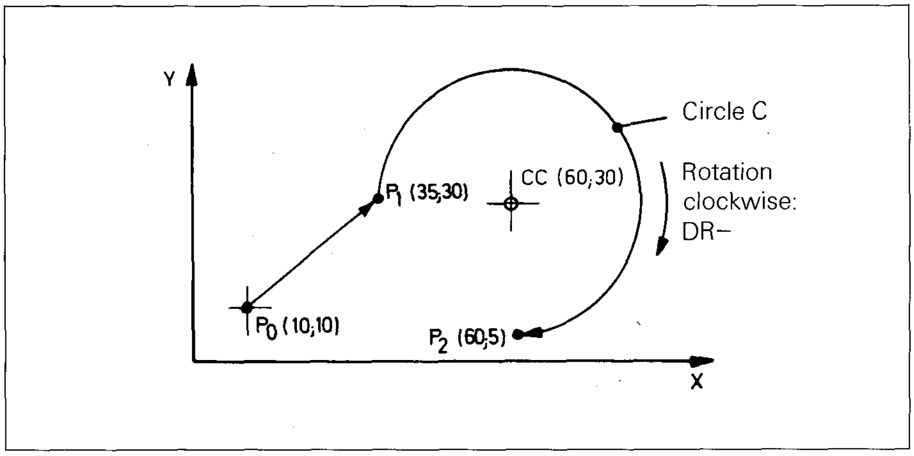
Examples:

Circular path programming in Cartesian co-ordinates

Point P_1 is defined in a positioning block. Then the **circle centre CC** (see section M 3.2.3.3) and the **end position of the arc P_2** are to be programmed.

Program blocks:

```
1 L     X+10.000    Y+10.000
                    R0  F100    M
2 L     X+35.000    Y+30.000
                    R0  F100    M
3 CC    X+60.000    Y+30.000
4 C     X+60.000    Y+5.000
                 DR-R0  F 100   M
```



Circular path programming in polar co-ordinates

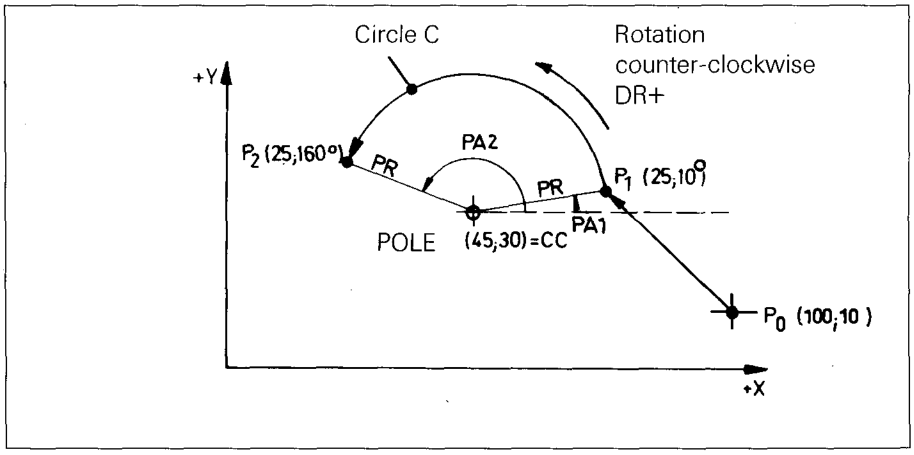
First, the centre point = Pole is entered in Cartesian co-ordinates (see section M 3.2.3.3). The points P_1 and P_2 are then programmed with radius PR (25) and angles PA 1 (10°) and PA 2 (160°).

Point P_2 may also be programmed incrementally:

PA_2 = + 150° (incremental).

Program blocks:

```
1 L     X+100.000   Y+10.000
                    R0  F100    M
2 CC    X+45.000    Y+30.000
3 LP    PR+25.000     PA+10.000
                    R0  F100    M
4 CP    PR+25.000     PA+160.000
                 DR-R0  F 100   M
```



A corrected contour cannot be commenced within a circular path.

M 3.2.3.5) Helical interpolation

Helical interpolation is mainly used for the manufacture of large diameter internal and external screw threads.

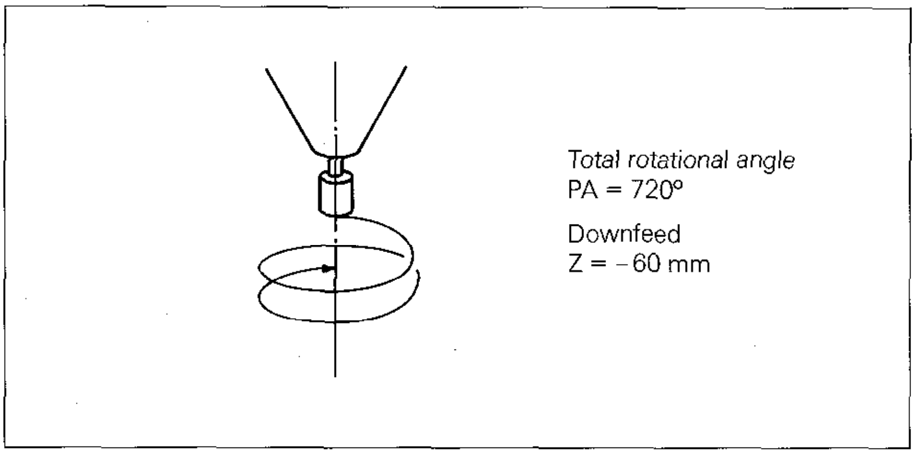
With this type of interpolation, **circular motion** is performed in the **working plane** (e.g. X-Y plane) while simultaneous **linear motion** of the **tool axis** takes place.

The helix is programmed in **polar co-ordinates** using the [C]-key and entering the total angle of revolution and an additional up or downfeed co-ordinate.

Please note:
The circle centre should be already established!

Example:

CC X + 0,000 Y + 0,000
LP PR + 50,000 PA + 0,000
RR F 120 M
CP IPA 720,000 IZ – 60,000
DR – RF M



Dialogue initiation: Press [P] and then [C]!

Dialogue question	Response
POLAR COORDINATES ANGLE PA?	.Enter polar co-ordinate angle. With entry values exceeding 360°, the polar co-ordinate angle must be entered incrementally. .Press axis key for linear motion axis
COORDINATES?	Enter co-ordinates for linear motion (in incremental or absolute dimensions)
ROTATION CLOCKWISE: DR–?	By pressing [+/–] -key, enter .rotation cw (clockwise): DR– (negative direction of rotation) or rotation ccw (anti-clockwise): DR+ (positive direction of rotation) .press [ENT]
TOOL RADIUS COMP.: RL/RR/ NO COMP.?	Enter tool radius compensation: **.press** [RR] **or** [RL] **.press** [ENT]
FEED RATE? F =	Enter feed rate of path, press [ENT]
AUXILIARY FUNCTION M?	Enter auxiliary function if reqd. press [ENT]

If dialogue questions are answered with [NO ENT], data entry is omitted; the next dialogue question is displayed.

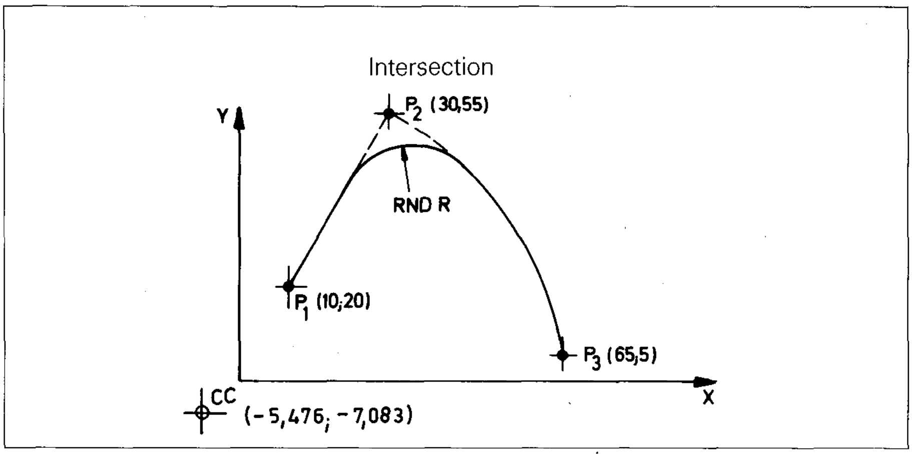
Helical interpolation allocates one program block.

230 CP IPA + 720,000 IZ – 60,000
DR– R0 F 100 M

Programming of the fourth within a helical interpolation block is only possible if the fourth axis is linear.

M 3.2.3.6) Rounding of corners (Arcs with tangential transitions) [RND]

Another way of programming a circular path is by insertion of tangential arcs with radius R into corners or into a path of contours. The insertion of "rounding off" radii is possible on all corners which are formed from straight/straight, straight/arc or arc/arc contours.



Example: plane X, Y

The corner which is formed by line P_1, P_2 and arc P_2, P_3 is to be "rounded off" with a radius R having tangential transitions.

Programming sequence:
.the contour P_1 P_2 (with tool offset RR or RL)
.the rounding off block with rounding off radius R
.the contour P_2 P_3 (with tool offset RR or RL)

The control **only** requires the **rounding off-radius** (all further data is calculated by the TNC 150 itself).

Dialogue initiation: press [RND]

Dialogue question	Response
ROUNDING OFF RADIUS R?	Enter numerical value or parameter (see section M 5); press [ENT].

.A rounding off block must be preceded or followed by a positioning block which contains both co-ordinates of the interpolation plane.

Dialogue question:
ROUNDING OFF RADIUS R?

Entry range: 0 – 19999.999 mm

"Rounding off" allocates one program block:

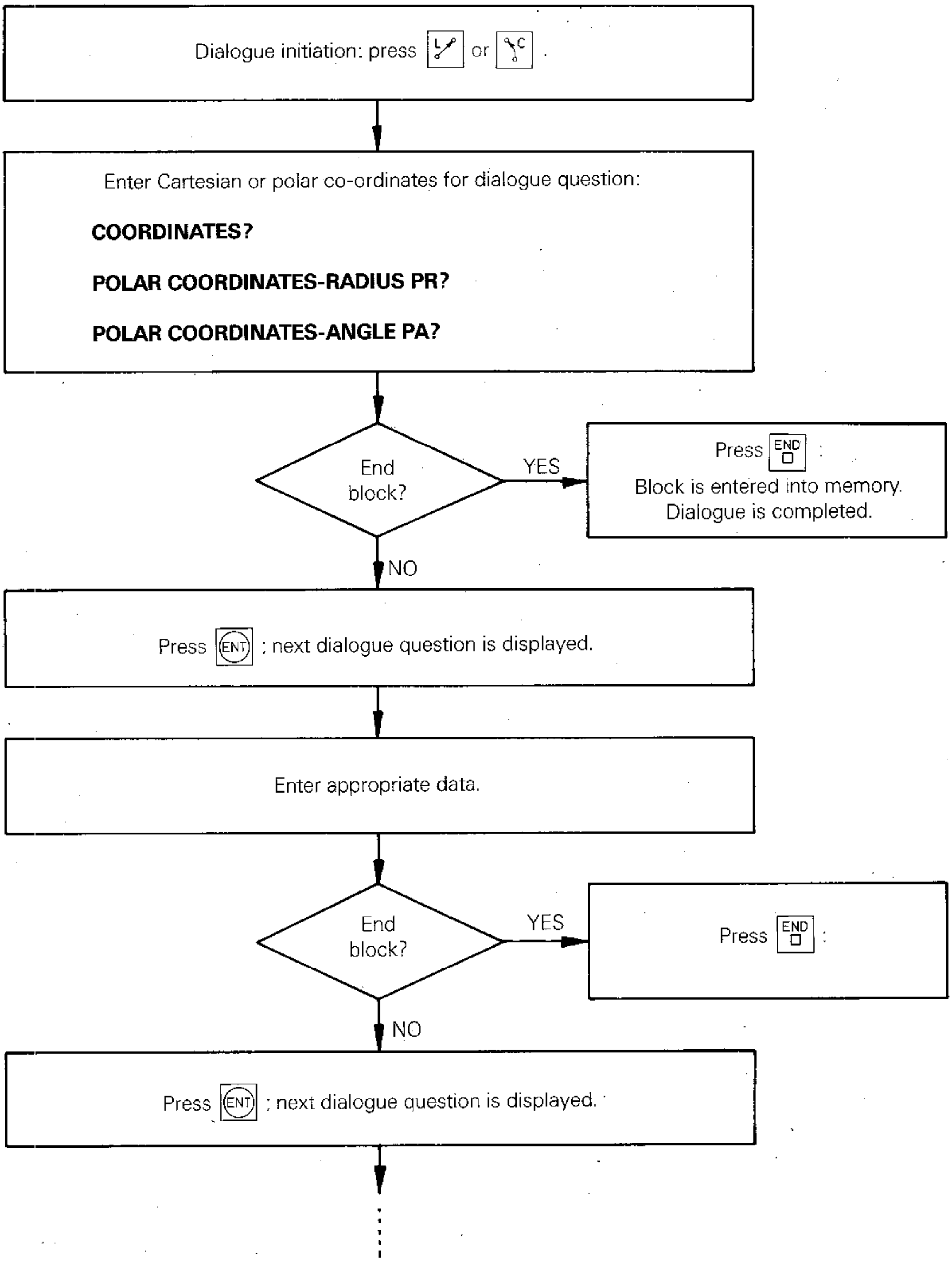
RND R 10,000

Program for previous example:

```
1 TOOL DEF 1   L+100.000
               R+10.000
2 TOOL CALL 1 Z
                S 1000
3 L     X+10.000 Y+20.000
               RL F100      M
4 CC    X-5.476  Y-5.000
5 L     X+30.000 Y+55.000
               RL F100      M
6 RND   R+10.000
7 C     X+65.000 Y+5.000
           DR- RL F100      M
```

M 3.2.4) Curtailed positioning block [END]

Within certain program sequences, it is often the case that the tool compensation (RR/RL/R0), feed rate and auxiliary function (M) remain unchanged for a series of blocks. With TNC 150, such data does not have to be re-entered for every individual block. This means that the block is ended immediately after entry of the nominal position co-ordinates. During program run, the tool radius compensation, feed rate and auxiliary function correspond to the data last entered.



With the [END] -key, the block can be ended after every entry.

The first block of a machining program must contain the required type of radius compensation and the feed rate otherwise the following error is displayed:
UNDEFINED PROGRAM START

M 3.2.5) Constant contouring speed at corners: M 90

The TNC 150 control checks whether the program contour can be traversed at the programmed feed rate. If there is a danger that the contour cannot be maintained (with external corners and small radii), the feed rate is automatically reduced. With internal corners, axis-standstill will always take place.

*If feed rate reduction is undesirable, a constant contouring speed can be impelled by programming the auxiliary function **M 90.** This can however, lead to small contour blemishes on external and internal corners.

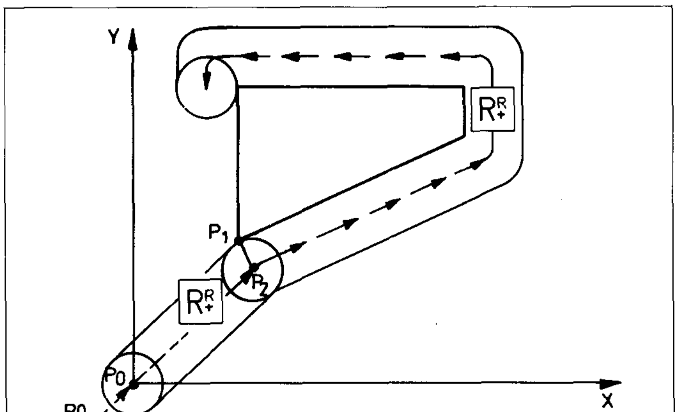
This M-function is only effective for operation with trailing axes and depends on the stored machine parameters. Please check with your machine tool manufacturer if your control operates in this mode.

M 3.2.6) Approach to – and departure from a contour

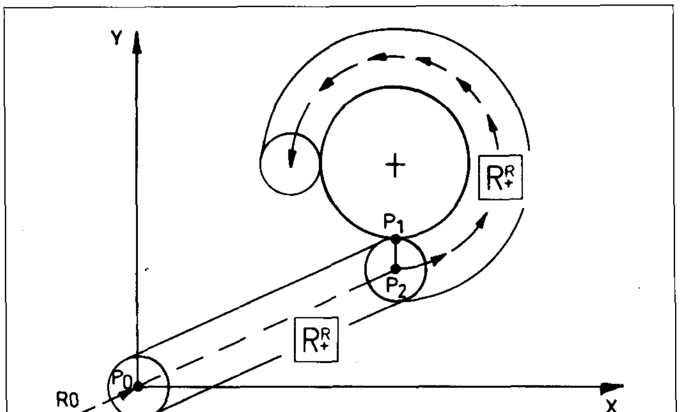
M 3.2.6.1) Contour approach and departure on a straight path

Approach to – and departure from a contour can take place in two ways:

Case 1:
The starting position P_0 is approached without radius compensation (R0). The following positioning block to point P_1 is programmed with radius offset RR or RL.
When approaching the contour the control automatically calculates the auxiliary point P_2 away from P_1. Point P_2 is calculated by constructing a perpendicular at the beginning of the contour. The distance between P_2 and P_1 corresponds to the radius programmed in the tool definition.

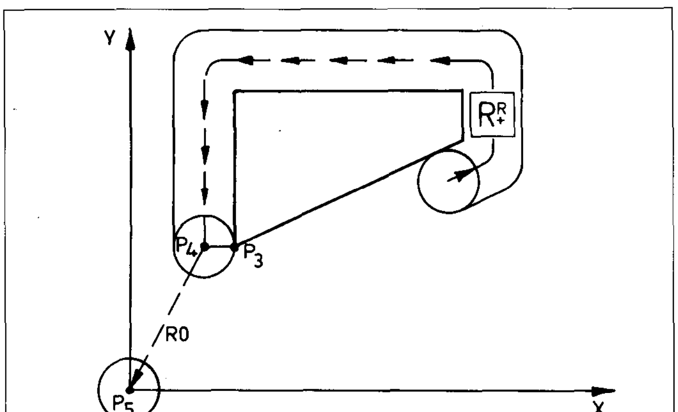


starting position P_0: without compensation

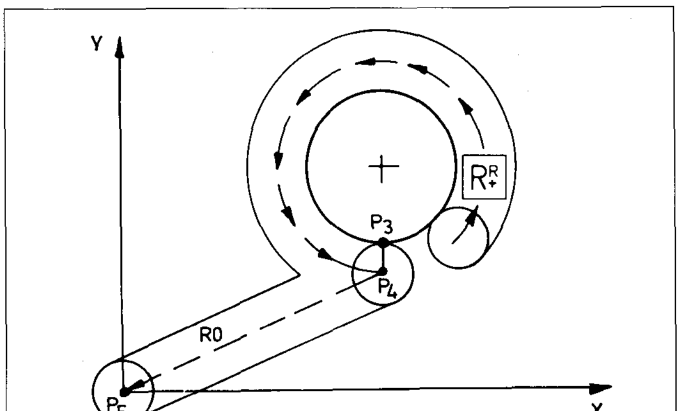


starting position P_0: without compensation

When leaving the contour by **approaching the end position P_5 without compensation (R0),** the control automatically calculates the end point P_4 of the contour by constructing a perpendicular to the final point of the contour P_3.

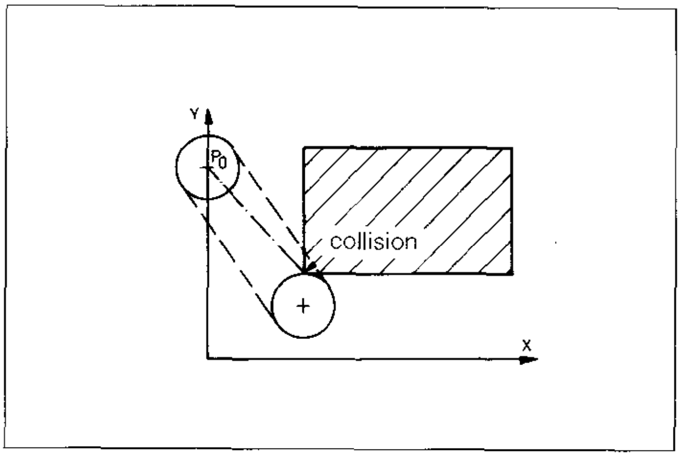


End position P_5: without compensation

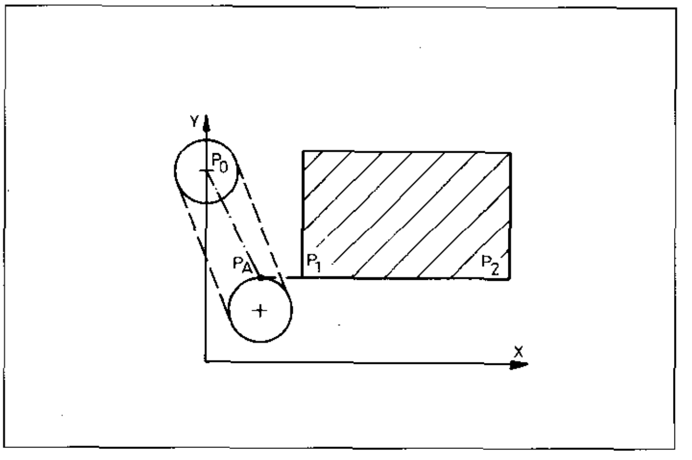


End position P_5: without compensation

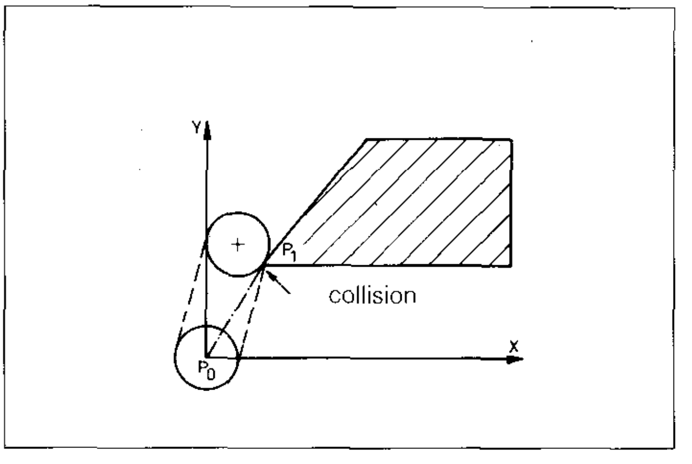
When **approaching a contour,** e. g. from a tool change position P_0, a collision with the workpiece must be prevented. This is also applicable to contour programming with contour offset.



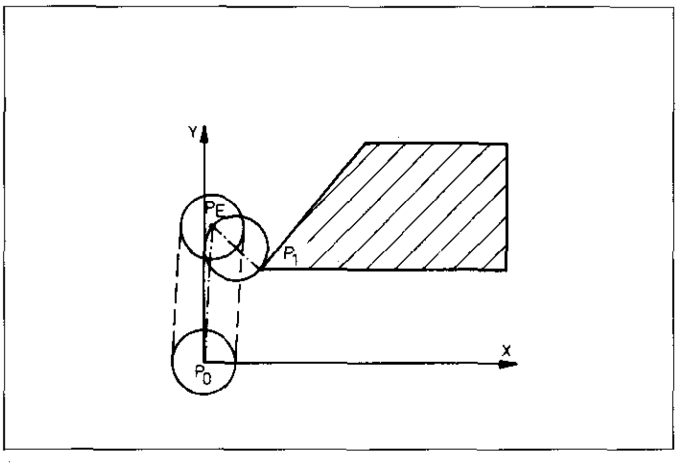
An auxiliary point P_A which lies on the extension of the line P_1 – P_2 must therefore be programmed. The distance of point P_A to the workpiece must be the tool radius R plus a certain safety clearance of e. g. 5 mm. The auxiliary point P_A is approached with contour offset.



When **leaving a contour,** a collision with the workpiece must also be prevented. If, after reaching point P_1, the tool change position P_0 is to be approached, a collision would certainly take place.



Therefore, an auxiliary point P_E must also be programmed at a safe distance from the workpiece. This point, however, is approached without contour offset. This also applies for the return traverse to the tool change position P_0.

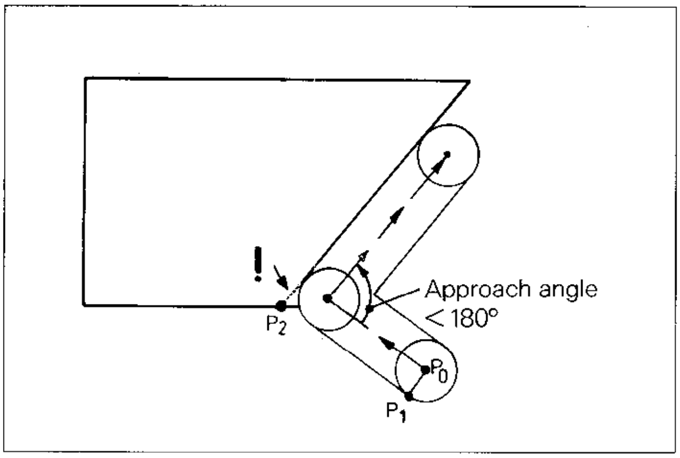


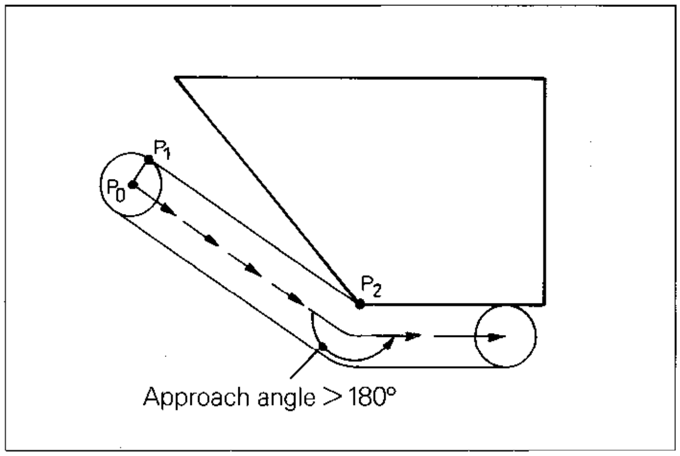
Case 2:

The machining program commences with the positioning block to point P_2 – with offset RR or RL; the control already considers point P_0 as being an auxiliary point for P_1 and positions to point P_2 as if it was a point within the contour; i. e.

.if the approach angle to the contour is less than 180°, the bisection of the angle is approached,
.if the approach angle to the contour is greater than 180°, a transitional arc is inserted.

It is not possible to make a corrected program start within a circular interpolation block.

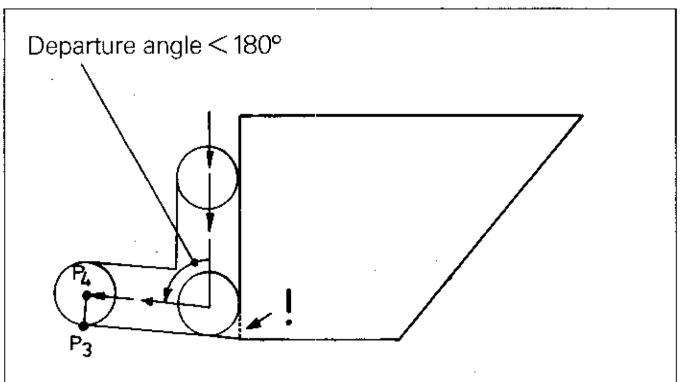




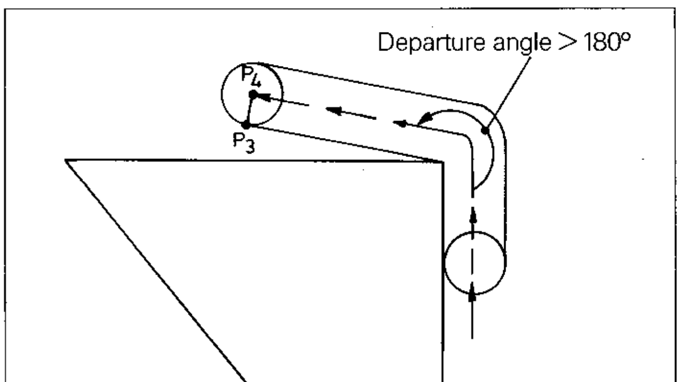
The program block for **leaving the contour also contains radius offset RR or RL.** Contour correction is terminated in this case with

.the auxiliary function **M 98** or
.a successive empty block or
.a TOOL CALL.

The control calculates the auxiliary end point P_4 by constructing a perpendicular to the final point of the contour P_3. The distance between points P_3 and P_4 corresponds to the tool radius.



End position P_3: compensated with RR



End position P_3: compensated with RR

If the approach angle is less than 180°, the workpiece will not be completely machined (see above sketch!)

Change of approach behaviour at beginning of contour: M 95, M 96

Instead of the normal approach behaviour, contour approach can be altered by the auxiliary functions M 95 or M 96 as follows:

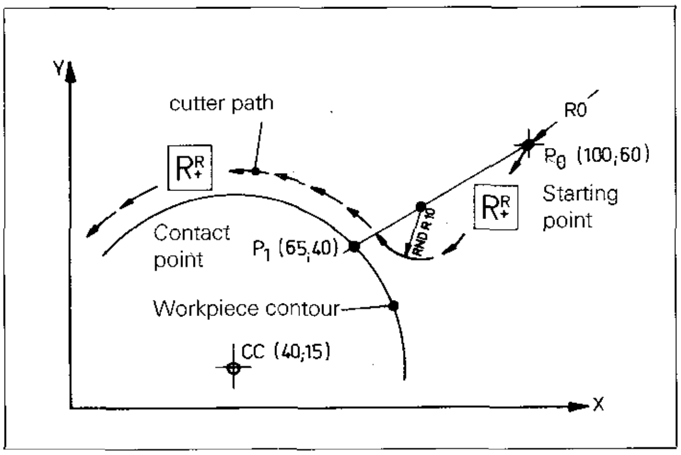
If normal approach corresponds to the first case, the second case can be impelled by programming **M 96.**

If normal approach corresponds to the second case, the first case can be impelled by programming **M 95.**

M 3.2.6.2) Tangential contour approach and departure

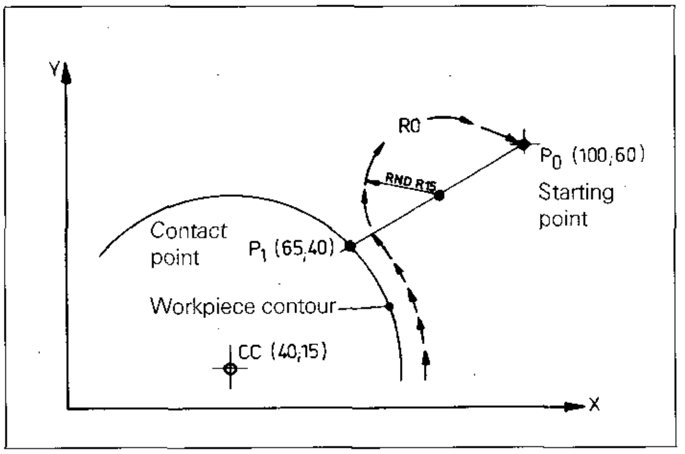
The [RND] -key serves in programming the smooth tangential approach to a contour and rounding of corners (see section M 3.2.3.5).
An arc or a straight line can be approached by means of a smooth tangential arc to a desired point of contact and at a determined contouring speed:

Approaching contour



Firstly, the **starting point P_0** is entered in a previous block with **tool offset R0.** The **next positioning block** – for the contact point P_1 – must contain a contour offset – **RR or RL** – (due to the transition between R0 to RR or RL, the control automatically recognizes that a contour is to have a smooth approach). Lastly, a rounding off-block is to be programmed with the [RND] -key.

Learing contour



The departure from the contour is programmed similarly: If the contour offset changes from **RR or RL to R0** the control automatically recognizes that the tool must leave the contour on the programmed auxiliary arc.

Program for the previous example:

Procedure	Program block display
Tool definition and tool call	1 TOOL DEF 1 L + 100.000 R + 10.000
	2 TOOL CALL 1 Z S 1000
Starting point is positioned	3 L X 100.000 Y + 60.000 R0 F 9999 M 03
Contact point and contouring speed are specified	4 L X 65.000 Y + 40.000 RR F 50 M
Rounding off-radius for smooth contour approach	5 RND R 10
Circle centre for workpiece contour	6 CC X 40.000 Y15.000
Programming of workpiece contour	7 C X 65.000 Y 40.000 DR+ RR F 50 M
Rounding off-radius for leaving contour	8 RND R 15
Return to starting point	9 L X 100.000 Y 60.000 R0 F50 M 05

A rounding off-block must be preceded or followed by a positioning block which contains both co-ordinates of the interpolation plane.

M 4) Programmed stop [STOP]

Dialogue initiation: press [STOP] .

Dialogue question	Response
AUXILIARY FUNCTION M?	Enter required M-function; press [ENT] or [NO ENT] if no M-function is required.

A programmed stop [STOP] allocates one program block:

STOP

M

A programmed STOP via the [STOP]-key does not activate a "spindle stop" and "coolant off" as per auxiliary function MOD.

M 5) Parameter programming

With TNC 150, parameters (Q 0 to Q 99) may be programmed instead of co-ordinate and feed rate values. These parameters are then assigned via Q DEF to certain values or functions (mathematical or logical relationships).

The following entry values can be replaced by parameters:

1) with positioning blocks
   X-value, Y-value, Z-value, F-value, IV-value, PR-value, PA-value

2) with CC-blocks
   X-value, Y-value, Z-value, IV-value

3) with TOOL-DEF-blocks
   Tool raidus R, Tool length L
   (with a tool call, the current parameter value is effective)

4) with RND-blocks
   Rounding off radius R

5) with canned cycles
   Set-up clearance, Pecking depth, Total depth, Dwell time, Length and **width** of slots and rectangular pockets,
   Radius of circular pockets;
   Feed rates,
   Co-ordinate system rotation

Programs which contain parameter programming have slow machining speeds in most cases. Especially with the machining of contours which are described by means of mathematical formulae, the TNC-calculation time for co-ordinates has a great effect.
Contours derived from mathematical formulae are usually approximated by the use of polygons. This can also reduce the machining speed-especially with internal contours.

Parameters are entered with the [Q] -key in conjunction with a number 0 – 99.

The assignment of a certain value or function is performed with the [Q DEF] -key.

Parameter programming caters for:

.parametric programs
.contours described by mathematical formulae
and
.jump to label after parameter comparison.

M 5.1) Parameter entry [Q]

If the TNC 150 dialogue requires the entry of co-ordinates or feed rate values, parameters may be entered instead if numerical values.

Dialogue demands entry of numerical value.

↓

Press [Q] and enter required parameter No. (0 – 99)
press [+/-] if required and
enter into memory with [ENT].

↓

Parameter is programmed.

The display shows the following block:

L X Q 1 Y Q 2
RR F 100 M

Explanation: Co-ordinates X and Y have been programmed with parameters Q 1 and Q 2; the numerical values are defined separately by the parameter definition "Q DEF".

M 5.2) Parameter Definition [Q DEF]

The Parameter definition is used for assigning the parameters Q 0 to Q 99 with numerical values or functional relationships. A parameter definition may be located anywhere within the machining program; it must, however, always be located before parameter call-up.

The parameter definition is selected via [Q DEF]. The required parameter function can be selected by "paging" through the function library with the [↓] and [↑]-keys (repetitive pressing).

Programmable functions:
(FN = Abbreviation for "function")

FN 0:	**ASSIGN**
FN 1:	**ADDITION**
FN 2:	**SUBTRACTION**
FN 3:	**MULTIPLICATION**
FN 4:	**DIVISION**
FN 5:	**SQUARE ROOT**
FN 6:	**SINE**
FN 7:	**COSINE**
FN 8:	**ROOT SUM OF SQUARES**
FN 9:	**IF EQUAL, JUMP**
FN 10:	**IF UNEQUAL, JUMP**
FN 11:	**IF GREATER THAN, JUMP**
FN 12:	**IF LESS THAN, JUMP**

M 5.2.1) FN 0: Assign

The parameter assign function is used for assigning either a numerical value or another parameter to a certain parameter.

Dialogue initiation: press [Q DEF].

Dialogue question	Response
FNO: ASSIGN	Enter function by pressing [ENT].
PARAMETER NUMBER FOR RESULT?	Key-in parameter number: 0 – 99; press [ENT].
FIRST VALUE/PARAMETER?	Enter numerical value or parameter; press [ENT].

The display shows e. g. the following block:

FN 0: Q 12 = + 20.000 *Explanation:* A value of 20.000 has been assigned to parameter Q 12.

The "=" sign signifies an assignment!

M 5.2.2) FN 1: Addition

With parameter addition, the sum of two numerical values or parameters is assigned to a certain parameter.

Dialogue initiation: press [Q DEF] and then [↓].

Dialogue question	Response
FN 1: ADDITION	Enter function by pressing [ENT].
PARAMETER NUMBER FOR RESULT?	Key-in parameter number: 0 – 99; press [ENT].
FIRST VALUE/PARAMETER?	Key-in first value or parameter; press [ENT].
SECOND VALUE/PARAMETER?	Key-in second value or parameter; press [ENT].

The display shows e. g. the following block:

FN 1: Q 1 = + 20.000
+ + Q 2

Explanation: The sum of 20.000 + parameter Q 2 is assigned to parameter Q 1. The numerical value for Q 2 is located in another parameter definition.

M 5.2.3) FN 2: Subtraction

With parameter subtraction, the difference between two numerical values or two parameters is assigned to a certain parameter.

Dialogue initiation: press [Q DEF] and then [↓] until the function
FN 2: SUBTRACTION is displayed.

Programming is similar to the parameter addition routine (see section M 5.2.2).

The display shows e. g. the following block:

FN 2: Q 5 = Q 3 **− + 20.000**	*Explanation:* The difference between parameter Q 3 – 20.000 is assigned to parameter Q 5. The numerical value for Q 3 can be found in another parameter definition.

M 5.2.4) FN 3: Multiplication

With parameter multiplication, the product of two numerical values or parameters is a assigned to a certain parameter.

Dialogue initiation: press [Q DEF] and then [↓] until the function
FN 3: MULTIPLICATION is display.

Programming is similar to the parameter addition routine (see section M 5.2.2).

The display shows e. g. the following block:

FN 3: Q 21 = Q 2 *** + 5.000**	*Explanation:* The product of Q 2 and 5.000 is assigned to parameter Q 21. The numerical value for Q 2 can be found in another parameter definition.

M 5.2.5) FN 4: Division

With parameter division, the quotient of two numerical values or parameters is assigned to a certain parameter.

Dialogue initiation: press [Q DEF] and then [↓] until the function
FN 4: DIVISION is displayed.

Programming is similar to the parameter addition routine (see section M 5.2.2).

The display shows e. g. the following block:

FN 4: Q 63 = + 30.000 **DIV + Q 25**	*Explanation:* The result of the division calculation 30.000 : Q 25 is assigned to the parameter Q 63. The numerical value for Q 25 can be found in another parameter definition.

M 5.2.6) FN 5: Square root

With the square root function, the square root of a numerical value or a parameter is assigned to a certain parameter.

Dialogue initiation: press [Q DEF] and then [↓] until the function
FN 5: SQUARE ROOT is displayed.

Programming is similar to the parameter assignment routine
(see section M 5.2.1).

The display shows e.g. the following block:

FN 5: Q 6 = SQRT + 20.000

or

FN 5: Q 6 = SQRT + Q 74

Explanation: The square root of 20.000 is assigned to parameter Q 6
or
the square root of parameter Q 74 is assigned to parameter Q 6.
The numerical value for Q 74 can be found in another parameter definition.

SQRT is an abbreviation for "square root".

M 5.2.7) FN 6: Sine

With the sine function, the sine of an angle (programmed in degrees) is assigned to a certain parameter.

Dialogue initiation: press [Q DEF] and then [↓] or [↑] until the function
FN 6: SINE is displayed.

Programming is similar to the parameter assignment routine
(see section M 5.2.1).

The display shows e.g. the following block:

FN 6: Q 10 = SIN + 90.000

or

FN 6: Q 69 = SIN + Q 86

Explanation: The sine of 90° is assigned to parameter Q 10
or
the sine of parameter Q 86 is assigned to parameter Q 69.
The numerical value for Q 86 can be found in another parameter definition.

M 5.2.8) FN 7: Cosine

With the cosine function, the cosine of an angle (programmed in degrees) is assigned to a certain parameter.

Dialogue initiation: press [Q DEF] and then [↑] until the function **FN 7: COSINE** is displayed.

Programming is similar to the parameter assignment routine (see section M 5.2.1).

The display shows e. g. the following block:

FN 7: Q 12 = COS + 45.000

or

FN 7: Q 99 = COS + Q 11

Explanation: The cosine of 45° is assigned to parameter Q 12
or
the cosine of Q 11 is assigned to parameter Q 99.
The numerical value for Q 11 can be found in another parameter definition.

M 5.2.9) FN 8: Root of sum of squares

With the function "root of sum of squares" the square root of the sum of two squares is assigned to a certain parameter.

Dialogue initiation: press [Q DEF] and then [↑] until the function **FN 8: ROOT SUM OF SQUARES** is displayed.

Programming is similar to the parameter addition routine (see section M 5.2.2).

The display shows e. g. the following block:

FN 8: Q 20 = + 30.000
LEN + Q 45

Explanation: Parameter Q 20 is assigned to the following formula:

$$Q\,20 = \sqrt{30^2 + Q\,45^2}$$

The numerical value for Q 45 can be found in another parameter definition.

LEN is the abbreviation for "length".

M 5.2.10) FN 9: If equal, jump

This function activates a jump to a program mark when the parameter is equal to a certain numerical value.

Dialogue initiation: press [Q DEF] and then [↑] until the function
FN 9: IF EQUAL, JUMP is displayed.

Dialogue question	Response
FN 9: IF EQUAL, JUMP	Enter function by pressing [ENT].
FIRST VALUE?	Key-in first numerical value or parameter; press [ENT].
SECOND VALUE?	Key-in second numerical value or parameter; press [ENT].
LABEL NUMBER?	Key-in label number; press [ENT].

The display shows e. g. the following block:

FN 9: IF + Q 2
EQU + 20.000 GOTO LBL 30

Explanation: If parameter Q 2 is equal to the numerical value 20.000, a jump takes place to LBL 30.

"EQU" is an abbreviation for "equal".

M 5.2.11) FN 10: If unequal, jump

This function activates a jump to a program mark when the parameter is unequal to a certain numerical value.

Dialogue initiation: press [Q DEF] and then [↑] until the function
FN 10: IF UNEQUAL, JUMP is displayed.

Programming is similar to the function FN 9 (see section M 5.2.10)

The display shows e. g. the following block:

FN 10: IF + Q 3
NE + 10.000 GOTO LBL 2

Explanation: If parameter Q 3 is different to 10.000 a jump takes place to LBL 2.

"NE" is an abbreviation for "not equal"

M 5.2.12) FN 11: If greater than, jump

This function activates a jump to a program mark when the parameter exceeds a certain numerical value.

Dialogue initiation: press [Q DEF] and then [↑] until the function

FN 11: IF GREATER THAN, JUMP is displayed.

Programming is similar to function FN 9
(see section M 5.2.10).

The display shows e. g. the following block:

FN 11: IF + Q 3
GT + 30.000 GOTO LBL 5

Explanation: If parameter Q 3 is greater than 30.000, a jump takes place to LBL 5 during program run.

"GT" Abbreviation for "greater than"

M 5.2.13) FN 12: If less than, jump

This function activates a jump to a label number when the parameter is less than a certain numerical value.

Dialogue initiation: press [Q DEF] and then [↑]. The function

FN 12: IF LESS THAN, JUMP is displayed.

Programming is similar to function FN 9
(see section M 5.2.10).

The display shows e. g. the following block:

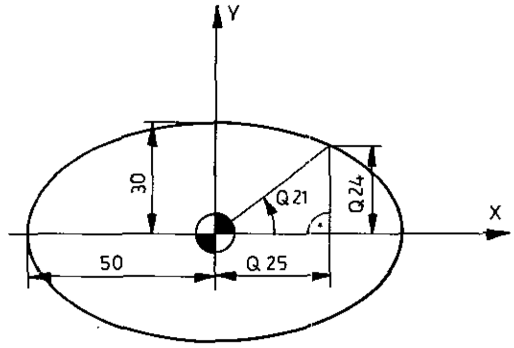
FN 12: IF + Q 6
LT Q 5 GOTO LBL 3

Explanation: If parameter Q 6 is smaller than Q 5, a jump takes place to LBL 3 during program run.

"LT" Abbreviation for "less than"

M 5.3) Example of parameter programming

Ellipse



Procedure	**Program block display**
Traverse to tool-change position	1 TOOL CALL 0 Z S 0,000 2 L Z+20,000 R0 F15999 M 3 L X+70,000 Y+70,000 R0 F15999 M
Tool definition 1, coarse-fine mill (4 flutes) Ø 20 mm programmed stop and tool call 1	4 TOOL CALL 1 L + 0,000 R + 10,000 5 STOP M 6 TOOL CALL 1 Z S 250,000
Positioning blocks to starting position	7 L Z–15,000 R F M 8 L Y+ 0,000 R F M
Parameter definition Q20 = angular pitch Q21 = initial angle Q22 = Y-semi-axis Q23 = Y-semi-axis	9 FN 0: Q20 = + 2,000 10 FN 0: Q21 = + 0,000 11 FN 0: Q22 = + 30,000 12 FN 0: Q23 = + 50,000
The co-ordinates of the ellipse are calculated with the following formulae: Y = Q24 = Q22 x sin Q21 X = Q25 = Q23 x cos Q21 Q24 and Q25 are used as co-ordinates for linear interpolation	13 LBL 1 14 FN 6: Q24 = SIN + Q21 15 FN 7: Q25 = COS + Q21 16 FN 3: Q24 = + Q24* + Q22 17 FN 3: Q25 = + Q25* + Q23 18 L X + Q25 Y + Q24 RR F200 M
New angle Q21 = previous angle Q21 + angular step Q20	19 FN 1: Q21 = + Q21* +20
If the angle Q21 i smaller than 360,1° jump to LBL 1!	20 FN 12: IF + Q21 LT + 360.100 GOTO LBL 1
The ellipse is completely machined, a departure is made from the contour	21 L Y + 70,000 R F200 M 98
Traverse to tool-change position	22 TOOL CALL 0 Z S 0,000 23 L Z + 20,000 R0 F15999 M 24 L Z + 70,000 Y + 70,000 R F M 05 25 STOP M

Further examples of parameter programming can be found in the "Programming Examples" manual which is available upon request.

M 6) Subprograms and program part repeats

Program labels for marking subprograms or program part repeats can be set at any desired location within the program. These label numbers serve as so-called "jump addresses".

A jump command to a label number always ensures the finding of the correct location within the program even after program editing (insertion and deletion of blocks). Numbers 1 to 254 can be used for allocating labels. The label number "0" is always used as a mark for "end of subprogram".

If a subprogram is to be machined at different locations, there are two possibilities for programming:
.compile the whole subprogram in incremental dimensions (with incremental nominal position values)
or
.compile the subprogram in absolute dimensions (with absolute nominal position values) and define locations with datum shift routine (see section M 7.2.7).

M 6.1) Setting label numbers [LBL SET]

Dialogue initiation: press [LBL SET]

Dialogue question	Response
LABEL NUMBER?	Enter required number; press [ENT].

Dialogue question:
LABEL NUMBER?

Possible entry values: 0 – 254

The allocation of a label number requires one program block.

LBL 10

M 6.2) Jump to a label number [LBL CALL]

Dialogue initiation: press [LBL CALL]

Dialogue question	Response
LABEL NUMBER?	Enter label number to be called-up; press [ENT].
REPEAT REP?	Press [NO ENT] if the label is a marker for a **subprogram** or enter **number of repetitions** if the label signifies a **program part repeat;** press [ENT].

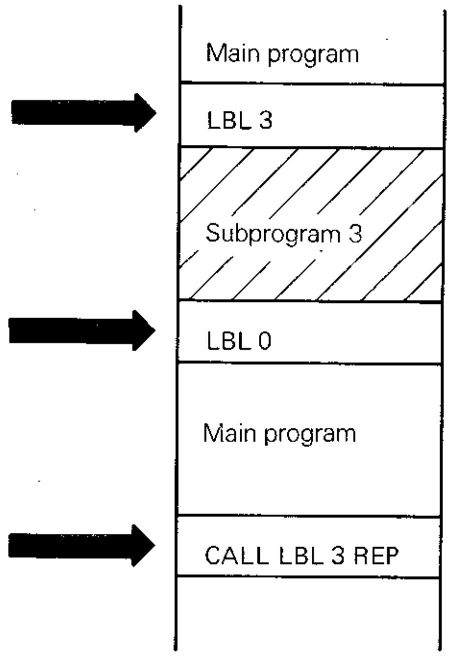
Dialogue question:
REPEAT REP?

Possible entry values: 1 – 65 534

A jump to a program label allocates one program block.

with call-up of a subprogram: **CALL LBL 12 REP**
or
with a program part repeat: **CALL LBL 18 REP 10/10**

M 6.3) Schematic diagram of a subprogram



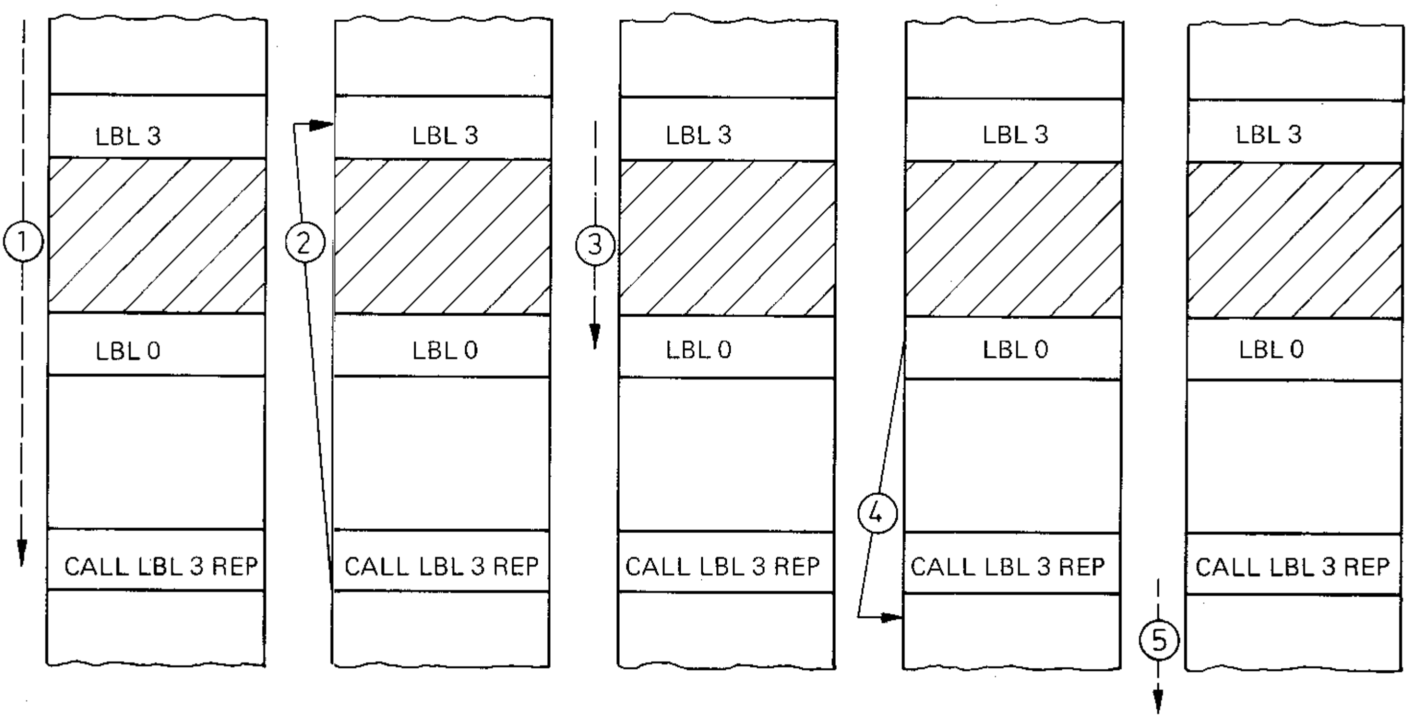
The beginning of the subprogram is labelled (e.g. LBL 3).

The end of the subprogram is labelled LBL 0.

By making a subprogram call-up, the subprogram can be retrieved at any location within the main program sequence (a jump is made to the desired program label). After the subprogram has been executed, the main program sequence is resumed.

After call-up, a **subprogram** can only be **executed once.**

Explanation of program procedure:

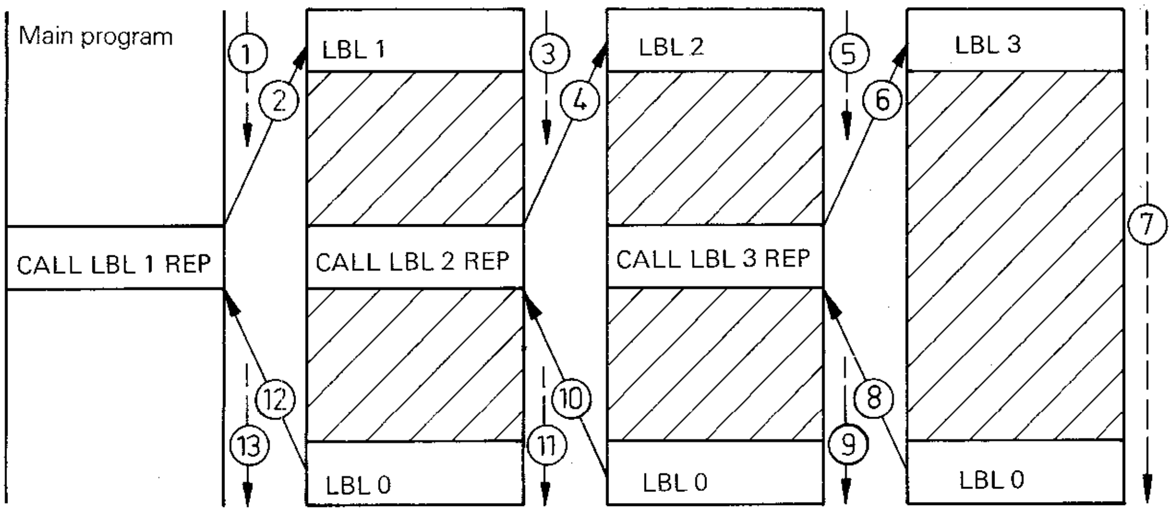


1. The main program sequence is worked through until the subprogram is called up.
2. Now a jump takes place to the label number of the call-up.
3. The subprogram is worked through until the end (LBL 0).
4. Return jump to the block immediately after the call-up.
5. The main program is continued.

Nesting of subprograms

Subprograms (sub-routines) can be nested up to 8 times, i.e. various subprograms can be interconnected with other subprograms via jump commands. Subprograms may also contain program part repeats. If the subprogram is nested more than 8 times, the error "EXCESSIVE SUBPROGRAMMING" is indicated.

Schematic diagram of subprogram "nesting":

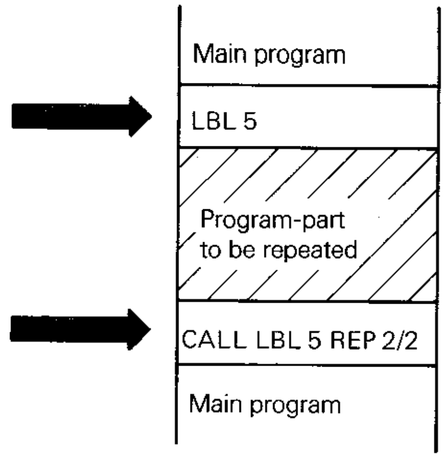


A subprogram may not be contained within a subprogram.

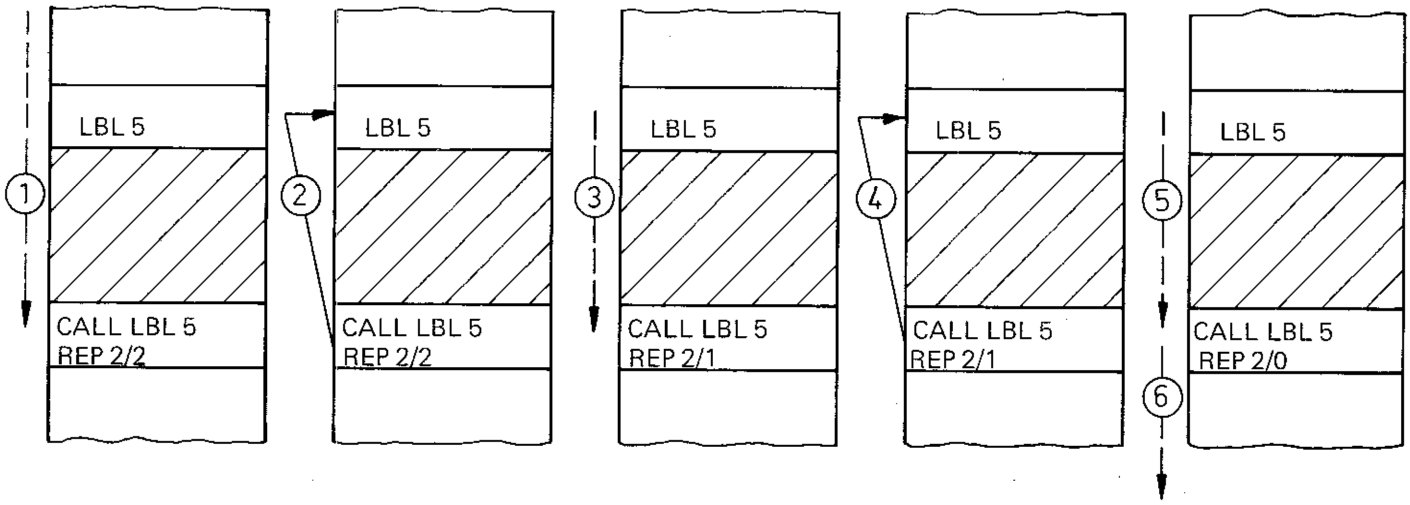
M 6.4) Schematic diagram of a program part repeat (Program loop)

The beginning of the program part which is to be repeated is labelled (e.g. LBL 5).

With a program part repeat, the number of repetitions is entered after the label number. A maximum of 65535 repeats may be entered.



Explanation of program procedure:



1. The main program is executed until call-up of the program part repeat. In the example, two repetitions have been programmed: CALL LBL 5 REP 2/2; the last figure (after the dash) indicates a count-down of the repetitions still to be executed.

2. Now a jump takes place to the label which has been called.

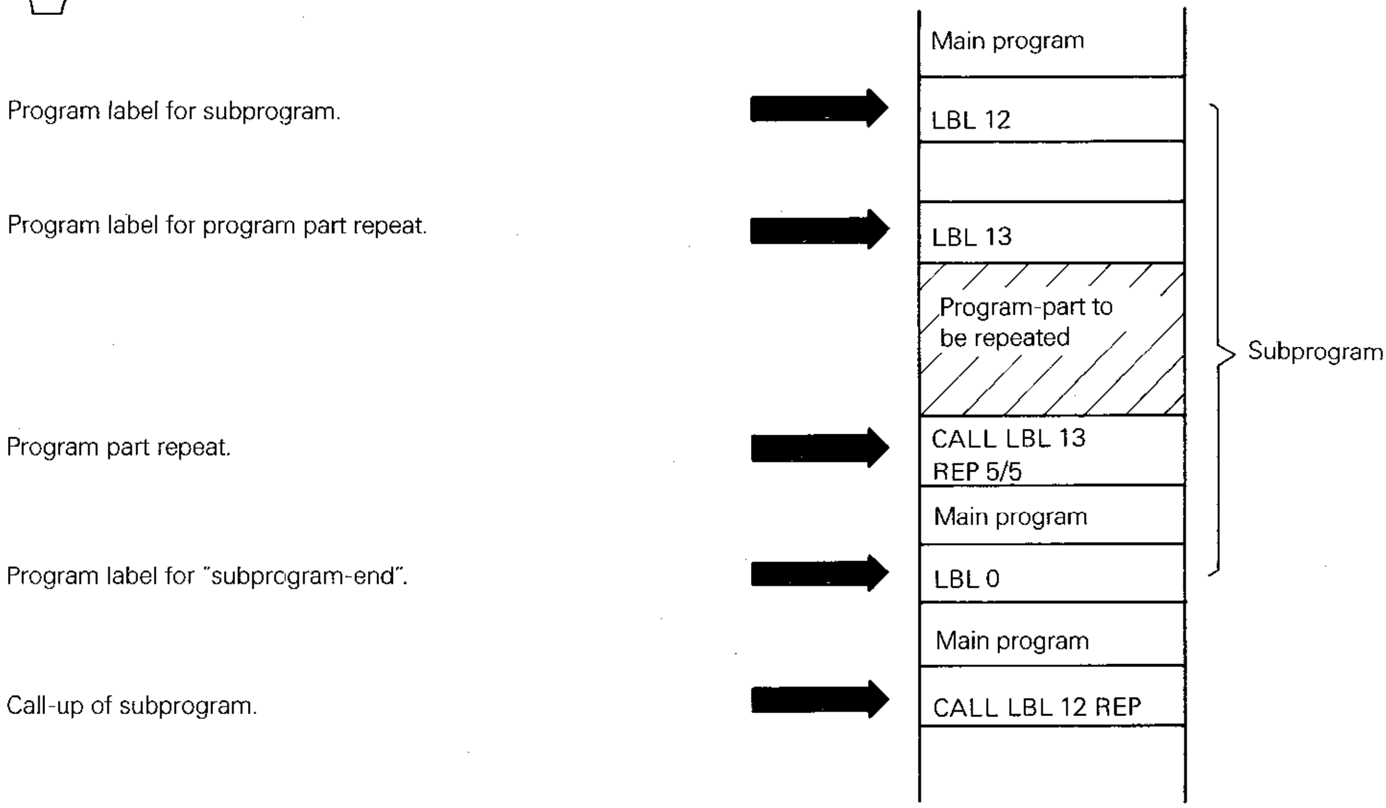
3. The part-program is now repeated. If a "label 0" is included within the part-program, this is ignored by the control.

4. New jump to label.

5. After completion of the second repetition, the display shows: CALL LBL 5 REP 2/0.

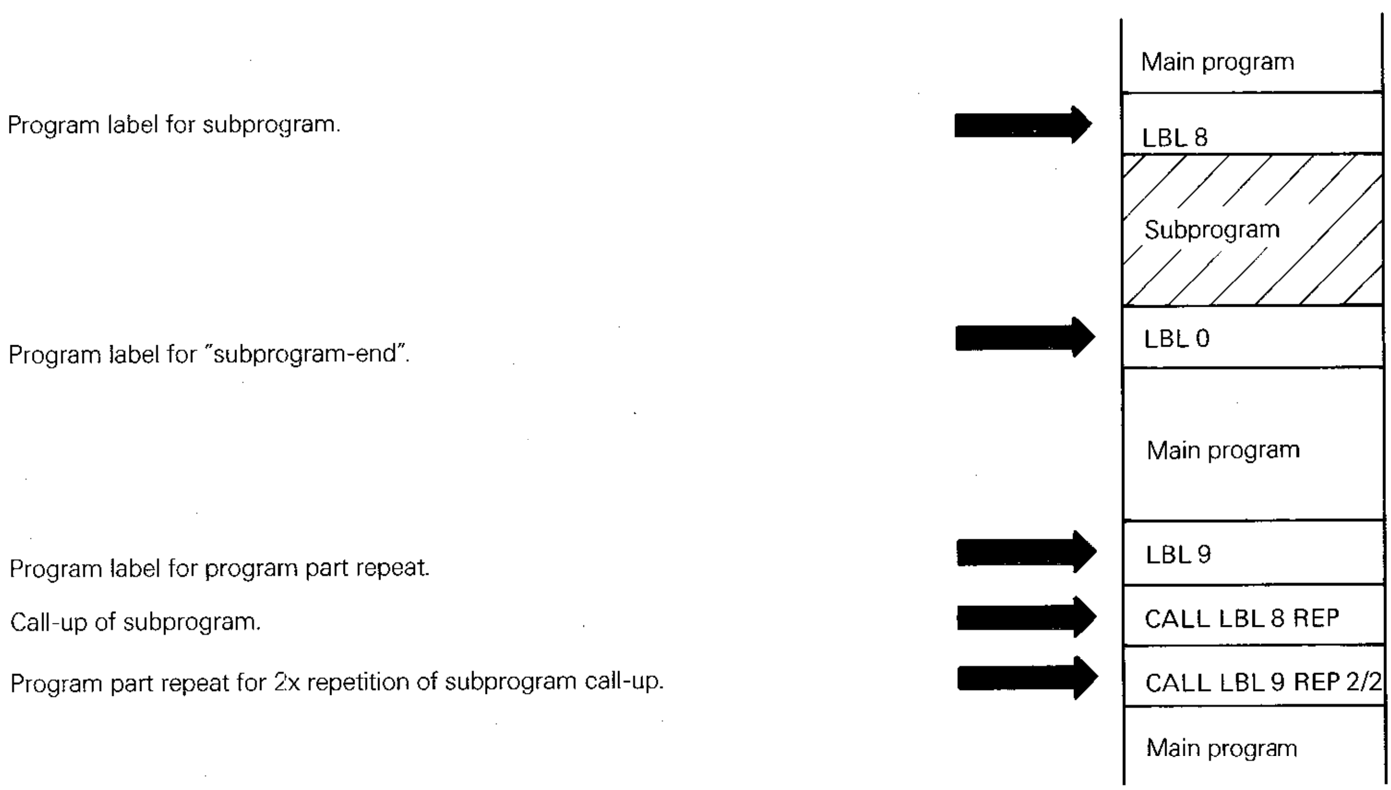
When all repetitions are completed, main program run is continued.

A **program part repeat** may also be programmed within a subprogram.



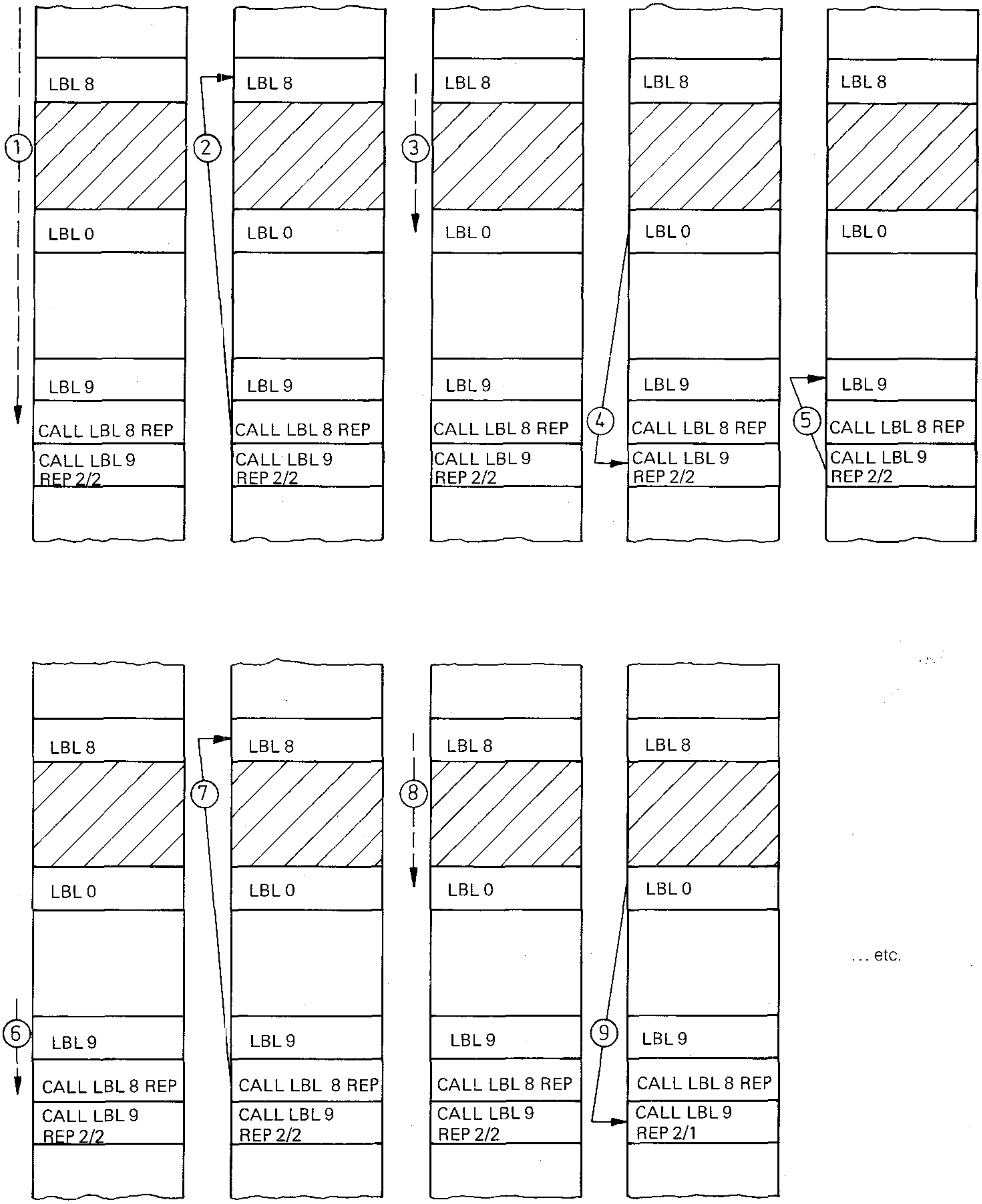
M 6.5) Schematic diagram of a multi-subprogram repetition

If a subprogram is to be repeated several times, programming should be performed in accordance with the following diagram:



If **two repeats** are programmed, the subprogram is **executed three times.**

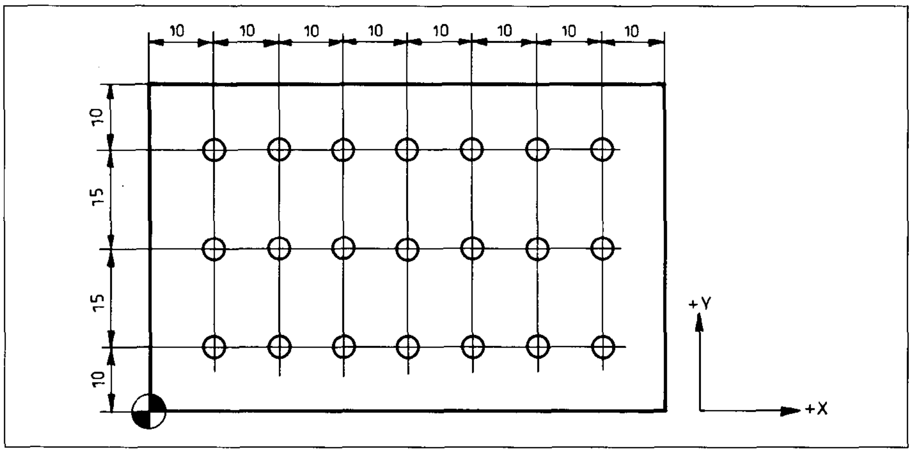
Explanation of program procedure

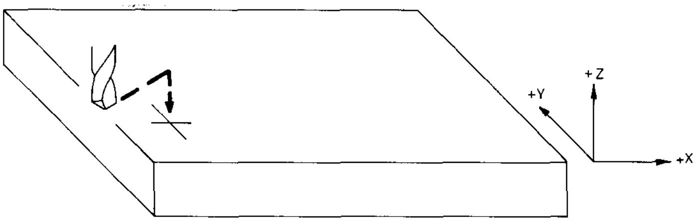
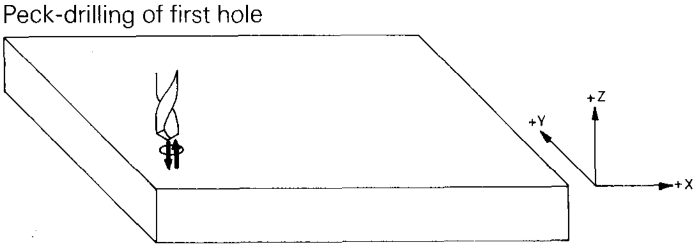
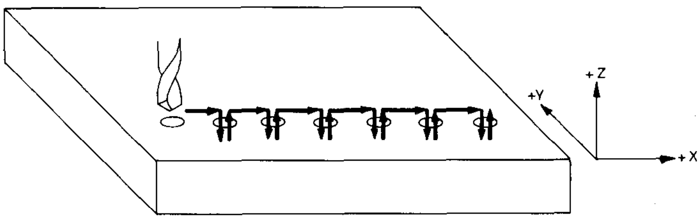


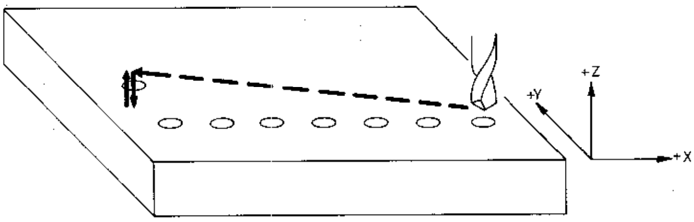
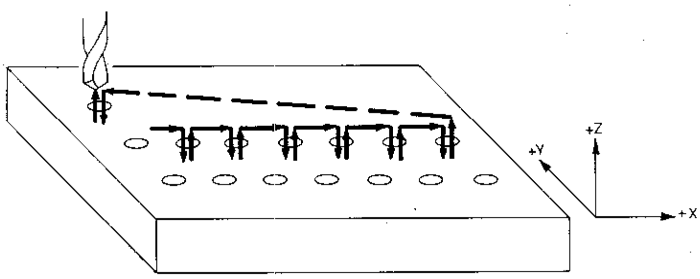
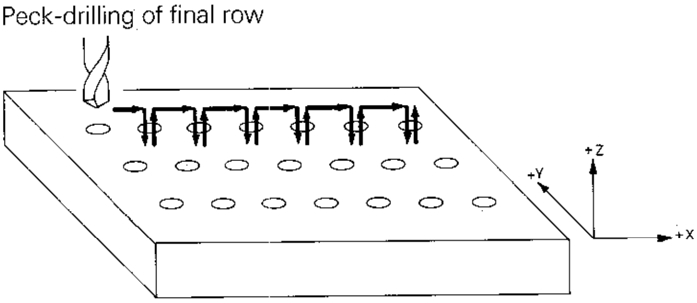
1. The main program is executed until call-up of the subprogram.
2. Return jump to label number which has been called.
3. Execution of subprogram.
4. Return jump to the block immediately after the call-up.
5. Return jump to label for program part repeat.
6. The subprogram call-up is located within the program part repeat.
7. Return jump to label number which has been called.
8. Execution of subprogram.
9. Return jump to the block immediately after the call-up.
10. This program procedure is repeated until all program part repeats, i. e. all subprogram call-ups have been executed.

M 6.6) Programming of hole patterns via subprograms and program part repeats

Time consuming programming of hole patterns is made more simple by using subprograms and program part repeats. The following example explains the method of programming.



Programming procedure:	Program block display
Select tool compensation and traverse to tool-change position Tool definition and Tool call	1 TOOL CALL 0 Z S 0,000 2 L Z +20,000 R0 F9999 M05 3 L X −20,000 Y −20,000 R0 F9999 M 4 TOOL DEF 1 L... R... 5 STOP M 6 TOOL CALL 1 Z S...
Definition of hole pattern	7 CYCL DEF 1.0 PECKING 8 CYCL DEF 1.1 SET-UP −2,000 9 CYCL DEF 1.2 DEPTH −25,000 10 CYCL DEF 1.3 PECKG −3,000 11 CYCL DEF 1.4 DWELL 0 12 CYCL DEF 1.5 F 200
Traverse to first hole of first row 	13 L X +10,000 Y +10,000 R0 F9999 M03 14 L Z +2,000 R0 F9999 M
Peck-drilling of first hole 	15 CYCL CALL M
Programming of first row in incremental dimensions with program part repeat and labelling of this program section as a subprogram 	16 LBL 1 17 L I X +10,000 R0 F9999 M 18 CYCL CALL 19 LBL CALL 1 REP 5/5 20 LBL 0

Programming procedure:	Program block display
Traverse to second hole row (the Y-co-ordinate is programmed incrementally) and peck-drill first hole of row 	21 L X +10,000 I Y +15,000 R0 F9999 M 22 CYCL CALL M
Peck-drilling of second row and subsequent rows and first hole of final row (if more than three rows are to be drilled, the number of repeats "REP" is to be changed). 	23 LBL CALL 1 REP 1/1
Peck-drilling of final row 	24 LBL CALL 1 REP
Traverse to tool-change position	25 TOOL CALL 0 Z S 0,000 26 L Z +20,000 R0 F9999 M05 27 L X −20,000 Y −20,000 R0 F9999 M

M 7) Canned cycles

For general purpose operation, TNC 150 possesses canned cycles for re-occuring machining operations. Moreover, for simplification of programming, a number of co-ordinate transformation routines are offered by the TNC 150 (datum shift, mirror image, co-ordinate system rotation, scaling). A dwell time can also be entered in form of a cycle.

Range of cycles:

Cycle 1 = Pecking
Cycle 2 = Tapping
Cycle 3 = Slot milling
Cycle 4 = Pocket milling
Cycle 5 = Circular pocket
Cycle 9 = Dwell time
Cycle 7 = Datum shift
Cycle 8 = Mirror image
Cycle 10 = Co-ordinate
Cycle 11 = Scaling

The following cycles are executed at the point of definition:
9 = Dwell time, 7 = Datum shift, 8 = Mirror image, 10 = Co-ordinate and 11 = Scaling.

It is therefore unnecessary to retrieve the cycle via the [CYCL CALL]-key. All other cycles require a cycle call.

M 7.1) Selecting a certain cycle

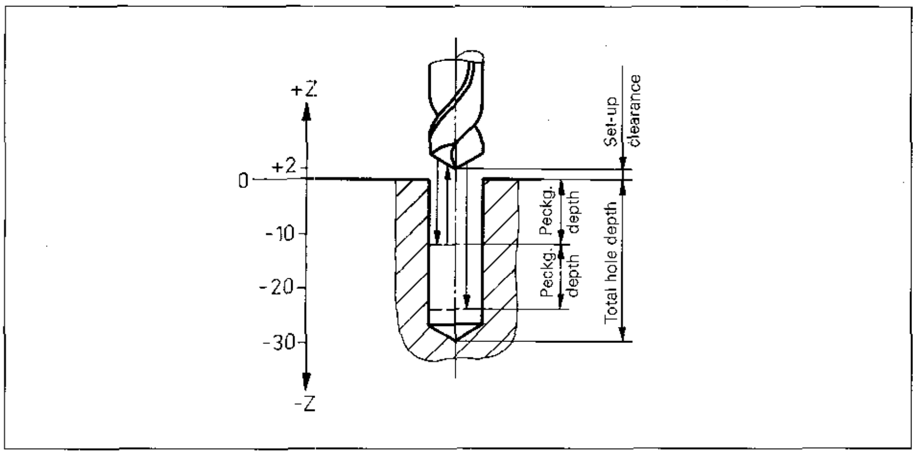
("Paging" of cycle library)

The cycle is called-up by means of [CYCL DEF] and [↓] (repetitive pressing if reqd.). By pressing [ENT] the cycle is transferred into the memory and defined as per the dialogue.

M 7.2) Explanation of canned cycles

M 7.2.1) Cycle: "Pecking"

Provisions for execution of cycle:
.A previous tool call (determination of drilling axis and spindle speed).
.The direction of spindle rotation must already have been determined with a previous program block (M 03 or M 04)
.The starting position (set-up clearance) must have been approached in a previous block.



Example:

Set-up clearance = −2.
(When the machine is traversed −2 in incremental mode, the tip of the tool must make contact with the workpiece surface at absolute value = 0)

Total hole depth = − 30

Pecking depth = − 12

1st Procedure: Drilling to depth − 12 and retraction of Z-axis to the + 2-position in rapid traverse. (This is necessary for breaking the swarf)

2nd Procedure: Rapid traverse to position − 11.4* and further peck-drill operation at programmed feed rate to position − 24. Now retraction of Z-axis to + 2-position in rapid traverse.

3rd Procedure: Rapid traverse to position − 23.4* and further peck-drill operation at programmed feed rate to position − 30. Upon reaching the total hole depth, the dwell time commences (the drill cuts-free) and then the axis retracts to the starting position + 2 in rapid traverse.

* The advanced stop distance before reaching the pecking depth is automatically calculated by the control.
.With a total hole depth of 30 mm the advanced stop distance is 0.6 mm.
.With a total hole depth exceeding 30 mm the advanced stop distance is calculated according to the following formula: $\frac{\text{Total hole depth}}{50}$
.The advanced stop distance never exceeds 7 mm.

Dialogue initiation: press CYCL DEF and ↓

Dialogue question	Response
CYCL DEF 1 PECKING	Press ENT
SET-UP CLEARANCE?	Enter set-up clearance with sign**; Press ENT. This position must already have been approached with a previous block.
TOTAL HOLE DEPTH?	Enter hole depth with sign**; Press ENT.
PECKING DEPTH?	Enter pecking-depth with sign**; Press ENT.
DWELL TIME IN SECS.?	Enter dwell time for cutting drill free; Press ENT.
FEED RATE? F = ...	Enter feed rate; Press ENT.

** The set-up clearance, the total hole depth and the pecking depth must all have the same arithmetical sign.

Dialogue question:
DWELL TIME IN SECS.?

Possible entry values: 0 – 19 999.999 s
The "pecking" cycle allocates six program blocks. When "paging" the program, the following blocks are displayed:

CYCL DEF 1.0 PECKING	
CYCL DEF 1.1 SET-UP − 2,000	Set-up clearance
CYCL DEF 1.2 DEPTH − 100,000	Total hole depth
CYCL DEF 1.3 PECKG − 20,000	Pecking depth
CYCL DEF 1.4 DWELL − 0,000	Dwell time
CYCL DEF 1.5 F 80	Feed rate

M 7.2.2) Cycle: "Tapping"

Provisions for execution of cycle:
.For tapping, a chuck with length compensation facility is to be used. The length compensation chuck must allow for the tolerances between the feed rate and the spindle speed as well as the spindle slow-down after reaching the final position.
.Previous tool call (definition of working spindle axis and spindle speed).
.The spindle rotating direction must have been determined with a previous block (M 03 for right-hand thread/M 04 for left-hand thread).
.The starting position (set-up clearance) must have been approached with a previous block.

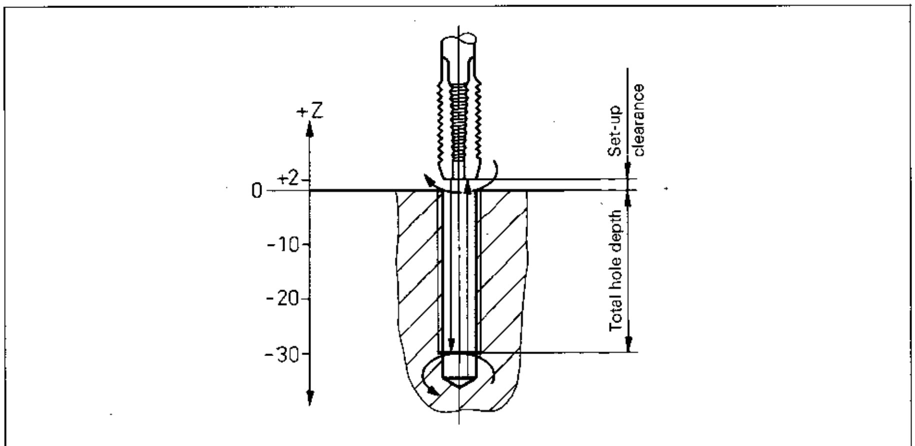
Calculation of feed rate for cycle definition "tapping":

Feed rate [mm/min.] = spindle speed [rpm] x thread pitch [mm]

Example:

Set-up clearance = – 2

Total hole depth = – 30



The thread is cut in **one single** operation. After the total depth has been reached, the rotating direction of the tool spindle is automatically switched over to the opposite direction after a delay which has been programmed via the machine parameters. Now the programmed dwell time takes effect. Finally, the tapping tool is retracted to the position of the set-up clearance.

If the "Tapping cycle" is called, the programmed feed rate can only be altered within a limited range with the override potentiometer. The range limits are determined by the machine manufacturer by entering certain machine parameters. This limited function of the override potentiometer is necessary for reasons of safety.

Dialogue initiation: press [CYCL DEF] and [↓] until the cycle "tapping" is displayed.

Dialogue question	**Response**
CYCL DEF 2 TAPPING	Press [ENT]
SET-UP CLEARANCE?	Enter set-up clearance with sign*; press [ENT]. This position must already have been approached in a previous block.
TOTAL HOLE DEPTH?	Enter hole depth with sign*; press [ENT].
DWELL TIME IN SECS.?	Program amount of dwell time required between rotation change-over and retraction of tapping tool; press [ENT].
FEED RATE? F = ...	Enter feed rate; press [ENT].

* The set-up clearance and the hole depth must have the same arithmetical sign and be programmed incrementally.

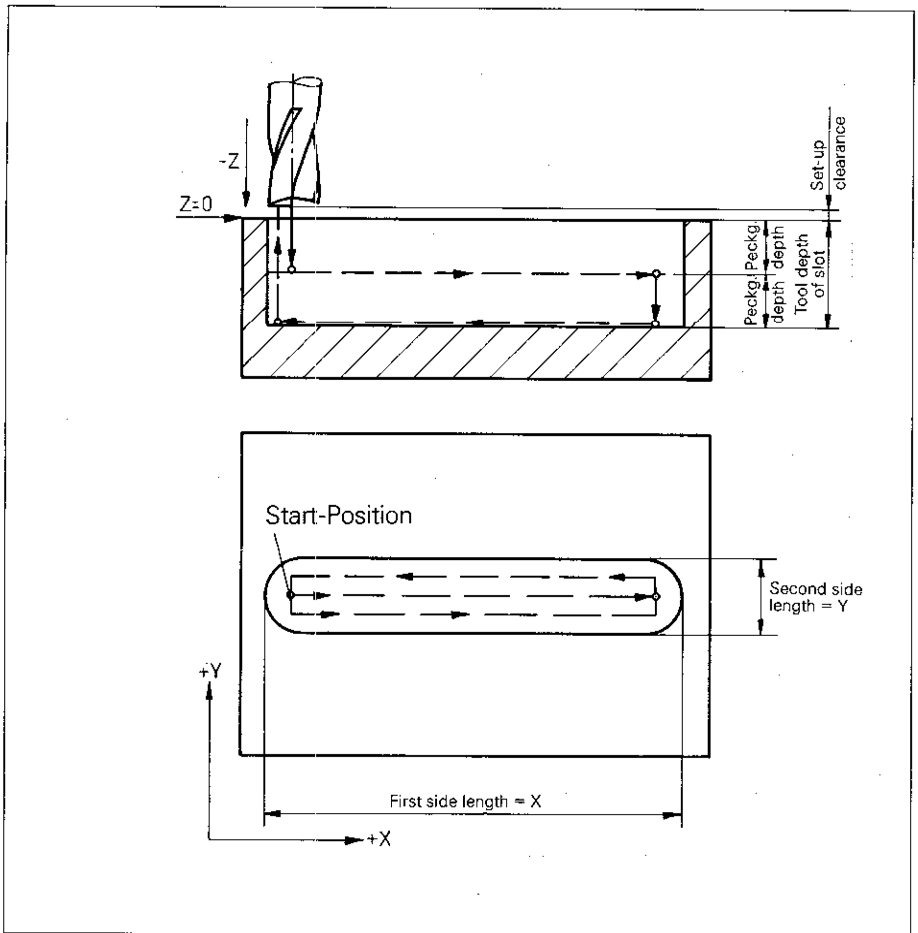
The "tapping" cycle allocates five program blocks. When "paging" the program, the following blocks are displayed:

CYCL DEF 2.0 TAPPING	
CYCL DEF 2.1 SET-UP – 2,000	Set-up clearance
CYCL DEF 2.2 DEPTH – 30,000	Total hole depth
CYCL DEF 2.3 DWELL 0,000	Dwell time
CYCL DEF 2.4 F 160	Feed rate

M 7.2.3) Cycle: "Slot milling"

Provisions for execution of cycle:
.The slot must be larger than the diameter of the milling cutter.
.Previous tool call (definition of working spindle axis and spindle speed).
.The spindle rotating direction must have been determined with a previous block (M 03 or M 04).
.The starting position (starting point of elongated slot and set-up clearance) must have already been defined with previous blocks.

Operating procedure:



1. Rough cut: The milling cutter penetrates the workpiece at the programmed feed rate until the first pecking depth is reached. Now the first rough cut is made into the material. The next pecking depth is milled out at the other end of the slot etc.

2. Finishing cut: The cutter now makes a finishing cut to the side limits of the slot and finally traverses the intended contour in down-cut* milling.

3. Return to starting position: The milling cutter returns to the set-up clearance position in rapid traverse. If the number of pecks is an odd number, the starting position is reached with an additional traverse along the slot.

The starting point of the slot can be established by means of two methods:
1. With an axis-parallel positioning block (dialogue initiation: key X, Y or Z)
with radius compensation R+ or R− by approaching the slot in longitudinal direction.

2. With a linear interpolation block (dialogue initiation: key) by approaching the slot perpendicular to linear direction with radius compensation RR or RL and by de-activating radius compensation with auxiliary function M 98.

* The terms "up-cut" and "down-cut" milling refer to right-hand rotation of the tool.

Dialogue initiation: press CYCL DEF and ↓ until the cycle "slot milling" is displayed.

Dialogue question	Response
CYCL DEF 3 SLOT MILLING	Press ENT
SET-UP CLEARANCE?	Enter set-up clearance with sign*; press ENT. This position must already have been approached in a previous block.
MILLING DEPTH?	Enter milling depth with sign*; press ENT.
PECKING DEPTH?	Enter pecking depth with sign*; press ENT.
FEED RATE FOR PECKING	Enter feed rate for pecking into workpiece; press ENT.
FIRST SIDE LENGTH?	The numerical value for the longitudinal direction of the slot is programmed with the correct sign. (It must be determined in which direction the slot lies with respect to the starting position.)
SECOND SIDE LENGTH?	The width of the slot is always programmed with a positive sign.
FEED RATE? F=...	Enter feed rate for milling of slot.

* The set-up clearance, milling depth and pecking depth must have the same arithmetical sign and be entered in incremental dimensions.

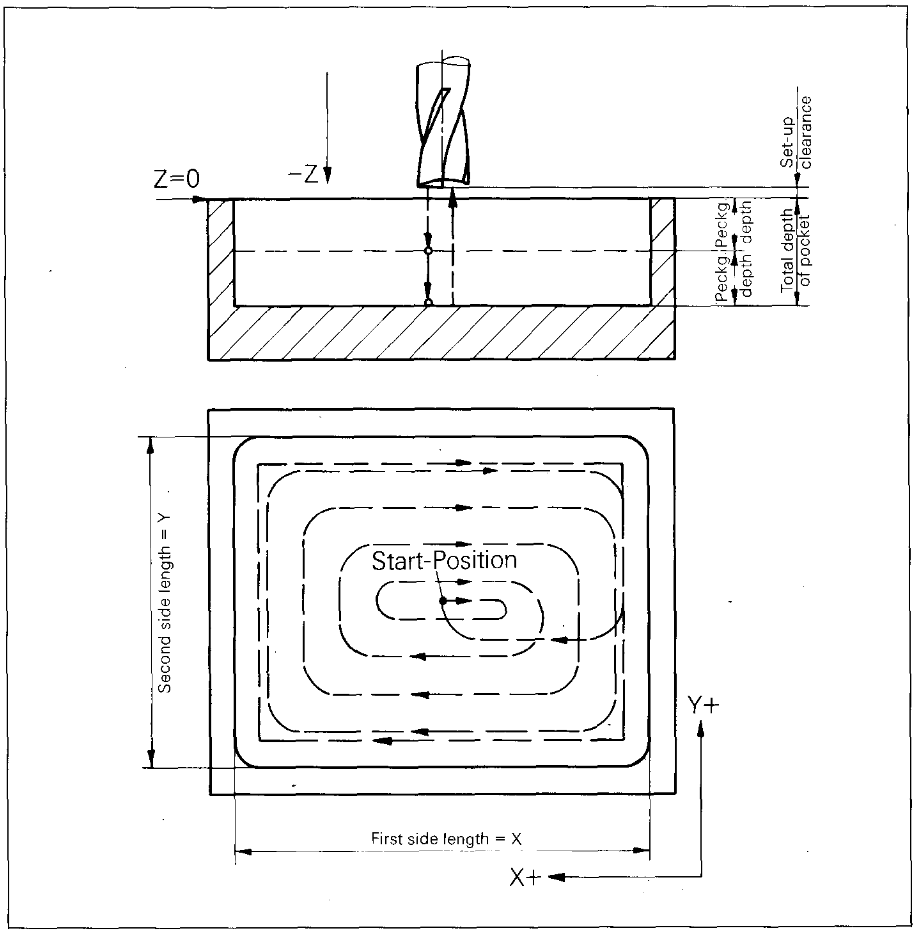
The "slot milling" cycle allocates seven program blocks. When "paging" the program, the following blocks are displayed:

CYCL DEF 3.0 SLOT MILLING	
CYCL DEF 3.1 SET-UP – 2,000	Set-up clearance
CYCL DEF 3.2 DEPTH – 40,000	Milling depth
CYCL DEF 3.3 PECKING – 20,000	Pecking depth
F 80	Feed rate for pecking
CYCL DEF 3.4 X + 80,000	Length of slot
CYCL DEF 3.5 Y + 20,000	Width of slot
CYCL DEF 3.6 F 100	Feed rate

M 7.2.4) Cycle: "Pocket milling" (Rough cut cycle)

Provisions for execution of cycle:
.Previous tool call (definition of working spindle axis and spindle speed).
.The spindle rotating direction must have been determined with a previous block.
.The starting position (centre of pocket and set-up clearance) must already have been defined with previous blocks.

Operating procedure:



After penetration into the workpiece, the milling cutter follows a path as shown above (either down-cut or up-cut milling) which is parallel to the edge limits of the pocket and which is traversed to a max. of K* x R (R = cutter radius) to the edge limits.
If the pocket is unable to be milled in one plunge (due to the cutting force being too great), a pecking depth has to be programmed .

The milling procedure is repeated until the final pocket depth is reached.

"Pocket milling" is a rough cut-cycle. If a finishing cut is required, this has to be programmed separately.

* The factor K is determined with a machine parameter by the machine tool manufacturer and can lie between 0.001 and 1.414.

Dialogue initiation: press CYCL DEF and ↓ until the cycle "pocket milling" is displayed.

Dialogue question	Response
CYCL DEF 4 POCKET MILLING	Press ENT
SET-UP CLEARANCE	Enter set-up clearance with sign*; press ENT. This position must already have been approached in a previous block.
MILLING DEPTH?	Enter milling depth with sign*; press ENT.
PECKING DEPTH?	Enter pecking depth with sign*; press ENT.
FEED RATE FOR PECKING	Enter feed rate for pecking into workpiece; press ENT.
FIRST SIDE LENGTH?	Enter first side length with positive sign*; press ENT.
SECOND SIDE LENGTH?	Enter second side length with positive sign*; press ENT.
FEED RATE? F = ...	Enter feed rate for milling of slot; press ENT.
ROTATION CLOCKWISE: DR–?	Use sign change-key for: clockwise rotation DR– (up-cut milling); or anti-clockwise rotation DR+ (down-cut milling); press ENT.

* The set-up clearance, milling depth and pecking depth must have the same arithmetical sign and be entered in incremental dimensions.

The "pocket milling" cycle allocates seven program blocks. When "paging" the program, the following blocks are displayed:

CYCL DEF 4.0 POCKET MILLING	
CYCL DEF 4.1 SET-UP – 2,000	Set-up clearance
CYCL DEF 4.2 DEPTH – 30,000	Milling depth
CYCL DEF 4.3 PECKING – 10,000	Pecking depth
F 80	Feed rate for pecking
CYCL DEF 4.4 X + 80,000	First side length
CYCL DEF 4.5 Y + 40,000	Second side length
CYCL DEF 4.6 F 100 DR+	Feed rate / Rotating direction

M 7.2.5) Cycle: "Circular pocket" (Rough cut cycle)

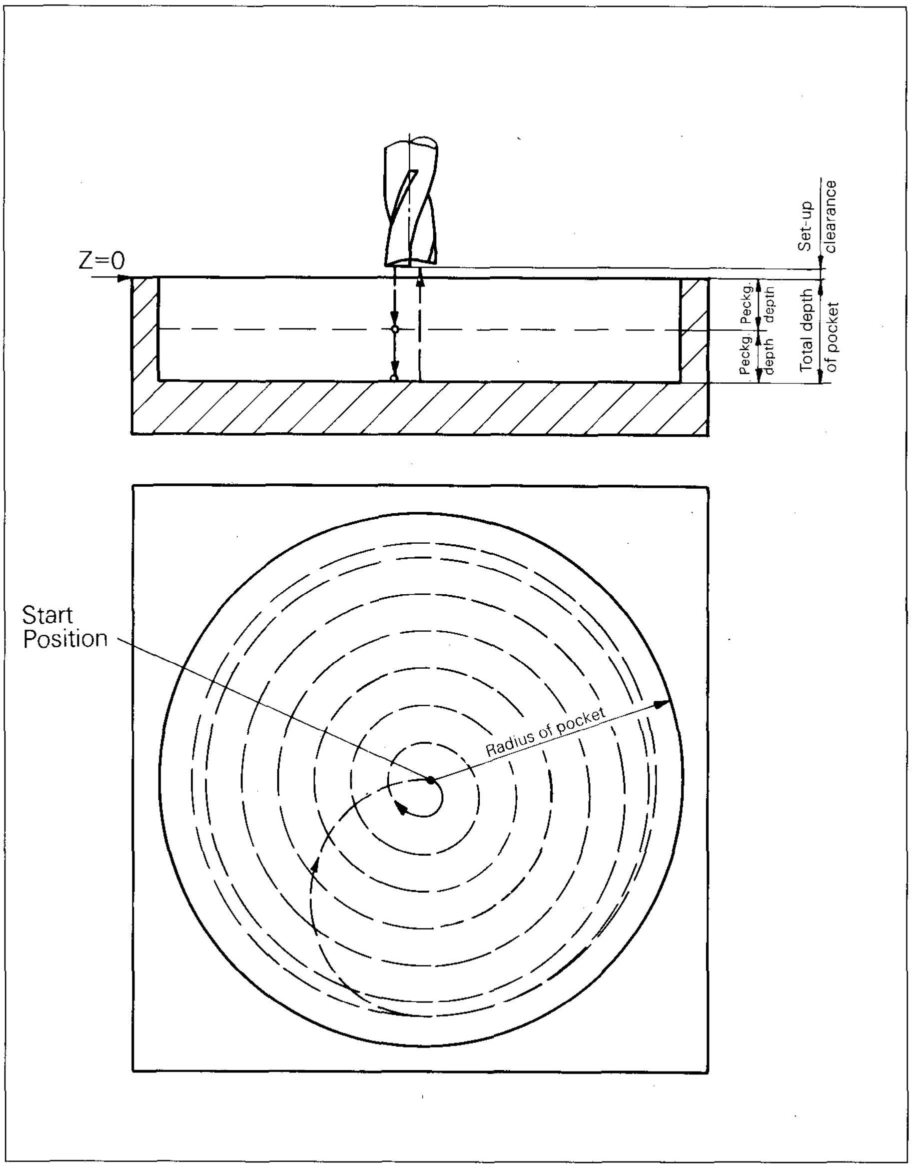
Provisions for execution of cycle:

.Previous tool call (definition of working spindle axis and spindle speed).

.The spindle rotating direction must have been determined with a previous block (M 03 or M 04).

.The starting position (centre of circular pocket and set-up clearance) must have already been defined with previous blocks.

Operating procedure:



After penetration into the workpiece, the milling cutter follows a path in a spiral direction towards the outer limit of the circular pocket, as shown above (either down-cut or up-cut milling). The pitch of the milling cutter is K* x R (R = cutter radius). If the pocket is unable to be milled in one plunge (due to the cutting force being too great), a pecking depth has to be programmed.

The milling procedure is repeated until the final pocket depth is reached.

The cycle "circular pocket" is a rough-cut cycle. If a finish cut is required, this is to be programmed separately.

* The factor K is determined by the machine tool manufacturer with a machine parameter and can lie between 0.001 and 1.414.

Dialogue initiation: press CYCL DEF and ↑ until the cycle "circular pocket" is displayed.

Dialogue question	Response
CYCL DEF 5 CIRCULAR POCKET	Press ENT
SET-UP CLEARANCE?	Enter set-up clearance with sign*; press ENT. This position must already have been approached in a previous block.
MILLING DEPTH?	Enter milling depth with sign*; press ENT.
PECKING DEPTH?	Enter pecking depth with sign*; press ENT.
FEED RATE FOR PECKING	Enter feed rate for pecking into workpiece; press ENT.
CIRCLE RADIUS?	Enter radius of circular pocket; press ENT.
FEED RATE? F = ...	Enter feed rate for milling of slot.
ROTATION CLOCKWISE: DR–?	Use sign change-key for: clockwise rotation DR– (up-cut milling); or anti-clockwise rotation DR+ (down-cut milling); press ENT.

* The set-up clearance, milling depth and pecking depth must have the same arithmetical sign and be entered in incremental dimensions.

Dialogue question:
CIRCULAR POCKET?

Possible entry values: 0 – 19 999,999

The "circular pocket" cycle allocates six program blocks. When "paging" the program, the following blocks are displayed:

CYCL DEF 5.0 CIRCULAR POCKET	
CYCL DEF 5.1 SET-UP – 2,000	Set-up clearance
CYCL DEF 5.2 DEPTH – 60,000	Milling depth
CYCL DEF 5.3 PECKING – 20,000	Pecking depth
F 80	Feed rate for pecking
CYCL DEF 5.4 RADIUS 120,000	Radius
CYCL DEF 5.5 F 100 DR–	Feed rate / Rotating direction

M 7.2.6) Cycle: "Dwell time"

By means of the "dwell time" cycle, a definite standstill time during the program sequence is determined (e.g. for chip breaking). Entry step: 0.001 s; Entry range 0 ... 19 999,99 s

A cycle call is unnecessary.

Dialogue initiation: press CYCL DEF and ↑ until the "dwell time" cycle is displayed.

Dialogue question	Response
CYCL DEF 9 DWELL TIME	Press ENT
DWELL TIME IN SECS.	Enter required dwell time.

The "dwell time" cycle allocates two program blocks. When "paging" the program, the following blocks are displayed:

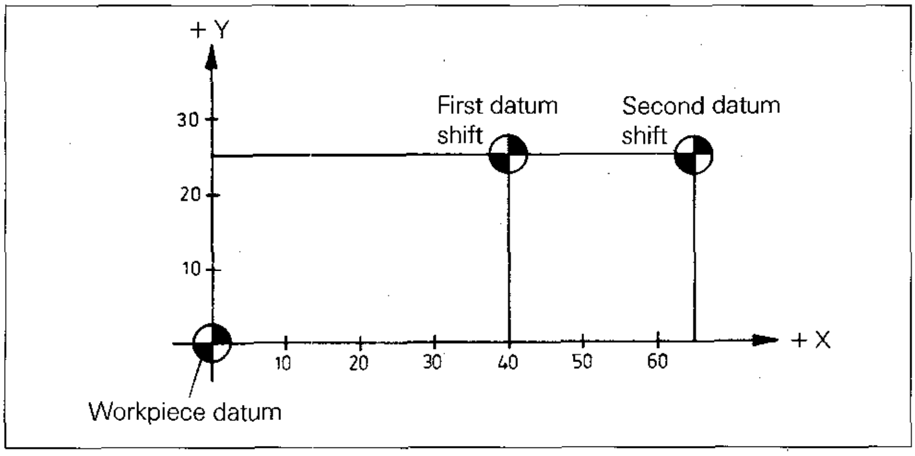
CYCL DEF 9.0 DWELL TIME	
CYCL DEF 9.1 DWELL10,000	Dwell time

M 7.2.7) Cycle: "Datum shift"

This cycle enables the shifting (displacement) of the workpiece datum **in all four axes** in either absolute or incremental dimensions. **The program section which is programmed after the cycle, is referenced to the new datum.** The workpiece datum which has been previously set with the preset facility is retained.

A cycle call is unnecessary.

Example: Datum shift in the X-Y-plane



Entry values:

First datum shift:
X 40.000
Y 25.000
Z 0.000

Second datum shift:
I X 25.000
I Y 0.000
I Z 0.000

Cancellation of the datum shift (i.e. positions are again referenced to the original workpiece datum which was preset) is performed by entering **a datum shift with the co-ordinates X 0.000, Y 0.000 and Z 0.000.**

Dialogue initiation: press CYCL DEF and ↑ until the cycle "datum shift" is displayed.

Dialogue question	Response
CYCL DEF 7 DATUM SHIFT	Press ENT
DATUM SHIFT ?	Enter datum shift in absolute or incremental dimensions. Press I if required. .Press first axis key and enter numerical value, .Press second axis key and enter numerical value, .Press third axis key and enter numerical value, .Press fourth axis key and enter numerical value, Press ENT or END □ (see section G 2)

The "datum shift" cycle allocates four program blocks. When "paging" the program, the following blocks are displayed:

CYCL DEF 7.0 DATUM SHIFT	
CYCL DEF 7.1 X+20,000	Datum shift X-Axis
CYCL DEF 7.2 Y+40,000	Datum shift Y-Axis
CYCL DEF 7.3 Z+10,000	Datum shift Z-Axis
CYCL DEF 7.4 C+90,000	Datum shift C-Axis

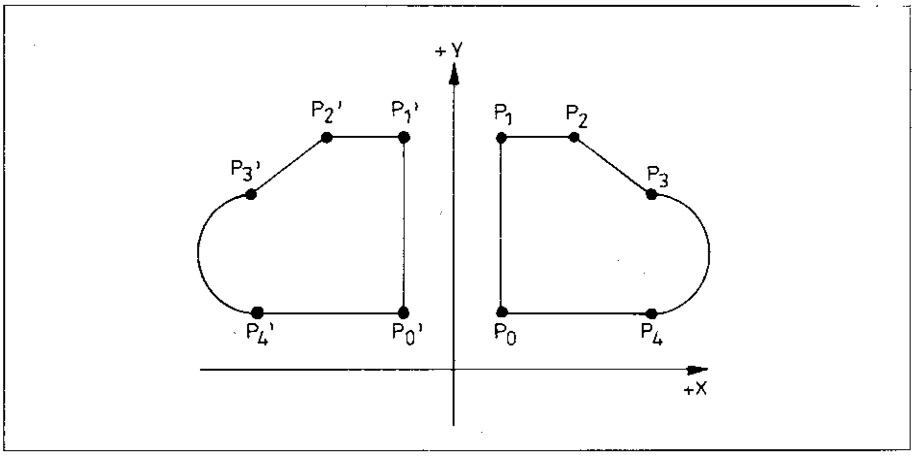
M 7.2.8) Cycle: "Mirror image"

This cycle enables the machining of a contour in mirror image, in the working plane. **The program section which falls within this cycle is produced in a mirror (reflected) image.** Simultaneous mirror image in two axes is also possible. Programmed co-ordinates of one axis or of two axes are multiplied by "–1"!

.The tool axis (working spindle axis) cannot be mirror imaged . . . (error indication: MIRROR IMAGE ON TOOL AXIS)
.A cycle call is unnecessary.

Example: Mirror image in the X-axis

The points P_0 to P_4 are the position values of a programmed contour. If mirror image is to take place in the X-axis, the arithmetical signs of all X-co-ordinates are inverted so that a reflected image of the points P_0' to P_4' is produced.



Dialogue initiation: press [CYCL DEF] and [↑] until the cycle "mirror image" is displayed.

Dialogue question	Response
CYCL DEF 8 MIRROR IMAGE	Press [ENT]
MIRROR IMAGE AXIS?	Enter mirror image axis: .Press first axis key .Press second axis key .Press [NO ENT] or [END □] (see section G 2)

Cancellation of mirror image

Mirror image is cancelled by

.programming the "mirror image" cycle and responding to all dialogue questions by pressing [NO ENT]
or by

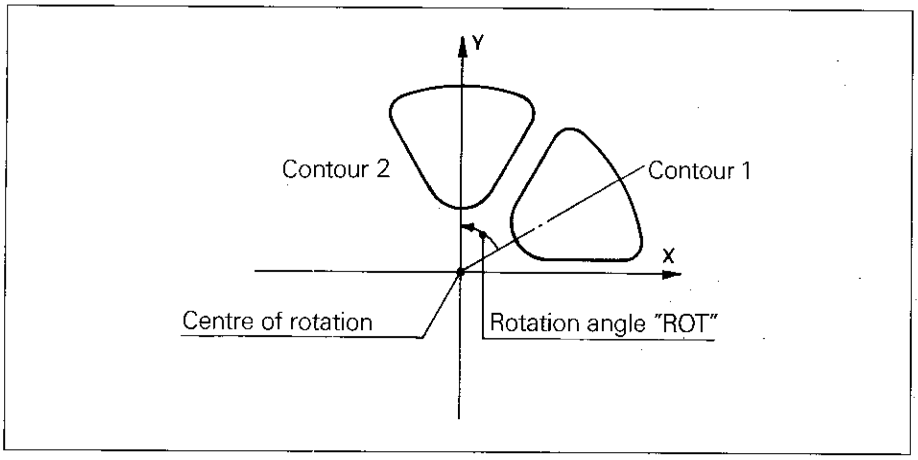
.programming of auxiliary function M 02 or M 03 (only possible if machine parameter 173 was set by the machine tool manufacturer).

The "mirror image" cycle allocates two program blocks. When "paging" the program, the following blocks are displayed:

CYCL DEF 8.0 MIRROR IMAGE	
CYCL DEF 8.1 X Y	Axis for mirror image

M 7.2.9) Cycle: "Co-ordinate rotation"

This cycle enables the rotation of a contour in the working plane and through a specific angle. The programm section which has been programmed within the cycle is rotated.



Example:

Contour 1 is programmed
Contour 2 is produced via the "Co-ordinate rotation" cycle.

Dialogue initiation: Press [CYCL DEF] and [↑] until the "co-ordinate rotation" cycle is displayed

Dialogue question	Response
CYCL DEF 10 ROTATION	Press [ENT] for cycle
ROTATION ANGLE?	Enter rotation angle and press [ENT] Entry range: 0° – 360°

Cancellation of "co-ordinate rotation"

Cancellation in performed as follows:

.Programming of rotation angle 0°
or
.Programming of auxiliary function M 02 or M 30 (only possilbe if machine parameter 173 has been set by the machine tool manufacturer).

The "co-ordinate rotation" cycle allocates two program blocks. When "paging" the program, the following blocks are displayed:

CYCL DEF 10.0 ROTATION
CYCL DEF 10.1 ROT + 20,000

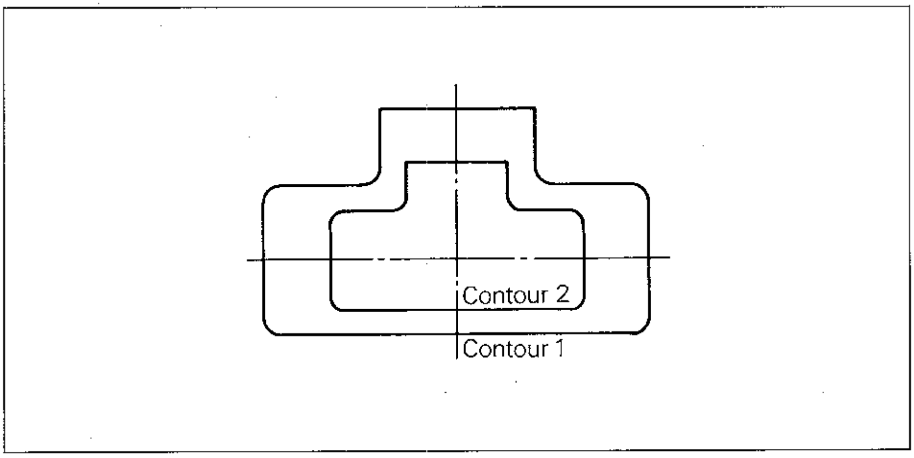
M 7.2.10) Cycle: "Scaling"

This cycle enables a contour to be geometrically increased or decreased in sized. The scaling factor is used for multipliying the co-ordinates either in the working plane
or
in the three main axes
– depending on the parameters entered.

The control can therefore take shrinkage dimensions into account and in the case of similar shapes on one workpiece, the shape need only be programmed once.

Example:

Contour 1 is programmed
Contour 2 is produced with the "scaling" cycle.



Dialogue intitiation: Press [CYCL DEF] and [↑] until the "scaling" cycle is displayed.

Dialogue question	Response
CYCL DEF 11 SCALING	Press [ENT]
FACTOR	Enter required factor and press [ENT] Entry range: 0.000000 –99.999999 Entry step: 0.000001

Cancellation of the "scaling" cycle

Cancellation is performed as follows:

.Programming of a scaling factor "1"
or
.Programming of auxiliary function M 02 or M 30 (only possible if machine parameter 173 has been set by the machine tool manufacturer).

The "scaling" cycle allocates two program blocks. When "paging" the program, the following blocks are displayed:

CYCL DEF 11.0 SCALING
CYCL DEF 11.1 SCL 0.980000

M 7.3) Cycle call [CYCL CALL]

There are two possibilities for cycle call:

1. Programming of a "CYCL CALL"-block

Dialogue initiation: press [CYCL CALL]

Dialogue question	Response
AUXILIARY FUNCTION M?	Enter M-function if reqd.; press [ENT] or press [NO ENT], if no auxiliary function is required.

The cycle call allocates one program block:

CYCL CALL
M 03

2. Programming of auxiliary function **M 99:** see section F 2

Example:
L X + 70.000 Y + 45.000
R0 F 9999 M 99

A cycle call is not required for the fixed machining cycles:
7 = Datum shift
8 = Mirror image
9 = Dwell time
10 = Co-ordinate system rotation
11 = Scaling

All other fixed machining cycles require a cycle call.

Only the **last defined cycle** within the program sequence can be retrieved with the [CYCL CALL] -key or auxiliary function M 99. Cycles which require no cycle call are ignored.

M 8) Program editing

M 8.1) Call-up of a program block

Select [⇉], [→] or [⇨].

↓

.Press [GO TO].
.Key-in desired block No. and press [ENT].

M 8.2) Program check blockwise

Select [⇉], [→] or [⇨].

↓

Press [GO TO] and enter block No. from which program-inspection is to commence: press [ENT].

↓

Check program either forwards or reverse by pressing the "paging keys" [↓] or [↑].

M 8.3) Deletion of blocks

Press [⇔].

↓

Select block **or last block of program part which is to be deleted.**

↓

Erase block(s) with [DEL □] -key.

In order to delete blocks for tool and cycle definition, the [DEL □] -key has to be pressed as many times, as individual blocks are required for the complete definition.

Block numbers for successive blocks are automatically amended.

M 8.4) Insertion of blocks into existing program

With TNC 150, new program blocks can be inserted into an existing program at any random location – only the block which immediately follows the location of insertion is to be selected and the new block may be entered. The numbers of the successive blocks are automatically shifted. If the storage capacity of the memory is exceeded, the dialogue display will show **PROGRAM MEMORY EXCEEDED.**

Press [⇔].

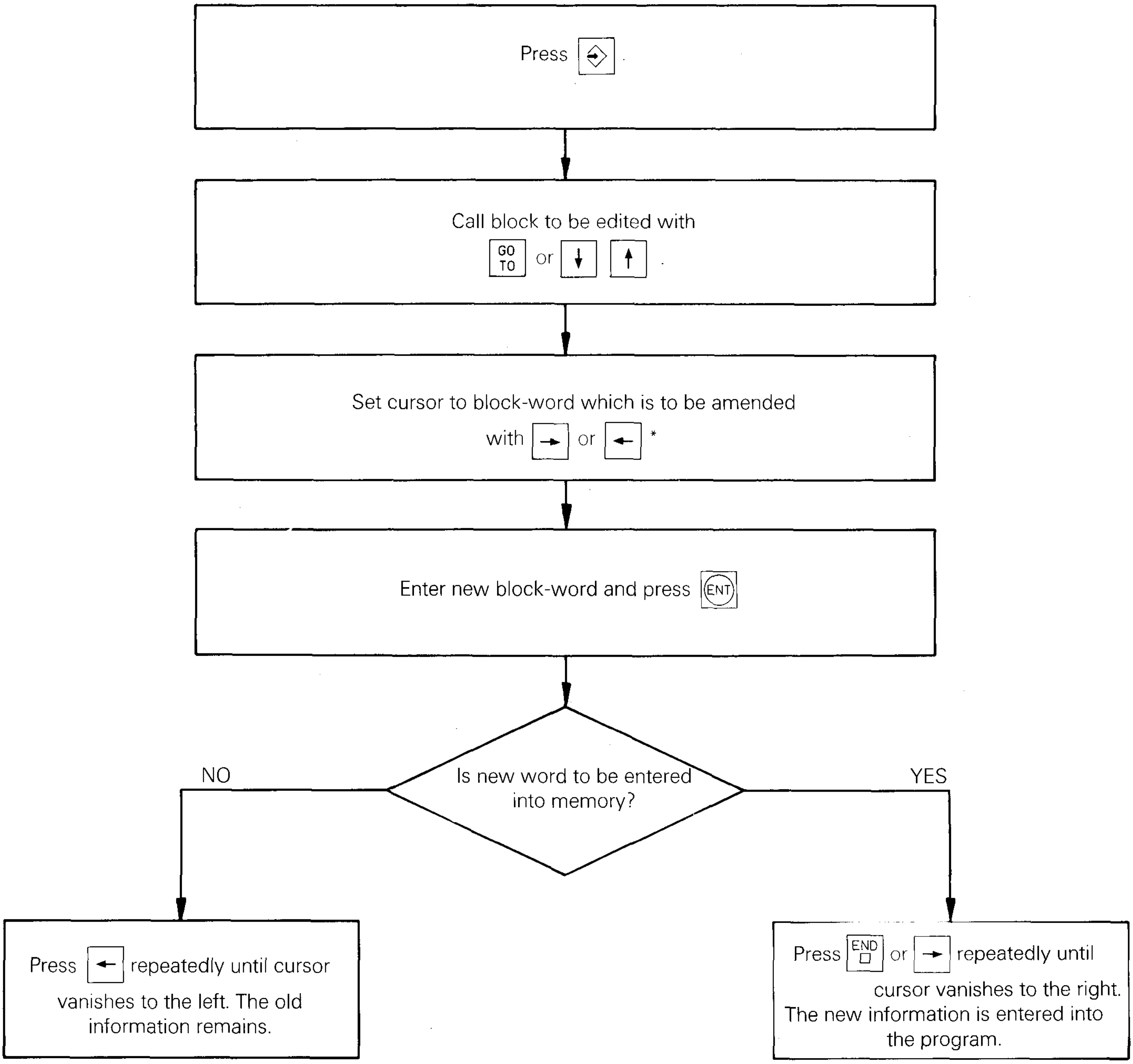
↓

Select block after which new block is to be inserted by pressing [GO TO] or [↓] or [↓].

↓

Enter new program blocks.

M 8.5) Editing within a block



If during the programming of a block the [←] -key is pressed, the word last entered is erased. With this, entry errors can be amended immediately. A block with an entry error therefore, does not have to be completely entered first and then edited afterwards.

*The setting of the cursor is initiated with the [→] -key!

M 8.6) Search routines for locating certain blocks

Press [⇨].

↓

Set cursor to particular word (search address) with [→] or [←]-key*.

↓

If [↓] or [↑] is pressed, only those program blocks which also contain the particular search address are displayed, and if necessary, amended.

* The setting of the cursor must be initiated with the [→]-key.

M 8.7) Clearing complete machining program

Press [⇨] and then [CL PGM]
The complete program list with cursor is displayed.

↓

Dialogue question:
CLEAR PROGRAM = ENT/END = NOENT

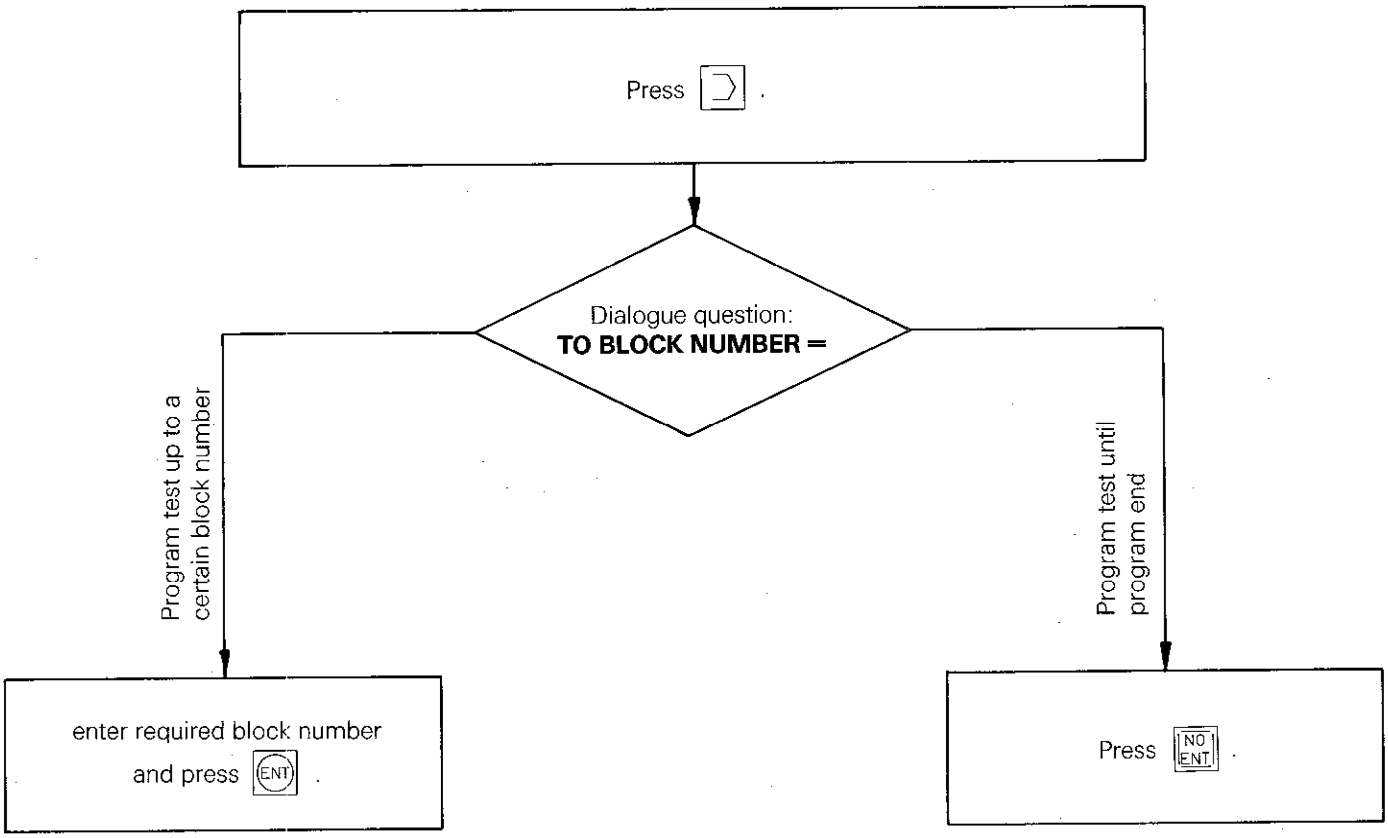
Do not clear program → Press [NO ENT].

Clear program → With [↓], [↑] and [←], [→]-keys, select program to be cleared. → Press [ENT].

By pressing the [CL PGM] and [ENT]-keys only one program is cleared. If several programs or the complete program memory is to be cleared, the keys have to be pressed the corresponding number of times.

M 9) Program test without machine movement

A stored program may be checked without machine movement. The control will display all recognizable errors in plain language dialogue.



Program test is automatically interrupted with a programmed stop, an empty block or fault/error display.

Program test can be terminated at any desired location by pressing the internal STOP -key.

N) Single axis positioning (non-simultaneous)

N 1) Programming single axis positioning blocks via keyboard [X] [Y] [Z] [IV]

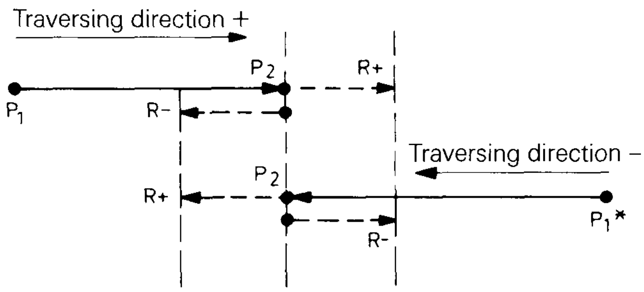
Single axis positioning routines may be programmed as a special case within the linear interpolation mode – as described in section M 3.2 (dialogue initiation with [L])

TNC 150 also enables another method of entering single axis programs by immediately initiating the dialogue on pressing the **axis key** [X], [Y], [Z] or [IV]. This method of entry is used on the TNC 131/135 and 145 controls.

A distinct difference between contour programming (dialogue initiation with [L] or [C]) and single axis programming as described in this section (dialogue initiation via axis keys) is constituted by the **tool radius compensation.**

[R+] must be pressed if the traversing distance is to be **extended** due to tool radius compensation.

[R–] must be pressed if the traversing distance is to be **shortened** due to tool radius compensation.

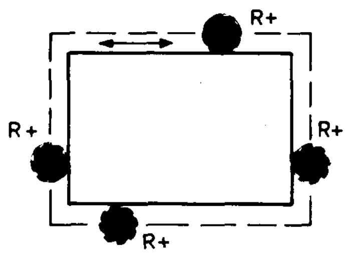


The adjacent sketch indicates how the compensation R+ and R– is implemented for positive and negative directions.

The designations [R+], [R–] are due to the double function of these keys!

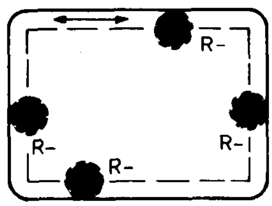
Example of tool radius compensation on an external contour.

R+ ... Traversing distance is **greater** than dimension on drawing.



Example of tool radius compensation on an internal contour.

R– ... Traversing distance is **smaller** than dimension on drawing.



All other programming is as per the procedure initiated with [L].

Dialogue initiation with axis key [X], [Y], [Z] or [IV].

Dialogue question	Response
POSITION VALUE?	.Press [I] if required .Enter numerical value or parameter (see section M 5). .Press [ENT].
TOOL RADIUS COMP. R+/R–/ NO COMP.?	Enter tool radius compensation if required: .Press [R+] or [R–]. .Press [ENT].
FEED RATE? F	Enter feed rate; press [ENT].
AUXILIARY FUNCTION M?	Enter auxiliary function; press [ENT].

Block entry can be terminated by pressing [END] (see section M 3.2.4).
If dialogue questions are responded to with [NO ENT], no data entry takes place; the next dialogue question is displayed.
A dialogue question relating to tool radius compensation is also displayed for the axis which has been allocated to the tool spindle with tool call. Calculation of the radius compensation value does not take place in this axis, no matter whether R+, R– or R0 has been entered.

The positioning block allocates one program block:

X + 46,000
R+ F 60 M 03

In a machining program, single axis positioning blocks which have been initiated via axis keys may not be mixed with blocks which have been initiated with [L], [CC] or [RND].

Example of incorrect programming:

```
L X + 50.000 Y + 20.000
        RR F 100 M
  X + 50.000
        R– F 100 M
L X +180.000 Y + 35.000
        RR F 100 M
```

Exception:
Only with contouring blocks without tool radius compensation **and positioning blocks for the tool axis** is it possible to insert single axis positioning blocks (dialogue initiation [X], [Y], [Z]) into a contour.

Positioning blocks with [IV]-key

The fourth axis can control either a rotary table or a linear axis. This axis is programmed with the [IV]-key.

When responding to dialogue questions, the following should be noted:
.When using the fourth axis for a rotary table:
Entry of nominal position value in (°) and feed rate in (°/min.).
.Tool radius compensation is not calculated in the fourth axis.

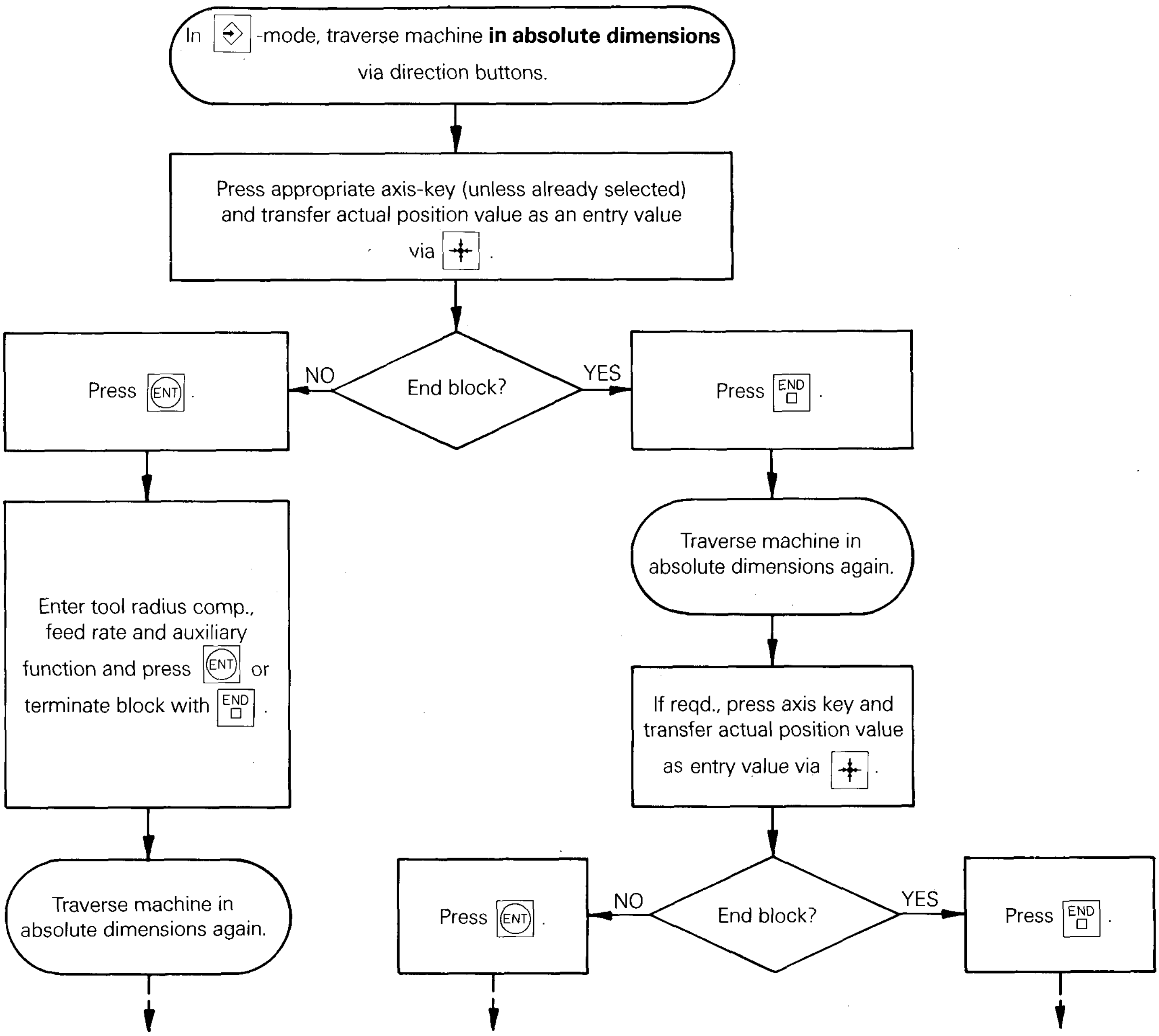
If the function M 94 is programmed within a positioning block for the fourth axis (rotary axis), the display is automatically reduced to a value below 360°.

The positioning block for [IV] requires one program block.

C + 90,000
R0 F 20 M

N 2) Programming with playback-key

The machine is **traversed manually** and the actual position data is programmed as a nominal position value. This method of programming is advisable only for single axis operation. The programming of complex contours using playback is **not possible!**



Programming of tool radius compensation
With playback programming, the machine is traversed manually (handwheel, axis-key) to the actual position which is to be stored. This actual value already contains the length and radius compensation for the tool being used. In the tool definition for this tool No. 1, the values L1 = 0 and R1 = 0 are to be entered and the actual radius R1 of the tool being used is to be noted. Programming of positioning blocks in playback takes place with entry of the appropriate tool radius compensation:
R+, R–, R0.

In the event of a tool break and insertion of a new tool the radius R2 of which differs to the radius R1, only the difference between the two radii has to be entered:

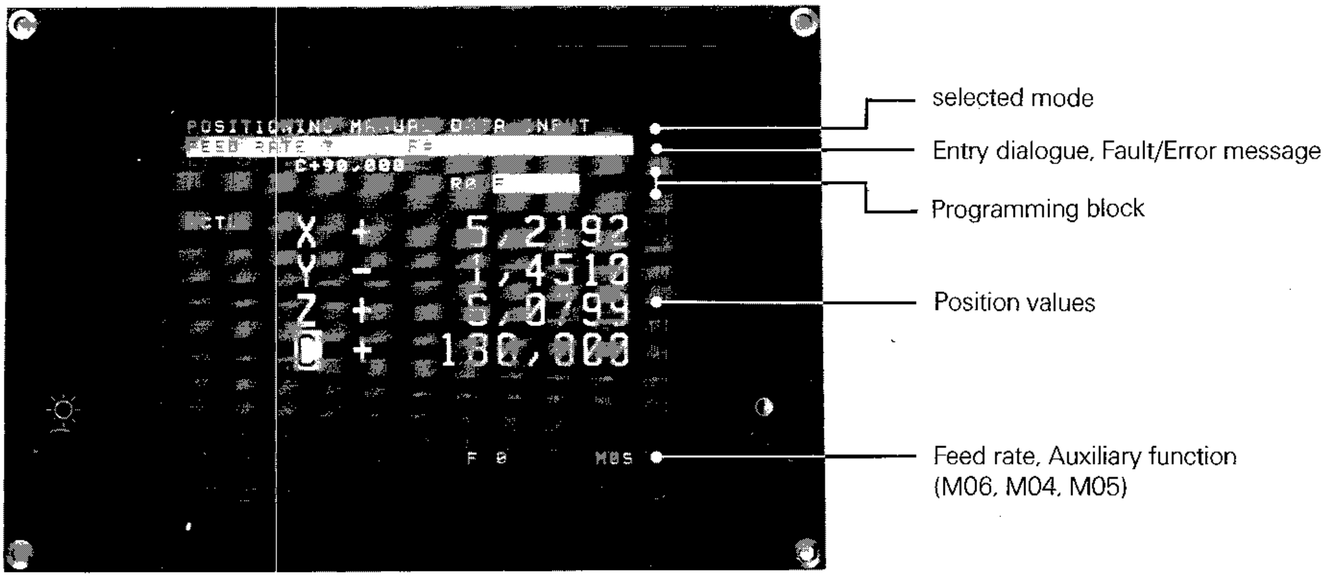
Radius compensation = R2 – R1

This radius compensation value may be **positive or negative** and is to be entered into the tool radius definition for R1 including the calculated arithmetical sign. The tool length compensation should also be entered.

O) Positioning with manual data input (MDI) (single block automatic) [MDI]

In this mode, the entered blocks are executed immediately after entry; the blocks are not stored.

VDU-display:



Every block is executed immediately after entry:

Press [MDI] "positioning with MDI"

↓

Initiate dialogue with [X], [Y], [Z], [IV] or [TOOL CALL].

↓

Enter data and press [ENT].
Dialogue display: **BLOCK COMPLETE**

↓

Press external START-button –
workpiece machining.

The programmed feed rate can be altered either a) via the override potentiometer of the control or
b) via an external potentiometer
depending on how the control has been adapted to the machine by the machine tool manufacturer.

If a block has been programmed incrementally, the block can be started as often as is required by pressing the external (START)-button.

A **tool call** can only be effective when:

.the tool has been previously definied, i.e. the compensation values (length and radius) have already been entered into the **program memory.**

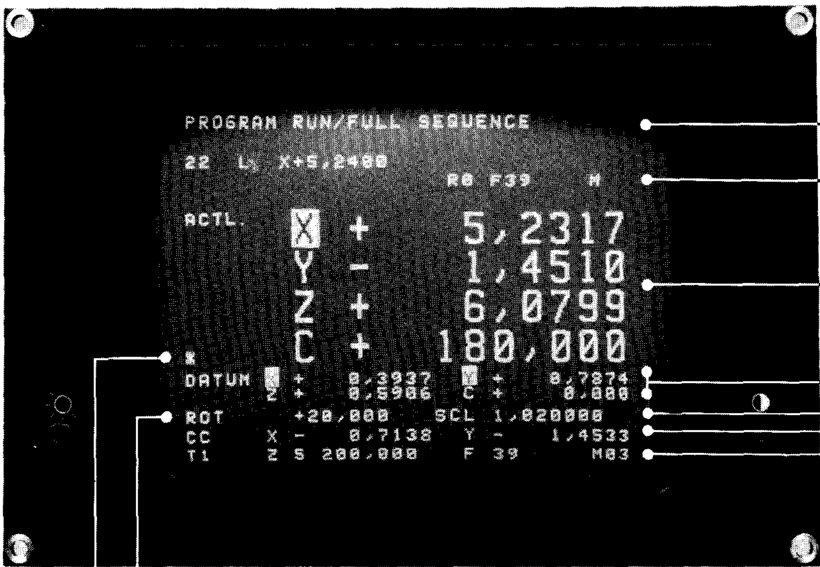
.in the [MDI]-mode, the tool call has been activated with the external (START)-button.

Interruption of a program block is performed as explained in section O 2) for automatic program run with the external (STOP) -button and internal [STOP] -key.

P) Automatic program run

In the operating modes "single block program run" and "automatic program run" stored programs are executed.

VDU-display (large display):

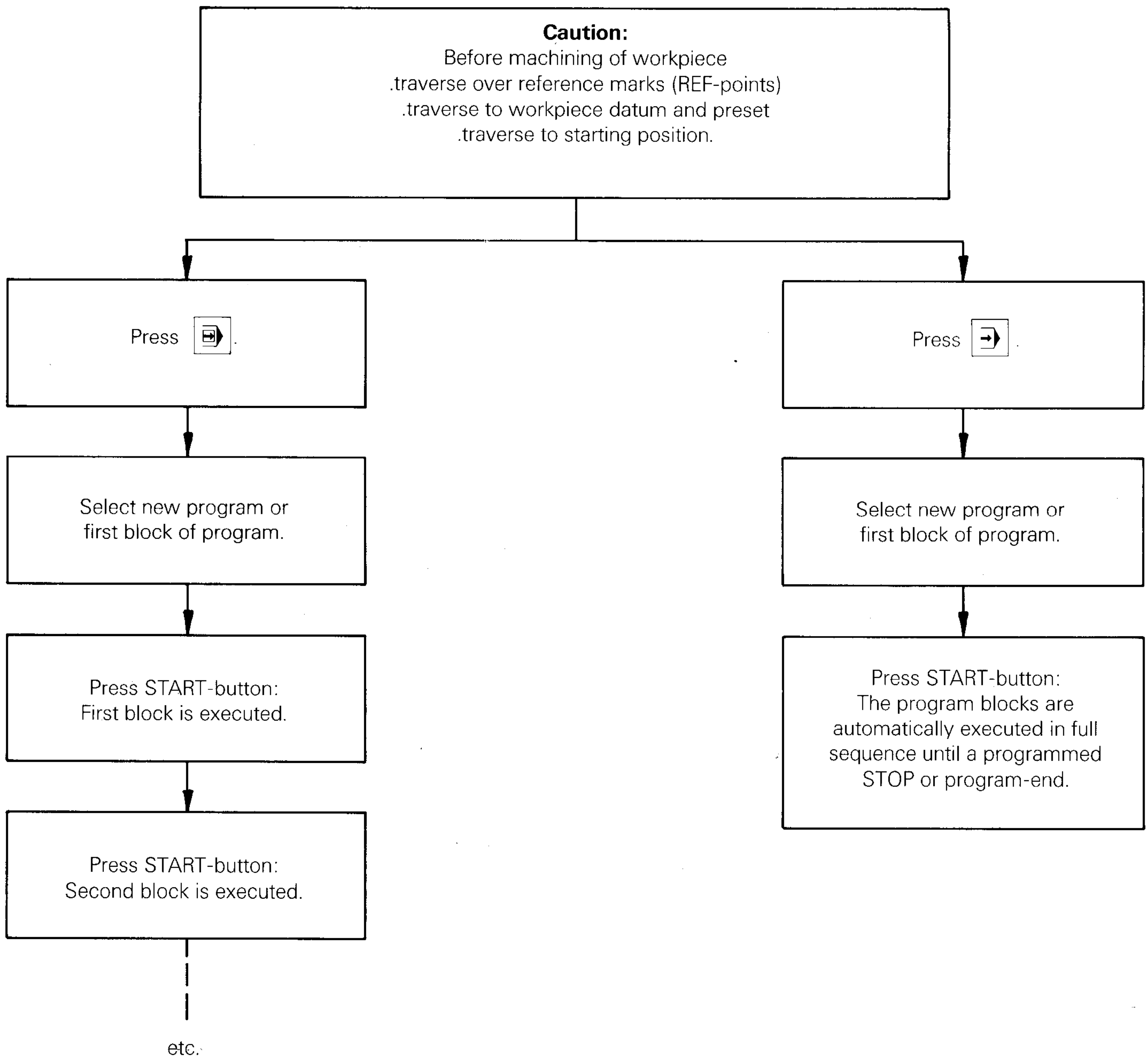


Status display for datum shift and mirror image

The status display for datum shift (see section M 7.2.7) and mirror image (see section M 7.2.8) indicates the number of datum shifts and mirror image which have been called-up:

Display in normal characters: No mirror image
Display inverted with orange background: **mirror image**

P 1) Starting program run

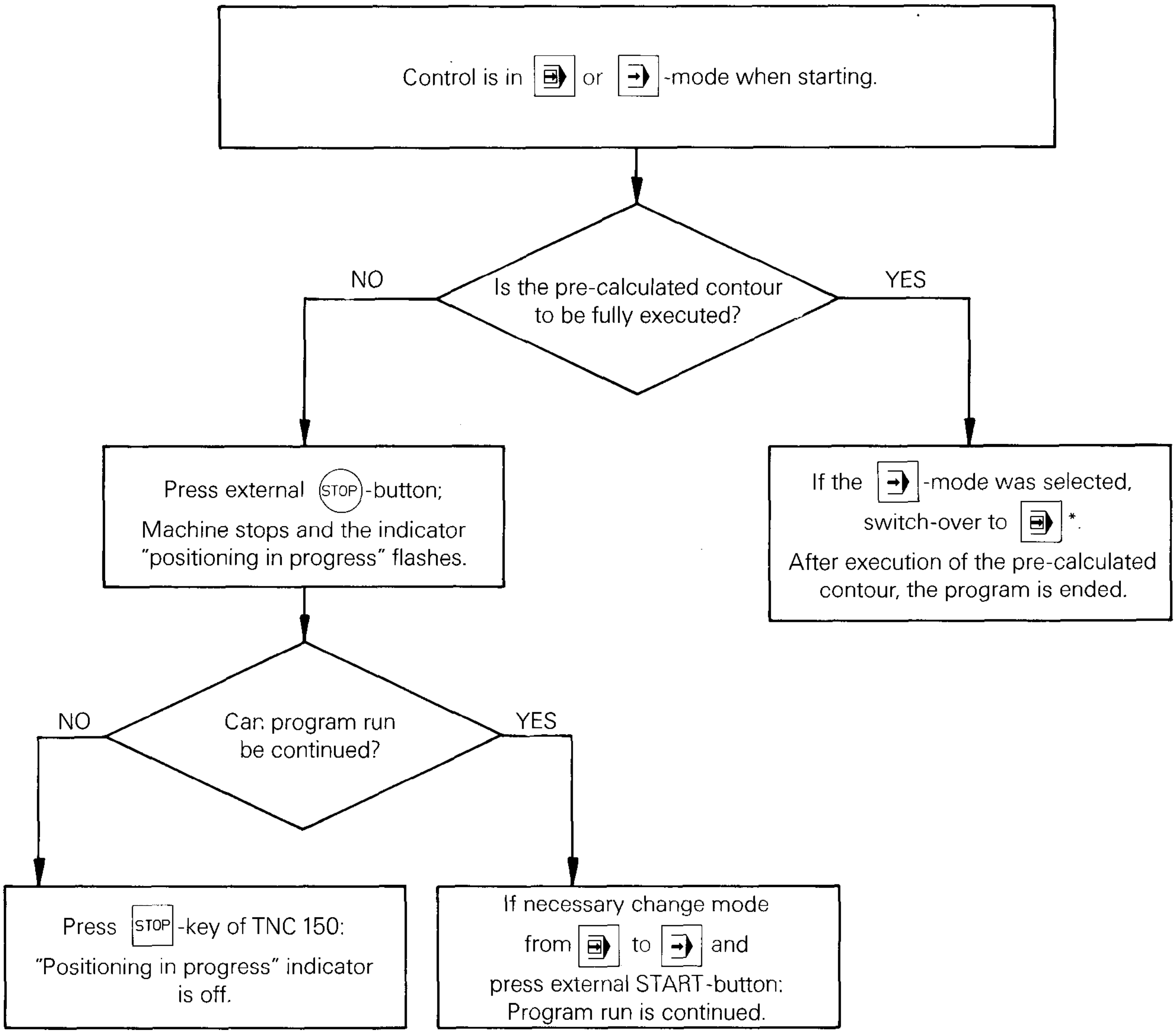


The programmed feed rate can be altered either a) via the override potentiometer of the control or
b) via an external potentiometer
depending on how the control has been adapted to the machine by the machine tool manufacturer.

If in the operating modes "single block" [⇒] or "Automatic" the [Q]-key is pressed after interruption of program run, and a parameter number is keyed-in and entered with [ENT] the value of the parameter is displayed (parameter programming, see section M 5.2)

Paging of the parameter list is performed with the [↓] [↑]-keys.

P 2) Interruption of program run



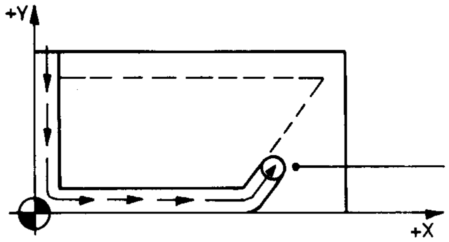
* With a subprogram call-up and program part repeat program run is only terminated after complete execution of the program part which has been called-up or repeated.

P 3) Re-entry into an interrupted program

If automatic program run is interrupted and the operating mode switched to "manual" – e.g. with a tool break or to take a measurement of the work – the control retains the following data:

.the last tool called
.the number of executed mirror images and datum shifts
.the absolute values of the datum shifts in three axes
.the last circle centre CC in absolute dimensions
.the last defined machining cycle
.the current stage with program part repeats
.the return address with subprograms

Interruption of automatic program run and re-entry into interrupted program:



Automatic program run is to be interrupted here, e.g. for tool change.

Press external (STOP)-button and [STOP]-key of TNC (see section P 2).

↓

If tool data (length and radius) remains unchanged:
Read-off position values of X, Y and Z-axes and note down!
Note down current block number!

↓

In operating mode [ ], [ ] or [ ] first traverse tool axis Z and, if necessary, X- and Y-axes to tool-change position and switch off main spindle and coolant.

↓

Insert new tool.

↓

Must tool data be changed?

NO:

Switch on main spindle and coolant

↓

In operating mode [ ], first traverse the X- and Y-axis and finally the Z-axis to the previous departure position.

YES:

If the tool data is to be amended: Address appropriate tool definition in [ ] -mode and enter new total data.

↓

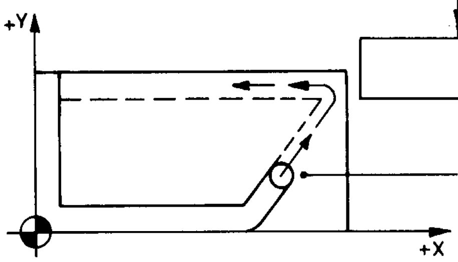
Call-up new tool in [ ] -mode

↓

In operating mode [ ] or [ ]; switch on main spindle and coolant again. Traverse to contour with new tool and touch workpiece.

↓

Start program run in [ ] or [ ] -mode.



Program run is continued with complete tool compensation.

The following points must be remembered when interrupting program run:

a) If an interruption takes place within a subprogram or program part repeat, and a block is then addressed with the [GO TO] -key, the countdown for the program part repeats is reset and the return jump address for the subprogram is erased. If the countdown or the return jump address is to be retained, program blocks may only be selected with the [↓] and [↑] -keys.

b) If, after termination of program run, the program is "paged" with the [↓] and [↑] -keys and a re-start does not take place at the block which was interrupted, the following error is displayed:

SELECTED BLOCK NOT ADDRESSED

Program run can be continued:

.by addressing the block which was interrupted with the [↓], [↑] -keys.
.by addressing any desired block with [GO TO] ; however, the countdown for program part repeats is reset or the return jump address for a subprogram is erased.

c) If, after interruption of program run, a block is inserted or erased, the last cycle definition and the corresponding display on the VDU-screen is erased.
With a new program run-start, the desired cycle definition must be executed before the next cycle call, otherwise the following error is displayed:

CYCL INCOMPLETE

Cycle definition selection must take place with [GO TO] , however, the countdown for program part repeats and the return jump address for a subprogram is erased.

d) If,
.with an amended incremental block or
.with linear block with one co-ordinate or
.within a cycle
program run is interrupted and re-started, the following error is displayed:

PROGRAM START UNDEFINED

The program must be amended accordingly or the previous block is to be addressed via [GO TO] – with this however, the countdown for program point repeats and the return jump address for a subprogram is erased.

e) If, when returning to the contour, the tool is not located in the position which was reached when leaving – the TNC considers the actual position value for program run re-start as amended. When returning to the contour, proceed as explained in section M 3.2.6.1 (case 2).

P 4) Positioning to program without tool

For checking a program without tool, all tool call blocks within the program are to be amended to number 0 (= no tool). It is advantageous to note down the tool number of each tool call (or note down the number of one tool call and then change the other tool calls by means of the search routine facility).

When running the program with the machine, the position displays always show the absolute values of the programmed positions (drawing dimensions) without tool radius compensation.

After this check, all tool call blocks are to be reverted to the appropriate tool numbers!

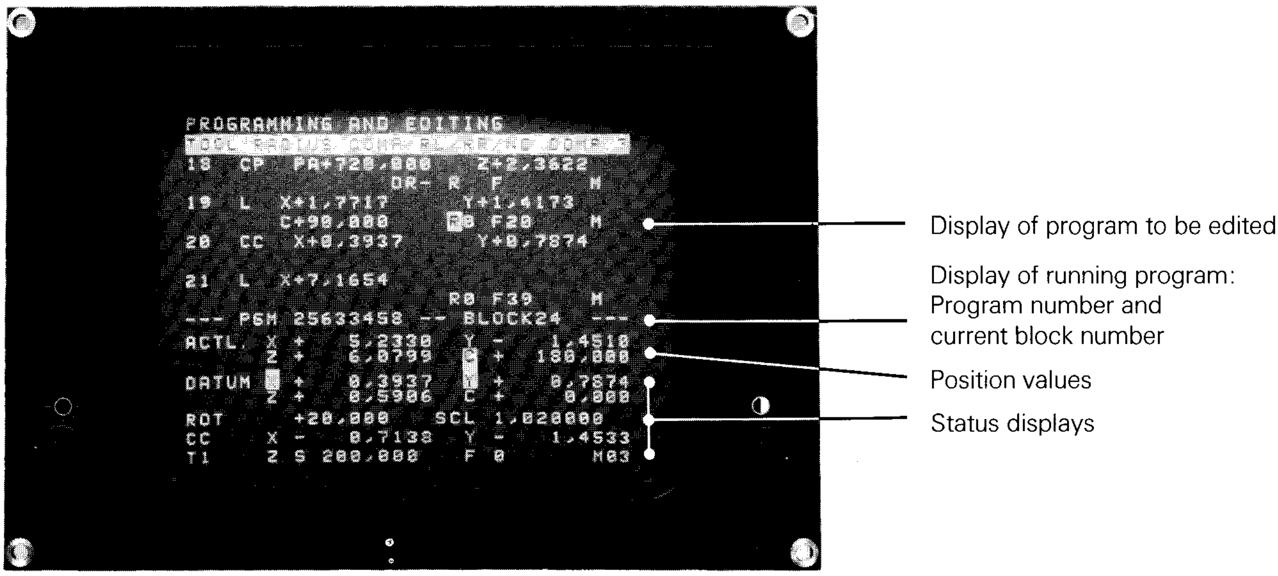
P 5) Program run with simultaneous programming and editing

The 150 permits the execution of a machining program with simultaneous programming amd editing of a new program.

A program is called up and started in the "program run" mode. The operating mode is then edited over to "programming" and the [PGM NR] -key is pressed.

A new program number or a stored program – which is not being machined – can now be called up.

The VDU-display indicates which program has been started and which block is currently being executed.



Q) External data input and output [EXT]

Q 1) Interface

The TNC 150 is equipped with a standard interface connection according to

CCITT-recommendation V.24
or EIA-standard RS-232-C

This data input/output interface permits connection of the HEIDENHAIN-magnetic tape cassette units ME 101 (portable unit) or ME 102 (pendant type).

However, other programming or peripheral units (e. g. tape punching/reading unit, telex, printer) which have V.24-compatibility may be also connected to the TNC 150.

Peripheral units with a 20 mA-interface may not be connected!

Q 2) HEIDENHAIN-magnetic tape cassette units ME 101 and ME 102

HEIDENHAIN supplies special magnetic tape cassette units for external program storage.

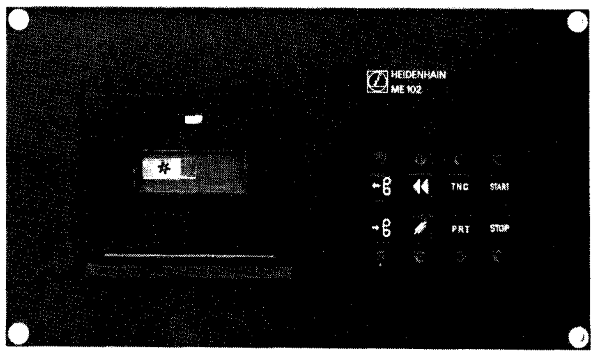
ME 101 – portable unit for alternate use on several machines.
ME 102 – pendant type housing for direct installation into machine control panel.

ME 101 and ME 102 are both fitted with 2 data input and output connectors:

In addition to the TNC 150, a commercially available peripheral unit can be connected to the V.24 (RS-232-C)-output of the ME-unit (connector PRT).

The data transfer rate between control and ME is fixed at 2400 Baud. The transfer rate between the ME and a peripheral unit can be adapted by means of a stepping switch (110, 150, 300, 600, 1200, 2400 Baud).
Exact details of ME operation are given in the ME 101 and ME 102 operating manuals.

ME 101



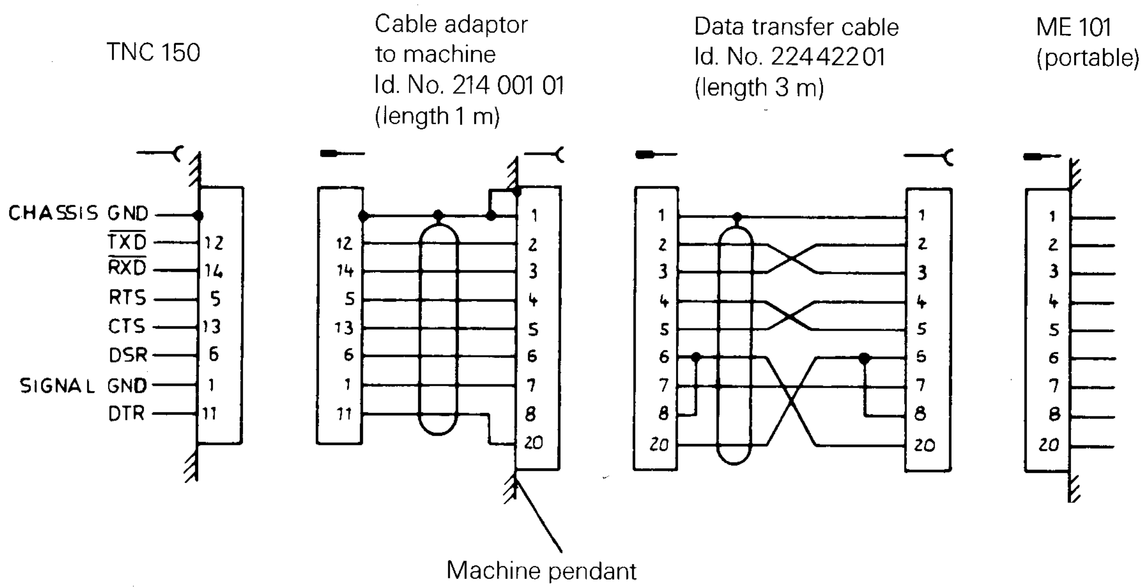
ME 102

Q 3) Connecting cables

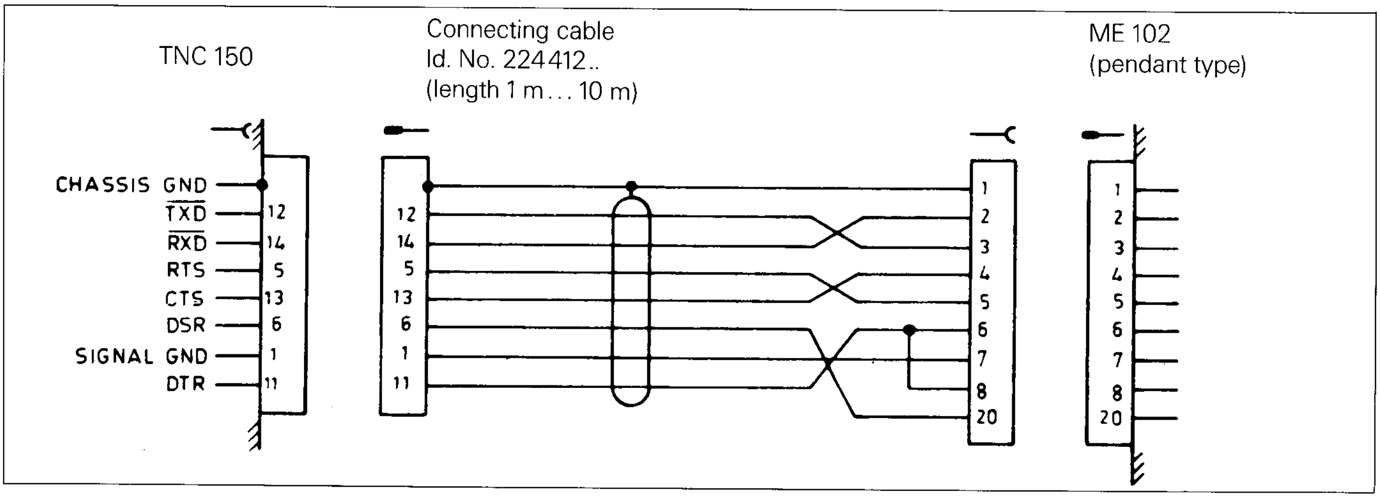
HEIDENHAIN supplies the following connecting cables:

a) Cable adapter for extension of V.24-connection of TNC – to – machine housing in which the TNC is installed.

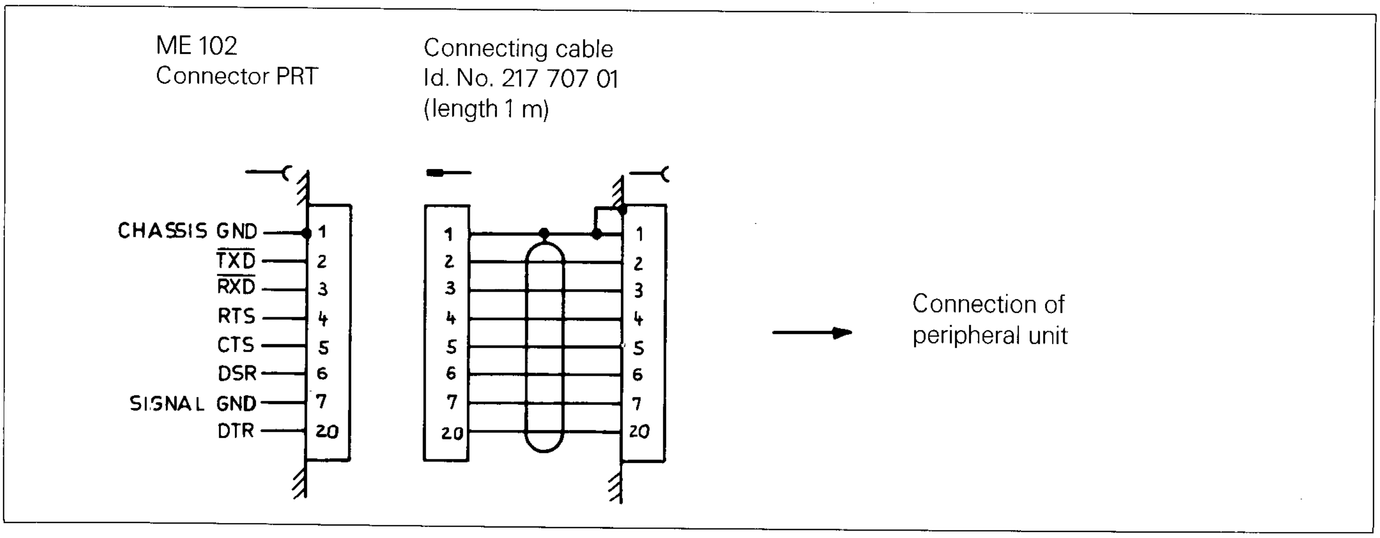
b) Data transfer cable for connection to ME 101.



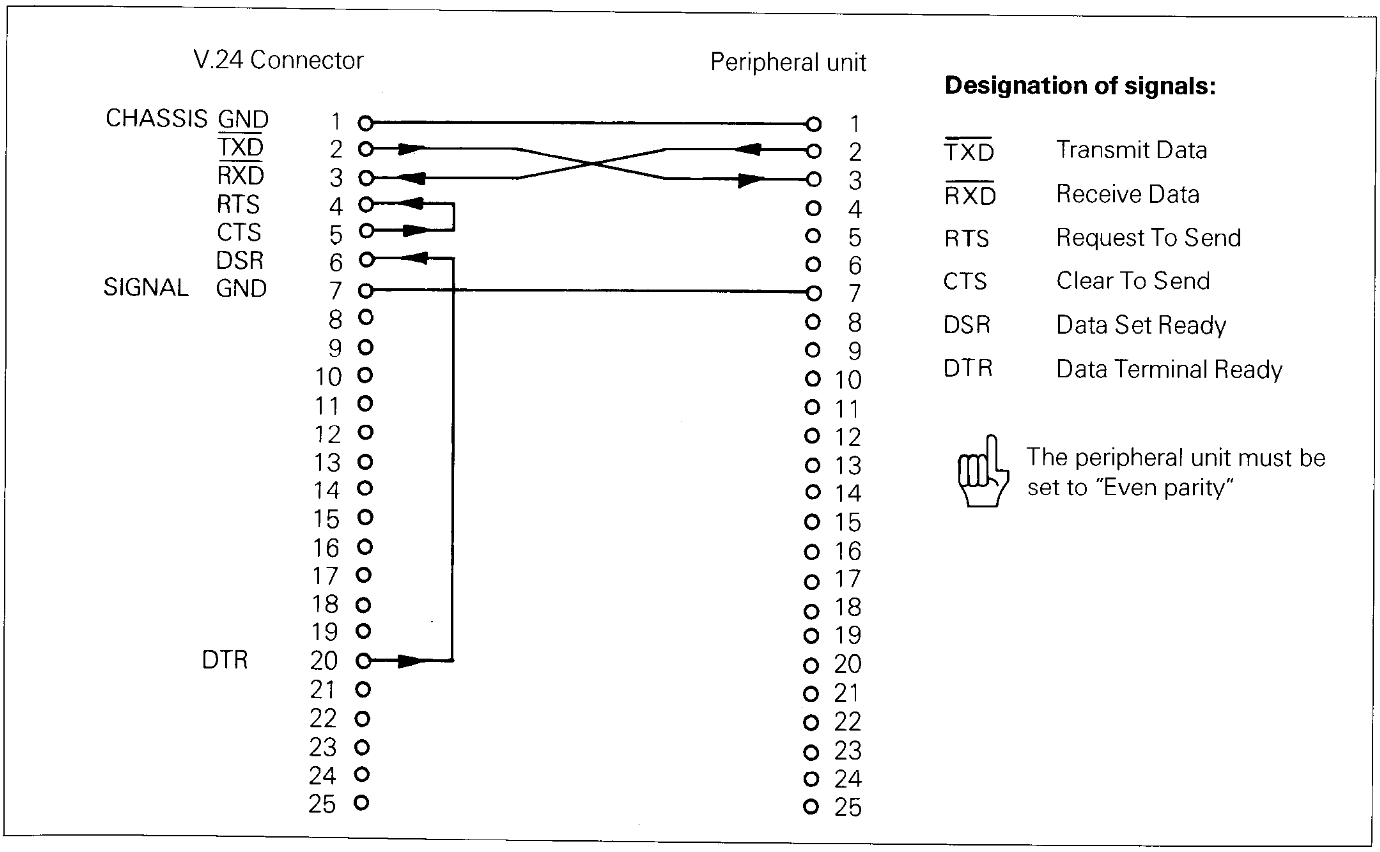
c) Connecting cable for direct connection of ME 102 (pendant type) to TNC 150.



d) Connecting cable for extension of the V.24 connection of the ME 102 to the housing in which the control and the ME 102 are installed (machine control panel).



The following **connector layout** has proved favourable for the **connection** of a commercially available **peripheral unit** (e. g. printer with tape reader and puncher).



Q 4) Entry of Baud rate

The transfer rate for the V.24-interface of the TNC 150 is automatically set to 2400 Baud (adapted to the HEIDENHAIN Magnetic Tape Cassette Units ME 101/ME 102).

If the TNC 150 is to be connected to a peripheral unit with another Baud rate (without intermediate connection of the ME), the Baud rate may be altered via the MOD-function (see section J 2.4).

The following transfer rates are possible: 110, 150, 300, 600, 1200 or 2400 Baud.

Control switch-off with discharged or missing buffer batteries automatically erases the programmed Baud rate. A control re-start then automatically sets the value to 2400 Baud.

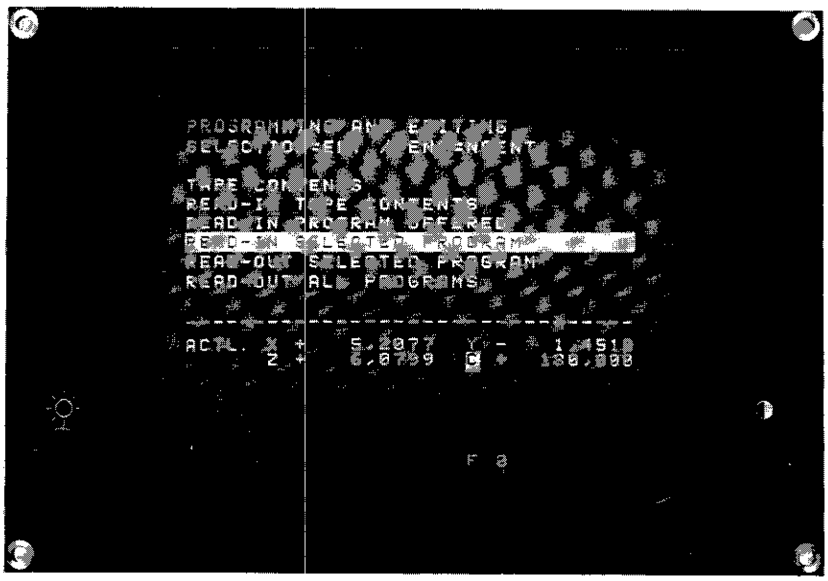
Q 5) Operating procedure for data transfer

Data output on printer, tape puncher or magnetic tape cassette units ME 101/ME 102.

The TNC 150 program organisation facility enables up to 24 different programs to be stored on one side of an M 101/ME 102 – cassette. As required, programs can be called up directly and transferred into the TNC 150.

If a program which exceeds the magnetic tape capacity is being read-in or read-out, the dialogue message **EXCHANGE CASSETTE – ME START** appears. After changing the cassette and re-starting of ME, the remaining program blocks are read-in or read-out.

If the [EXT]-key is pressed in the "programming"-mode, the following modes are displayed for selection on the VDU.



The required operating mode can be selected via the [↓], [↑]-keys. Data transfer is started by pressing the [ENT]-key.

By pressing the [NO ENT]-key, the operating mode for external data input/output can be cancelled.

Data transmission which has been already started can be interrupted by pressing [DEL] and the [STOP]-key on the ME-unit.
After interruption of data transmission, the following error is displayed:
ME: PROGRAM INCOMPLETE
After clearing the error display with [CE], the operating mode menu for data transmission is displayed.

When using an older TNC 150 version or a TNC 145 – program (without a number);

Enter new PGM-NR (only with first and last block) and select "editing".
Finally, "Read-in tape contents" as explained in section Q 5.2.1).

Q 5.1) Tape contents

The **TAPE CONTENTS** mode indicates which programs are stored on a cassette.

Insert cassette into ME an press [←8] and [TNC] .

↓

Select [⇨] on TNC .

↓

Press [EXT] and selcet **TAPE CONTENTS**
via the [↓] are [↑] -key.

↓

Now press [ENT] ; the VDU displays
EXTERNAL DATA INPUT
and the magnetic tape cassette is started.

↓

All numbers which are stored in the cassette are displayed in the VDU.
However, the programs have not been transferred to the TNC 150.

↓

Press [NO ENT] ; the **TAPE CONTENTS** mode is cancelled.

Q 5.2) External program input

Programs can be transferred from the ME to the TNC in different ways:

READ-IN TAPE CONTENTS all programs which are stored on the magnetic tape are transferred into the TNC.

READ-IN PROGRAM OFFERED the programs which are stored on the magnetic tape are offered for transfer one after the other.

READ-IN SELECTED PROGRAM a certain program number is entered; the corresponding program is seached for in the ME and finally transferred into the TNC.

If a program number which is already stored in the TNC is entered for transfer from the ME to the TNC, the following dialogue is displayed:

PROGRAM NUMBER ALLOCATED
ERASE = ENT/OVERREAD = NOENT

.Should the program in the TNC be erased?: press [ENT]
.Should the selected program not be transferred form the ME into the TNC?: press [NO ENT].

Q 5.2.1) Read-in of tape contents

With the **READ-IN TAPE CONTENTS** mode, all programs which are stored on the magnetic tape are transferred into the TNC 150.

Insert cassette containing required programs into the ME
Press [←6] and [TNC] on ME .

↓

Select [⇒] on the TNC.

↓

Press [EXT] and select **READ-IN TAPE CONTENTS**
via [↓] or [↑] -key.

↓

Press [ENT], the VDU-display shows
EXTERNAL DATA INPUT
and the magnetic tape is started.

↓

All programs on the tape are transferred into the TNC-memory.
The last program is automatically displayed.

Q 5.2.2) Read-in of program offered

In the **READ-IN PROGRAMM OFFERED** mode certain programs can be called-up from the magnetic tape.

Insert cassette containing required programs into the ME
Press [←8] and [TNC] on ME.

↓

Select [⇨] on TNC.

↓

Press [EXT] and select **READ-IN PROGRAM OFFERED**
via [↓] or [↑] -key.

↓

Press [ENT], the VDU-display shows
EXTERNAL DATA INPUT
and the magnetic tape is started

↓

The VDU-displays the number of the first program of the magntic tape.
The dialogue line displays:
ENTRY = ENT/OVERREAD = NOENT

↓

If the offered program is to be transferred into the TNC-memory
Press [ENT].
If the offered program is not to be transferred into the TNC-memory
Press [NO ENT]

↓

The control indicates all programs which are stored on the magnetic tape one after the other. By pressing [ENT] or [NO ENT] the operator can decide whether each program is to be transferred into the TNC or not.

Q 5.2.3) Read-in of selected program

With the **READ-IN SELECTED PROGRAM** – mode, a certain program on the magnetic tape can be transferred into the TNC.

Insert cassette containing the required program into the TNC
Press [←⊙] and [TNC] on the ME.

↓

Select [⇔] on the TNC.

↓

Press [EXT] and select **READ-IN SELECTED PROGRAM**
via the [↓] or [↑] -key.

↓

Press [ENT] ; the VDU-displays shows
PROGRAM NUMBER =

↓

Enter required Program number and press [ENT] .

↓

The required program is searched for on the
magnetic tape and then transferred to the TNC-memory.

Q 5.3) External program output

Programs can be transferred from the TNC to the ME in two different ways:

.READ-OUT SELECTED PROGRAM	the programs stored in the TNC can be individually selected and output.
.READ-OUT ALL PROGRAMS	all programs stored in the TNC are output.

Q 5.3.1) Output of selected program

In the **READ-OUT SELECTED PROGRAM** mode, the programs stored in the TNC can be individually selected and output.

Insert emply cassette (with write release plug) into ME and press [→8] and [TNC] on ME.

↓

Select [⇔] on the TNC.

↓

Press [EXT] and select **READ-OUT SELECTED PROGRAM** via the [↓] or [↑] -key.

↓

Press [ENT] ; the VDU-display shows

EXTERNAL DTA OUTPUT

After a short formand winding cycle of the tape, the VDU-diaplay indicates all stored program numbers. The dialogue line displays:

OUTPUT=ENT/END=NOENT

↓

A program number can be selected with [↓] , [↑] and the cursor keys [←], [→]. The selected program is transferred by pressing [ENT].

The operating mode is cancelled by pressing [NO ENT].

Q 5.3.2) Output of all programs

In the operating mode **READ-OUT ALL PROGRAMS** all programs stored in the TNC are transferred to the ME.

Insert empty cassette (with write release plug) into ME and press [→B] and [TNC] on ME.

↓

Select [⇔] on TNC.

↓

Press [EXT] and select **READ-OUT ALL PROGRAMS** via [↓] or [↑] -key.

↓

Press [ENT], the VDU-display shows **EXTERNAL DATA OUTPUT** and data transfer begins.

Q 6) External programming at a terminal

Whilst developing the TNC 150, a great deal of emphasis was made on operator convenience. For this reason, programming format purposely deviates from programming standards which were originally devised for program input via punched tape (e. g. G-functions do not have to be programmed).
However, programs can be prepared externally e. g. on a terminal with tape puncher.

The following points must be observed:

a) A program must be commenced with the signals CR (carriage return) and LF (line feed). Both signs must be located **before** the first block, otherwise this will be ignored with program entry.

b) Each program block must be completed with CR, or LF or FF.

c) ETX (Control C) is to be entered after the last program block (or a random ASCII-character, depending on the machine parameter entered).

d) The number of spaces between the signs is optional.

e) In order to recognize data-transfer errors, the TNC 150 checks for "even parity". The external programming unit must therefore be set to "even parity".

Further information concerning the V.24 interface and external programming are given in the following manuals: "Information on V.24 Data Transfer Connection".

R) Programming of machine parameters

Machine parameters are determined by the machine tool manufacturer and entered into the control during the initial setting-up procedure via an external data medium (ME/cassette containing machine parameters) or via key-in. After interruption of power with empty or missing batteries, the control asks for the machine parameters which have to be re-entered either manually or by using the HEIDENHAIN magnetic tape cassette unit as per the checklist below on page 119.

R 1) List of machine parameters

Code number	Entry value (to be filled in by machine tool manufacturer)
MP 00 MP 01	
MP 02 MP 03	
MP 04 MP 05	
MP 06 MP 07	
MP 08 MP 09	
MP 10 MP 11	
MP 12 MP 13	
MP 14 MP 15	
MP 16 MP 17	
MP 18 MP 19	
MP 20 MP 21	
MP 22 MP 23	
MP 24 MP 25	
MP 26 MP 27	
MP 28 MP 29	
MP 30 MP 31	
MP 32 MP 33	
MP 34 MP 35	
MP 36 MP 37	
MP 38 MP 39	
MP 40 MP 41	
MP 42 MP 43	
MP 44 MP 45	
MP 46 MP 47	
MP 48 MP 49	
MP 50 MP 51	
MP 52 MP 53	
MP 54 MP 55	
MP 56 MP 57	
MP 58 MP 59	
MP 60 MP 61	
MP 62 MP 63	
MP 64 MP 65	
MP 66 MP 67	
MP 68 MP 69	
MP 70 MP 71	

Code number	Entry value (to be filled in by machine tool manufacturer)
MP 72 MP 73	
MP 74 MP 75	
MP 76 MP 77	
MP 78 MP 79	
MP 80 MP 81	
MP 82 MP 83	
MP 84 MP 85	
MP 86 MP 87	
MP 88 MP 89	
MP 90 MP 91	
MP 92 MP 93	
MP 94 MP 95	
MP 96 MP 97	
MP 98 MP 99	
MP 100 MP 101	
MP 102 MP 103	
MP 104 MP 105	
MP 106 MP 107	
MP 108 MP 109	
MP 110 MP 111	
MP 112 MP 113	
MP 114 MP 115	
MP 116 MP 117	
MP 118 MP 119	
MP 120 MP 121	
MP 122 MP 123	
MP 124 MP 125	
MP 126 MP 127	
MP 128 MP 129	
MP 130 MP 131	
MP 132 MP 133	
MP 134 MP 135	
MP 136 MP 137	
MP 138 MP 139	
MP 140 MP 141	
MP 142 MP 143	

Code number	Entry value (to be filled in by machine tool manufacturer)
MP 144 MP 145	
MP 146 MP 147	
MP 148 MP 149	
MP 150 MP 151	
MP 152 MP 153	
MP 154 MP 155	
MP 156 MP 157	
MP 158 MP 159	
MP 160 MP 161	
MP 162 MP 163	
MP 164 MP 165	
MP 166 MP 167	
MP 168 MP 169	
MP 170 MP 171	
MP 172 MP 173	
MP 174 MP 175	
MP 176 MP 177	
MP 178 MP 179	
MP 180 MP 181	
MP 182 MP 183	
MP 184 MP 185	
MP 186 MP 187	
MP 188 MP 189	
MP 190 MP 191	
MP 192 MP 193	
MP 194 MP 195	
MP 196 MP 197	
MP 198 MP 199	
MP 200 MP 201	
MP 202 MP 203	
MP 204 MP 205	
MP 206 MP 207	
MP 208 MP 209	
MP 210 MP 211	
MP 212 MP 213	
MP 214	

R 2) Entry of machine parameters using a magnetic tape cassette unit ME

Switch on power.
Dialogue display: **OPERATION PARAMETERS ERASED**

↓

Press CE.
Dialogue display: **EXCHANGE BUFFER BATTERY**
MACHINE PARAMETER MP 0XX?
MP 0:

↓

Insert new battery: press CE.
Dialogue display: **MACHINE PARAMETER PROGRAMMING**
MACHINE PARAMETER MP 0XX?
MP 0:

↓

.Connect magnetic tape unit ME (connection TNC)
.Insert cassette and rewind to tape start,
.Select ME-modes ←ß and TNC.

↓

Press TNC-key EXT.
Dialogue display: **MACHINE PARAMETER PROGRAMMING**
EXTERNAL DATA INPUT
MP 0:

↓

After loading machine parameters.
Dialogue display: **POWER INTERRUPTED**

↓

Press CE.
Dialogue display: **PROGRAM MEMORY ERASED**

↓

Press CE.
Dialogue display: **RELAY EXT. DC VOLTAGE MISSING**

↓

Switch on external DC voltage and traverse over reference marks.
The control is now operational.

R 3) Manual entry of machine parameters

Switch on power.
Dialogue display: **OPERATION PARAMETERS ERASED**

↓

Press CE.
Dialogue display: **EXCHANGE BUFFER BATTERY**
MACHINE PARAMETER MP 0XX?
MP 0:

↓

Insert new battery: press CE.
Dialogue display: **MACHINE PARAMETER PROGRAMMING**
MACHINE PARAMETER MP 0XX?
MP 0:

↓

Enter machine parameters as per checklist.
Press ENT after every parameter.

↓

Machine parameter entry completed.
Dialogue display: **POWER INTERRUPTED**

↓

Press CE.
Dialogue display: **PROGRAM MEMORY ERASED**

↓

Press CE.
Dialogue display: **RELAY EXT. DC VOLTAGE MISSING**

↓

Switch on external DC voltage and traverse over reference marks.
The control is now operational.

S) Typical operating errors and fault/error messages.

The TNC 150 indicates programming and operating errors in plain language dialogue. In most cases, the cause of error can be found by means of these messages. However, we would like to give some hints concerning a few typical errors.

Error	**Cause of error and remedy**
Control voltage cannot be switched on	.Emergency stop buttons was pressed: Release button .One axis is located on emergency stop limit switch: Back-off axis
VDU-screen is dark	Potentiometer for brightress is turned down: Set potentiometer to required brightness
VDU-screen only shows a portion of the data	Potentiometer for contrast is turned down. Set potentiometer to required contrast
Program run cannot be started	Feed rate override is set to 0: Turn override to required setting
Dialogue display: **BUTTON NON-FUNCTIONAL**	.The pressing of the key last pressed is not permitted. .Same key pressed severeal times
Dialogue display: **PROGRAM START UNDEFINDED**	Tool radius compensation or feed rate was not defined in the first block of machining program.
Dialogue display: **SPINDLE?**	Call-up of a cycle without M03 or M04

T) Technical specifications

Control versions

TNC 150 with interface for external machine PLC

Transducer inputs: sinusoidal signals
TNC 150 B
TNC 150 F (without 3D-movement)

Transducer inputs: square wave signals
TNC 150 BR
TNC 150 FR (without 3D-movement)

TNC 150 with PLC-board(s)

Transducer inputs: sinusoidal signals
TNC 150 Q
TNC 150 W (without 3D-movement)

Transducer inputs: square wave signals
TNC 150 QR
TNC 150 WR (without 3D-movement)

T 1) Technical specifications, General

Control type	Shop-floor-programmable contouring control for 4 axes Linear interpolation in 3 out of 4 axes. Circular interpolation in 2 out of 4 axes, Helical interpolation in 3 out of 4 axes. mm/inch instant conversion for entry values and displays Entry step up to 0.001 mm or 0.0001 inch or 0.001° Display step 0.005 mm or 0.0002 inch or optionally 0.001 mm or 0.0001 inch
Operator-prompting and displays	Visual display screen (9 inch or 12 inch) with max. 18 x 32 alphanumeric characters: Plain language dialogue and fault/error indication (in various languages); Display of current program block including previous block and two successive blocks Actual position/Nominal position/Target distance/Trailing error (lag) display and status display for all important program data
Program memory	Buffered semiconductor store for 24 programs with a total of 1200 program blocks
Operating modes	**Manual operation:** the control operates as a digital readout **Automatic positioning with MDI:** positioning block is keyed-in without entry into memory and immediately positioned **Automatic program run in single blocks:** block-by-block positioning with individual press of button **Automatic:** after press of button, complete run of program sequence until "programmed STOP" or program end **Programming:** a) with linear or circular interpolation: Manually to program sheet or workpiece drawing or externally via the V.24/ CRS-232-C data transfer connection (e.g. via Magnetic Tape Cassette Unit ME 101/ME 102 from HEIDENHAIN, or other compatible peripheral unit) b) with single axis operation: additionally by entering actual position data (actual values) from position display (playback) during conventional manual machining Supplementary operating modes mm/inch, Actual position/Nominal position/Target distance/Trailing error (lag) – display, Baud rate, Working range, Vacant blocks, NC-Software number, PLC-Software number, Code number, Fourth axis on/off
Programmable functions	Nominal position values – (absolute or incremental dimensions) entry in Cartesian co-ordinates or in polar co-ordinates Tool length and radius compensation Rounding of corners Tangential approach and departure of contours Spindle speeds Feed rate Rapid traverse Subprograms, Program part repeats Canned cycles for: Pecking, Tapping, Slot milling, Rectangular pocket milling, Circular pocket, Dwell time, Mirror image, Datum shift, Co-ordinate rotation, Scaling Auxiliary functions M Program stop
Parameter programming	Mathematical functions (=, +, −, x, : , sine, cosine, $\sqrt{\ }$, $\sqrt{a^2 + b^2}$) Parameter comparisons (=, =, >, <)
Program editing	Through editing of block-word information, inserting of program blocks, deletion of program blocks; Search routines or finding blocks with common criteria
Monitoring system	The control monitors the functioning of important electronic subassemblies including positioning system, transducers and important machine functions. If a fault is discovered via this monitoring system, it is indicated in plain language on the visual display screen (VDU) and the machine emergency stop is activated.
Program run continuation after interruption	The control simplifies continuation of program run by storing all important program data

Reference mark evaluation	After a power failure, automatic re-generation of datum value by traversing over transducer reference marks
Max. traversing distance	+/− 30000.000 mm or 1181.1024 inches
Max. traversing speed	15999 mm/min. or 629.9 inches/min.
Feed rate and spindle override	Two potentiometers on TNC 150-control panel
Transducers	HEIDENHAIN incremental linear transducers or rotary encoders Grating pitch 0.02 mm or 0.01 mm
Limit switches	Software-controlled limit switches for axis movements (X+, Y−, Y+, Y−, Z+, Z− and IV+, IV−)
Control inputs (TNC 150 B/150 Q with standard PLC-program)	Transducers X, Y, Z, IV 1 electronic handwheel Start, Stop, Rapid traverse Feedback signal: "Auxiliary function completed" Feed rate release Manual activation (opens positioning loop) Feedback signal; emergency stop-supervision Reference end position X, Y, Z, IV Reference pulse suppressor X, Y, Z, IV Direction buttons X, Y, Z, IV External feed rate potentiometer
Control outputs (TNC 150 B/TNC 150 Q with standard PLC-program)	1 analogue output each for X, Y, Z, IV, S Axis release, X, Y, Z, IV Control in operation M-strobe signal S-strobe signal T-strobe signal 8 outputs for M-, S- and T-functions coded "Coolant off" "Coolant on" "Spindle counter-clockwise" "Spindle halt" "Spindle clockwise" Spindle lock on Control in automatic operating mode Emergency stop
Integrated PLC Control version TNC 150 Q	1000 user-markers (without power failure protection) 1000 user-markers (with power failure protection) 1024 fixed allocated markers 63 (+63) inputs (24 V =, ca. 10 mA) 31 (+31) outputs (24 V =, ca. 1.2 A) 16 counters 32 timers External power supply for PLC: 24 V = + 10 %/− 15 % max. 40 A (depending on outputs connected)
Mains power supply	Selectable 100/120/140/200/220/240 V + 10 %/− 15 %, 48 ... 62 Hz
Power consumption	ca. 60 W (9" VDU) or (12" VDU)
Ambient temperature	Operation 0 ... + 45° C (+ 32 ... + 113° F) Storage − 30 ... + 70° C (− 22 ... + 158° F)
Weight	Control: 11.5 kg 9" Visual display unit: 6.8 kg 12" Visual display unit: 10 kg PLC-power board: 1.2 kg (TNC 150 Q)

T 2) Transducers

The TNC 150-control regulates the actual position with a step of 0.001 mm. It subdivides the grating pitch of the linear transducers 20x or 10x. *Incremental linear transducers with 20µm or 10µm grating pitch (constant) are to be used such as:*

.LS 107 (measuring lengths 240 mm up to 3040 mm)
.LS 703 (measuring lengths 170 mm up to 3040 mm)
.LS 903 (measuring lengths 70 mm up to 1240 mm)
.LID 300, LID 310 (measuring lengths 50 mm up to 3000 mm).

For angular measurement the incremental rotary encoders ROD 250 and ROD 700 with 18000 or 36000 lines are available.

If the accuracy requirements are justified, indirect measurement may be performed with a rotary encoder ROD 450 which is connected to the machine leadscrew. The required number of lines is calculated with the following formula:

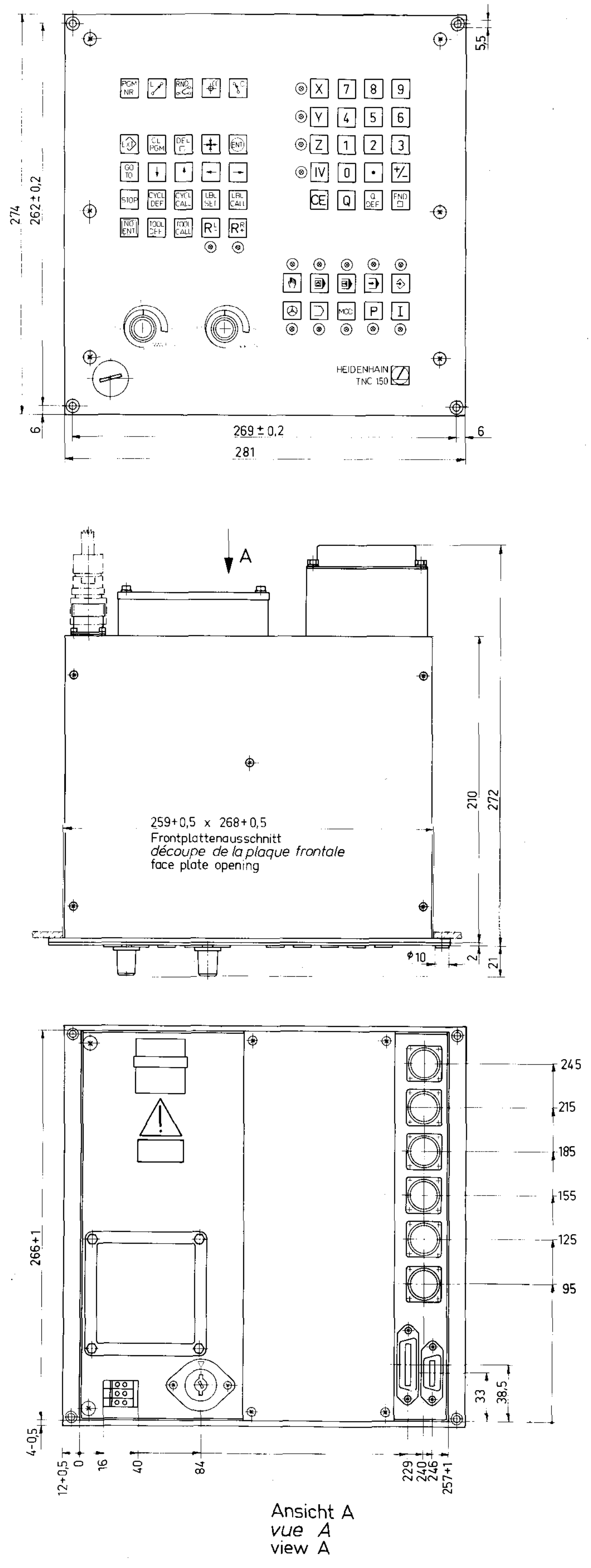
Lines/revolution = 50 x leadscrew pitch (in mm).

Since the cable length between the linear transducer and the TNC 150 must not exceed **20 m, a special version TNC 150 R** has been developed for larger cable lengths between transducer and control. This control version has inputs for transducers with square wave signals and can therefore only be installed in conjunction with an external pulse shaping electronics unit EXE. The output signal of the EXE is evaluated by 2x or 4x within the control.

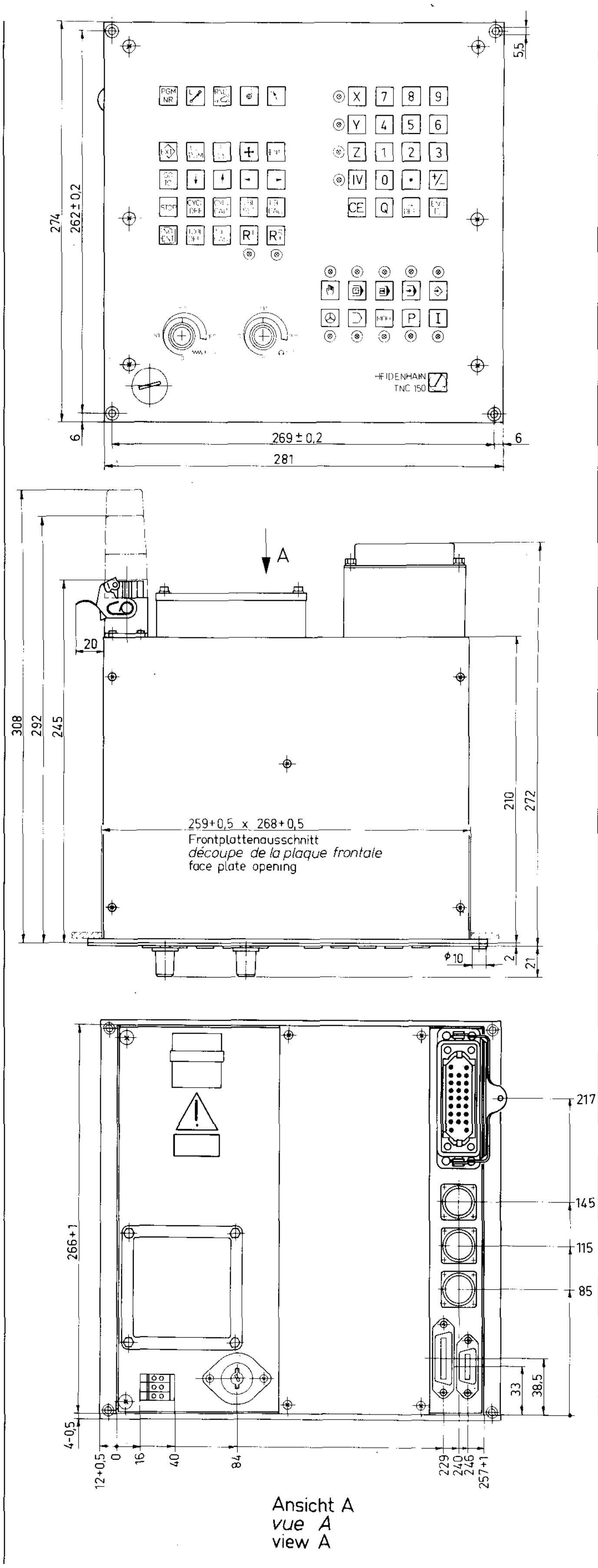
The max. cable length between transducer and EXE is 20 m. The max. cable length between EXE and TNC is 50 m. The total cable length is therefore 70 m.

For direct length measurement the transducer type LB 326 (Measuring length approx. 30 m, grating pitch 0.1 mm) can be used together with an EXE 829 (3-axis input and 25x interpolation).

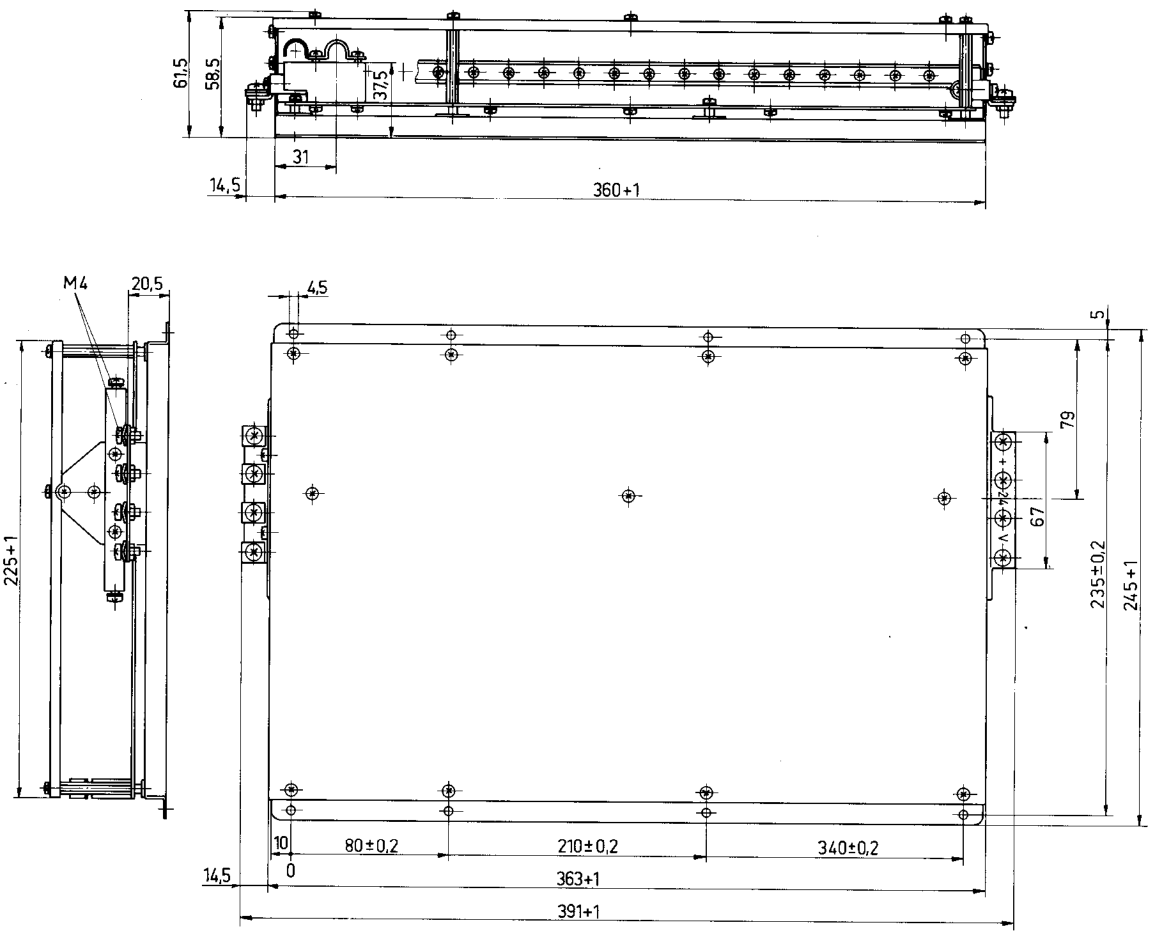
TNC 150 B/F
TNC 150 Q/W



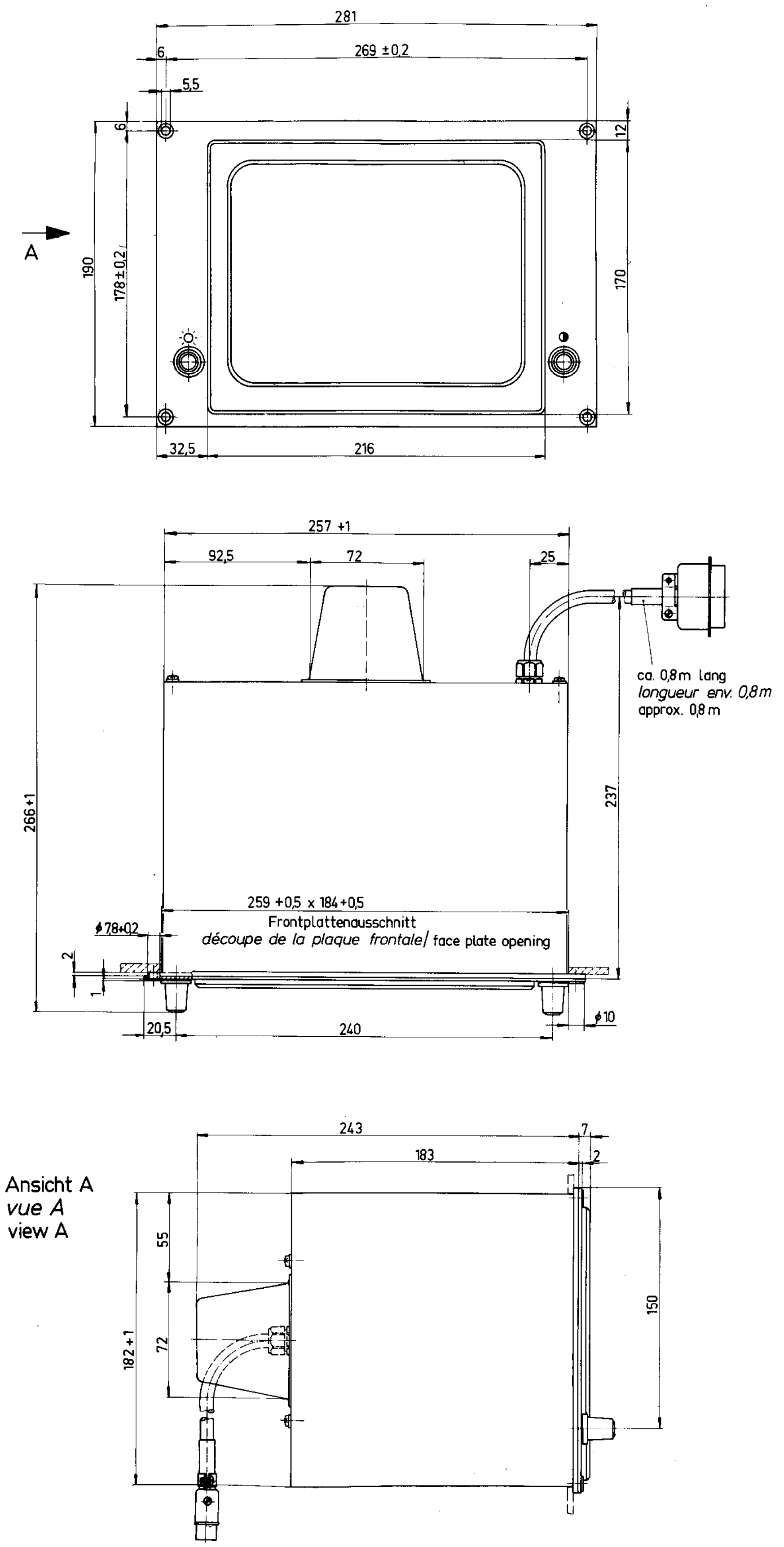
TNC 150 BR/FR
TNC 150 QR/WR



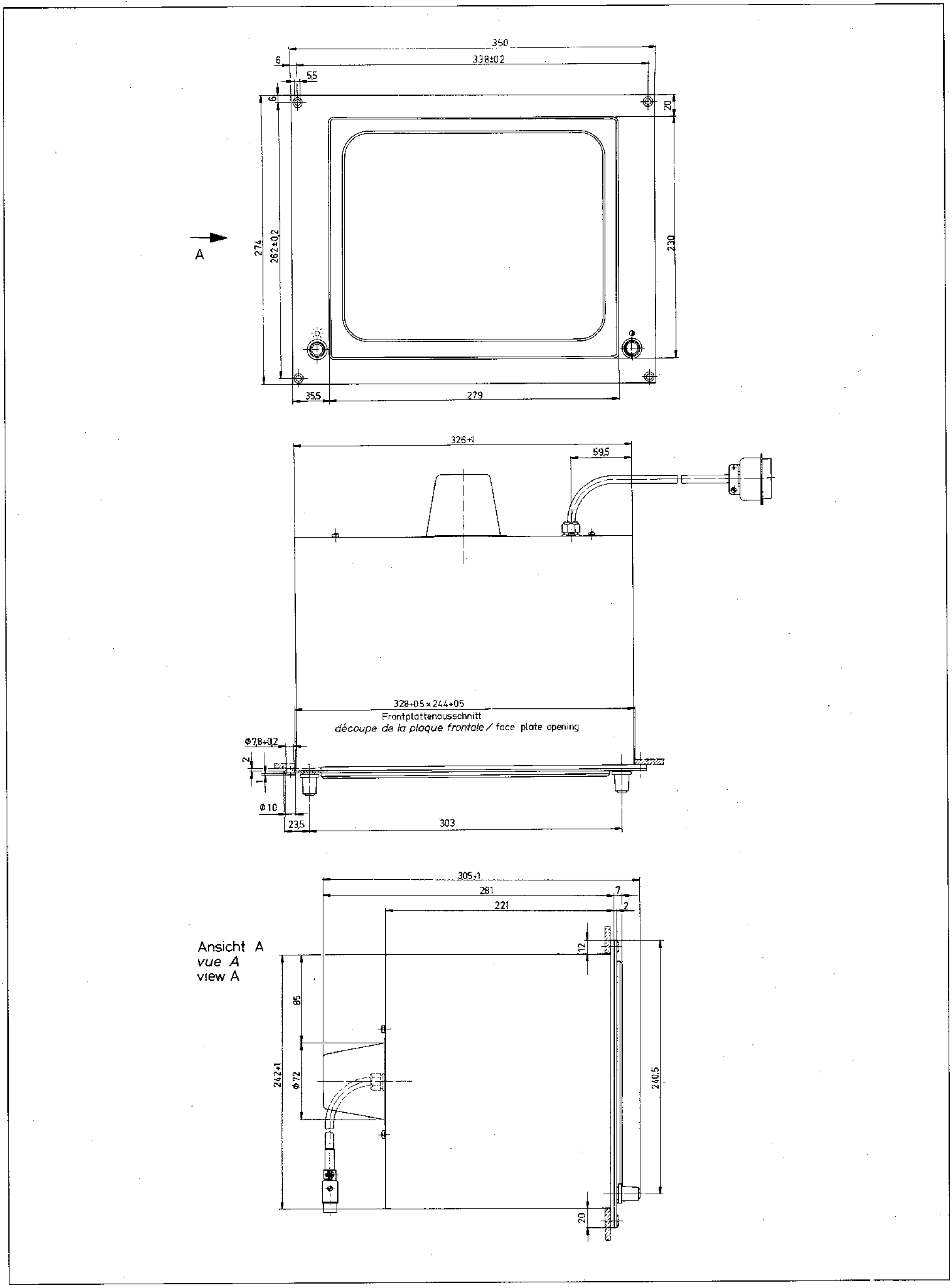
PC-Power board PL 100 B
PL 110 B



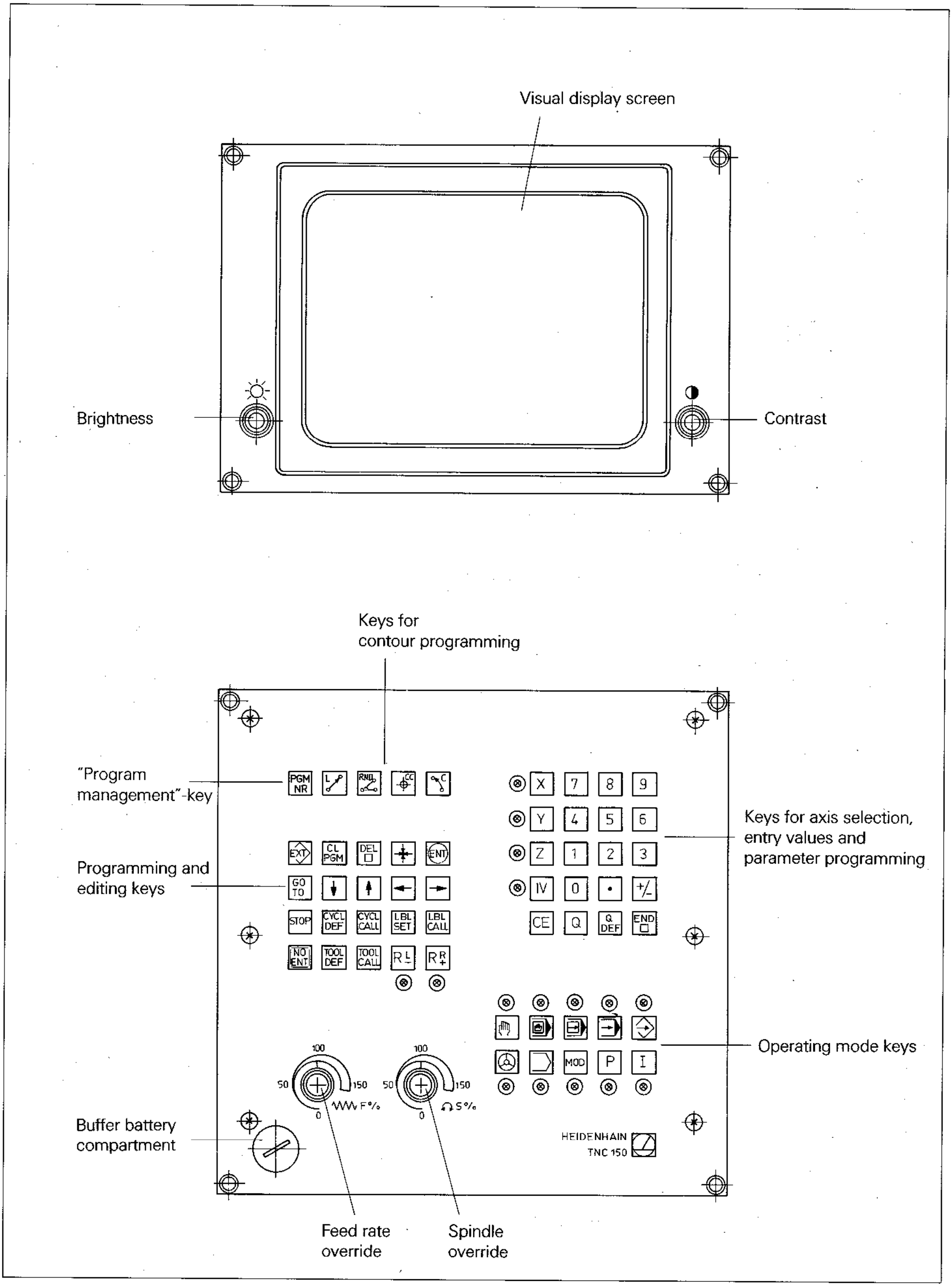
Visual display unit BE 111 (9")



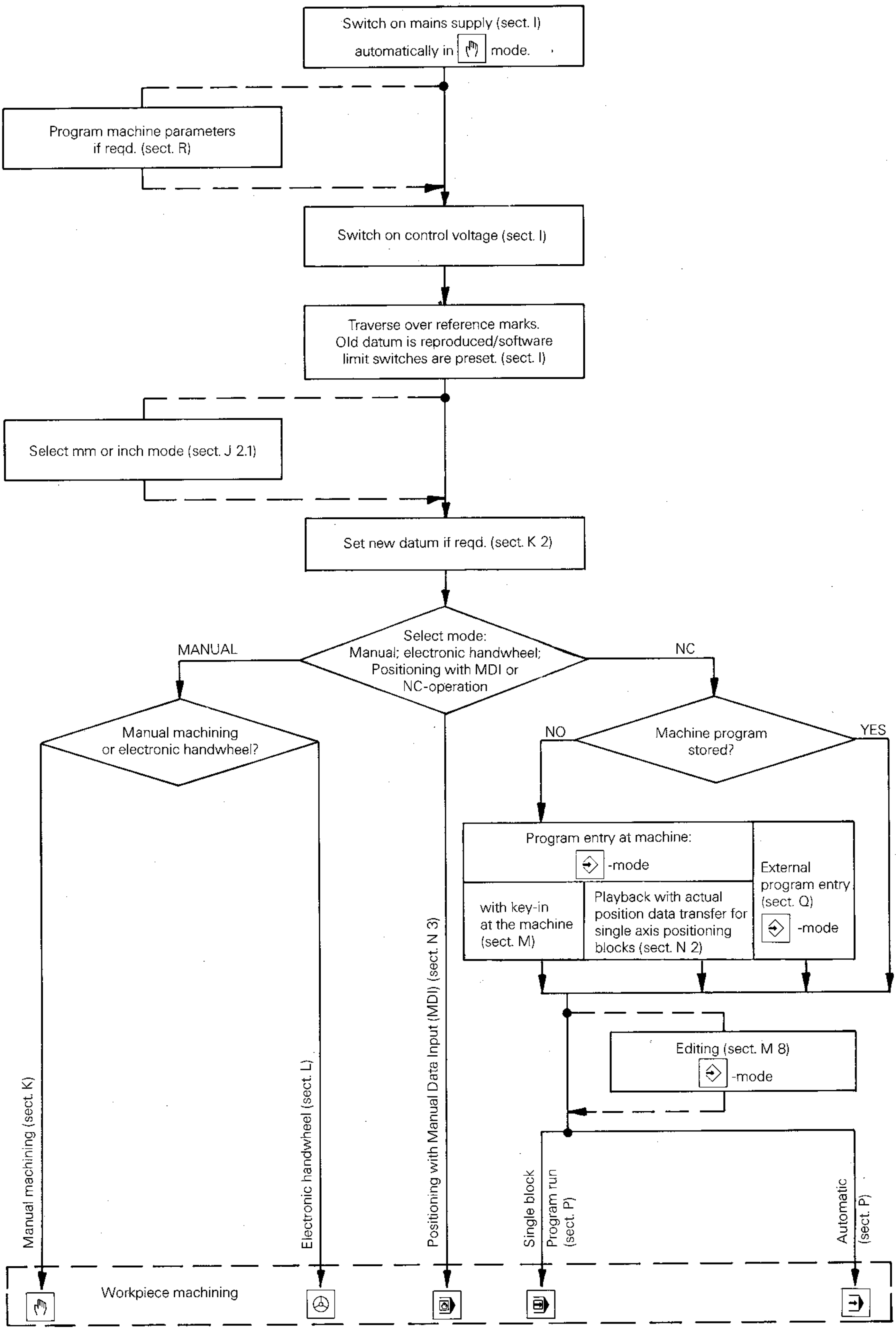
Visual display unit BE 211 (12")



Operating panel



V) Diagram for TNC 150-operation



HEIDENHAIN
DR. JOHANNES HEIDENHAIN GmbH
Postfach · D-8225 Traunreut · ☎ (0 86 69) 31-0
Telex 56 831 · Telefax (0 86 69) 59 75

223 217 26 · 5 · 9/85 · H · Printed in West Germany · Subject to technical modifications