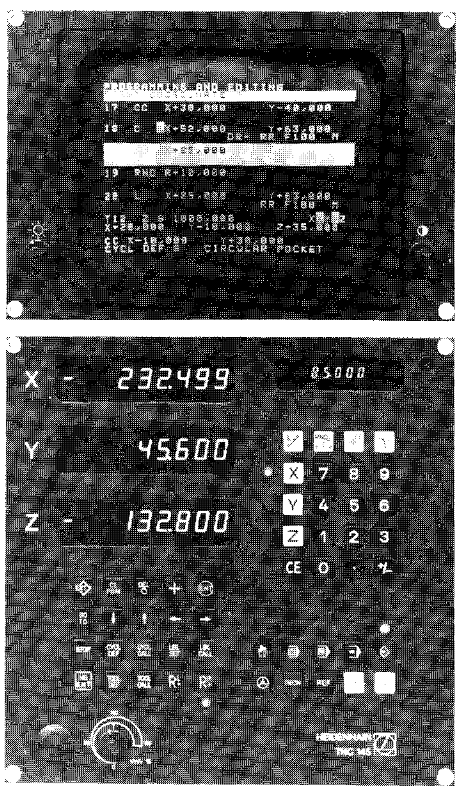
HEIDENHAIN
Optics and Electronics
Precision Graduations

Operating Manual

HEIDENHAIN TNC 145 C
Contouring Control



Issue 11/84

Dialogue initiation

Dialogue initiation with key	Operating mode: Manuel	Single block MDI	Programming and editing	Program run	See section	Page
X Y Z	Setting datum point	Single axis positioning (positioning block)	Programming for single axis positioning (positioning block)		G 2.1	24, 29
L			Straight line		I 4.3	41
RND			Rounding of corners/smooth contour approach		I 4.5 I 4.6	46 47
CC			Circle centre or Pole		I 4.2 I 4.4.1	40 43
C			Circular arc		I 4.4	43
STOP		.Confirmation of an external STOP for discontinuation of program run .Output of an M-function	Programmed STOP	Confirmation of an external STOP for discontinuation of program run	I 7, L 3	72, 87
TOOL DEF			Tool Definition		I 1.1	29
TOOL CALL		Tool Call	Tool Call		I 1.2	31
LBL SET			Setting Label		I 5.1	51
LBL CALL			Label Call		I 5.2	51
CYCL DEF			Cycle Definition		I 6	60
CYCL CALL			Cycle Call		I 6	60
CL PGM			Clear Program		I 8.7	76
INCH	mm/inch conversion	mm/inch conversion	mm/inch conversion	mm/inch conversion	F 3	23
REF	Display of displacements to reference marks				G 3	27
EXT	Programming of transfer rate for data interface		Entry of program via data interface	Output of program into external peripheral unit	J	78

Basic-Symbols	Meaning
	Machine traverse "controlled"
	Block *
	Memory for machining program (store)

*The machining program consists of individual program blocks; every program block comprises so-called "words".

Keys for programming of contours

Key Symbol	Abbreviation for	Meaning	See section	Page
L	LINEAR PATH	Straight cut traverse (simultaneously in 2 axes or only in 1 axes)	I 4.3	41
RND	ROUND	."Rounding off" corners (programming of arcs with tangential transitions) .Tangential contour approach	I 4.5 I 4.6	46 47
CC	CIRCLE CENTRE	.Circle centre for circular path .Pole for nominal value input in polar co-ordinates	I 4.2 I 4.4.1	40 43
C	CIRCLE	Circular arc (simultaneous traverse in 2 axes)	I 4.4	43

Keys for entry values and axis selection

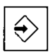
Key Symbol	Abbreviation for	Meaning		See section	Page
0 . . . 9	- - -	Entry value and axis selection keys	Decimal keyboard for numerical values	E 3	19
·	- - -		Decimal point	E 3	19
+/−	- - -		Sign change	E 3	19
X Y Z	- - -		Axis selection for datum set with single axis traversing and programming of nominal position values	G 2, I	24 29
CE	CLEAR ENTRY	For deletion of entry values or cancellation of fault/error indications within dialogue		D 2, E 3	16, 19

If, in the selected operating mode, a button is inadvertently pressed which has no function, the error "BUTTON NON-FUNCTIONAL" is indicated.

This error code can be cancelled by pressing the CE -key.

TNC 145 C Keyboard

Operating mode-keyboard

Key Symbol	Meaning	See section	Page
	**Manual** operation **1.** The control operates as conventional digital readout. The machine can be traversed via the axis-direction buttons. **2.** Datum set **3.** After switch-on of power, the reference marks must be traversed over	G	24
 REF	**4.** The position displays indicate the displacement to the transducer reference marks.		
	Traversing of machine axes via the **electronic handweel**	H	28
	**Single block positioning with MDI** (manual data input) Single axis automatic traversing. One single block can be entered only (single axis positioning block or tool call). The entered program is not influenced contouring operation, with cycles, subprograms or program part repeats is not possible in this operating mode. A **tool call** is only effective when .the tool has been previously defined, i.e. the compensation values (length and radius) have been entered into the program memory .after the tool call, the external START-button is activated.	K	84
	**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 or "linear path" programming) .Circle centre .Circular path .Tool definition .Tool call .Cycle definition .Cycle call .Label (program mark) set .Label (program mark) call: subprogram or program part repeat .Program STOP	I	29
	**Program run, single block** (block-by-block) A press of the START-button is required to execute each individual program block.	L	85
	**Automatic program run** (complete program sequence) with single press of START-button, the stored program sequence is run to a programmed STOP or to the end. Program blocks are automatically called up and executed in sequence.	L	85
INCH	**Inch** Entry and display of .position values in imperial mode .feed rate, contouring speed (speed of tool centre point) in 0.1 inch/min. If a machining program is to be entered in inch, the INCH -key must be pressed before program entry. INCH off: entry and display in mm	F 3	23
I	**Incremental dimension (chain dimensions);** when off: absolute dimensions	B 2, I 2	11 33
P	Entry of nominal values in **polar co-ordinates;** when off: right angled (Cartesian) co-ordinates	B 3, I 4.2	12 40

Programming and editing keys

Key Symbol	Abbreviation for	Meaning		See section	Page
EXT	- - -		External data input or output	J	78
✛	- - -		Actual position value: Transfer of actual machine position data as entry value for programming (Playback)	I 3.2	36
CL PGM	CLEAR PROGRAM	Editing keys	Clear complete program content	I 8.7	76
DEL □	DELETE BLOCK	Editing keys	Delete previously entered block	I 8.3	73
ENT	ENTER		Enter into dialogue or memory	E 2, E 3,	18, 19
GO TO	GO TO BLOCK	Editing keys	Block search key	I 8.1	73
↓ ↑	- - -	Editing keys	"Paging" of program content forwards or reverse	I 8.2 I 8.6	73 76
← →	- - -	Editing keys	Cursor movement for program word selection	I 8.5 I 8.6	75 76
STOP	STOP (HALT)		Programmed STOP or interruption of positioning	I 7, L 3	72, 87
CYCL DEF	CYCLE DEFINITION	Cycle keys	Definition of fixed programmed machining cycle (canned cycle)	I 6	60
CYCL CALL	CYCLE CALL	Cycle keys	Call-up of fixed programmed machining cycle (canned cycle)	I 6	60
LBL SET	LABEL . SET	Subprogram keys	Allocation of program label (For subprogram or program part repeat)	I 5.1	51
LBL CALL	LABEL CALL	Subprogram keys	Call-up of program label (Jump to label No.)	I 5.2	51
NO ENT	NO ENTRY		No entry: The parameter requested by the dialogue is not required.	E 2	18
TOOL DEF	TOOL DEFINITION	Tool keys	Tool or cutter definition (Tool No., length, radius)	I 1.1	29
TOOL CALL	TOOL CALL	Tool keys	Call-up of required tool	I 1.2	31
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.	I 3 I 4.1	33 37
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.	I 3	33

Contents

This operating manual is valid for TNC 145 C-Controls with the following Ident.-numbers:

TNC 145 CS
from Ident.-No. 221 201 ..

TNC 145 CR
from Ident.-No. 221 638 ...

Please note:
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

The HEIDENHAIN TNC 145 C is a 3-axis continuous path control with 2 1/2 D circular path and straight cut interpolation. (2 1/2 D-interpolation means that the interpolation may be programmed in any two axes).

As all HEIDENHAIN TNC-controls, the TNC 145 C is designed for direct programming at the machine. Therefore for ease of operation, some of its details purposely deviate from established programming standards.

The programming procedure is "dialogue-guided", i.e. the necessary data required for program entry is asked for by the TNC 145 C - in the correct sequence - via the plain language dialogue display.

The TNC 145 C includes a CRT-screen (VDU = visual display unit) for display of dialogue and machining program, as well as further displays for entry values and actual position readout.

The finest step for entry of position values in Cartesian co-ordinates ist 0.001 mm or 0.0001 inch. With entry in polar co-ordinates, the polar radius PR may be 0.001 mm or 0.0001 inch and the polar angle PA in degrees down to 0.001°.
The resolution is 0.005 mm/0.0002 inch or 0.001 mm/0.0001 inch.

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

.with stationary machine (to program sheet or workpiece drawing)

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). A machining program which has been externally stored, or prepared on an external programming unit, can be read in.

with **single axis programming:**
manually with key-in.

with stationary machine (to program sheet or workpiece drawing)

.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 interface as explained above.

The HEIDENHAIN ME 101/ME 102 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 145 C and e.g. a printer unit, may be simultaneously connected.
Programs which have been entered externally can be edited or optimised if required.

Main functions of the TNC 145 C

The TNC 145 C controls the automatic machining of a workpiece in accordance with a program which is entered into and stored within the TNC 145 C. 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.).
A machining program comprises individual "blocks" which consist of word — information. This will be dealt with later.

For execution of a stored program, the operating mode **"automatic program run"** ([→] -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** ([⊗] -key).
Datum-set and reference mark approach are performed in the **"manual"** mode ([☞] -key). This mode is otherwise only for conventional digital readout operation.
The **"program entry"** mode is initiated with the [→] -key (the respective lamp is then on). From the range of **tools** entered with the tool definition blocks ([TOOL DEF] -key) the required tool is to be selected with a tool call block before commencement of the machining procedure ([TOOL CALL] -key).
For programming of the **tool path** only the workpiece contour and the dimensions according to the drawing have to be entered: length and radius of the tool are automatically taken into account with the TNC 145 C. In order to describe the contour, the path (linear [L] or arc [CC] [C]) and to which nominal position the machine is to traverse must be entered. In comparison to other numerical controls, only the [RND] -key has to be pressed for automatic insertion of tangential transition radii. This also applies to the automatic rounding of corners.
If the machine has to be traversed with single axis positioning only — which is often the case — programming can also be carried out via the axis-keys [X], [Y], [Z] as per the HEIDENHAIN-controls TNC 131/135 i.e. greater simplicity.
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.

Furthermore, with single axis positioning, transfer of actual position (display) values as nominal values is also possible (Teach-in).

The **traversing speed** is programmed in mm/min. or 0.1 inch/min.

A substantial reduction of programming is made possible by **fixed (canned) programmed cycles**:
.Three dim. line (for fast positioning of workpiece and tool)
.Pecking (Deep hole drilling)
.Tapping
.Slot milling
.Pocket milling
.Circular pocket
.Datum shift
.Mirror image

Required cycle parameters are specified 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 145 C 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.
Program editing i.e. corrections or optimisation of programs through amendment of block-word information, 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.

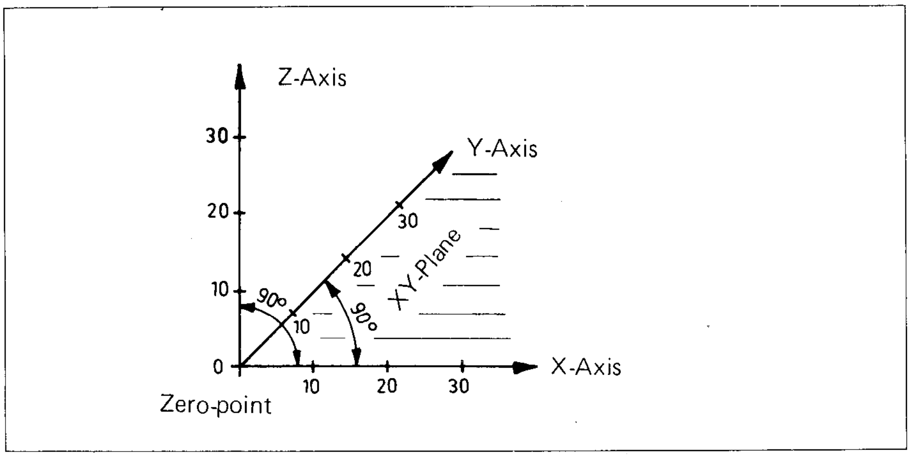
After control switch-off or power failure, a buffer battery takes care that the memory content i.e. machining program and machine parameters (control functions suited to the machine characteristics) is not erased. In order that this data is not lost, a **BATTERY CHANGE MUST BE MADE WITH THE CONTROL SWITCHED 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 D 3) and F 2).

B) Dimensions, Positions, Co-ordinates

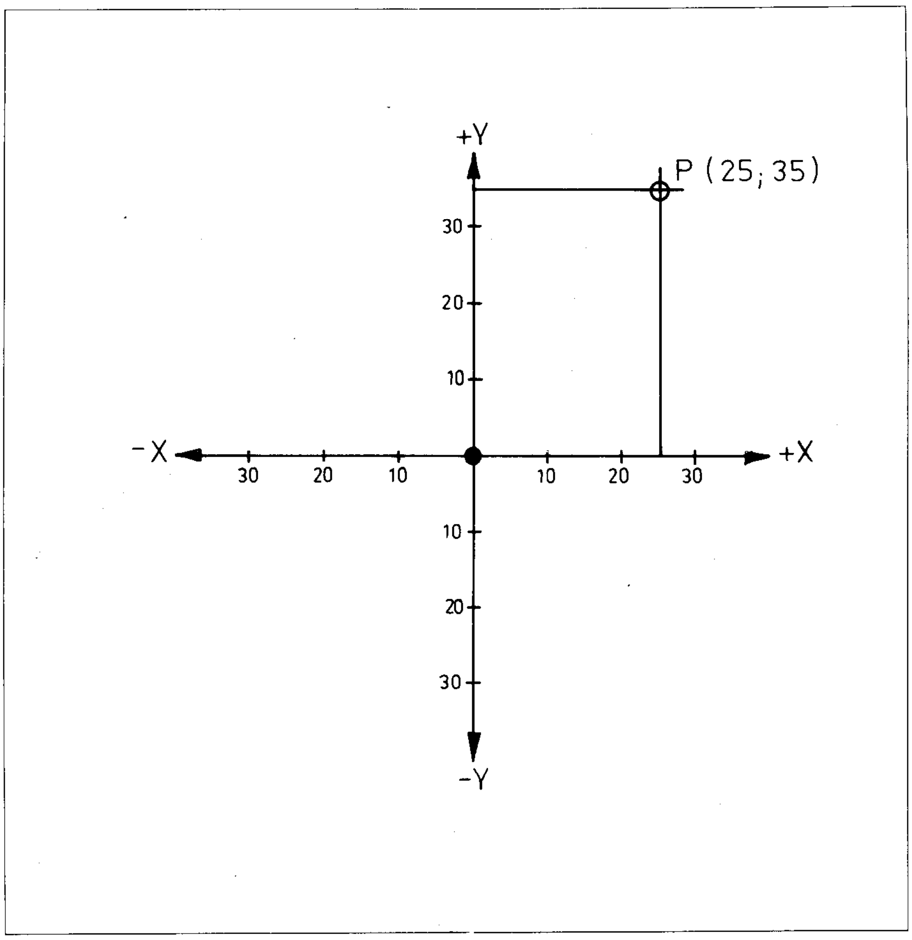
B 1) Cartesian co-ordinates and workpiece datum

One must differentiate between the "actual position" of machine and workpiece, i.e. the momentary position, and the "nominal (intended) 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 145 C 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 B 3).

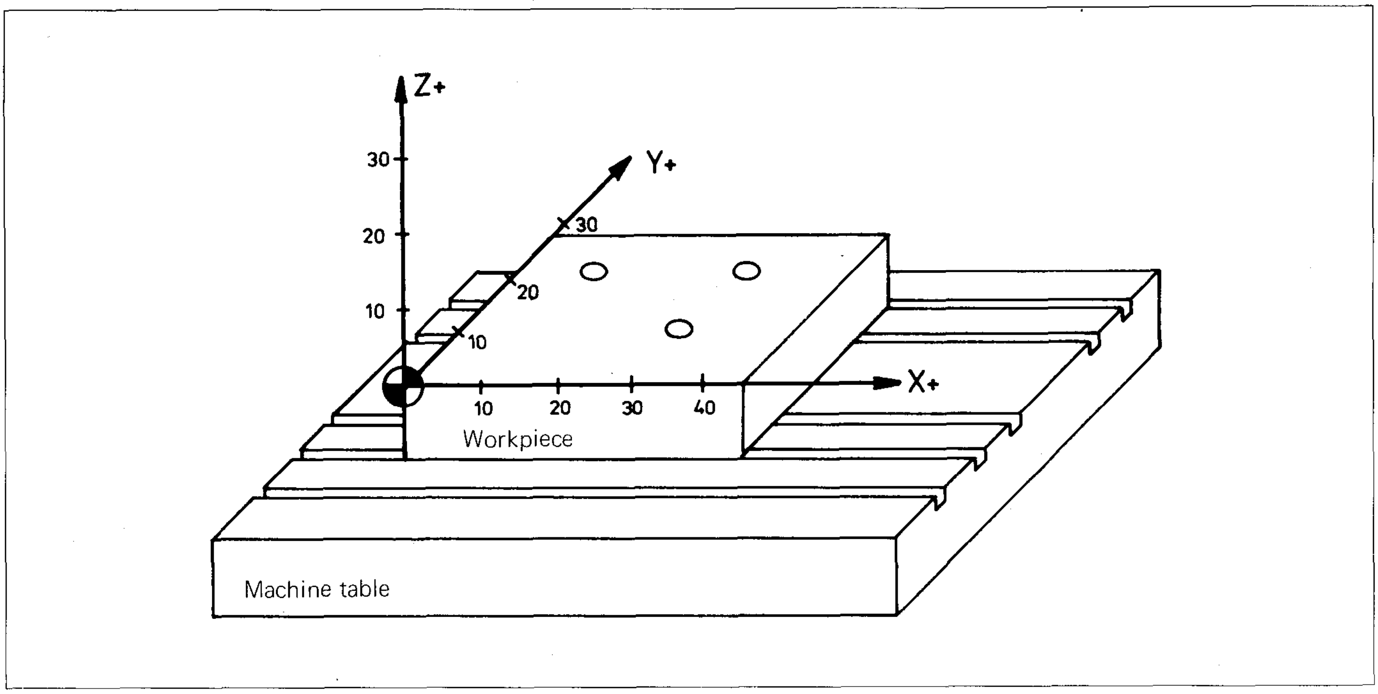


A right-angled co-ordinate system (grid system) is formed by three co-ordinate axes X, Y and Z which are perpendicular to each other. The two axes X and Y constitute the XY-plane. All three axes have a common point of intersection the so-called zero-point (or "origin").



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 the values of the three co-ordinates X, Y and Z.

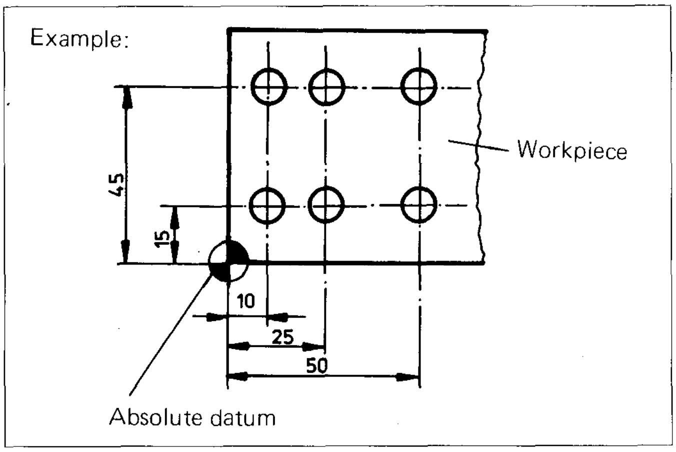
To determine positions in a machining program, the co-ordinate system is established such that program preparation is easy and convenient. E.g. the co-ordinate axes can coincide with the workpiece edges (the workpiece is clamped to the machine table such that its co-ordinate axes are parallel to the 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 ⊕ .



B 2) Absolute dimensions / Incremental dimensions

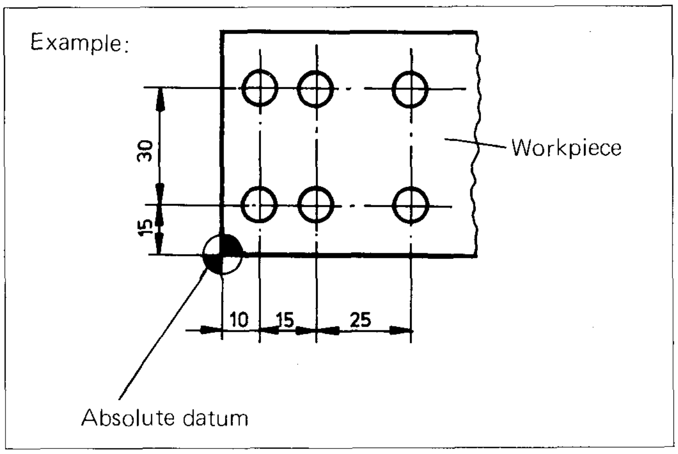
Workpiece dimensions are either absolute or incremental.

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 keyed-in nominal position value.

Incremental dimensioning



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

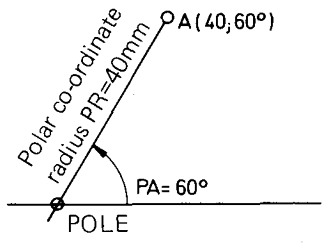
Programming in absolute dimensions offers the advantage of making geometric amendments of single positions without affecting other positions. Re-entry into an interrupted program after power failure or any other defect is also more simple with absolute programming. Furthermore, a suitable location of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

On the other hand, **incremental programming** reduces calculation work.

B 3) Polar co-ordinates

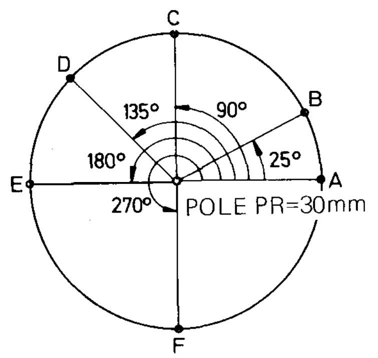
TNC 145 C also offers the possibility of entering nominal position values by using polar co-ordinates.

With polar co-ordinates, points are determined **in one plane only.**
These points are referenced to a polar co-ordinate datum – **the "pole"** – 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:

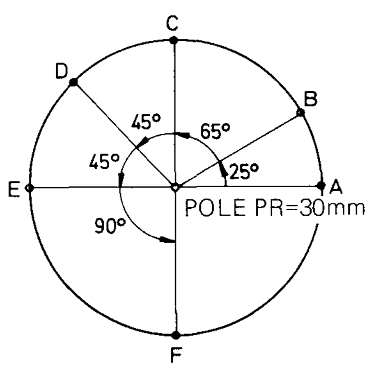


Polar co-ordinates of points A, B , . . . , F;
PR and PA absolute

A	$(30_A;\ 0^\circ_A)$
B	$(30_A;\ 25^\circ_A)$
C	$(30_A;\ 90^\circ_A)$
D	$(30_A;\ 135^\circ_A)$
E	$(30_A;\ 180^\circ_A)$
F	$(30_A;\ 270^\circ_A)$

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

Example:



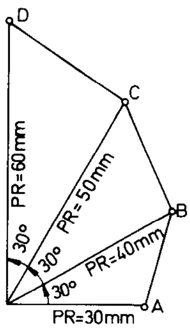
Please note:
The first point A must be defined in absolute A $(30_A;\ 0_A^\circ)$

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

B	$(30_A;\ 25^\circ_I)$
C	$(30_A;\ 65^\circ_I)$
D	$(30_A;\ 45^\circ_I)$
E	$(30_A;\ 45^\circ_I)$
F	$(30_A;\ 90^\circ_I)$

c) Radius and directional angle programmed in incremental dimensions

Example:

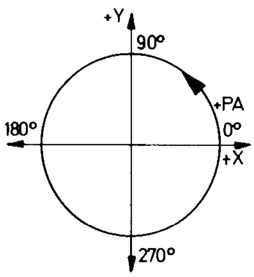


Please note:
The first point A must be defined in absolute: A (30_A; $0°_A$)

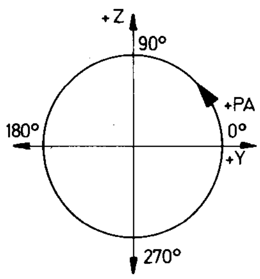
Polar co-ordinates of points B, C, D
PR and PA incremental:

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

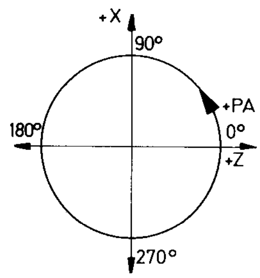
Definition of planes and 0° - axes



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



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

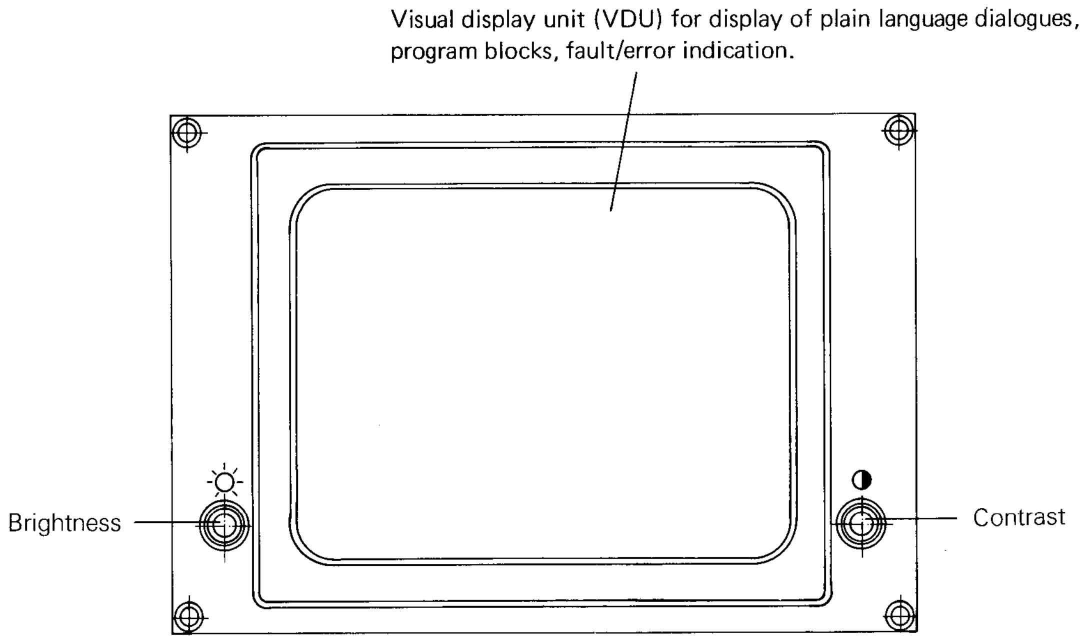


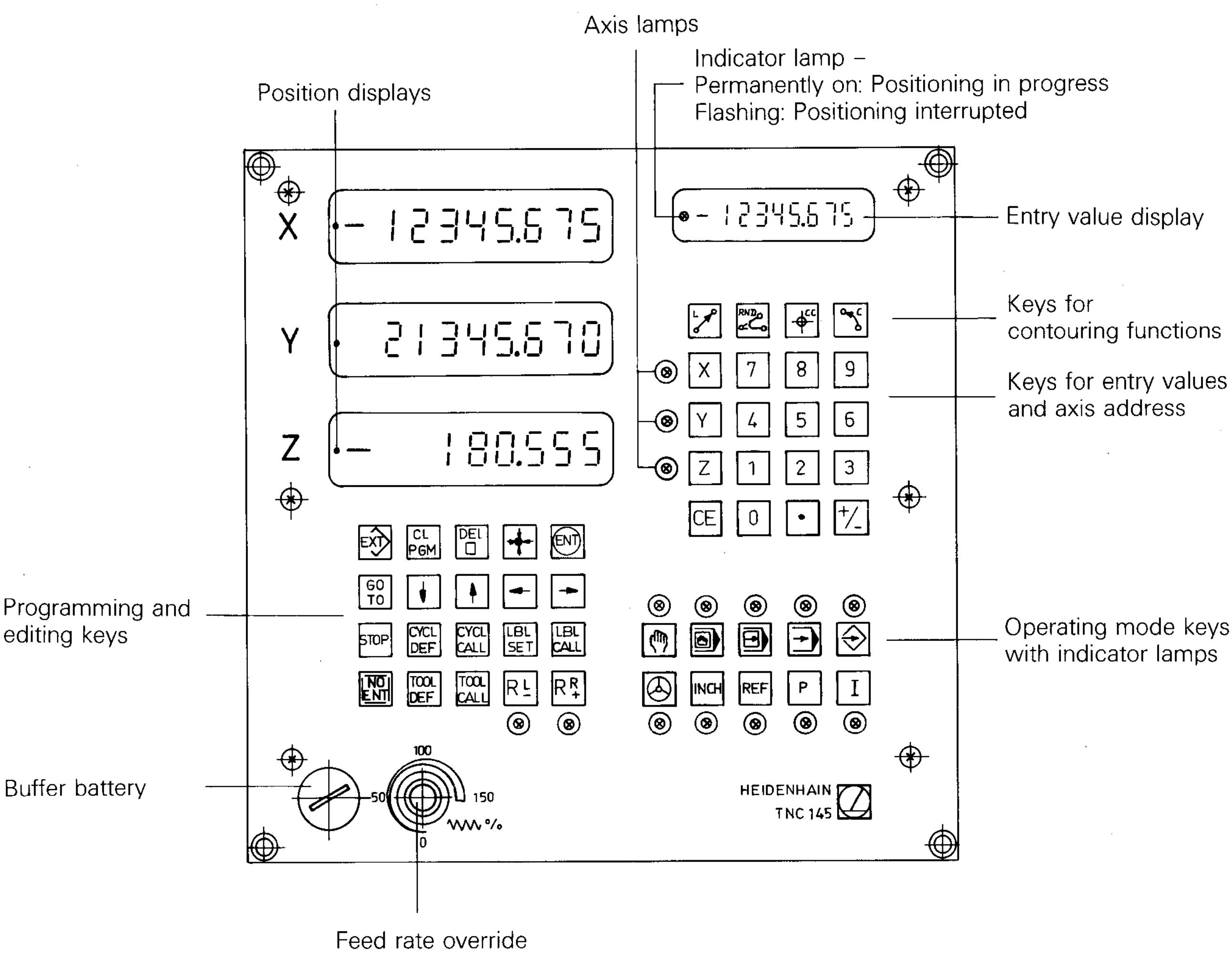
In the **Z/X**-plane the 0°-axis lies on Z+

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

C) Keyboards and displays of TNC 145 C

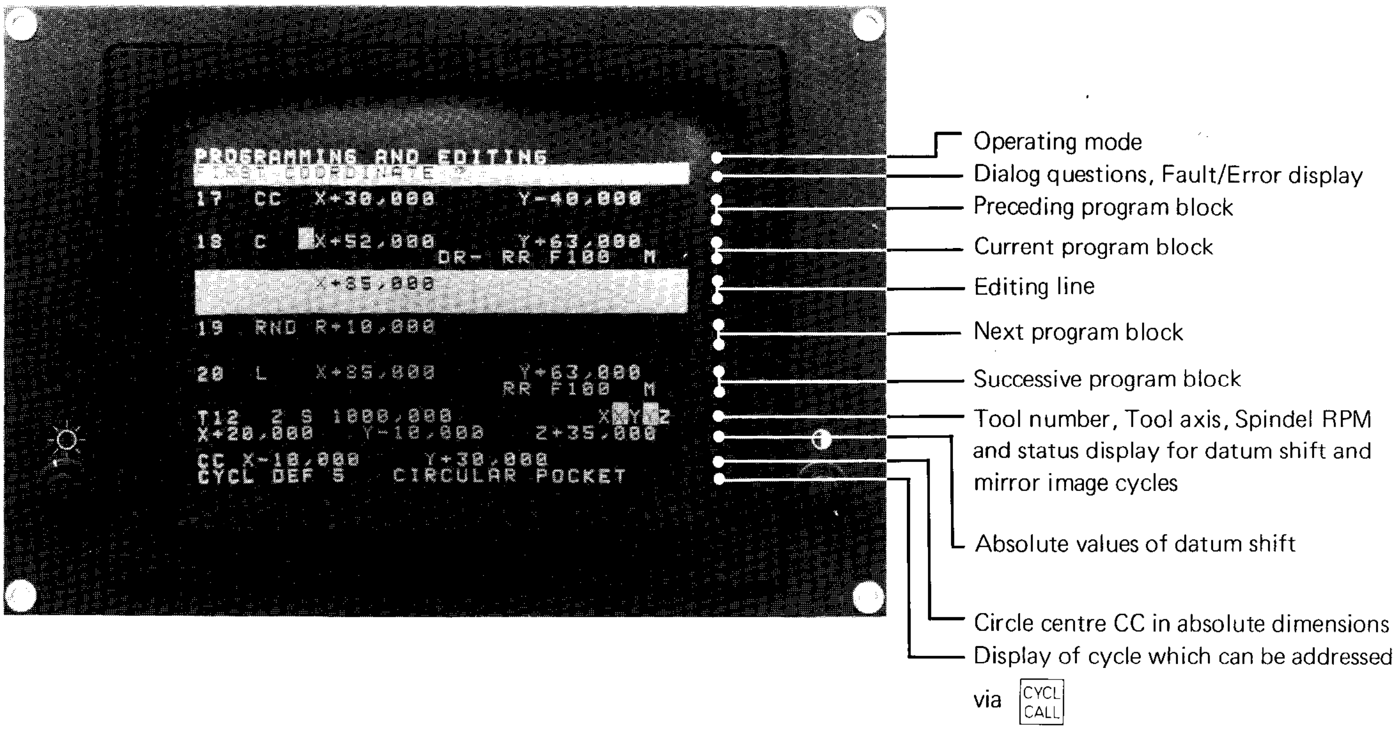
Full layout





Visual display unit

The VDU-Screen of the TNC 145 C displays the following information:



Status display for datum shift and mirror image

The status display for datum shift (see section I 6.3.7) and mirror image (see section I 6.3.8) indicates the datum shifts and mirror images called up:

Axis display in normal characters: Datum shift
Axis display in inverted characters: Mirror image

Example:

If mirror image is to take place in the Y-axis and datum shift in the X-Y-plane, the following is displayed:
X Y Y

D) Fault/error prevention and diagnosis

D 1) Fault/error indication

The TNC 145 C possesses an extensive monitoring system for prevention of 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
BUTTON 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

.Certain machine functions
e.g. fault indication
GROSS POSITIONING ERROR
X-MEASURING SYSTEM DEFECTIVE
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 failures in control electronics). This simultaneously activates an automatic machine switch-off via the **EMERGENCY STOP** contact of the control.

D 2) Cancellation of fault/error indication

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

D 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 PG-screw-cap in the lower left-hand corner of the operating panel (see section C). 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 Mallory Alcaline batteries type "MN 1500".

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 (machining program and machine parameters) will become erased in the event of a mains power failure: **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 F 2)!**

E) Dialogues of TNC 145 C

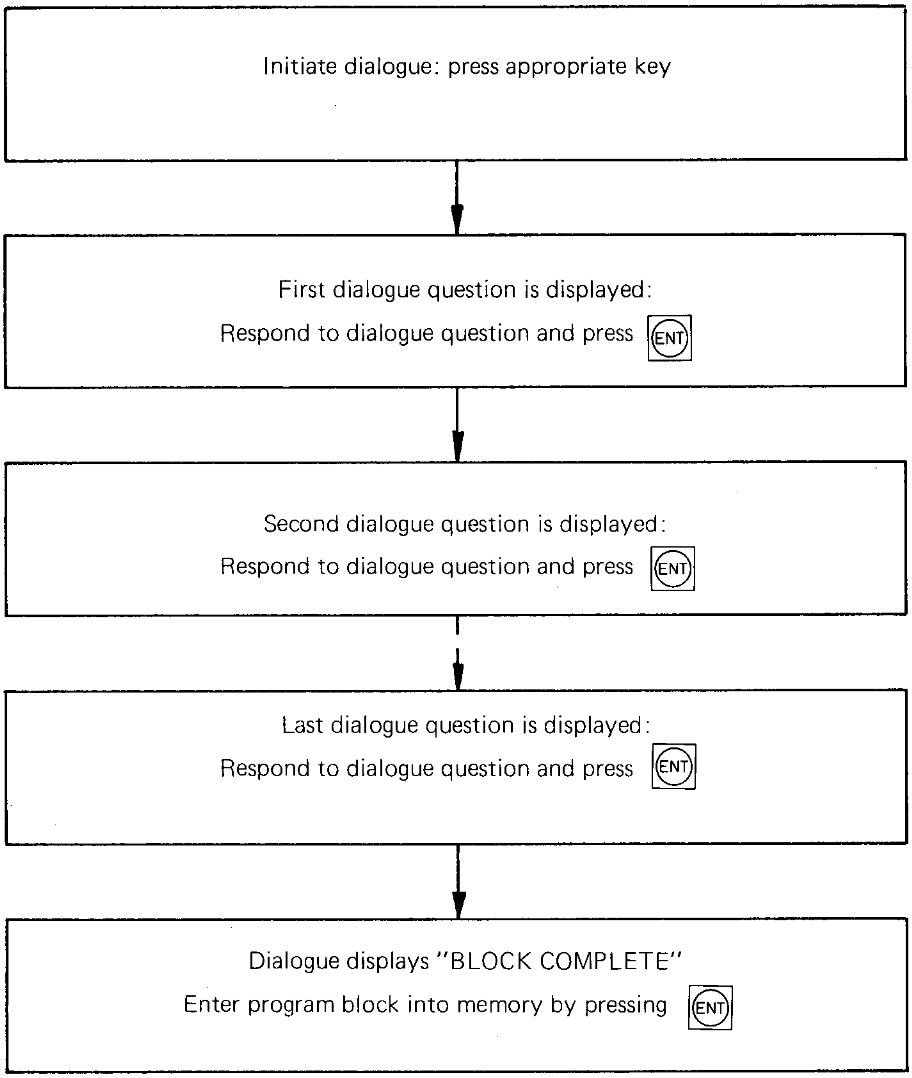
Operating and programming of the HEIDENHAIN TNC 145 C-Control is characterised by the plain language dialogue. After the operator has initiated a dialogue, the control takes over the full guidance with respect to program entry by means of direct question.

Examples of plain language fault/error display, see section D.

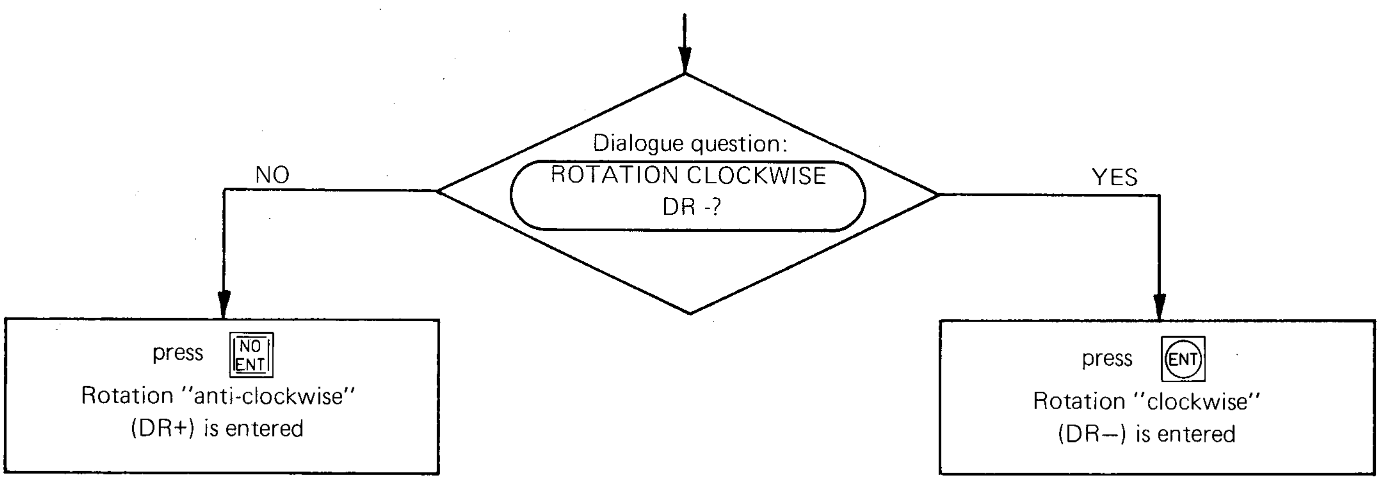
E 1) Dialogue initiation

Keys for dialogue initiation are explained on page 2.

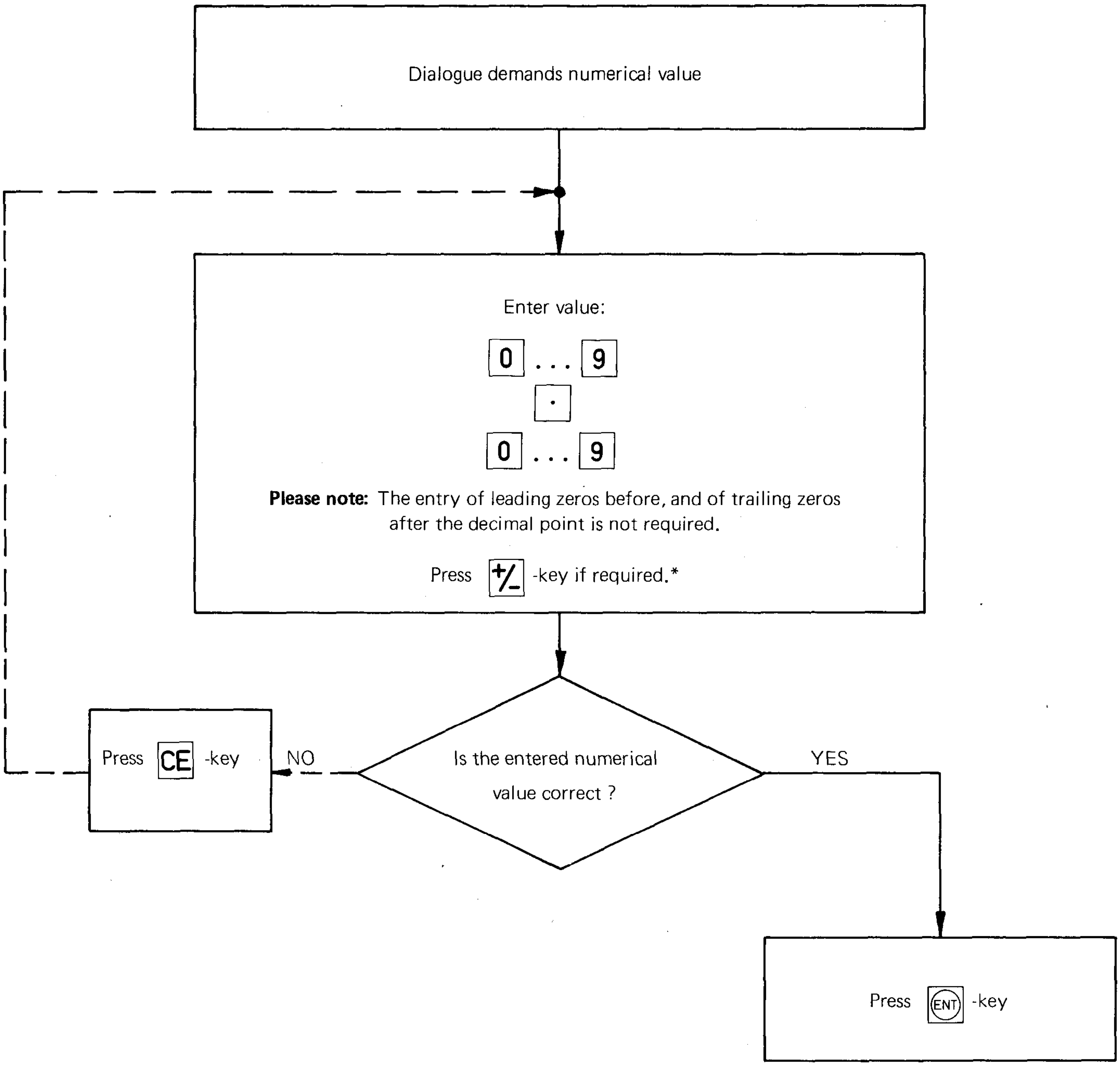
E 2) Rules for responding to dialogue questions



Certain dialogue questions can be responded to – without entry of a numerical value – by pressing the ENT or NO ENT -key:
These types of dialogue questions are specially dealt with in the individual sections.



E 3) Entry of numerical values



***Please note:**
When pressing the sign change-key the arithmetical sign of the value in the entry value-store is altered.
When entering negative values:
First key-in numerical value and then the negative sign!

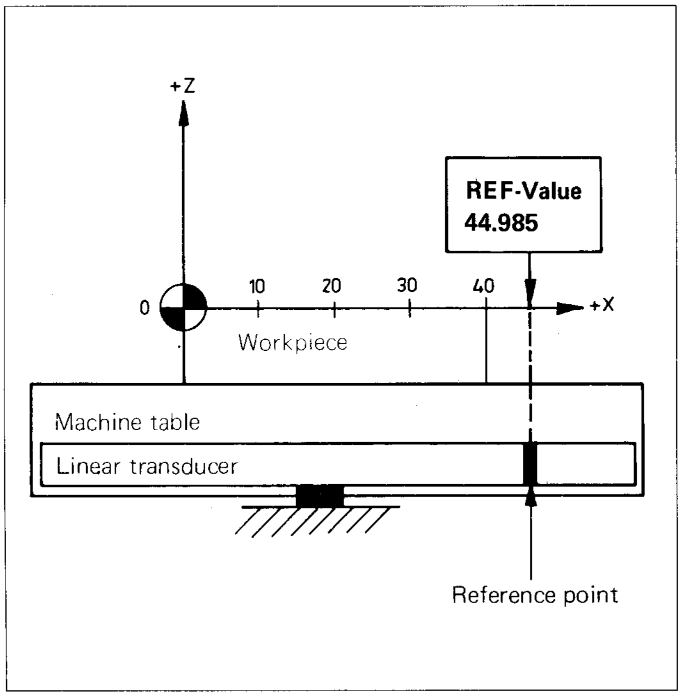
Entry step of position values
For entry in Cartesian co-ordinates:
down to 0.001 mm or 0.0001 inch

For entry in polar co-ordinates:
Polar radius PR up to 0.001 mm or 0.0001 inch,
polar angle PA in degrees up to 0.001°.

F) Preparatory tasks

F 1) TNC 145 C switch-on and traversing over the reference marks

The transducers of all machine axes possess reference marks. These marks, when traversed 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 the position 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 marks must be traversed over after every interruption of power (due to the TNC 145 C 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 B 1) and G 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 145 C switch-on and traversing over of reference marks is performed as follows:

Switch on supply voltages

Dialogue display: POWER INTERRUPTED

Press CE : Fault display is cancelled

Dialogue display: RELAY EXT.D C VOLTAGE MISSING

The EMERGENCY STOP cut-out 1) has been checked.
This has led to power switch - off
Switch on power again!

Dialogue display: TRAVERSE OVER X-REFERENCE MARK
TRAVERSE OVER Y REFERENCE MARK
TRAVERSE OVER Z-REFERENCE MARK

- Control is automatically set to operating mode
- Position displays are blocked and show the "REF-values" which were previously set

Is automatic traversing over reference points possible? 2)

NO

Key-in code value **84159** 0 ... 9

ENT press

Dialogue display: CAUTION: SOFTWARE LIMITS INACTIVE
TRAVERSE OVER X-REFERENCE MARK
TRAVERSE OVER Y-REFERENCE MARK
TRAVERSE OVER Z-REFERENCE MARK

Traverse over reference points in any desired sequence via external direction buttons (the advanced limit switches are inactive).

The position displays show actual position values; the dialogue display of the appropriate axis is erased when the external direction button is released.

YES

Traverse over reference points one after the other: 3)

The position displays show actual position values.

Dialogue display: MANUAL OPERATION

Now the desired operating mode may be selected.

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 F 2).

3) Automatic traverse over the reference points is activated via the external START-button. For reasons of safety, each axis must be individually re-started. The position displays only commence counting when the reference points have been passed; the dialogue display of each axis is then erased.

F 2) Programming of Machine Parameters

Machine parameters are determined by the machine tool manufacturer and entered into the control during the initial starting procedure via data medium (ME/cassette loaded with machine parameters) or by key-in procedure.
After an interruption of power with empty or missing buffer batteries, the parameters are demanded again by the control after the fault indication "RELAY EXT.D C VOLTAGE MISSING" has been cancelled. These parameters must be re-entered either by manual key-in (as per the list below) or e.g. by means of a HEIDENHAIN magnetic tape unit ME 101/102.

Code number	**Entry value (to be entered below by machine tool manufacturer)**
MACHINE PARAMETER 00 MACHINE PARAMETER 01	
MACHINE PARAMETER 02 MACHINE PARAMETER 03	
MACHINE PARAMETER 04 MACHINE PARAMETER 05	
MACHINE PARAMETER 06 MACHINE PARAMETER 07	
MACHINE PARAMETER 08 MACHINE PARAMETER 09	
MACHINE PARAMETER 10 MACHINE PARAMETER 11	
MACHINE PARAMETER 12 MACHINE PARAMETER 13	
MACHINE PARAMETER 14 MACHINE PARAMETER 15	
MACHINE PARAMETER 16 MACHINE PARAMETER 17	
MACHINE PARAMETER 18 MACHINE PARAMETER 19	
MACHINE PARAMETER 20 MACHINE PARAMETER 21	
MACHINE PARAMETER 22 MACHINE PARAMETER 23	
MACHINE PARAMETER 24 MACHINE PARAMETER 25	
MACHINE PARAMETER 26 MACHINE PARAMETER 27	
MACHINE PARAMETER 28 MACHINE PARAMETER 29	
MACHINE PARAMETER 30 MACHINE PARAMETER 31	
MACHINE PARAMETER 32 MACHINE PARAMETER 33	
MACHINE PARAMETER 34 MACHINE PARAMETER 35	

Code number	**Entry value (to be entered below by machine tool manufacturer)**
MACHINE PARAMETER 36 MACHINE PARAMETER 37	
MACHINE PARAMETER 38 MACHINE PARAMETER 39	
MACHINE PARAMETER 40 MACHINE PARAMETER 41	
MACHINE PARAMETER 42 MACHINE PARAMETER 43	
MACHINE PARAMETER 44 MACHINE PARAMETER 45	
MACHINE PARAMETER 46 MACHINE PARAMETER 47	
MACHINE PARAMETER 48 MACHINE PARAMETER 49	
MACHINE PARAMETER 50 MACHINE PARAMETER 51	
MACHINE PARAMETER 52 MACHINE PARAMETER 53	
MACHINE PARAMETER 54 MACHINE PARAMETER 55	
MACHINE PARAMETER 56 MACHINE PARAMETER 57	
MACHINE PARAMETER 58 MACHINE PARAMETER 59	
MACHINE PARAMETER 60 MACHINE PARAMETER 61	
MACHINE PARAMETER 62 MACHINE PARAMETER 63	
MACHINE PARAMETER 64 MACHINE PARAMETER 65	
MACHINE PARAMETER 66 MACHINE PARAMETER 67	
MACHINE PARAMETER 68 MACHINE PARAMETER 69	
MACHINE PARAMETER 70 MACHINE PARAMETER 71	
MACHINE PARAMETER 72	

Machine parameter input via ME-Magnetic tape unit

Switch on power, cancel dialogue POWER INTERRUPTED
and switch on control voltage (see section F 1).

↓

Dialogue question: MACHINE PARAMETER PROGRAMMING
MACHINE PARAMETER-MPO?

.Connect Magnetic tape unit (connector TNC),
.Insert cassette and rewind to Band-start,
.Select modes [←8] and [TNC] .

↓

Press [EXT]

Dialogue question: BAUD RATE: 2400?

↓

Enter Baud rate (the magnetic tape unit requires 2400 Baud)
and transfer via [ENT]

Dialogue display: MACHINE PARAMETER PROGRAMMING
EXTERNAL DATA INPUT

Magnetic tape unit is automatically started,
machine parameters are read-in.

↓

Read-in procedure is concluded:

Dialogue display: PROGRAM MEMORY ERASED

Press [CE] and traverse over reference marks.

F 3) mm/inch conversion [INCH]

The control can be programmed either in the metric or in the imperial mode. Switchover from "mm" to "inch" must be performed before commencement of program entry. Switchover with dialogue-supervision:

Press [INCH] (in desired operating mode).

↓

Dialogue question:
NEW OPERATING MODE:
YES = ENT / NO = NO ENT

NO → Press [NO ENT]
"INCH" is not selected.

YES → Press [ENT]
"INCH" is selected.

Please note:
The inch-mode can be cancelled by re-pressing the [INCH] and [ENT] -keys.

G) Manual operation

G 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 145 has been adapted to the machine.

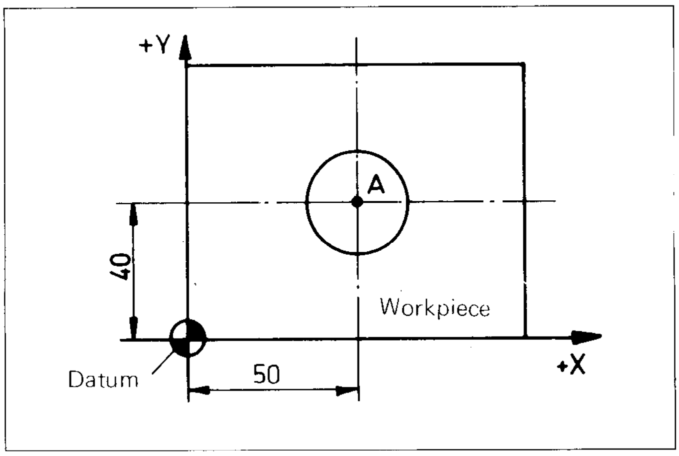
The machine axes can be traversed in two ways:

.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.

.continous operation
If, after pressing the axis direction button, the START-button ist 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.

G 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.

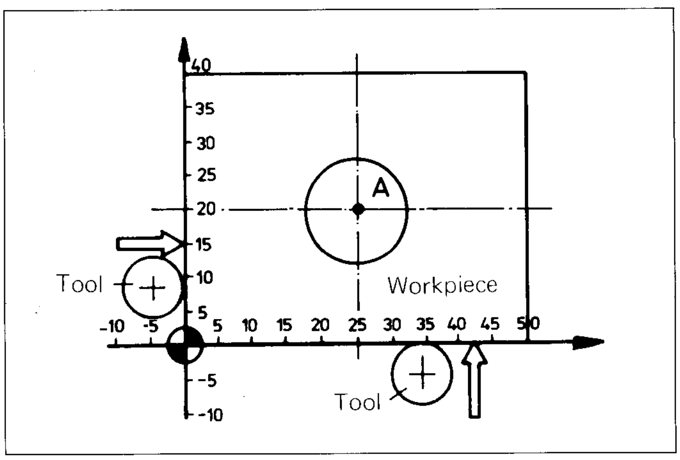


For this, either

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.

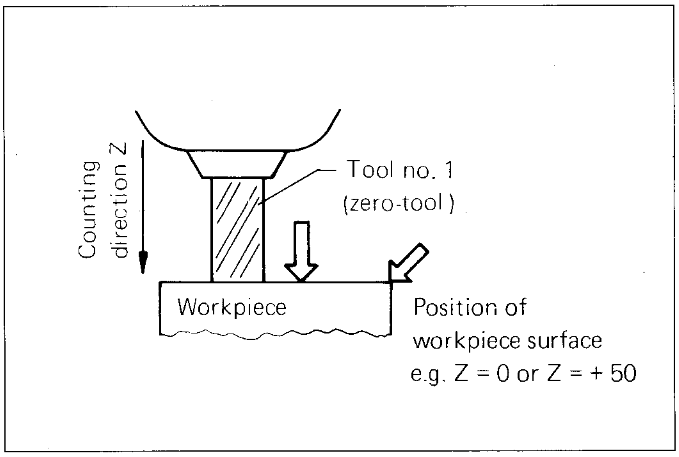
The presetting of both axes corresponds to case b) (instead of 50 and 40, the value -5 is to be entered).

In this example, the Z-axis corresponded to the tool-axis. The 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 (Zero-tool, see also section I 1.1 "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).

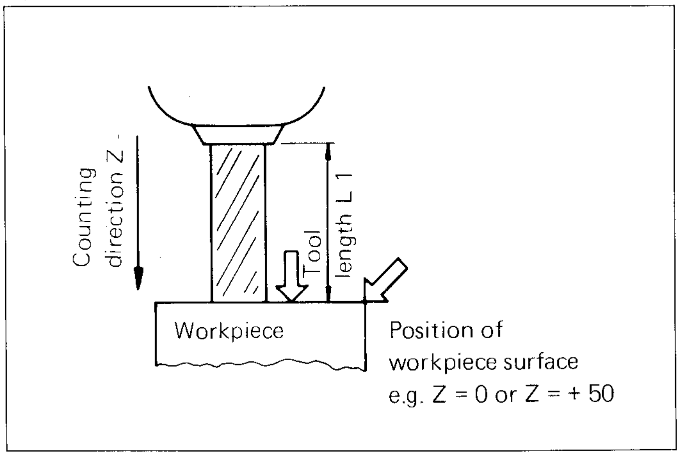
If the workpiece surface is to have a value other than 0, then the tool axis must be pre-set to this value e.g. +50.



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:



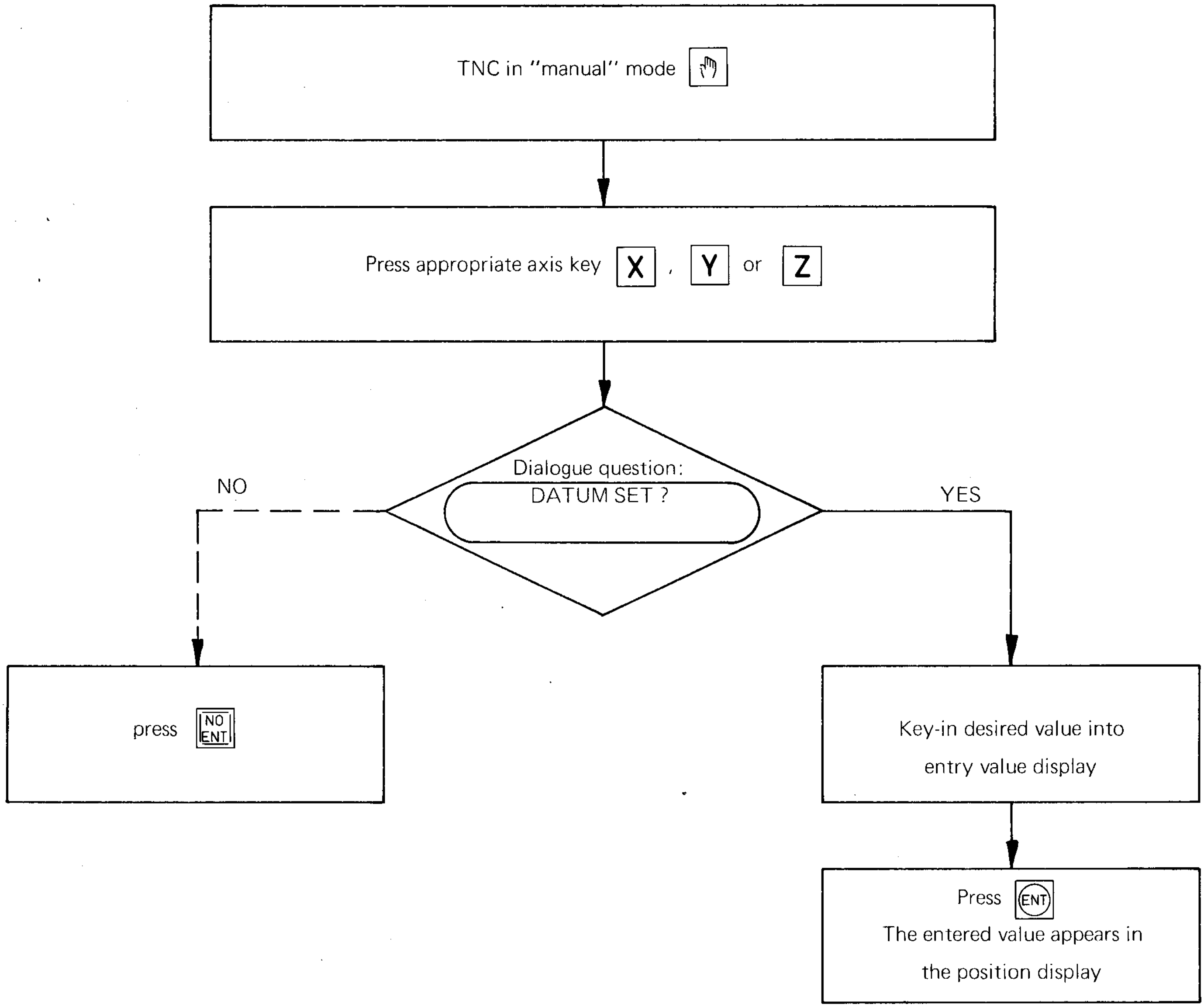
(Datum value Z) = (tool length L 1) + (surface position)

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:



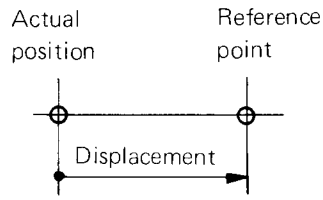
Please note:

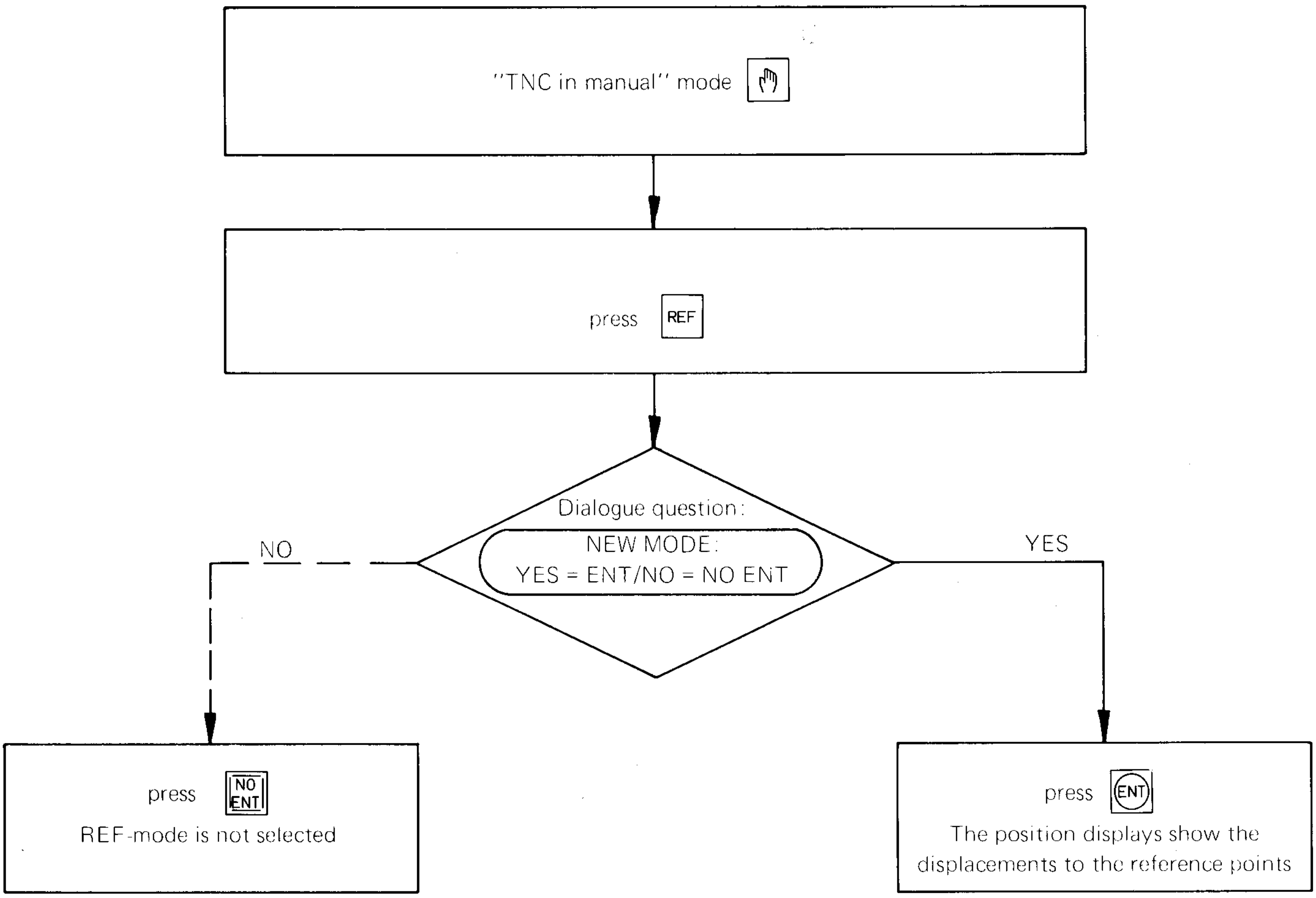
If REF is on, the datum cannot be set (see section G 3) !

G 3) Operating mode REF

The displayed actual position values are referred to a pre-determined datum. The position displays will only show the displacement (difference between actual value co-ordinates and REF-values) when the "REF" operating mode is ON.

Switching on of the REF -function is as follows:





Please note:
The "REF" mode is switched off by re-pressing REF and ENT .

H) Operating mode "Electronic handwheel"

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

The handwheel is made active when the -mode is on. Switch-over between axes is performed by pressing the appropriate TNC-axis key X, Y or Z.

The traversing speed is determined by the subdividing factor.
The required subdividing factor is keyed-in and transferred by pressing the ENT -key.
Available entry values: 1 . . . 10.

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

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

Please note:
The external axis direction buttons remain active in this mode!

I) Operating mode "Programming" [⊕]

I 1) Tool compensation

I 1.1) Tool definition [TOOL DEF]

The TNC 145 C allows for tool compensation – therefore the entry of a machining program can be made directly from the drawing dimensions or 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. The conventient search routine facility enables a certain tool definition to be easily called up for inspection or amendment (see section I 8.6).

Please note:
Whilst the length of the tool is being automatically taken into account after call-up. The radius to be considered it **must be determined** in every positioning block. A strict distinction must be made between **single axis positioning** (as with TNC 135) and **contouring operation.**

As explained in section I 3 ("Single axis positioning block") and I 4 ("Contour programming") the radius compensation works differently in both modes.

Dialogue initiation: [TOOL DEF] -key

(Dialogue question: TOOL NUMBER ?)

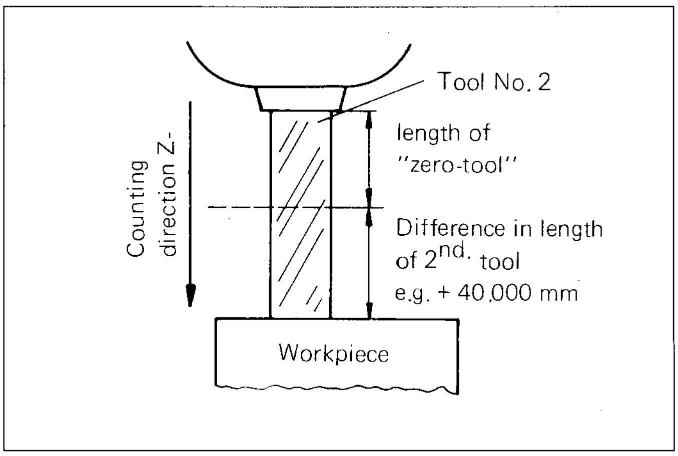
Possible entry values:
.for machines **without** automatic tool change: 1 - 255
.for machines **with** automatic tool change: 1 - 99
(the control only provides the tool numbers 1-99 in coded form).

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

(Dialogue question: TOOL LENGTH L ?)

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

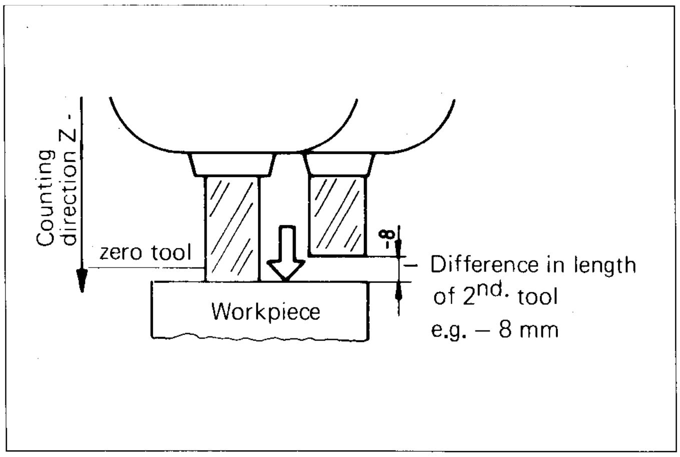
a) Tools in chuck without length stop
Firstly, the datum of the tool axis must be defined (see section G 2). The surface of the workpiece is touched with the tip of the first tool and the position display of the appropriate axis (e.g. Z-axis) is pre-set. The first tool is defined as "zero-tool", i.e.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



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-setter.

Dialogue question: TOOL RADIUS R ?

The tool radius is entered as a positive value. The compensation **direction** is determined within the positioning block.

Special case:
Programming of axis-parallel positioning blocks in the Playback-mode (see section I 3.2).
In this particular case, the machine is manually traversed (handwheel, axis-key) to the position which is to be stored. This position value incorporates the length and radius compensation for the tool being used. In the tool definition for this tool 1, the values L 1 = 0, R 1 = 0 are to be entered and the radius R 1 of the tool being used is to be noted down. The programming of the positioning block in Playback-mode takes place in each case with the entry of the appropriate radius compensation R+, R-, R0.

In the event of a tool breaking and insertion of a new tool which has a radius R 2 different to that of R 1, only the difference in radius has to be entered as compensation.

Radius compensation = R 2 — R 1

This radius compensation value can be **positive or negative** and is to be entered into the tool radius definition for R 1 including the calculated arithmetical sign. The length compensation must also be re-entered.

Please note:
The tool definition allocates a program block.

... TOOL DEF ... L ...
R ...

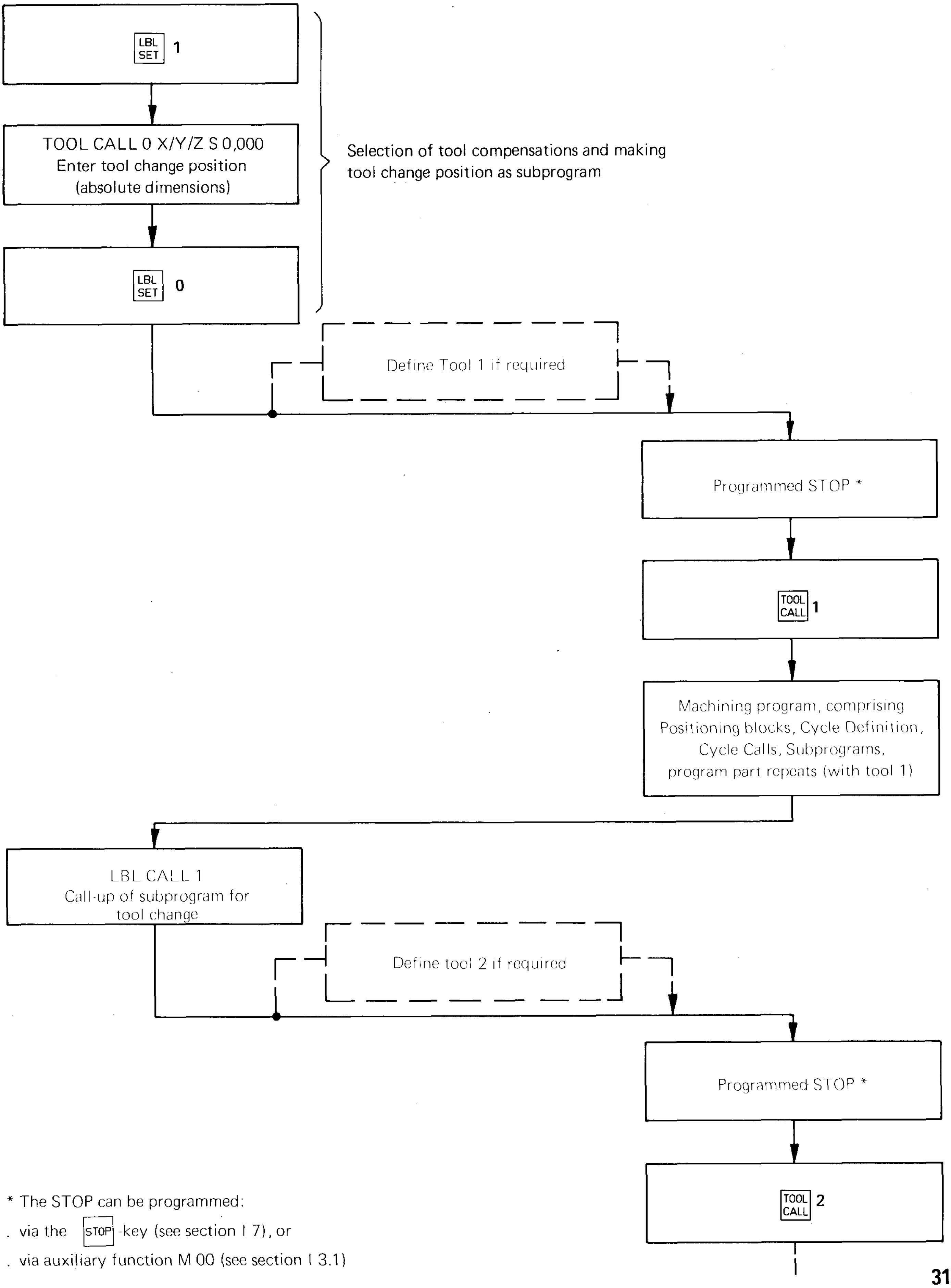
I 1.2) Tool Call / Tool Change TOOL CALL

With tool change, the data (length and radius) for the new tool must be called-up via the TOOL CALL -key.

Please note:

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



* The STOP can be programmed:

. via the STOP -key (see section I 7), or

. via auxiliary function M 00 (see section I 3.1)

Dialogue initiation: TOOL CALL -key

Dialogue question: TOOL NUMBER ?

Possible entry values: 0 - 255 (with automatic tool change 0 - 99)

Please note:
If, after a tool call the machine is to be traversed without compensation (as an exception), a tool call with the number 0 has already been pre-programmed for "no tool", i.e. length L = 0 and radius R = 0).

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

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.).

Dialogue question: SPINDLE SPEED S (RPM) ?

Entry is with a maximum of 4 digits in rev./min.
If necessary, the control rounds-off the value to the next standard value.

Please note:
When entering the machine data, the machine tool manufacturer lays down a series of spindle speeds. If an rpm is programmed which is not within this range, the fault signal "WRONG RPM" is indicated during program run.

The tool call only requires one program block.

... TOOL CALL ... X/Y/Z S ...

The following spindle speeds are programmable:

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

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.

... TOOL CALL 0 X/Y/Z S ...

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.

I 2) Entry of nominal position values in absolute or incremental dimensions I

Nominal position values are either in absolute or incremental (chain) dimensions (see section B 2). This applies to Cartesian co-ordinates (X, Y, Z) and polar co-ordinates (PR, PA).
With an absolute dimension, the indicator lamp of the I -key must be off and with incremental dimensions the I -key must be on. For change-over absolute/incremental, press the I -key.

The I -key can be pressed either before or after dialogue initiation of a positioning block (until the ENT -key is pressed).

I 3) Single axis positioning block

I 3.1) Programming via keyboard

The TNC 145 C is a contouring control with linear and circular interpolation. Correspondingly, single axis positioning blocks may be programmed in linear interpolation as a special case. On the other hand – as a unique feature – the TNC 145 C can be used for single axis programming as with the HEIDENHAIN-controls TNC 131/135.

The dialogue is initiated with an axis key X, Y or Z.

Dialogue question: POSITION VALUE ?

Enter nominal position value (absolute or incremental)

Dialogue question: TOOL RADIUS COMP. R+/R-/ NO COMP. ?

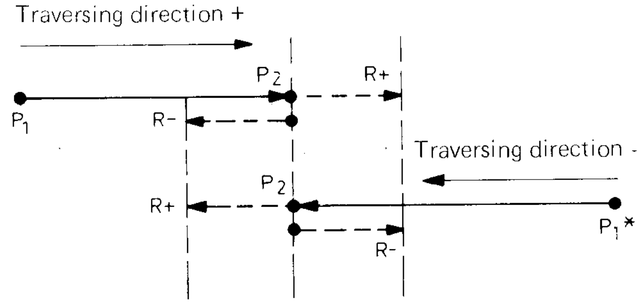
It should now be determined whether the traversing distance is to be extended (key R^R_+) or shortened (key R^L_-) due to radius compensation.

Press the appropriate key – the indicator lamp is then on.

Please note:
The designation R^R_+ and R^L_- is due to the double function of these keys.
With single, the keys merely serve as "R+" and "R-" compensation keys.

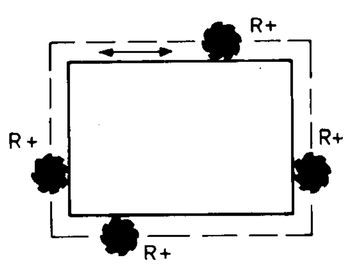
With contouring operation, entry via these keys determines whether the tool is located to the right = R^R or to the left = R^L of the contour in direction of tool motion (see section I 4 "Contour programming").

However, with a **single axis positioning block**, the pressing of the R^R_+ and R^L_- -keys determines the compensation as shown in the adjacent sketch.



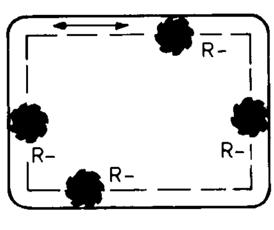
Example of tool radius compensation on an external contour.

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



If no tool radius compensation is required, both lamps are to be off.

R+ and R- are switched off by pressing the key, the lamp of which, is on.

Please note:
The dialogue question which refers to tool radius compensation also appears when a positioning block is entered for the axis which has been defined as the working spindle axis. Calculation of tool radius compensation does not take place in this axis even if R+, R- or R0 is entered.

(Dialogue question: FEED RATE ? F = . . .)

Entry of required feed in mm/min. or 0.1 inch/min. with a maximum of four digits. The max. feed rate (rapid traverse) is programmed with F 9999.

(Dialogue question: AUXILIARY FUNCTION M ?)

An auxiliary (miscellaneous) or relay function is programmed with an M-word. For instance, the working spindle can be switched on or off.

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.
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 anti-clockwise" and "coolant ON".
M 25	"Spindle HALT", Reference mark approach and "program run STOP". **The function of the M 25 depends on the machine parameters entered.** More exact information can be obtained from the machine tool manufacturer.
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 starting procedure.** More exact information can be obtained from the machine tool manufacturer.
M 95	Change of approach behaviour at beginning of contour (see section I 4.6.1)
M 96	Change of approach behaviour at beginning of contour (see section I 4.6.1)
M 97	Correction of tool path intersection for external corners (see section I 4.6.1)
M 98	Contour offset completed (see section I 4.1).
M 99	Same functions as "CYCL CALL".

If several M-functions are required for one block and these have not been accomodated in previous blocks, this can be overcome by programming each individual M-function with a positioning block which has been programmed in the incremental mode and to the position value "zero". The number of positioning blocks should correspond to the required number of M-functions.

If an M-function is not required in a programmed block, press ENT or NO ENT -key when M-question is displayed.

Please note:
Vacant M-functions which are still unassigned can be utilized by the machine tool manufacturer. These will be described in the machine operating manual.

M-Function (M-Functions which affect program run are indicated)	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 or	X
M 26	X	
M 27	X	
M 28	X	
M 29	X	
M 30		X

M-Function	At block beginning	end
M 31	X	
M 32		X
M 33		X
M 34		X
M 35		X
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-Function	At block beginning	end
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 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

*The output of M 25 is at block-beginning if no special function has been allocated with machine parameter 49 (MP 49 = 0). The output is at block-end if the function "reference mark approach" has been allocated with machine parameter 49 (MP 49 ≠ 0). The function of this machine parameter has been laid down by your machine tool manufacturer.

A positioning block allocates one program block:

. . . (I) X/Y/Z . . .
R+/R-/R0 F . . . M . . .

Please note:
Single axis positioning blocks which have been initiated with the axis keys [X], [Y] or [Z] cannot be inserted into a sequence of contours comprising linear path-blocks (dialogue initiation with [L] -key) or circular path blocks (dialogue initiation with [C] or [RND] -key).

Example of wrong 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:
Single axis positioning blocks (dialogue initiation with [X], [Y] or [Z]) can be inserted into a sequence of contours comprising contouring blocks which do not have tool radius compensation and positioning blocks for the tool-axis.

I 3.2) Programming with the transfer key for actual position data (Playback)

This type of programming is only applicable to single axis positioning. Programming of complex contours is not possible in the Playback-mode.

In -mode, traverse machine manually – via direction buttons – with the actual position displays having absolute position values.

↓

[X] [Y] or [Z].

↓

Transfer actual position value into entry display via -key, and enter into memory with [ENT].

↓

Specify tool radius compensation [R+ R] or [R- L] (see section I 3), Feed rate and Auxiliary function, and enter completed block into memory with [ENT].

↓

Traverse machine again in absolute.

I 4) Contour programming

I 4.1) Tool contouring offset $\boxed{R^R_+}$ $\boxed{R^L_-}$

The tool data (length and radius) are entered into the tool definition "TOOL DEF" and called-up via "TOOL CALL".

In addition to the tool radius compensation value, the control requires information as to how the radius is to be considered. For this, the $\boxed{R^R_+}$ and $\boxed{R^L_-}$ -keys are used.

For **contour programming**

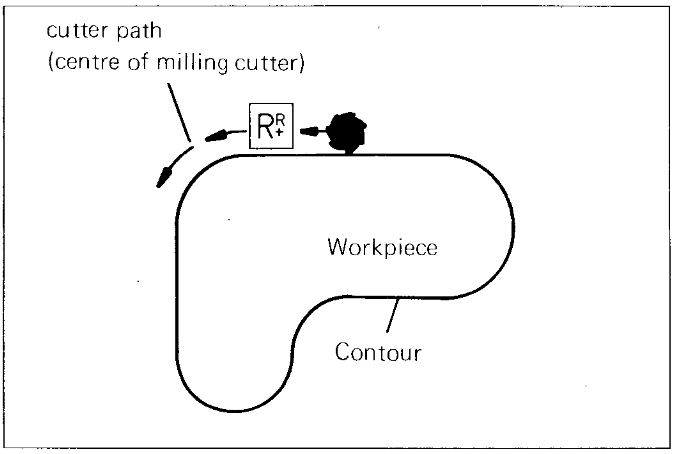
$\boxed{R^R_+}$ -key : In the traversing direction, the centre of the milling cutter travels on the right-hand side of the contour at a distance corresponding to the cutter radius.

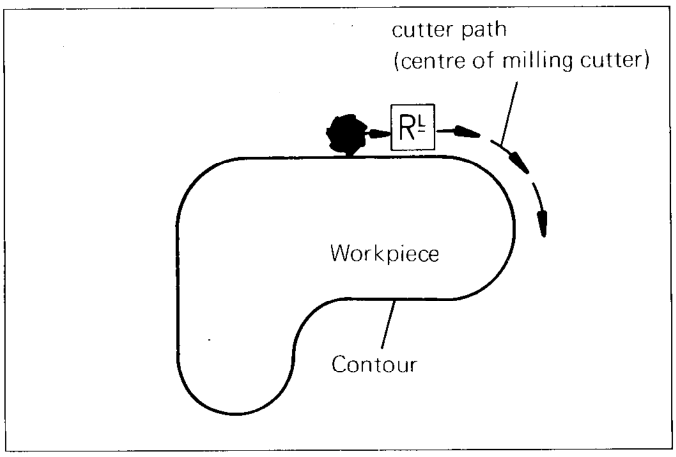
$\boxed{R^L_-}$ -key : In the traversing direction, the centre of the milling cutter travels on the left-hand side of the contour at a distance corresponding to the cutter radius.

(For **single axis positioning blocks**, the function of these keys is explained in section I 3).

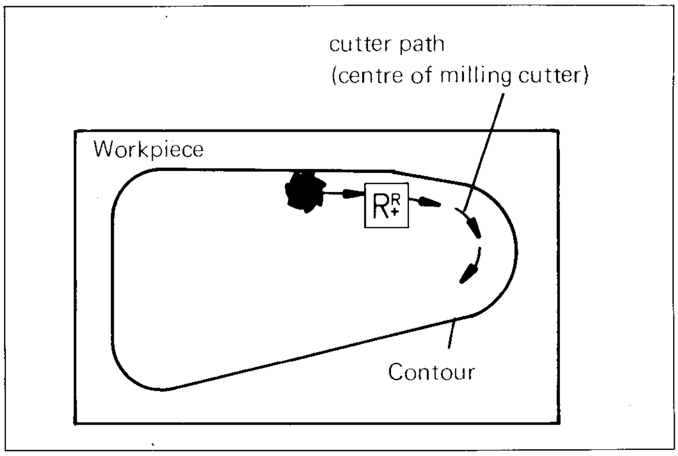
Examples:

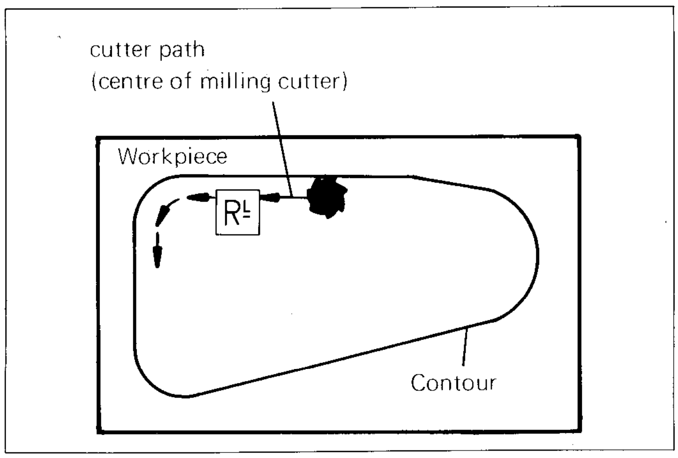
Milling of an external contour



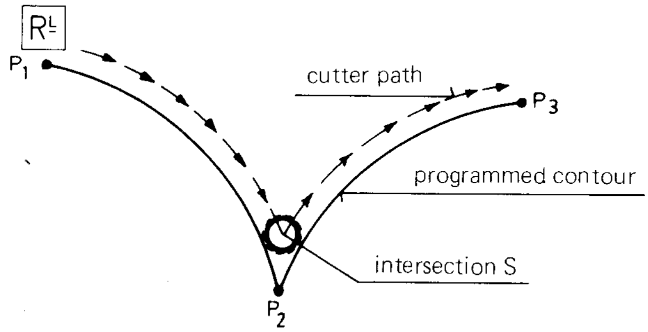


Milling of an internal contour



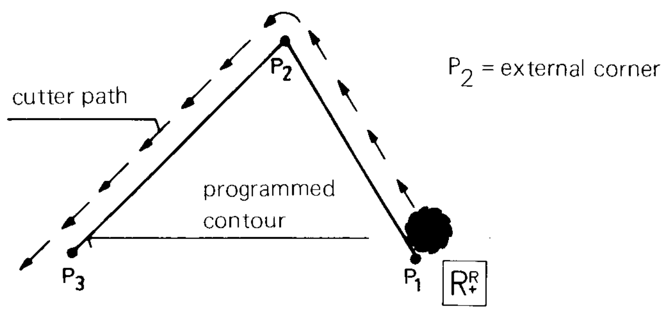


Determination of contour path intersection for internal corners



The TNC 145 C 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 cutter does not form a recess within the contour (thus damaging the workpiece).

Transitional arcs on external corners



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

Please note:
If a constant contouring speed has been impelled by programming the auxiliary function M 90, the programmed contouring speed is not reduced (see section I 4.7).

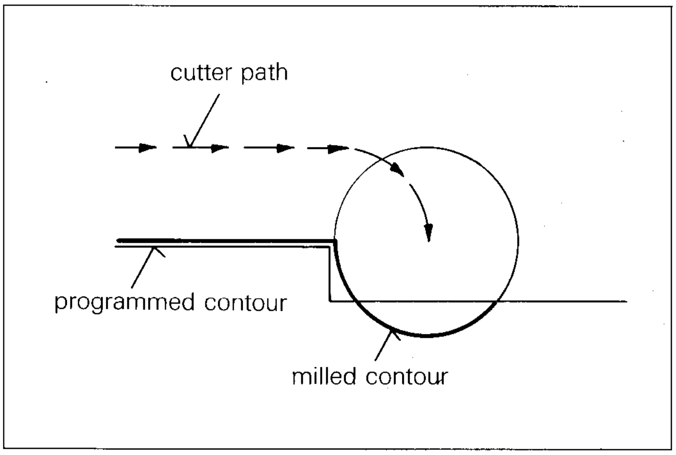
Correction of tool path intersection for external corners: M 97

In contouring operation with tool compensation, the TNC 145 C automatically inserts a transitional radius for external corners. If insertion of a transitional radius is not possible, the following error is displayed:

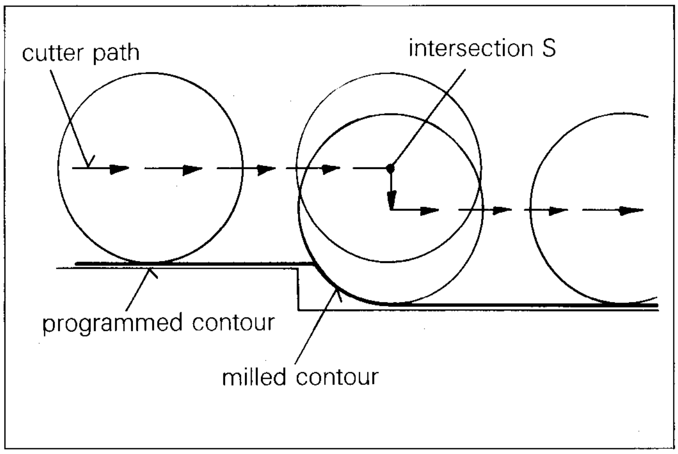
TOOL RADIUS TOO LARGE

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:

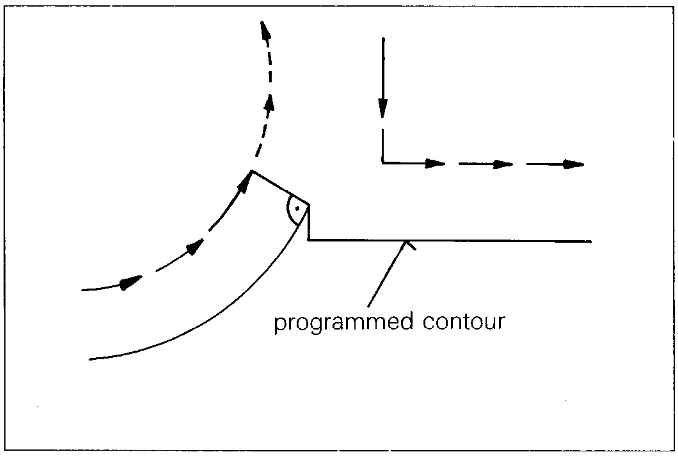


Without M 97: The transitional radius would damage the workpiece; the error "TOOL RADIUS TOO LARGE" is displayed.

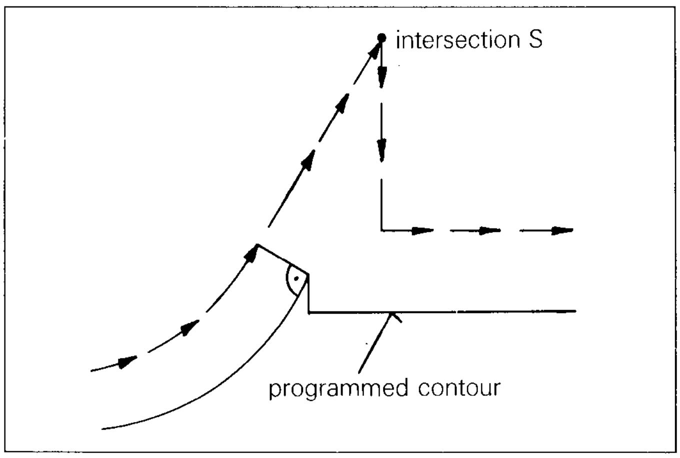


With M 97: No transitional radius is inserted; the control determines the tool path intersecting point S thus preventing damage to the workpiece.

Special case:



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



Remedy: A block is inserted:
L I X0,000 I Y0,000
RL/RR F . . . M 97
The control determines the point of intersection S and the contour can be milled.

I 4.2) Nominal position value entry in polar co-ordinates

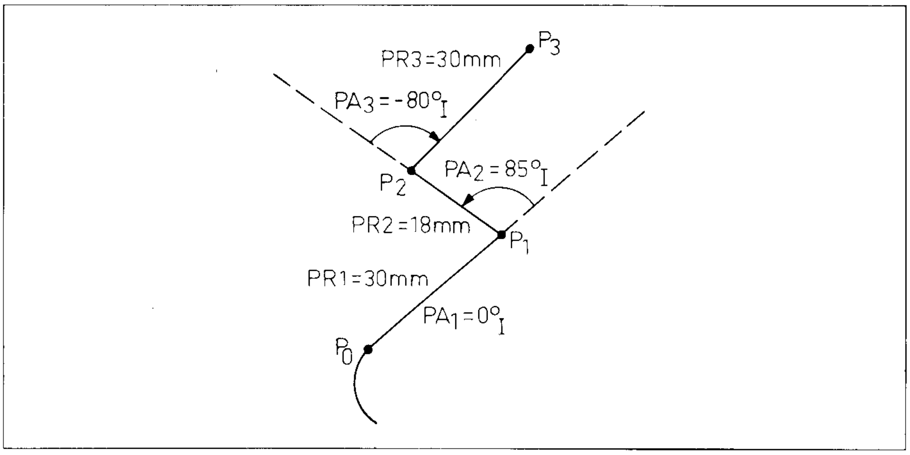
With contour-programming, nominal position values can also be entered in polar co-ordinates (see section B 3). First of all, the pole (co-ordinate origin) must be defined:

The pole 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.

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.

Please note:
With incremental programming of the polar co-ordinate angle, the angle is dereferenced to the previous uncorrected angle.

Pole programming is as follows:

Dialogue initiation: press [CC]

(Dialogue question: ENTER LAST POSITION ?)

If the last nominal position value is to be used as a pole, press [ENT] otherwise press [NO ENT].

If [NO ENT] is pressed, the following dialogue is displayed:

(Dialogue question: FIRST COORDINATE ?)

Enter first Cartesian co-ordinate of pole (absolute or incremental).

(Dialogue question: SECOND COORDINATE ?)

Enter second Cartesian co-ordinate or pole (absolute or incremental).
The definition of the pole allocates one program block.
. . . CC (I) X/Y/Z . . . (I) X/Y/Z . . .

If a positioning block is to be entered in polar co-ordinates, the [L] -key must be pressed before initiation of the dialogue with the [C] or [P] -key (see section I 4.3 or I 4.4)!

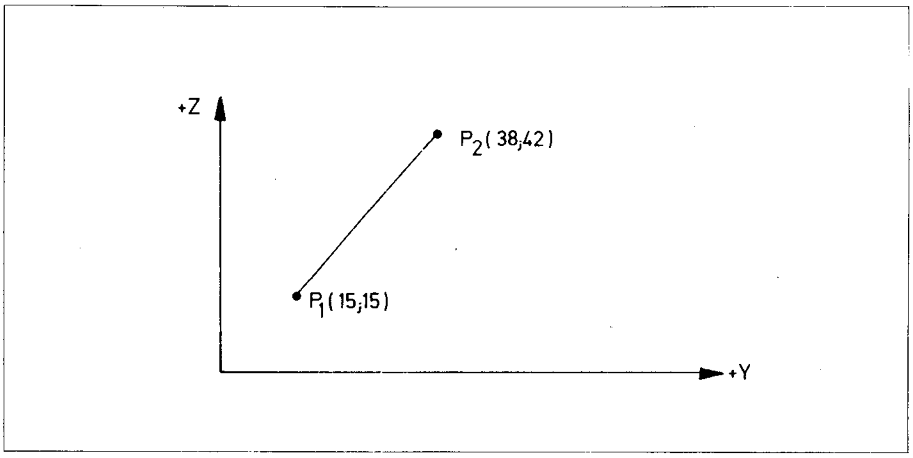
The polar co-ordinate-radius PR is entered in mm or inch (entry step 0.001 mm or 0.0001 inch) and the polar co-ordinate-angle in angular degrees (entry step down to 0.001°).
After the entry of polar co-ordinate-blocks has been finalised, the Cartesian datum which was set prior to the change-over to polar co-ordinate operation is still available for further programming in Cartesian co-ordinates.

I 4.3) Linear interpolation [L↗]

I 4.3.1) Linear interpolation with right-angled (Cartesian) co-ordinates

Example: Planes Y, Z

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.



Dialogue initiation: press [L↗]

Dialogue question: FIRST COORDINATE ?

Enter first co-ordinate of nominal position (absolute or incremental).

Dialogue question: SECOND COORDINATE ?

Enter second co-ordinate of nominal position (absolute or incremental).

Dialogue question: TOOL RADIUS COMP. R+/R-/NO COMP. ?

Enter contouring offset (see section I 4.1).

Dialogue question: FEED RATE ? F =

Enter contouring feed (see section I 3.1).

Dialogue question: AUXILIARY FUNCITON M ?

Enter auxiliary function (see section I 3.1)

The linear interpolation allocates one program block.

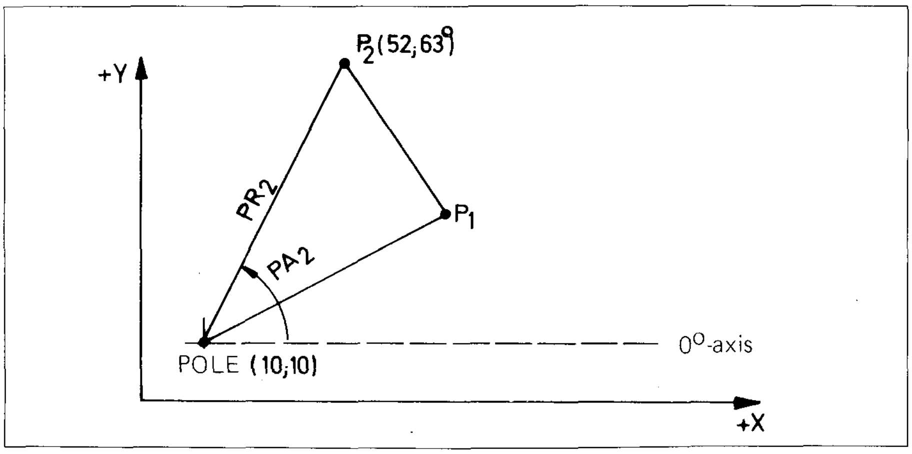
. . . L (I) X/Y/Z . . . (I) X/Y/Z . . .
RL/RR/R0 F . . . M . . .

Please note:
If single axis positioning is programmed in the contouring mode, the dialogue question which asks for the second co-ordinate is responded to by pressing [NO ENT].

A single axis positioning block, – the dialogue of which is initiated with the axis keys [X], [Y] or [Z] – may not be programmed within a sequence of blocks which describes a closed contour in the contouring mode (see section I 3.1).

I 4.3.2) Linear interpolation with polar co-ordinates

Polar co-ordinates are selected when the lamp beneath the $\boxed{P}$ -key is on.



Example: plane X, Y

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 $PA_2 = 63^o$. The machine will traverse in a straight path from point P_1 to point P_2.

Please note:
Before initiation of dialogue for linear interpolation the "pole" must be defined in **Cartesian co-ordinates** (see section I 4.2)!

Dialogue question: POLAR COORDINATES-RADIUS PR ?

Enter radius "PR" of nominal position (absolute or incremental)

Dialogue question: POLAR COORDINATES-ANGLE PA ?

Enter angle "PA" of nominal position (absolute or incremental)

Dialogue question: TOOL RADIUS COMP. R+/R-/NO COMP. ?

Enter contouring offset (see section I 4.1).

Dialogue question: FEED RATE ? F =

Enter contouring feed (see section I 3.1)

Dialogue question: AUXILIARY FUNCTION M ?

Enter auxiliary function (see section I 3.1)

Linear interpolation in polar co-ordinates allocates one program block.

. . . LP (I) PR . . . (I) PA . . .
RL/RR/R0 F . . . M . . .

I 4.4) Circular path

I 4.4.1) Definition of circle centre CC

The CC -key is used for defining the circle centre (CC).

Dialogue initiation: press CC (see also section I 4.2)

(Dialogue question: ENTER LAST POSITION ?)

If the last position is to be used as circle centre, press ENT otherwise press NO ENT

If NO ENT is pressed, the following dialogue is displayed:

(Dialogue question: FIRST COORDINATE ?)

Enter first Cartesian co-ordinate of CC (absolute or incremental).

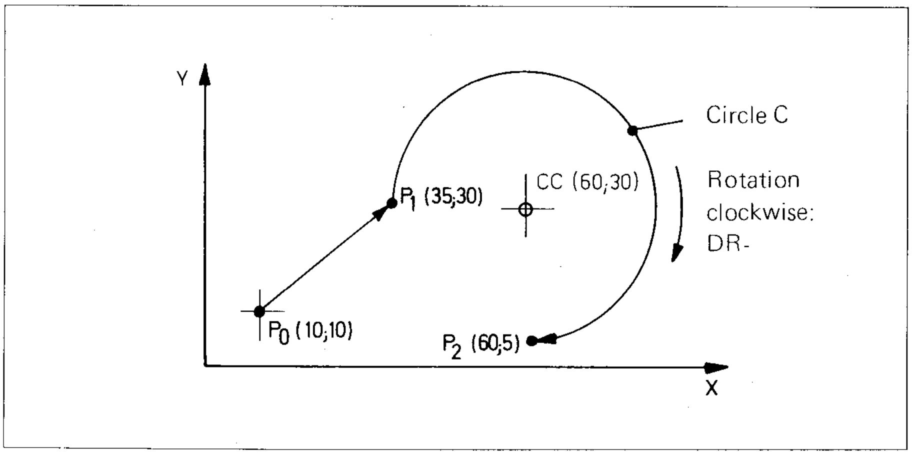
(Dialogue question: SECOND COORDINATE ?)

Enter second Cartesian co-ordinate of CC (absolute or incremental).

The definition of the circle centre allocates one program block:

... CC (I) X/Y/Z... (I) X/Y/Z ...

I 4.4.2) Circular path programming with right-angled (Cartesian co-ordinates)



Example: plane X, Y

Point P_1 is defined in a positioning block. Then the circle centre CC and the end position of the arc P_2 are to be programmed.

Dialogue initiation: press [C key]

Dialogue question: FIRST COORDINATE ?

Enter first co-ordinate of end-position of arc (absolute or incremental)

Dialogue question: SECOND COORDINATE ?

Enter second co-ordinate of end-position of arc (absolute or incremental).

Dialogue question: ROTATION CLOCKWISE: DR - ?

Response by pressing [ENT] rotation clockwise (right-hand rotation = DR -) is selected

Response by pressing [NO ENT] rotation anti-clockwise = DR + is selected

Dialogue question: TOOL RADIUS COMP. R+/R-/NO COMP. ?

Enter contouring offset (see section I 4.1).

Dialogue question: FEED RATE ? F =

Enter contouring feed (see section I 3.1).

Dialogue question: AUXILIARY FUNCTION M ?

Enter auxiliary function (see section I 3.1)

Circular interpolation allocates one program block.

... C (I) X/Y/Z ... (I) X/Y/Z ...
DR-/DR+ RL/RR/R0 F ... M ...

I 4.4.3) Circular path programming with polar co-ordinates

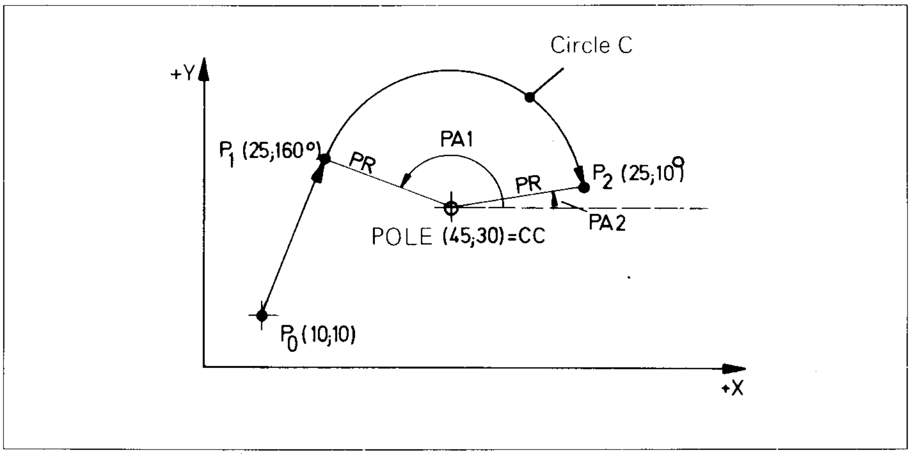
Polar co-ordinates are selected when the lamp beneath the [P] -key is on.

Example: plane X, Y

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

Point P_2 may also be programmed incrementally:

PR_2 = 0 (incremental)
PA_2 = -150° (incremental)



Dialogue initiation: press [C]

(Dialogue question: POLAR COORDINATES-RADIUS PR ?)

Enter radius "PR" of arc end position (absolute or incremental).

(Dialogue question: POLAR COORDINATES-ANGLE PA ?)

Enter angle "PA" of arc end position (absolute or incremental).

(Dialogue question: ROTATION CLOCKWISE: DR - ?)

Response by pressing [ENT] clockwise rotation (DR-) is selected.

Response by pressing [NO ENT] anti-clockwise rotation (DR+) is selected.

(Dialogue question: TOOL RADIUS COMP. R+/R-/NO COMP. ?)

Enter contouring offset (see section I 4.1)

(Dialogue question: FEED RATE ? F =)

Enter feed for path of tool centre (see section I 3.1).

(Dialogue question: AUXILIARY FUNCTION M ?)

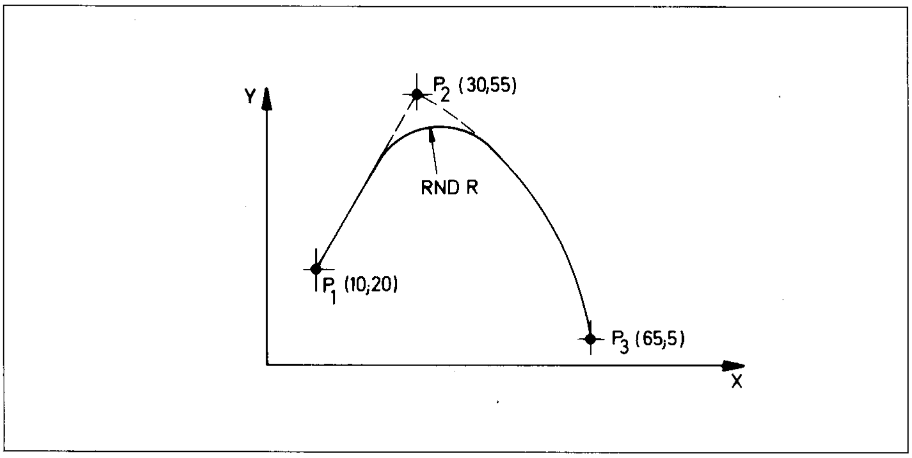
Enter auxiliary function (see section I 3.1)

Circular interpolation allocates one program block.

... CP (I) PR ... **(I) PA ...**
DR+/DR- **RL/RR/R0** **F ... M ...**

I 4.5) Rounding of corners RND (Arcs with tangential transitions)

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

Programming sequence:

.the contour $\overline{P_1 P_2}$ (with tool offset RR or RL)
.the rounding off block with rounding off radius
.the contour $\overset{\frown}{P_2 P_3}$ (with tool offset RR or RL)

Please note:
The control **only** requires the **rounding off-radius** (all further calculations are performed by the TNC 145 C itself).

Dialogue initiation: press RND

(Dialogue question: ROUNDING-OFF RADIUS R ?)

Enter required rounding off radius.

"Rounding of corners" allocates one program block.

... RND R ...

Please note:
.The rounding off block requires **contour programming** — it is not possible with single axis positioning blocks which have been initiated by the X , Y or Z -key. For rounding of corners single axis positioned contours must also be programmed in the contouring mode.
(Dialogue initiation with L -key).
.A rounding off block must be preceded or followed by a positioning block which contains both co-ordinates of the interpolation planes.

I 4.6) Approach to – and departure from a contour

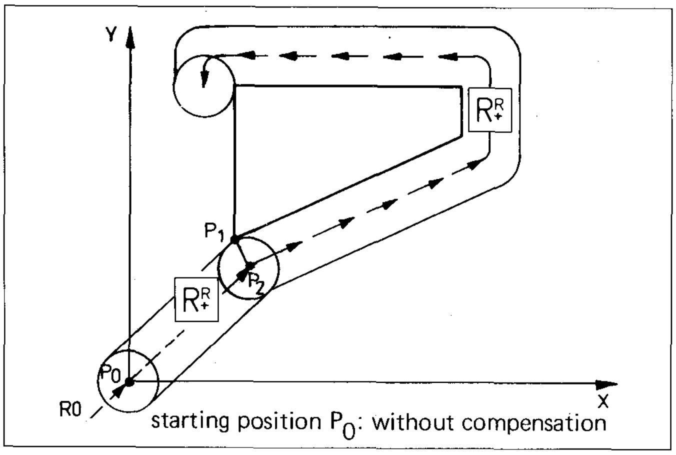
I 4.6.1) Perpendicular approach to – and departure from a contour

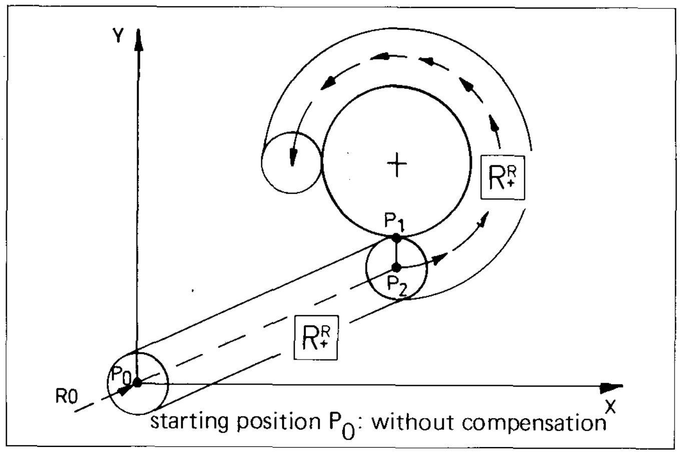
Approach to – and departure from a contour can take place in two ways:

Case 1:

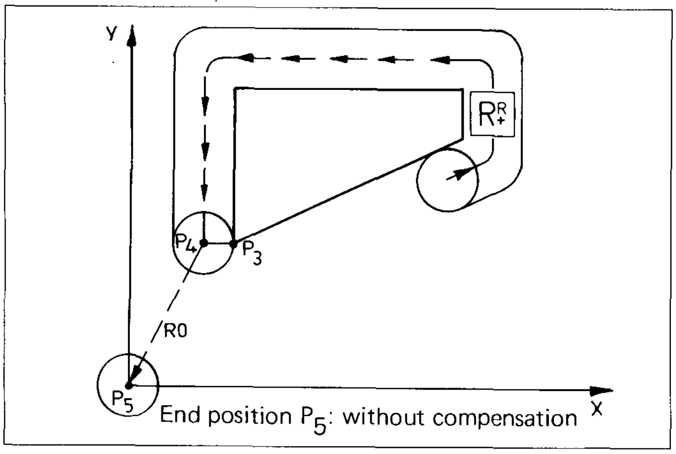
The starting position P_0 is approached without radius compensation (R0). The following positioning block to point P_1 is programmed with radius offset RR or RL.

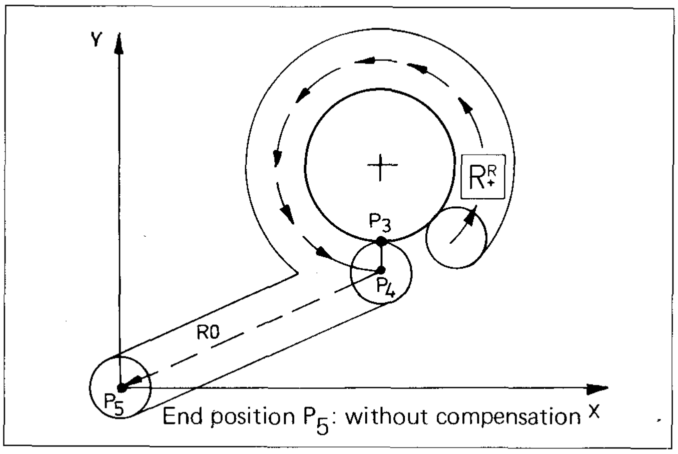
When approaching the contour the control automatically calculates the auxiliary point P_2 away from P_1. Point P_2 is calculated by constructing a perpendicular at the beginning of the contour. The distance between P_2 and P_1 corresponds to the radius programmed in the tool definition.

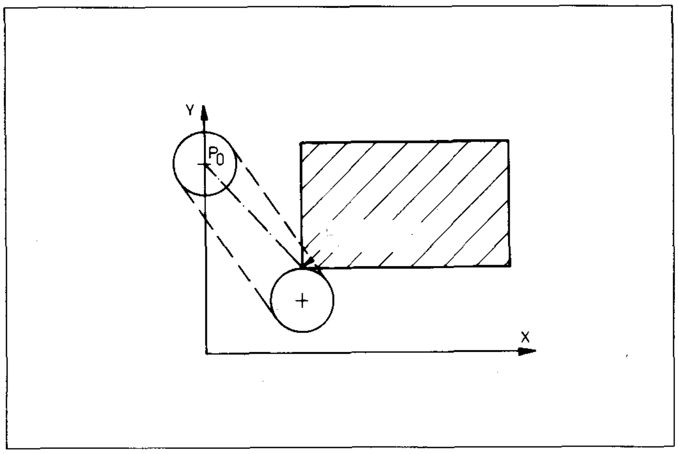




When leaving the contour by approaching the end position P_5 without compensation (R0), the control automatically calculates the end point P_4 of the contour by constructing a perpendicular to the final point of the contour P_3.

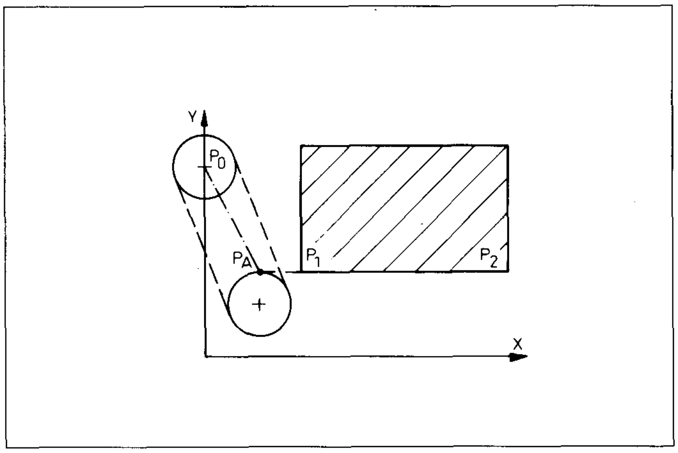




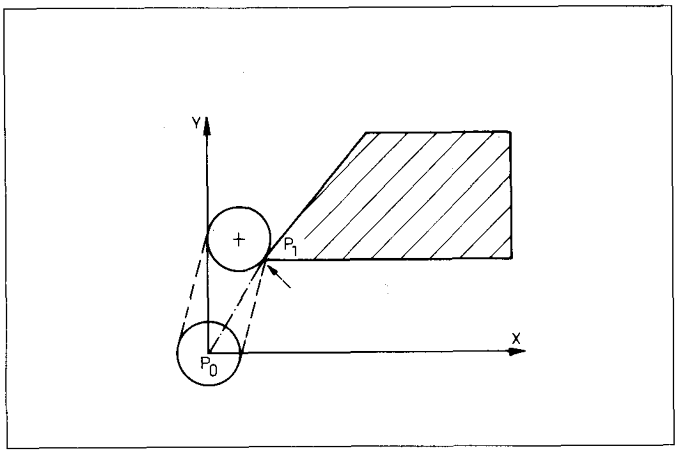


Please note:

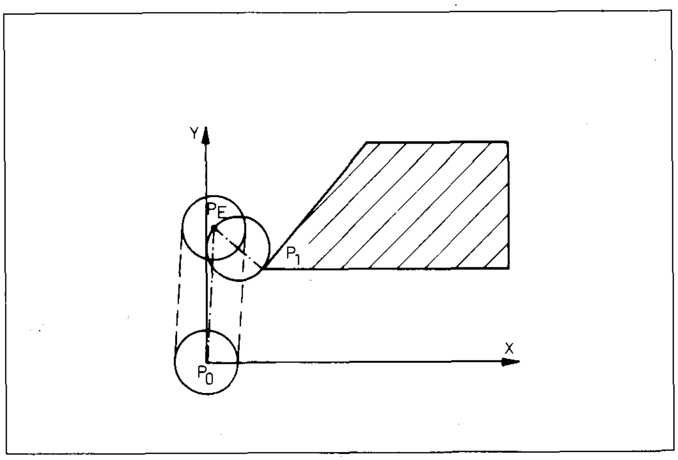
When **approaching a contour,** e.g. from a tool change position P_0, a collision with the workpiece must be prevented. This is also applicable to contour programming with contour offset.



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.



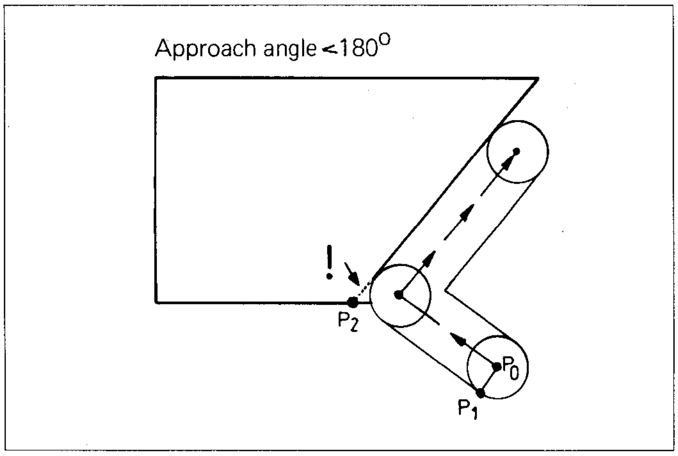
Therefore, an auxiliary point P_E must also be programmed at a safe distance from the workpiece. This point, however, is approached without contour offset. This also applies for the return traverse to the tool change position P_0.

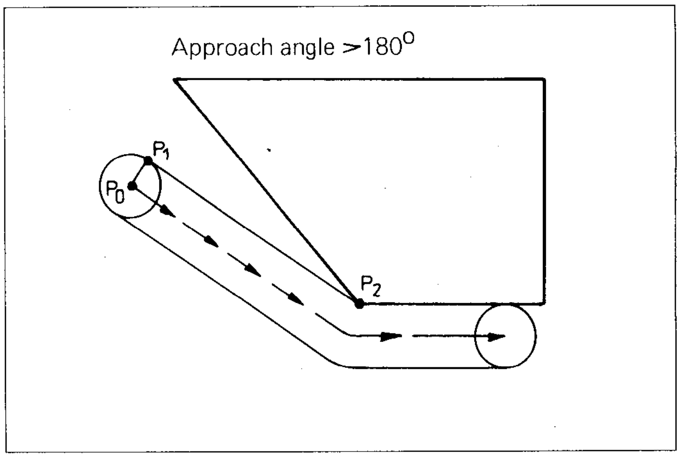
Case 2:

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 180^0, the bisection of the angle is approached.

.if the approach angle to the contour is greater than 180^0, a transitional arc is inserted.





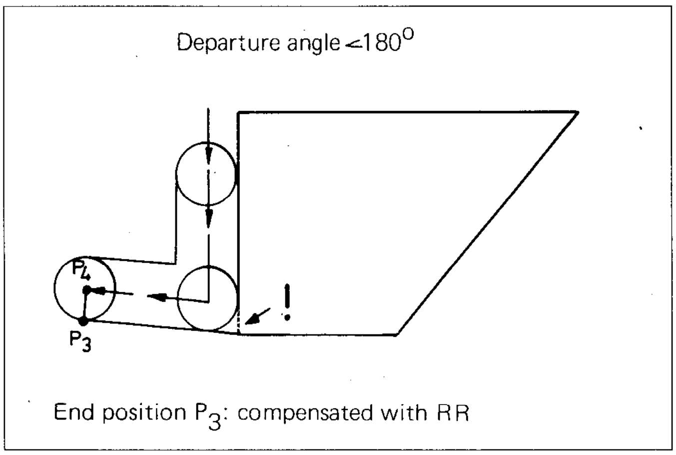
The program block for leaving the contour also contains radius offset RR or RL. Contour correction is terminated in this case with

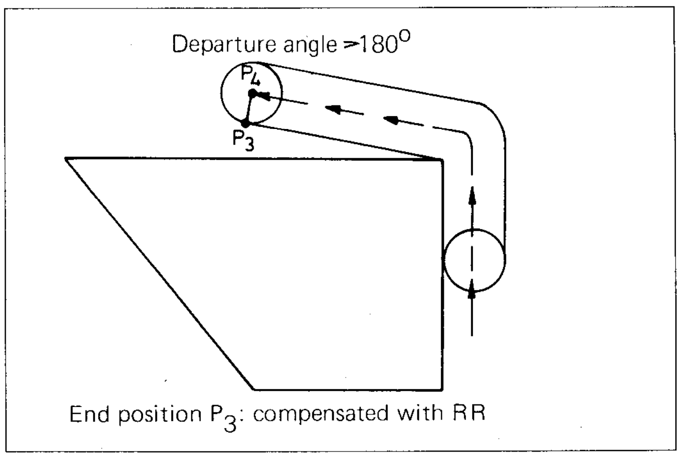
.the auxiliary function M 98 or

.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.





Please note:

If the approach angle is less than 180^0, the workpiece will not be completely machined (see! in the 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:

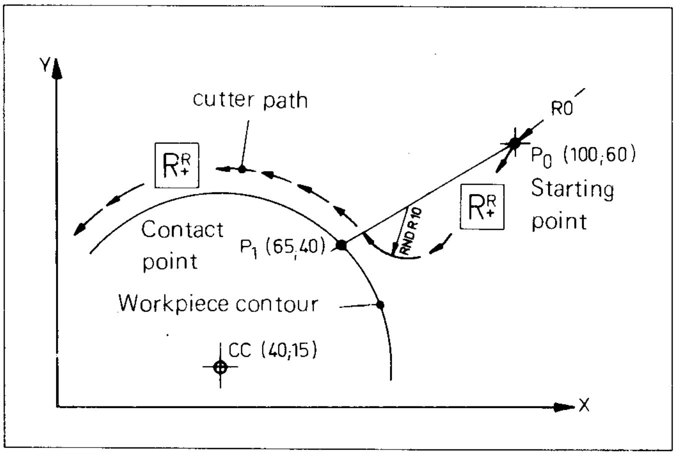
If normal approach corresponds to the first case, the second case can be applied by programming **M 96.**

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

I 4.6.2) Tangential contour approach and departure

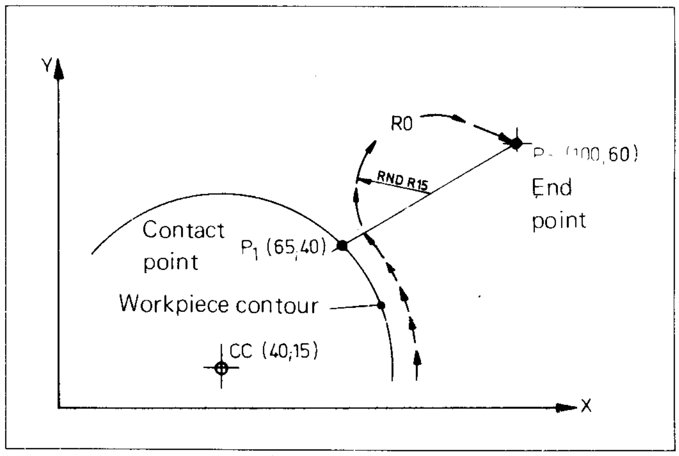
The [RND] -key serves in programming the smooth tangential approach to a contour and rounding of corners (see section I 4.5). An arc or a straight line can be approached by means of a smooth tangential arc to a desired point of contact and at a determined contouring speed:

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.

Leaving 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:

1	L	X	100.000	Y	60.000		
			R0	F	9999	M 03	Starting point is positioned
2	L	X	65.000	Y	40.000		Contact point
			RR	F	50	M	and contouring speed are specified
3	RND	R 10					Rounding off-radius for smooth contour approach
4	CC	X	40.000	Y	15.000		Circle centre for workpiece contour
5	C	X	65.000	Y	40.000		
			DR+RR	F	50	M	Programming of workpiece contour
6	RND	R 15					Rounding off-radius for leaving contour
7	L	X	100.000	Y	60.000		
			R0	F	50	M 05	Return to starting point

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

I 4.7) Constant contouring speed at corners: M 90

The TNC 145 C 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.

Please note:
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.

I 5) 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 255 can be used for allocating labels. The label number "0" is always used as a mark for "end of subprogram".

Please note:
If a subprogram is to be machined at different locations, the subprogram must be programmed incrementally (with **incremental** nominal position values).

I 5.1) Setting label numbers [LBL SET]

Dialogue initiation: press [LBL SET]

(Dialogue question: LABEL NUMBER ?)

Enter desired number.

The setting of a label number allocates one program block.

... LBL ...

I 5.2) Jump to a label number [LBL CALL]

Dialogue initiation: press [LBL CALL]

(Dialogue question: LABEL NUMBER ?)

Enter label number which is to be called-up.

(Dialogue question: REPEAT REP ?)

Key-in **0** when the selected program label designates a **subprogram.**

Enter number of **repeats** when the selected program label designates a program part repeat.

A jump to a program label allocates one program block.

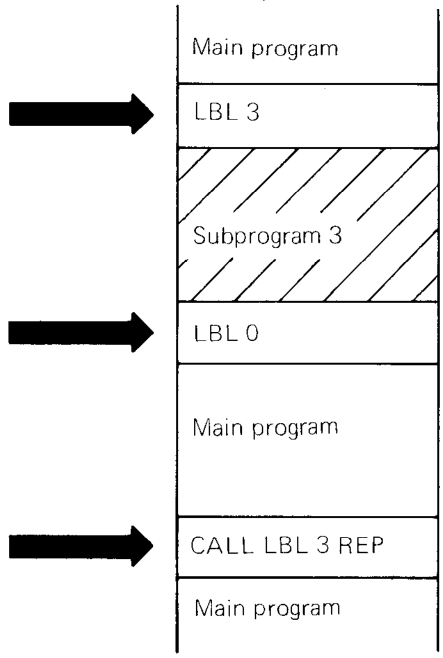
with call-up of a subprogram: **... CALL LBL ... REP**
or
with a program part repeat: **... CALL LBL ... REP .../...**

Schematic diagram of a subprogram:

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

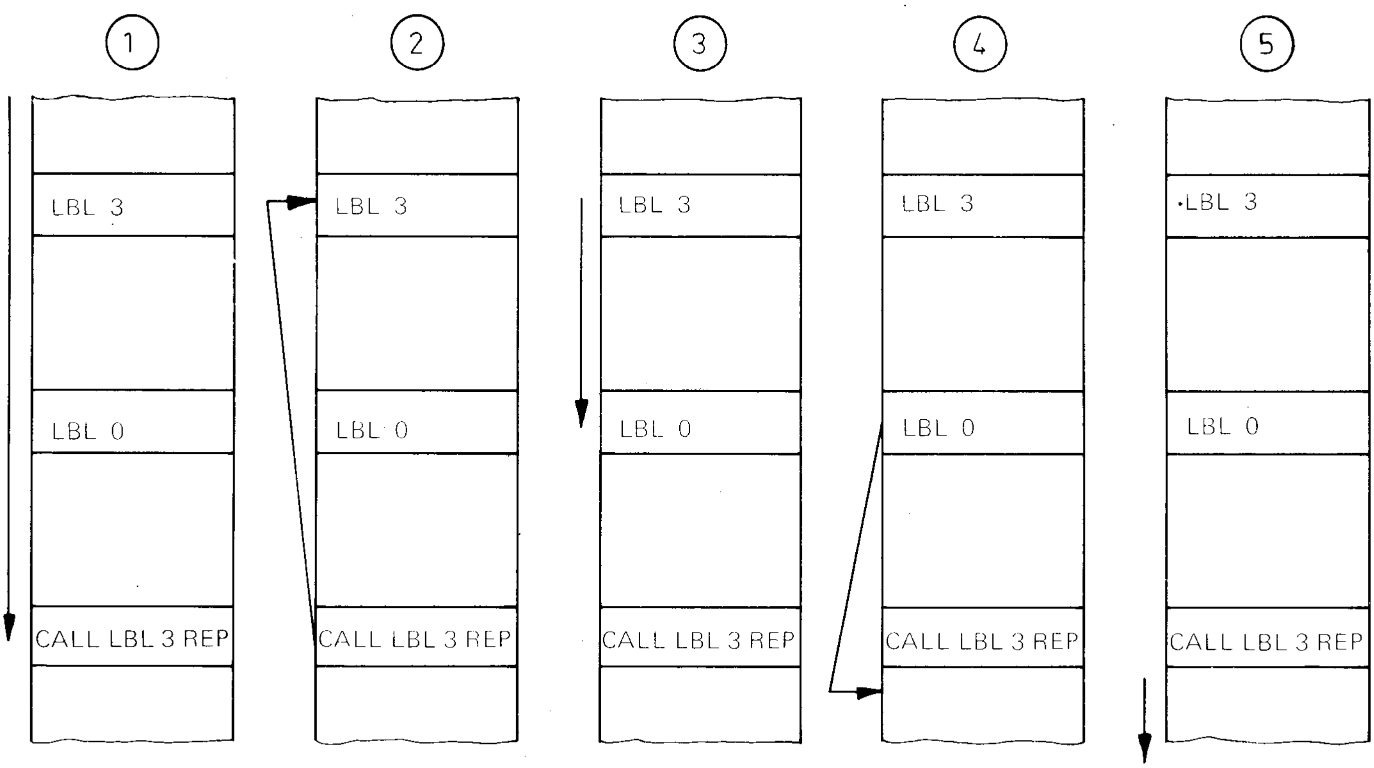
The end of the subprogram is labelled LBL 0.

By making a subprogramm call-up, the subprogramm 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 worked through, the main program sequence is resumed.



Please note:
After call-up, a **subprogram** can only be **worked through 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.

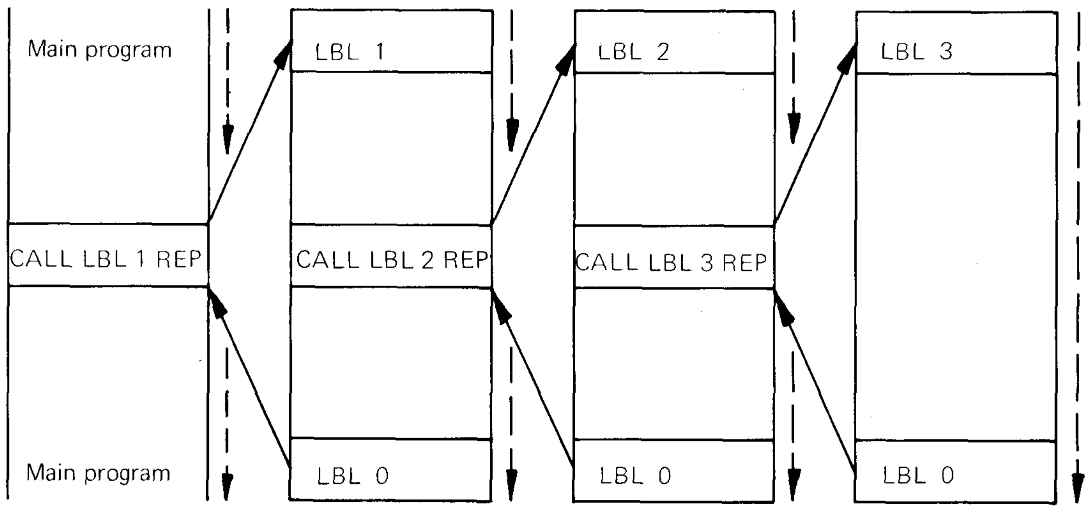
5. The main program is continued.

Please note:
A subprogram may not be defined within a subprogram!

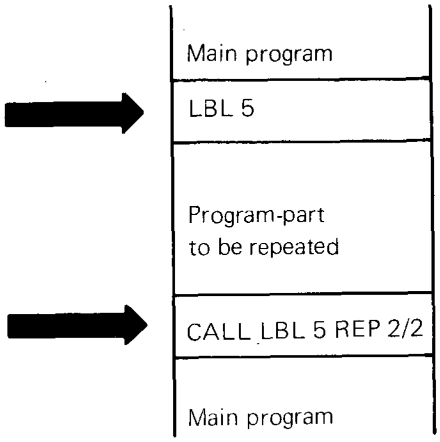
Nesting of subprograms

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":



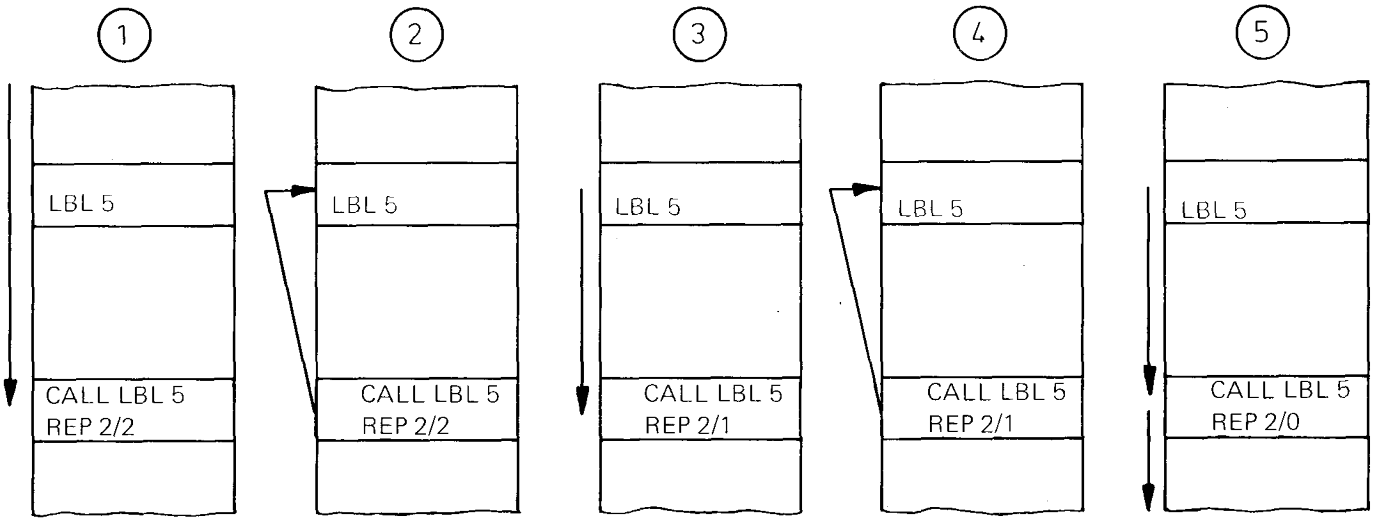
Schematic diagram of a program part repeat



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 worked through 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.

Please note:
A **program part repeat** may be programmed within a subprogram (see section I 5.3)

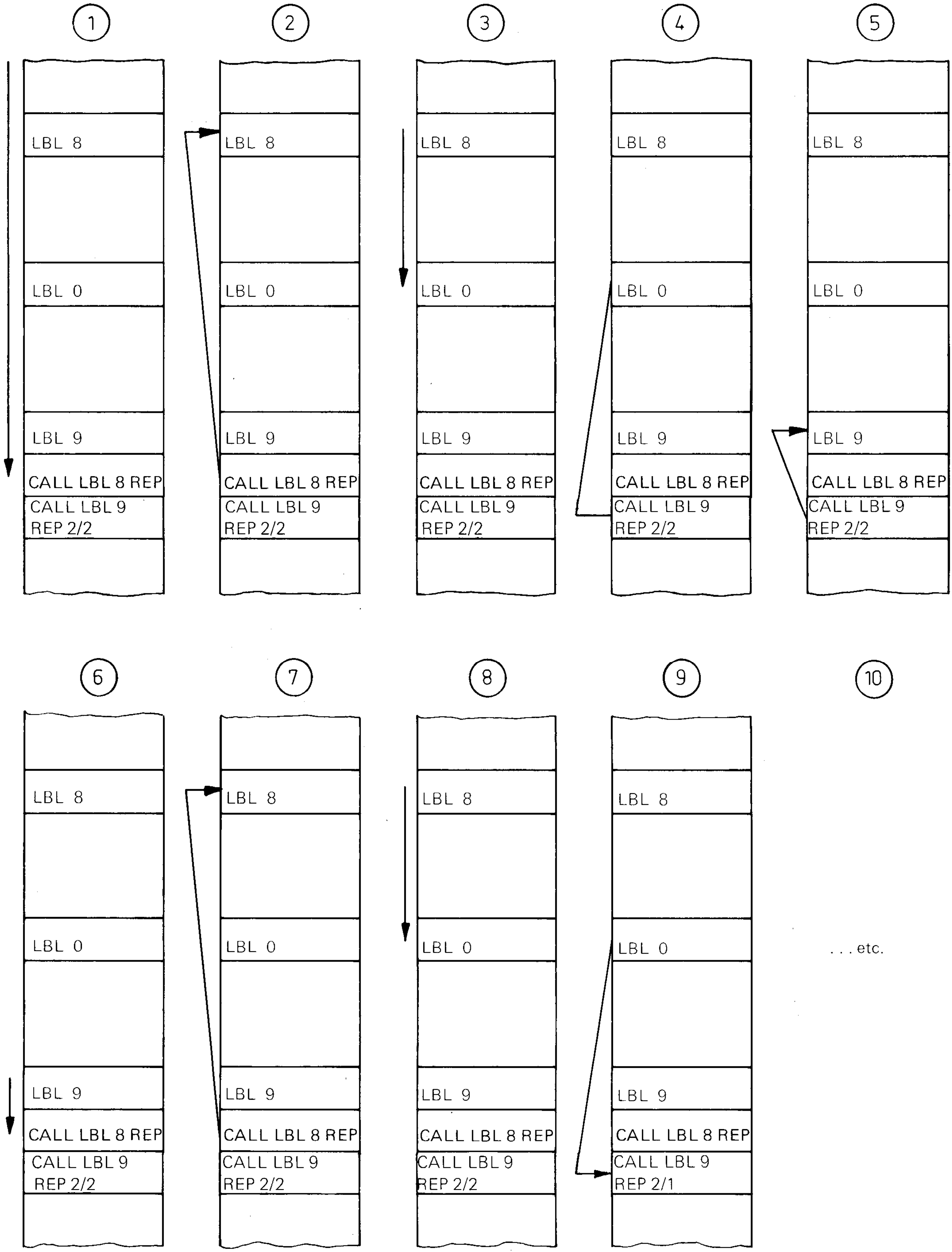
	Main program	
Program label for subprogram	LBL 12	
Program label for program part repeat	LBL 13	
	Program part to be repeated	Subprogram
Program part repeat	CALL LBL 13 REP 5/5	
Program label for "subprogram-end"	LBL 0	
	Main program	
Call-up of subprogram	CALL LBL 12 REP	
	Main program	

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:

	Main program
Program label for subprogram	LBL 8
	Subprogram
Program label for "subprogram-end"	LBL 0
	Main program
Program label for program part repeat.	LBL 9
Call-up of subprogram	CALL LBL 8 REP
Program part repeat for 2 x repetition of subprogram call-up	CALL LBL 9 REP 2/2
	Main program

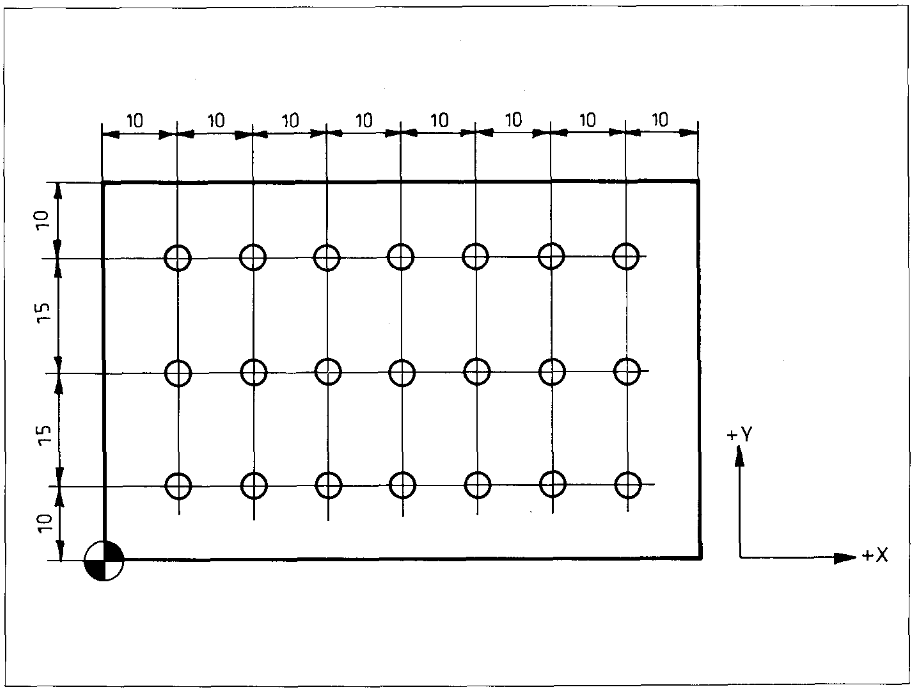
Please note:
If **two repeats** are programmed, the subprogram is **executed three times.**


1
LBL 8
LBL 0
LBL 9
CALL LBL 8 REP
CALL LBL 9 REP 2/2
2
LBL 8
LBL 0
LBL 9
CALL LBL 8 REP
CALL LBL 9 REP 2/2
3
LBL 8
LBL 0
LBL 9
CALL LBL 8 REP
CALL LBL 9 REP 2/2
4
LBL 8
LBL 0
LBL 9
CALL LBL 8 REP
CALL LBL 9 REP 2/2
5
LBL 8
LBL 0
LBL 9
CALL LBL 8 REP
CALL LBL 9 REP 2/2
6
LBL 8
LBL 0
LBL 9
CALL LBL 8 REP
CALL LBL 9 REP 2/2
7
LBL 8
LBL 0
LBL 9
CALL LBL 8 REP
CALL LBL 9 REP 2/2
8
LBL 8
LBL 0
LBL 9
CALL LBL 8 REP
CALL LBL 9 REP 2/2
9
LBL 8
LBL 0
LBL 9
CALL LBL 8 REP
CALL LBL 9 REP 2/1
10
. . . etc.

1. The main program is worked through 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 immeditately 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.

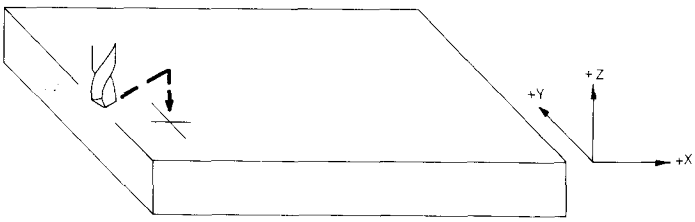
I 5.3) Programming of hole patterns via subprograms and program part repeats

Time consuming programming of hole patterns is made much more simple by using subprograms and program part repeats.

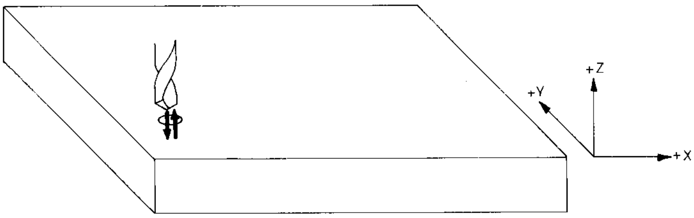


Programming procedure:

1. Select tool compensation	1 TOOL CALL 0 Z S 0,000
and traverse to tool-change position	2 L Z + 20,000 R0 F9999 M05
	3 L X - 20,000 Y - 20,000 R0 F9999 M
Tool definition and	4 TOOL DEF 1 LENGTH . . . RADIUS . . .
	5 STOP M
Tool call	6 TOOL CALL 1 Z S . . .
2. 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
3. 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

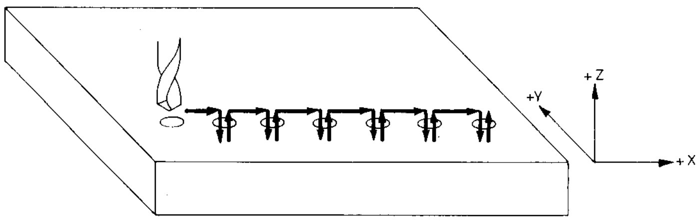


4. Peck-drilling of first hole



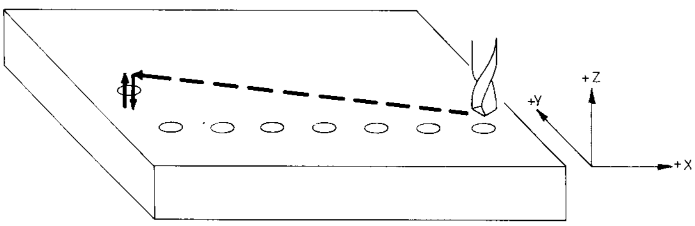

```
15 CYCL CALL
                                  M
```

5. Programming of first row in incremental dimensions with program part repeat and labelling of this program section as a subprogram



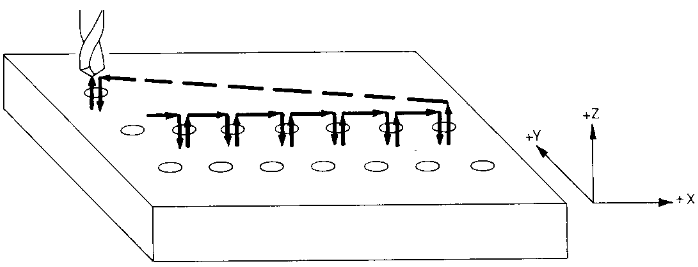

```
16 LBL 1
17 L    I  X + 10,000
                       R0  F9999  M
18 CYCL CALL

19 LBL CALL 1  REP 5/5
20 LBL 0
```

6. Traverse to second hole row (the Y-co-ordinate is programmed incrementally) and peck-drill first hole of row



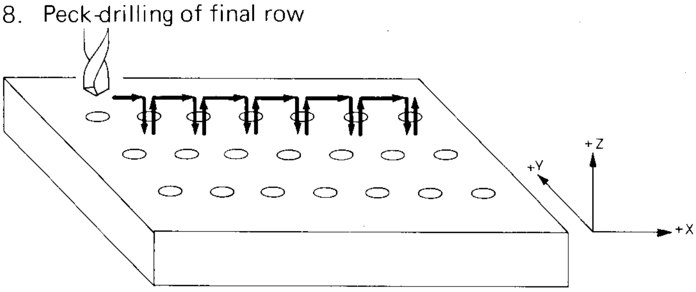

```
21 L       X + 10,000  I  Y + 15,000
                       R0  F9999  M
22 CYCL CALL
                                  M
```

7. 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
```

8. Peck-drilling of final row




```
24 LBL  CALL  1  REP
```

9. 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
```

I 6) Fixed program cycles (canned cycles)

I 6.1) Cycle definition CYCL DEF

For general purpose operation, TNC 145 C possesses "fixed" programmed cycles (canned cycles) for re-occuring machining operations.
Moreover, positioning of **3D-linear paths**, i.e. simultaneously in all three axes (without tool radius compensation) is also provided in form of a fixed programmed cycle. Additionally, a **datum shift** can be programmed and a **dwell time** can also be determined.

Range of cycles:

Cycle 0	=	Three-dimensional line
Cycle 1	=	Pecking
Cycle 2	=	Tapping
Cycle 3	=	Slot milling
Cycle 4	=	Pocket milling
Cycle 5	=	Circular pocket
Cycle 7	=	Datum shift
Cycle 8	=	Mirror image
Cycle 9	=	Dwell time

Please note:
The following cycles are executed at the point of definition:
0 = Three-dim. line 7 = Datum shift, 8 = Mirror image and 9 = Dwell time.

It is therefore unnecessary to retrieve the cycle via the CYCL CALL -key.

I 6.2) Selecting a certain cycle

("Paging" of cycle library)
When programming, the cycle definition-block is called-up by means of the CYCL DEF -key. Through repetitive pressing of the ↓ -key, the desired cycle is selected. By finally pressing the ENT -key, the cycle is transferred into the memory and defined as per the dialogue.

I 6.3) Explanation of Machining Cycles

I 6.3.1) Cycle "Three dim. line"

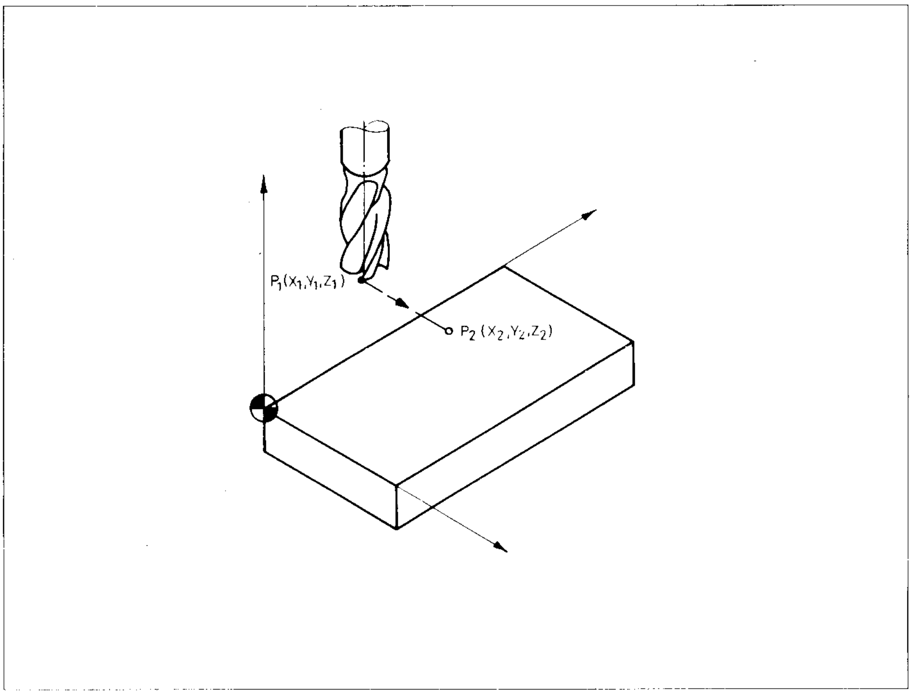
The 3D-Line cycle enables simultaneous traversing in all three axes. It is used for bringing tool and workpiece in a relative position to each other e.g. for start.

Please note:

.This cycle only requires tool length compensation. The tool radius is not considered for positioning.
.No "cycle call" is required.

Operating procedure:

The tool is located in the position P_1.
The co-ordinates X, Y and Z of the point P_2 are programmed in the "Three dim. line-cycle".
When machining, the tool then traverses in a straight path from the point P_1 to P_2.



Dialogue initiation: press CYCL DEF

Dialogue question	Response
CYCL DEF 0 THREE DIM.LINE	Transfer cycle by pressing ENT
FIRST COORDINATE ?	Enter nominal position coordinates (axes and position values) in absolute or incremental dimensions
SECOND COORDINATE ?	
THIRD COORDINATE ?	
FEED RATE ? F = . . .	Enter feed rate of linear path
AUXILIARY FUNCTION M ?	Enter auxiliary function

The "3D-Line-cycle allocates five program blocks. When "paging" the program, the following blocks are displayed:

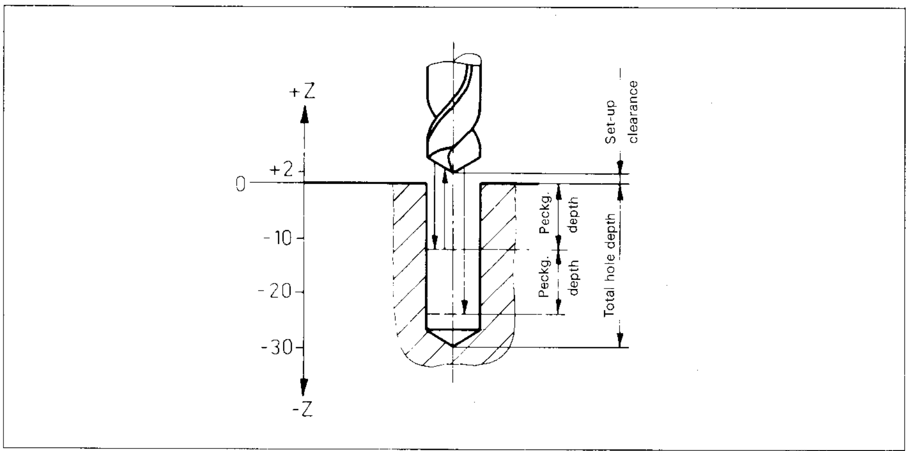
. . . CYCL DEF 0.0 THREE DIM.LINE	
. . . CYCL DEF 0.1 (I) X / Y / Z . . .	1st. Coordinate of nominal position
. . . CYCL DEF 0.2 (I) X / Y / Z . . .	2nd. Coordinate of nominal position
. . . CYCL DEF 0.3 (I) X / Y / Z . . .	3rd. Coordinate of nominal position
. . . CYCL DEF 0.4 F . . . M . . .	Feed Rate and auxiliary function

I 6.3.2) Cycle "Pecking"

Provision 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.
.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

Woodpecking 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.

Dialogue initiation: press CYCL DEF and ↓

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

* The set-up clearance, the total hole depth and the woodpecking depth must all have the same arithmetical sign and be entered in chain dimensions.

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 . . .	Set-up clearance
. . . CYCL DEF 1.2 DEPTH . . .	Total hole depth
. . . CYCL DEF 1.3 PECKG . . .	Woodpecking depth
. . . CYCL DEF 1.4 DWELL . . .	Dwell time
. . . CYCL DEF 1.5 F . . .	Feed rate

I 6.3.3) Cycle "Tapping"

Provisions for execution of cycle:

.For tapping, a chuck with length compensation facility is to be used.
.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.

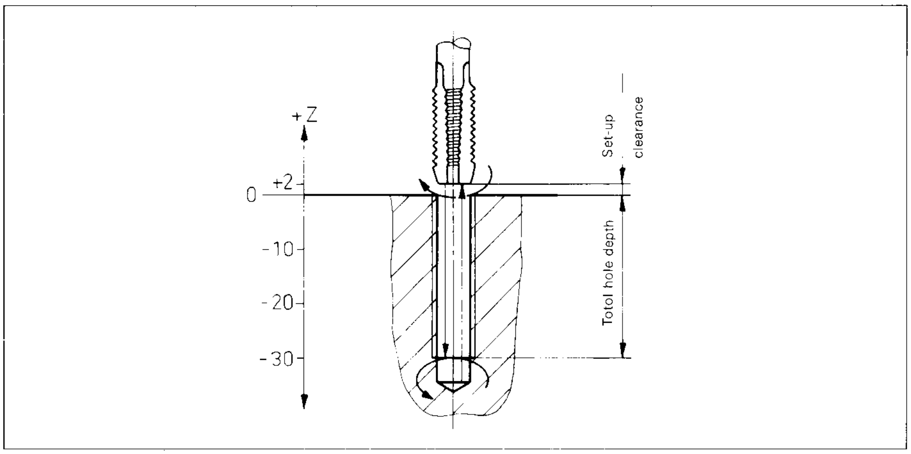
Calculation of feed rate for cycle definition "tapping":

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 of 1 sec. Now the programmed dwell time takes effect. Finally, the tapping tool is retracted to the position of the set-up clearance.

Caution: If the "Tapping cycle" is called, the programmed feed rate can only be altered within the range of 90 % and 110 % with the override potentiometer. This limited range on 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	Transfer cycle by pressing [ENT]
SET-UP CLEARANCE	Enter set-up clearance with sign *. This position must already have been approached in a previous block.
TOTAL HOLE DEPTH	Enter hole depth with sign *.
DWELL TIME IN SECS.	Enter amount of dwell time required between rotation change-over and retraction of tapping tool
FEED RATE ? F = . . .	Enter feed rate

* The set-up clearance and the hole depth must have the same arithmetical sign and be entered in chain dimensions.

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 . . .	Set-up clearance
. . . CYCL DEF 2.2 DEPTH . . .	Total hole depth
. . . CYCL DEF 2.3 DWELL . . .	Dwell time
. . . CYCL DEF 2.4 F . . .	Feed rate

I 6.3.4) Cycle "Slot milling"

Provisions for execution of cycle:

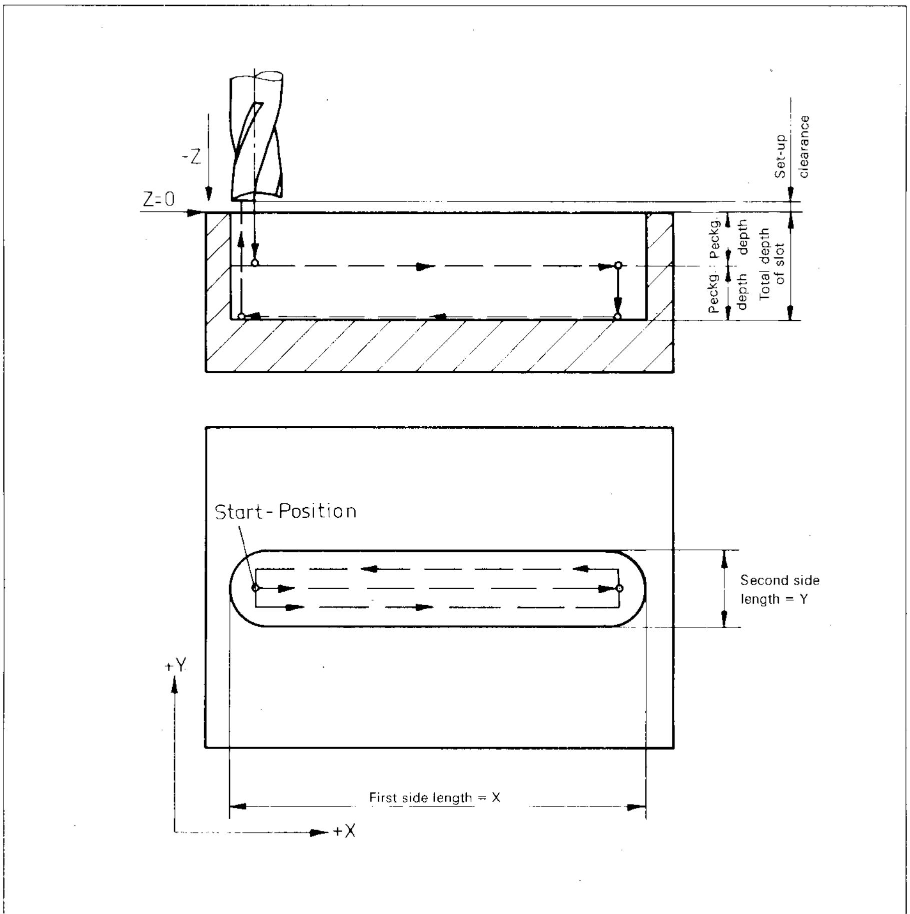
.The slot must be larger than the diameter of the milling cutter.

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

.The spindle rotating direction must have been determined with a previous block.

.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 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.

Please note:

The starting point of the slot must be approached **with tool radius compensation R+ or R-** in the longitudinal direction, i.e. with a **single axis positioning block** the contour offset RR or RL is not permitted for start position approach

*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	Transfer cycle by pressing ENT
SET-UP CLEARANCE ?	Enter set-up clearance with sign *. This position must already have been approached in a previous block.
MILLING DEPTH ?	Enter milling depth with sign *.
WOODPECKING DEPTH ?	Enter pecking depth with sign *.
FEED RATE FOR PECKING	Enter feed rate for pecking into workpiece
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 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 chain 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 . . .	Set-up clearance
. . . CYCL DEF 3.2 DEPTH . . .	Milling depth
. . . CYCL DEF 3.3 PECKING . . .	Woodpecking depth
F . . .	Feed rate for pecking
. . CYCL DEF 3.4 X / Y / Z . . .	Length of slot
. . . CYCL DEF 3.5 X / Y / Z . . .	Width of slot
. . . CYCL DEF 3.6 F . . .	Feed rate

Please note:
If machining programs which were compiled on the TNC 145 are executed with the TNC 145 C, the control will accept the cycle without the programmed feed rate for pecking. Pecking is carried out at half the programmed feed rate as with the previous TNC 145 model.

I 6.3.5) Cycle "Pocket milling" (Rough cut cycle)

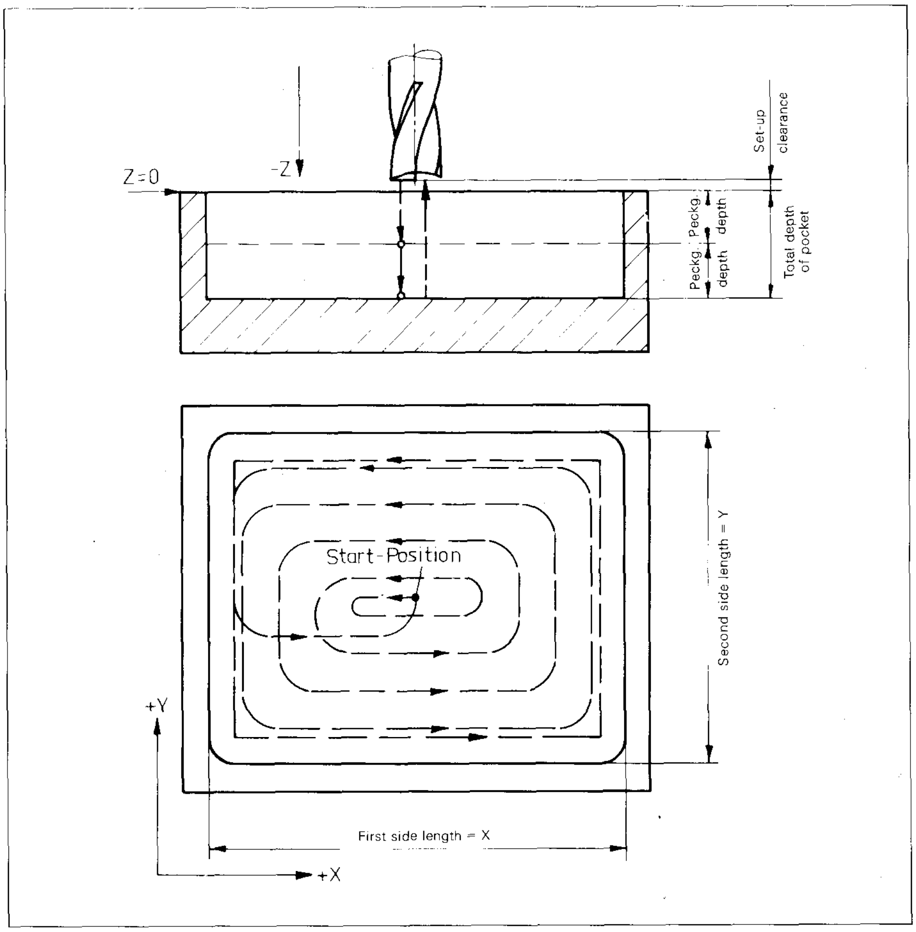
Provisions for execution of cycle:

.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 – this can be overcome by means of the pecking facility.

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

Please note:

"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 the machine parameter 54 and can lie between 0.1 and 1.414. The exact value can be obtained from your machine tool manufacturer.

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

Dialogue question	Response
CYCL DEF 4 POCKET MILLING	Transfer cycle by pressing ENT
SET-UP CLEARANCE	Enter set-up clearance with sign *. This position must already have been approached in a previous block.
MILLING DEPTH ?	Enter milling depth with sign *.
WOODPECKING DEPTH ?	Enter pecking depth with sign *.
FEED RATE FOR PECKING	Enter feed rate for pecking into workpiece
FIRST SIDE LENGTH ?	Enter first side length with positive sign.
SECOND SIDE LENGTH ?	Enter second side length with positive sign.
FEED RATE ? F = . . .	Enter feed rate for milling of slot.
ROTATION CLOCKWISE:	If clockwise rotation is required: press ENT (up-cut milling). If anti-clockwise rotation is required: press NO ENT (down-cut milling).

*The set-up clearance, milling depth and pecking depth must have the same arithmetical sign and be entered in chain 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 . . .	Set-up clearance
. . . CYCL DEF 4.2 DEPTH . . .	Milling depth
. . . CYCL DEF 4.3 PECKG . . .	Woodpecking depth
F . . .	Feed rate for pecking
. . . CYCL DEF 4.4 X / Y / Z . . .	First side length
. . . CYCL DEF 4.5 X / Y / Z . . .	Second side length
. . . CYCL DEF 4.6 . . . DR+ / DR-	Feed rate / Rotating direction

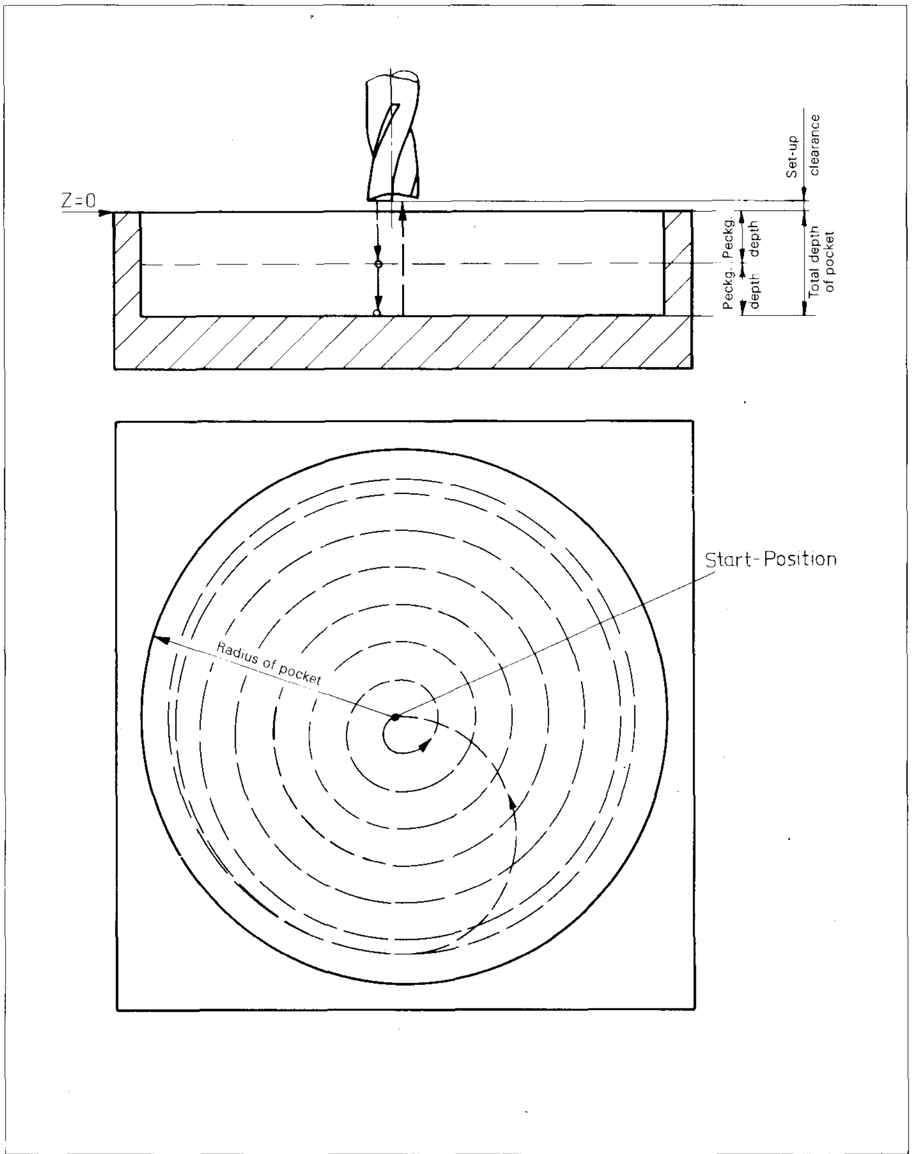
Please note:
If machining programs which were compiled on the TNC 145 are executed with the TNC 145 C, the control will accept the cycle without the programmed feed rate for pecking. Pecking is carried out at half the programmed feed rate as with the previous TNC 145 model.

I 6.3.6) Cycle "Circular pocket"

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 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 – this can be overcome by means of the pecking facility.

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

Please note:
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 with the machine parameter 54 and can lie between 0.1 and 1.414. The exact value can be obtained from your machine tool manufacturer.

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

Dialogue question	Response
CYCL DEF 5 CIRCULAR POCKET	Transfer cycle by pressing ENT
SET-UP CLEARANCE ?	Enter set-up clearance with sign *. This position must already have been approached in a previous block.
MILLING DEPTH ?	Enter milling depth with sign *.
WOODPECKING DEPTH ?	Enter pecking depth with sign *.
FEED RATE FOR PECKING	Enter feed rate for pecking into workpiece.
CIRCLE RADIUS ?	Enter radius of circular pocket.
FEED RATE ? F = . . .	Enter feed rate for milling of slot.
ROTATION CLOCKWISE: DR- ?	If clockwise rotation is required: press ENT (up-cut milling). If anti-clockwise rotation is required: press NO ENT (down-cut milling).

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

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 . . .	Set-up clearance
. . . CYCL DEF 5.2 DEPTH . . .	Milling depth
. . . CYCL DEF 5.3 PECKG . . .	Woodpecking depth
F . . .	Feed rate for pecking
. . . CYCL DEF 5.4 RADIUS . . .	Radius
. . . CYCL DEF 5.5 F . . . DR+ / DR-	Feed rate / Rotating direction

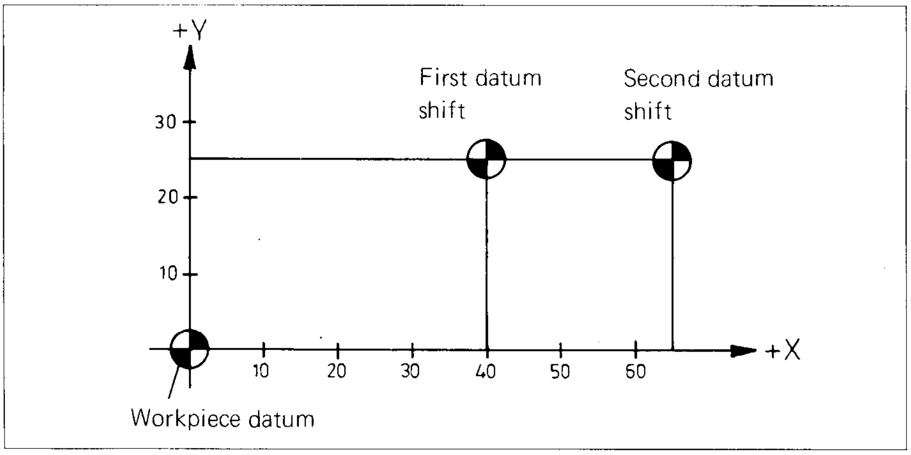
Please note:
If machining programs which were compiled on the TNC 145 are executed with the TNC 145 C, the control will accept the cycle without the programmed feed rate for pecking. Pecking is carried out at half the programmed feed rate as with the previous TNC 145 model.

I 6.3.7) Cycle "Datum shift"

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

Please note:
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
CYCLE DEF 7 DATUM SHIFT	Transfer cycle by pressing ENT
DATUM SHIFT ON X-AXIS ?	Enter datum shift in absolute or incremental dimensions.
DATUM SHIFT ON Y-AXIS ?	
DATUM SHIFT Z-AXIS ?	

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 (I) X . . . Datum shift X-Axis
. . . CYCL DEF 7.2 (I) Y . . . Datum shift Y-Axis
. . . CYCL DEF 7.3 (I) Z . . . Datum shift Z-Axis

I 6.3.8) 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.

Please note:

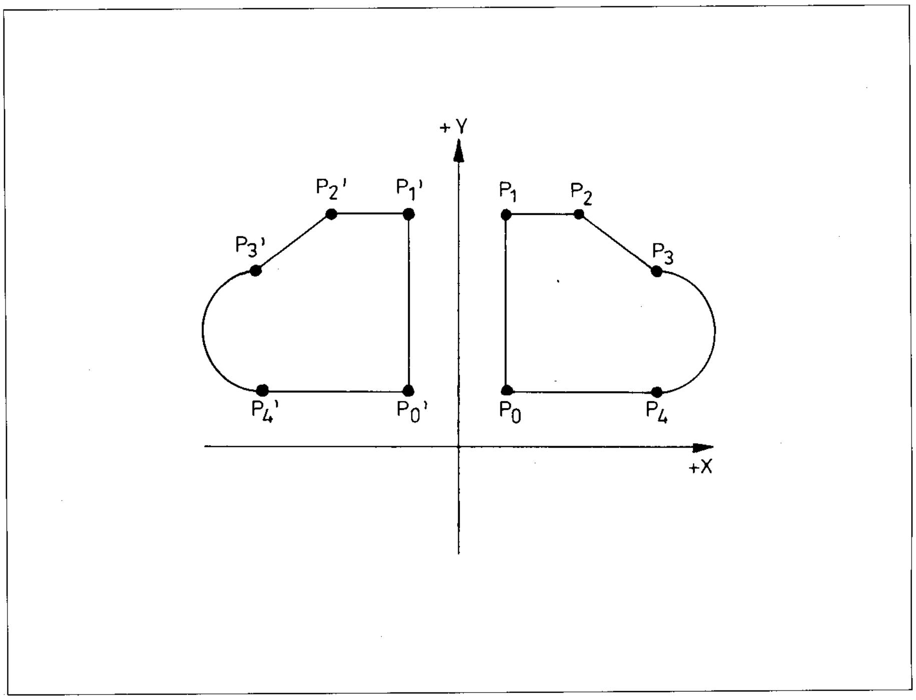
. The tool axis (working spindle axis) cannot be mirror imaged (error indication: MIRROR IMAGE ON TOOL AXIS)

. A cycle call is unnecessary.

. Before the cycle, tool compensation of the previous contour must be finalised.

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	Transfer cycle by pressing ENT
MIRROR IMAGE IN X-AXIS ?	Mirror image is required in the X-axis: press ENT . Mirror image is **not** required in the X-axis: press NO ENT . The next dialogue question is displayed.
MIRROR IMAGE IN Y-AXIS ?	Mirror image is required in the Y-axis: press ENT Mirror image is **not** required in the Y-axis: press NO ENT . The next dialgogue question is displayed.
MIRROR IMAGE IN Z-AXIS ?	Mirror image is required in the Z-axis: press ENT . Mirror image is **not** required in the Z-axis: press NO ENT .

Cancellation of mirror image

Mirror image is cancelled by programming the "mirror image" cycle and responding to all dialogue questions by pressing NO ENT .

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 / Z Axis for mirror image

I 6.3.9) 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).

Please note:
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	Transfer cycle by pressing 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 DWELL . . . Dwell time

I 6.4) 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: AUXILIARY FUNCTION M?

Enter auxiliary function.
The cycle call allocates one program block:

. . . CYCL CALL
M . . .

2. Programming of auxiliary function M 99 (see section I 3.1)

Please note:
A cycle call is not required for the fixed machining cycles:

0 = Three dim. line
7 = Datum shift
8 = Mirror image
9 = Dwell time

All other fixed machining cycles require a cycle call.

Please note:
Only the **last defined cycle** within the program sequence can be retrieved with the CYCL CALL -key. Cycles which require no cycle call are not taken into account.

I 7) Programmed STOP: key STOP

Dialogue initiation: press STOP

Dialogue question: AUXILIARY FUNCTION M?

If required, enter desired auxiliary function M.

A programmed stop, via the STOP -key, allocates one program block:

. . . STOP
M . . .

I 8) Program editing

I 8.1) Call-up of a program block

Select , or

↓

Key-In desired block No. and press GO TO

I 8.2) Program check blockwise

Select , or

↓

Enter block No. from which program inspection is to commence.

press GO TO .

↓

Inspect program either forwards or reverse

by pressing the "paging keys" ↓ or ↑ .

I 8.3) Deletion of blocks

press

↓

Go to block (or last block of program part)

which is to be deleted.

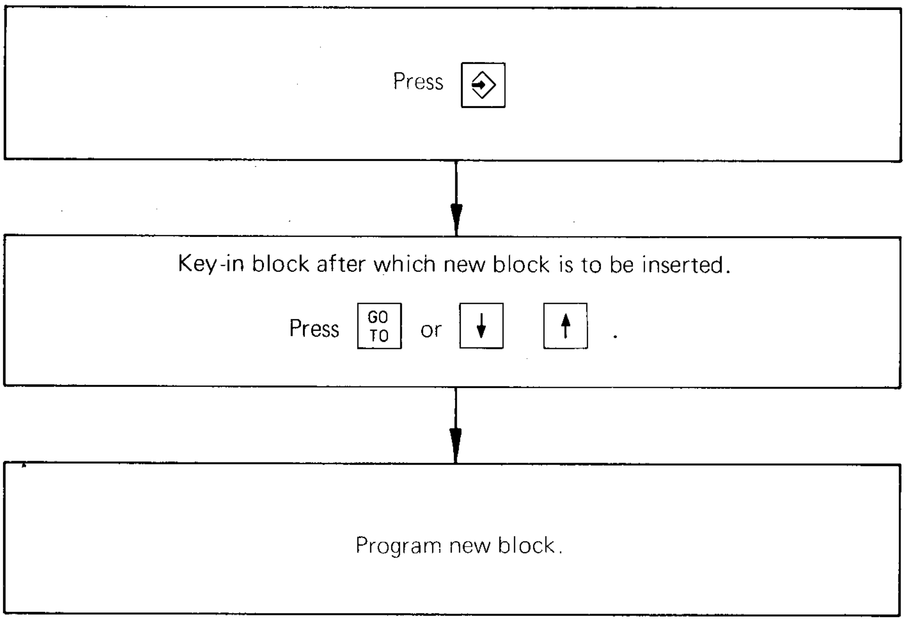
↓

Erase block(s) with DEL ☐ -key

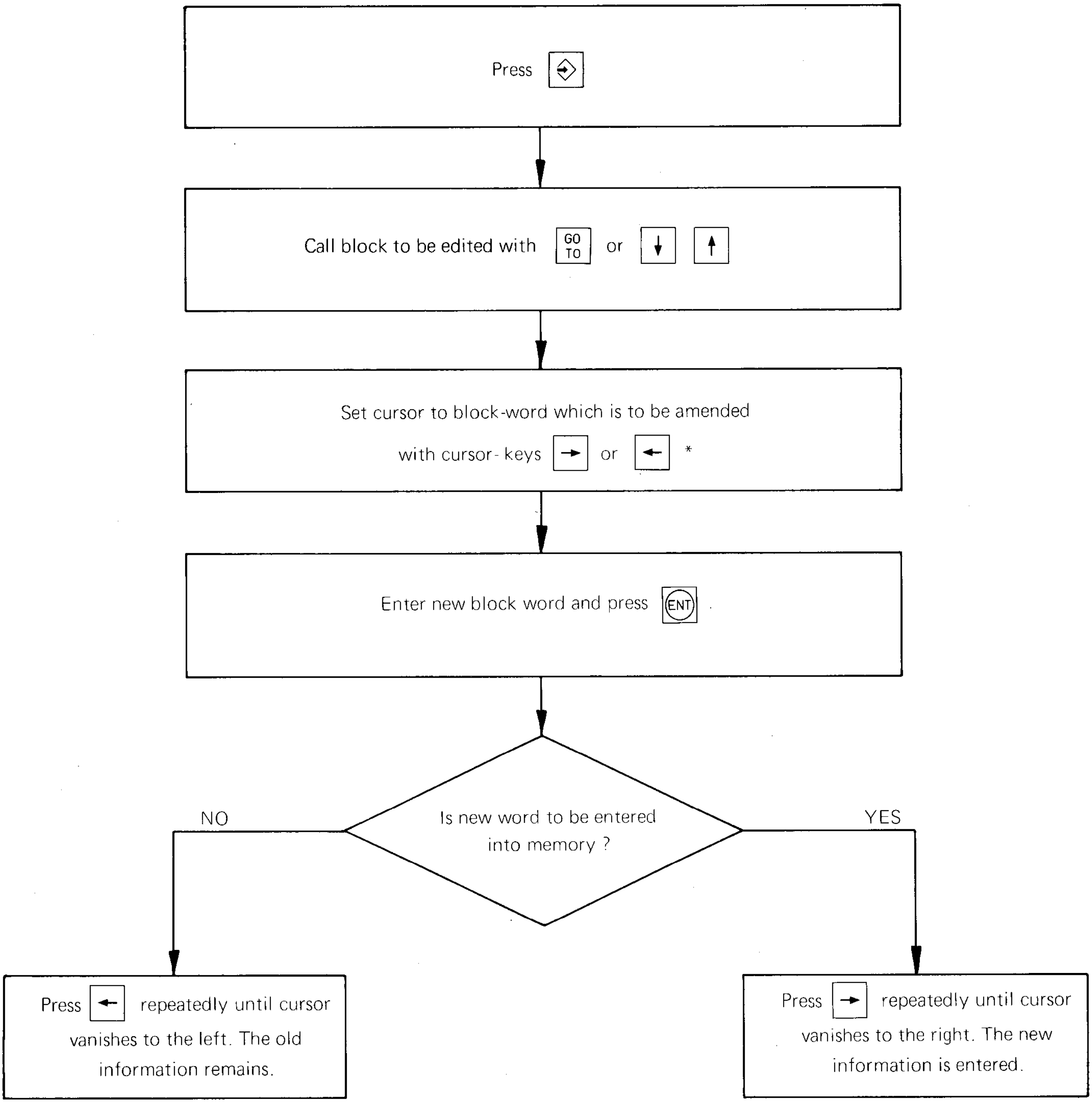
In order to delete blocks for tool and cycle definition, the DEL ☐ -key has to be pressed as many times, as individual blocks are required for the complete definition. Block numbers for successive block are automatically amended.

I 8.4) Insertion of blocks into existing program

With the TNC 145, 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".



I 8.5) Editing within a block



*** Please note:**

The setting of the cursor is initiated with the → -key.

Special case:

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.

I 8.6) Search routines for locating certain criterea

Press [⟐] .

↓

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

↓

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

I 8.7) Clearing of complete program memory

Press [⟐] .

↓

Press [CL PGM] .

↓

Dialogue question:
CLEAR PROGRAM ?
YES = ENT / NO = NO ENT

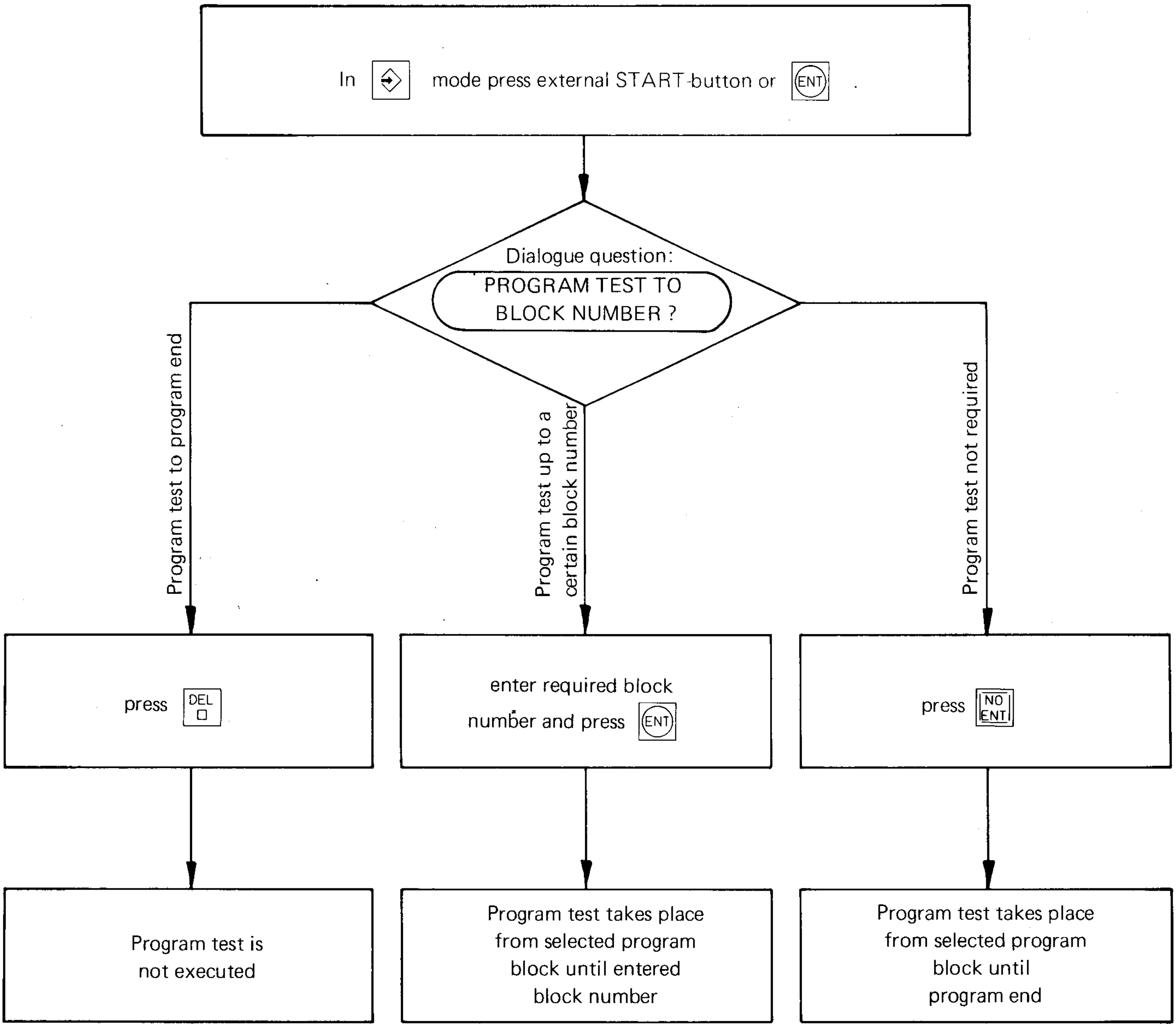
NO – Program is to be retained → Press [NO ENT] .

YES – Program is to be cleared → Press [ENT] .

*** Please note:**
The setting of the cursor is initiated with the [→] -key.

I 8.8) 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.

J) External data input and output EXT

J 1) Interface

The TNC 145 C 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 145 C. (Peripheral units with a 20mA-Interface may not be connected).

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

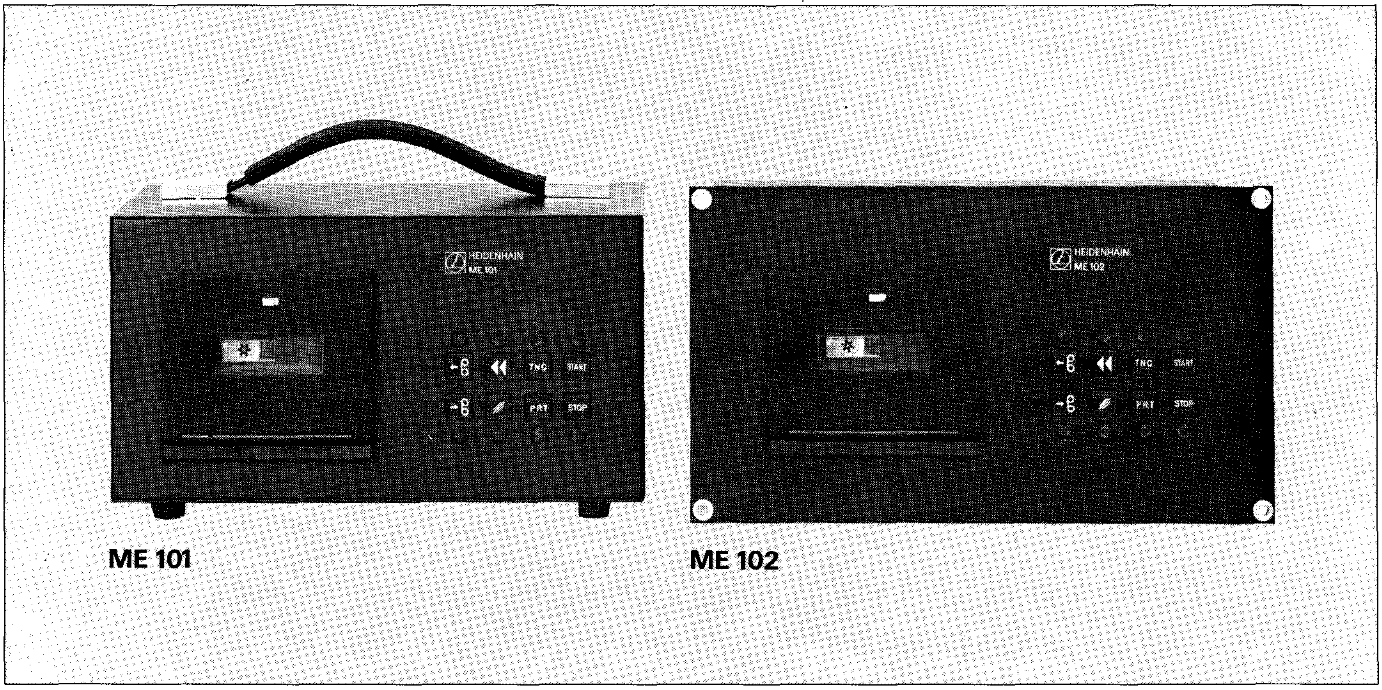
HEIDENHAIN supplies special magnetic tape cassette units for external program storage.

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 145 C, 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 can be obtained in the ME 101 and ME 102 operating manuals.

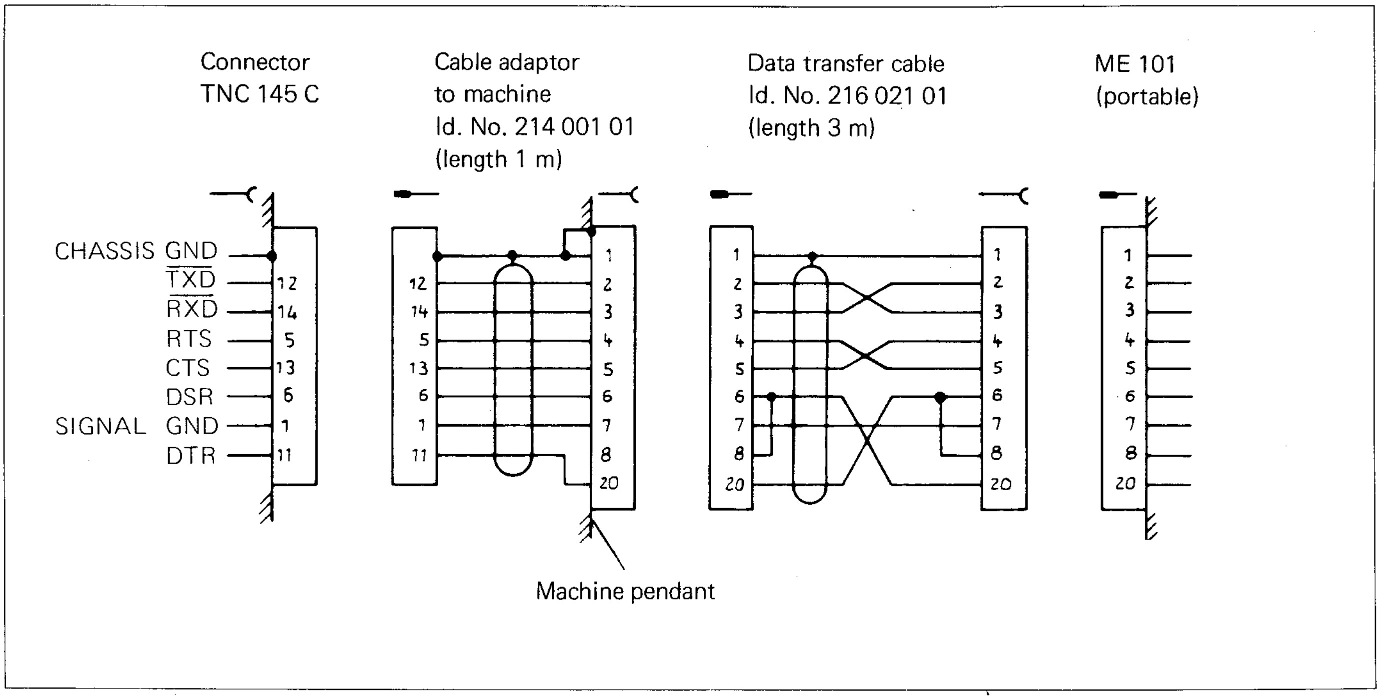


J 3) Connecting cables

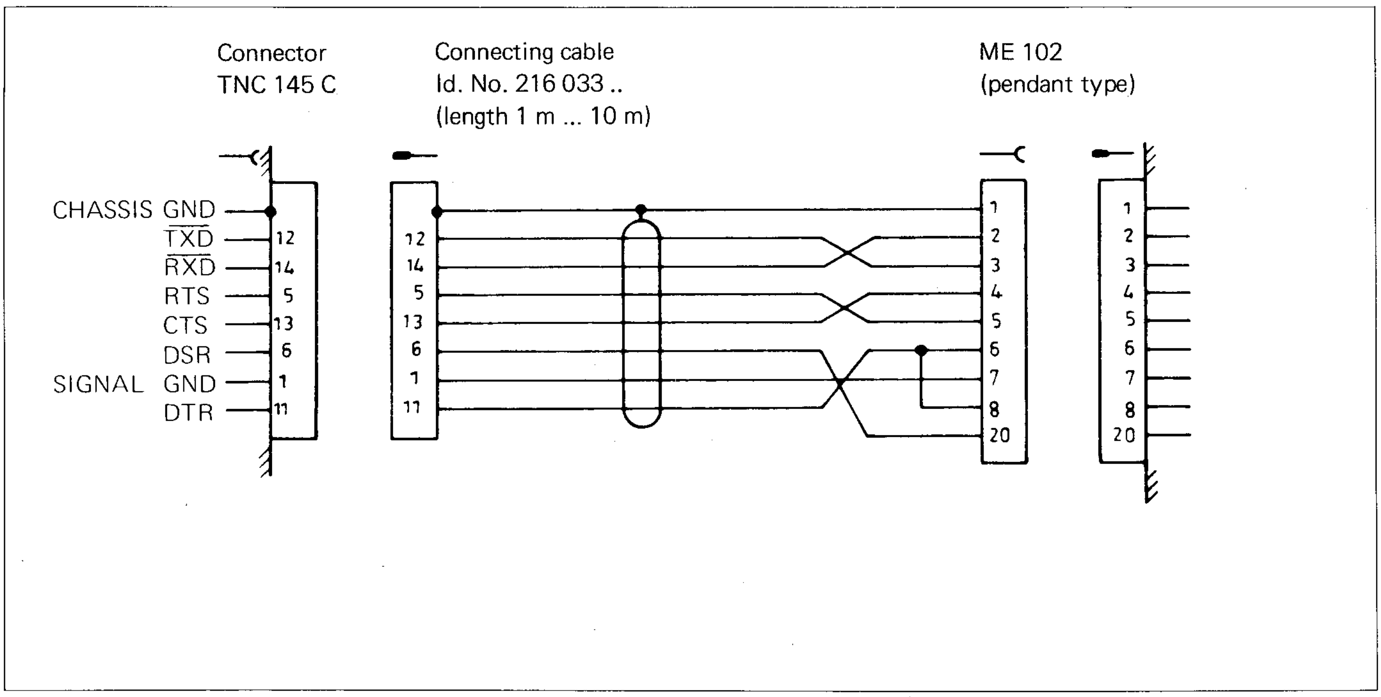
HEIDENHAIN supplies the following connecting cables:

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

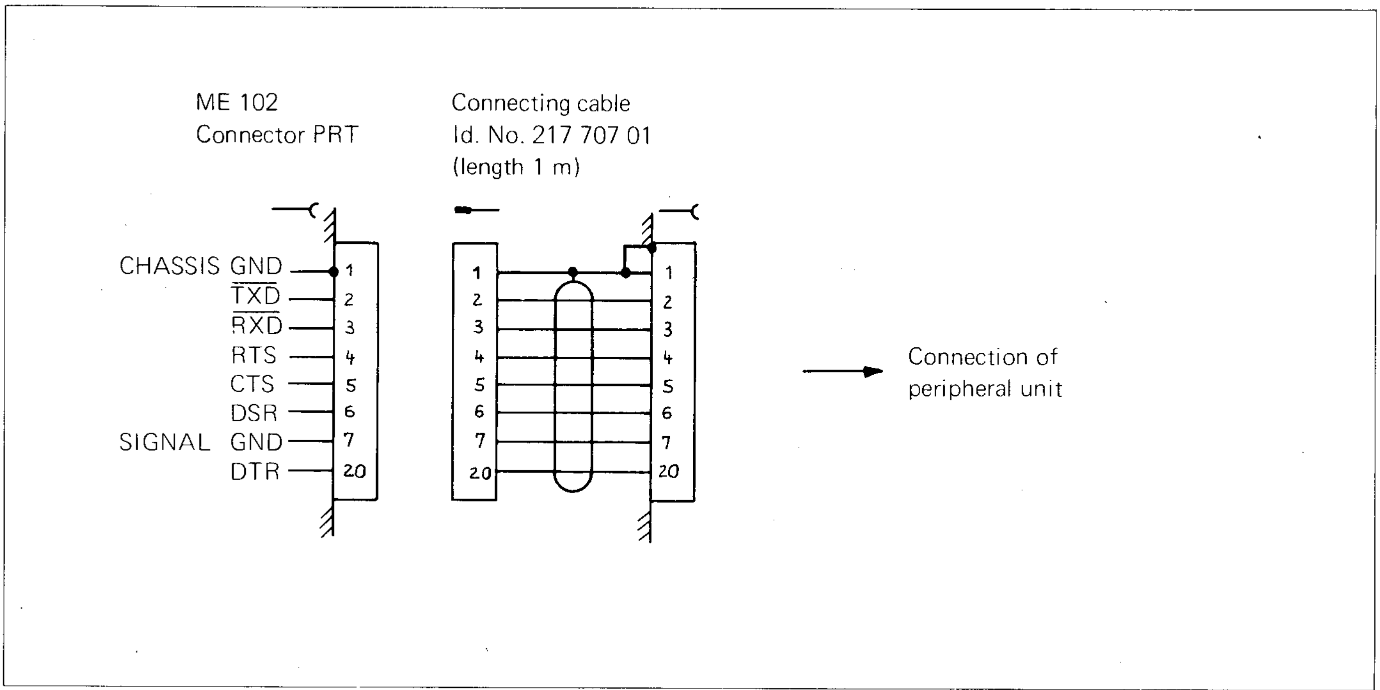
b) Data transfer cable for connection to ME 101.



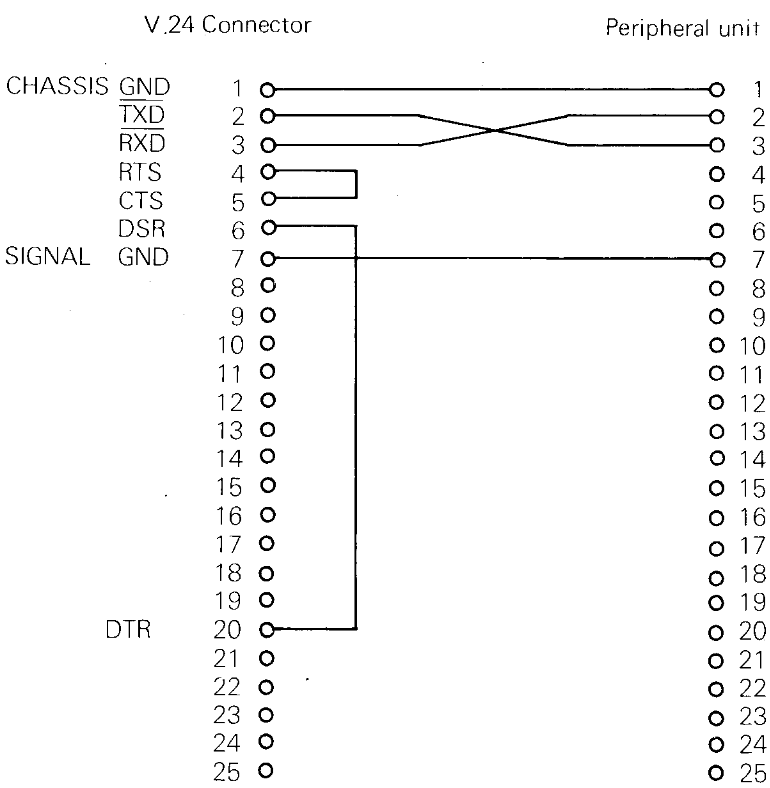
c) Connecting cable for direct connection of ME 102 (pendant type) to TNC 145 C.



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).



Designation of signals:

TXD	Transmit Data
RXD	Receive Data
RTS	Request To Send
CTS	Clear To Send
DSR	Data Set Ready
DTR	Data Terminal Ready

Please note:
The peripheral unit must be set to "Even parity"

J 4) Entry of Baud rate

The transfer rate for the V.24-interface of the TNC 145 C is automatically set to 2400 Baud (adapted to the HEIDENHAIN Magnetic Tape Cassette Units ME 101/ME 102).

If the TNC 145 C is to be connected to a peripheral unit with another Baud rate (without intermediate connection of the ME), the Baud rate is adapted as follows:

Firstly switch to [hand key] -mode!

Dialogue initiation: press [EXT]

Dialogue question: BAUD RATE = . . . ?

If required, key-in new Baud rate (110, 150, 300, 600, 1200 or 2400 Baud) and press [ENT] for entry into memory.

If, after pressing the [EXT] -key, the Baud rate is to be displayed (for inspection) only, the dialogue display can be cancelled by pressing the [EXT] or [DEL] -key.

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.

J 5) Operating procedure for data transfer

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

The TNC 145 C provides the following commands automatically (for print-out with line shift):

CR	Carriage Return
LF	Line Feed (shift)
SP	Space
ETX	End of Text

With program storage on a tape puncher, the punched tape contains these characters, with storage on the ME 101/102 they are on the magnetic tape.

Start of data output

Press [key] or [key]

↓

Select program beginning or block from which data-transfer should begin.

↓

Switch on peripheral unit

↓

Start data-output by pressing [EXT]

External input of a machining program

Before program entry, clear TNC memory to prevent portions of the "old program" being retained. With external input, existing program blocks are overwritten with new information.

Press [key]

↓

Clear TNC 145 C program memory (see section I 8.7)

↓

Start data-transfer by pressing [EXT]

↓

Switch on peripheral unit

Please note:

If a program which exceeds the magnetic tape capacity on one side 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 (only in connection with an ME which has program No. 212 902 07).

J 6) External programming at a terminal

Whilst developing the TNC 145 C, 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, so that idle time on the machine is saved.

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 and LF.

c) ETX (text end) is to be entered after the last program block.

d) Each block must contain the **full** information which would be asked for by the dialogue during manual key-in programming.

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

f) In order to recognize data-transfer errors, the TNC 145 C checks for "even parity". Therefore, the external programming unit must be set to "even parity".

Further information concerning the V.24 interface and external programming can be obtained from the following manuals:
"Information on V.24 Data Transfer Connection"
"Block format for the HEIDENHAIN TNC 145 control"
Please ask for these publications!

K) Positioning with manual data input (MDI) (single block controlled)

press "positioning with MDI"

↓

Initiate dialogue with X , Y , Z or TOOL CALL .

↓

Enter into memory ENT .
Dialogue display: "BLOCK COMPLETE"

↓

Press external START-button

↓

Machine workpiece

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

Please note:
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.**

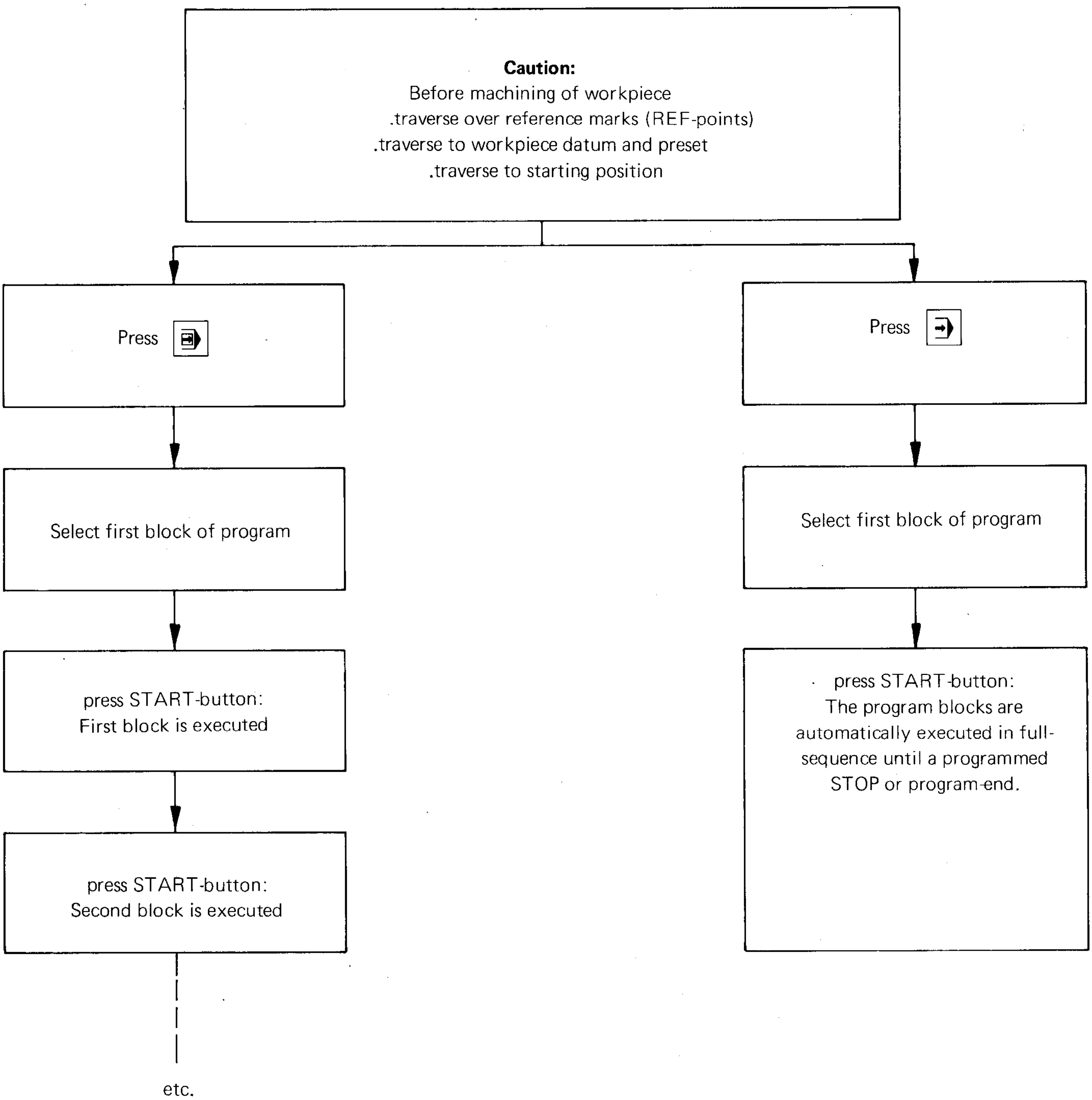
.in the -mode, the tool call has been activated with the external START -button.

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

L) Automatic program run

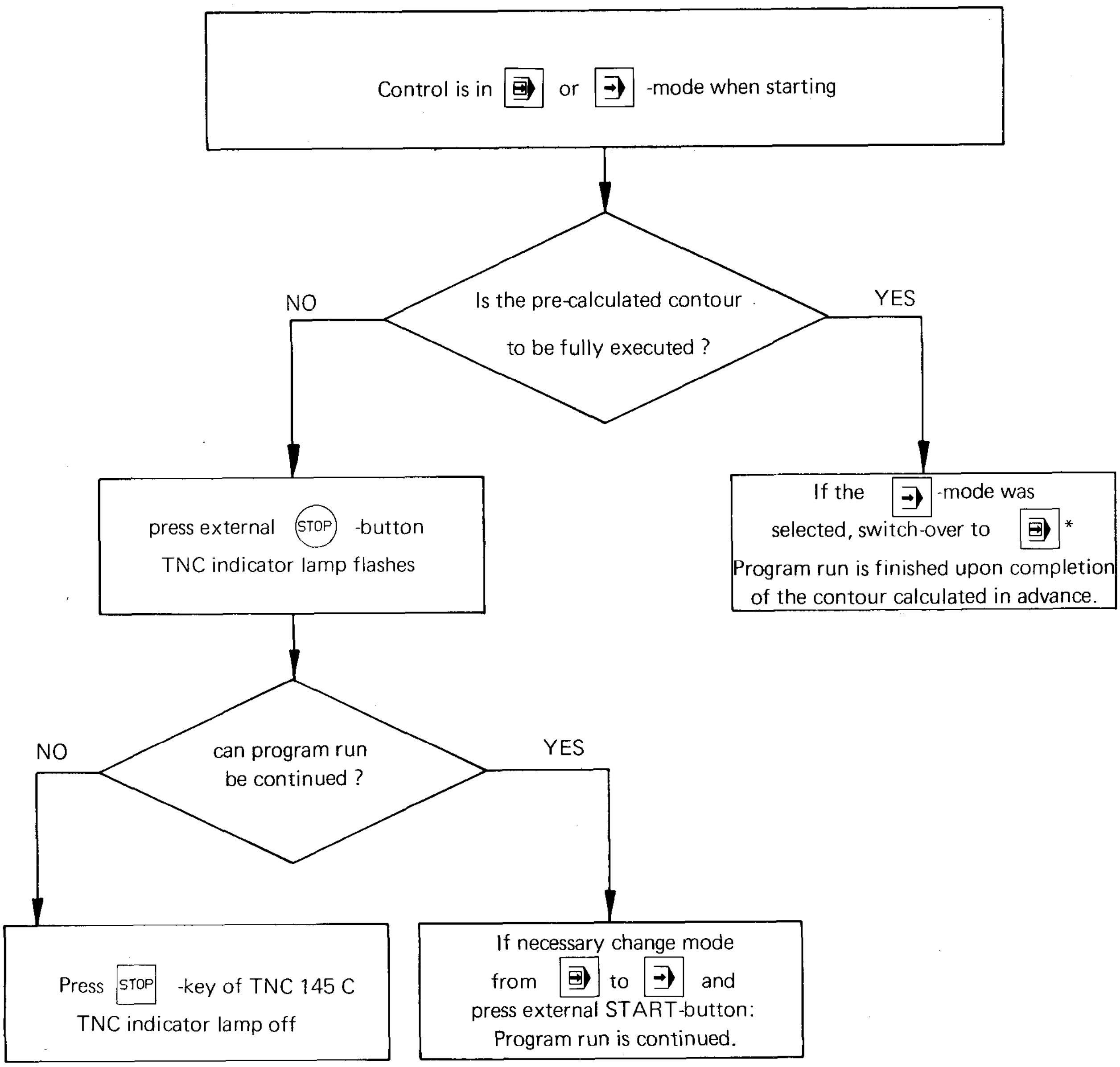
In the operating modes "single block program run" and "automatic program run" stored programs are executed.

L 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.

L 2) Interruption of program run



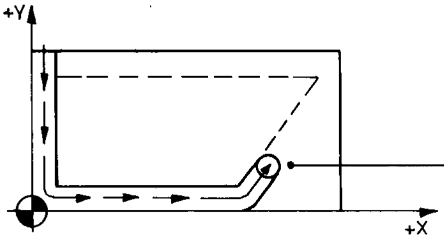
***Please note**
with a subprogram call-up and program part repeat program run is only terminated after complete execution of the call-up or series of repetitions.

L 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 L 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 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

↓

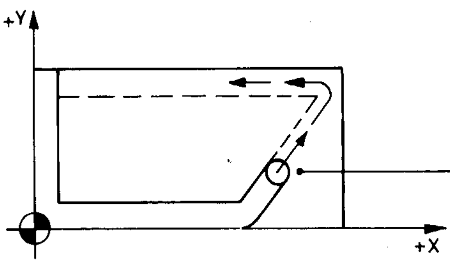
If the tool data is to be amended:
address appropriate tool definition with [GO TO] in -mode and enter new data.
Address interrupted block with [GO TO] .
If interruption takes place within a program part repeat or subprogram, read following section with heading "Please note"

↓

Switch on main spindle and coolant again.
.tool data unchanged: first traverse the X- and Y-axes and finally the Z-axis to the position of departure.
In operating mode , first traverse the X- and Y-axis and finally the Z-axis to the previous departure position.
.tool data amended: traverse to contour with new tool and touch workpiece.

↓

Start program run in or -mode



Program run is continued with complete tool compensation.

Please note:
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 repeat 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-start 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 I 4.6.1 (case 2).

L 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!

M) Technical specifications

M 1) Technical specifications, general

Control type	Shop floor programmable continuous path control for 3 axes (2 1/2 D)
Operator-prompting and displays	Visual display screen (9 inch) with max. 14 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; display of tool and fixed cycle last called, absolute values of datum shift, co-ordinates of circle centre CC in absolute dimensions and status display for datum shift and mirror image cycles. Additional displays for position values X, Y, Z and entry values.
Program memory	Buffered semiconductor store for 1000 program blocks
Operating modes	Manual operation: the control operates as a digital readout. Controlled operation with single block entry: each block is immediately positioned after key-in without entry into memory. Program run/single block: block-by-block positioning with individual press of START-button. Automatic program run: after press of START-button, complete run of program sequence until "programmed STOP" or program end. Programming: with **linear or circular interpolation** manually with stationary machine to program sheet or workpiece drawing or externally via the V.24-compatible data input/output (e.g via Magnetic Tape Cassette Unit ME 101/ME 102 from HEIDENHAIN or other commercially available peripheral unit). with **single axis positioning** additionally by entering actual position data from position display (playback) during conventional manual machining.

Programmable functions	Nominal position values - (absolute or incremental dimensions), entry in Cartesian co-ordinates or in polar co-ordinates; Straight path 2 1/2 D Circular arcs Rounding of corners derived from straights or arcs Tangential approach, on a circular path, to a defined point on a contour. 3 D-straight path, without tool radius compensation; Tool numbers, Tool length and radius; Offset direction for tool radius compensation; Rapid traverse/feed rate (in mm/min. or 0.1 inch/min.), Auxiliary functions M 00 . . . M 99, Spindle speeds (2 decades BCD), programmed stop, Subprograms (8 x nesting), Program part repeats Fixed Machining (canned) cycles: Pecking, Tapping, slot milling, Pocket milling, Circular pocket, Mirror image, Datum shift, Dwell time.
Program editing	through editing of block-word information, insertion of program blocks, deletion of program blocks; search routines for finding blocks with common criterea.
Monitoring system	the control monitors the functioning of important electronic subassemblies including positioning systems, transducers, spindle locking mechanism and EMERGENCY STOP circuit. 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.
Reference mark evaluation	After a power failure, automatic re-generation of datum value by traversing over reference marks of transducers.
Max. traversing speed	10 m/min.
Feed rate and rapid traverse override	Potentiometer on TNC-control panel
Transducers	HEIDENHAIN incremental linear transducers or angle encoders, grating pitch 0.02 mm , 0.01 mm or 0.1 mm (see section M 2).

Control inputs	Transducers X, Y, Z 1 electronic handwheel START, STOP, rapid traverse Verification: "Auxiliary function completed" Feed rate release Manual activation (opens positioning loop) Verification: EMERGENCY STOP-supervision Reference end position X, Y, Z Reference pulse suppressor X, Y, Z
Control outputs	1 analogue output each for X, Y, Z M-strobe signal S-strobe signal T-strobe signal 8 outputs for M-, S- and T-functions coded "Coolant OFF" "Coolant ON" "Spindle counterclockwise" "Spindle HALT" "Spindle clockwise" Spindle lock on Control in automatic operation mode EMERGENCY STOP
Mains power supply	Selectable 100/120/140/200/240 V +10% / -15%, 48 . . . 62 Hz
Power consumption	ca. 60 W (including VDU)
Ambient temperature	Operation 0 . . . 45^{o} (+32 . . . + 113^{o}F) Storage -30 . . . + 70^{o}C (-22 . . . + 158^{o}F)
Weight	TNC 145 C: 11.2 kg (VDU 6.8 kg)

M 2) Transducers

M 2.1) Transducers for TNC 145 CS

The TNC 145 CS-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 20 µm or 10 µm grating pitch (constant) are to be used such as:

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 (only in metric system) the incremental rotary encoders ROD 250 and ROD 700 with 18 000 or 36 000 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).

The cable length between the linear transducer and the TNC 145 C must not exceed **20 m.**

M 2.2) Transducers for TNC 145 CR

Since the cable length between the control TNC 145 CS and the transducer may not exceed 20 m, a special control, TNC 145 CR has been developed for greater distances.
This TNC-control has a transducer input for square-wave signals only and can therefore only be used in conjunction with an external pulse shaping electronics unit EXE. The EXE-signal is evaluated by a 4x or 2x multiplication in the TNC 145 CR-unit.

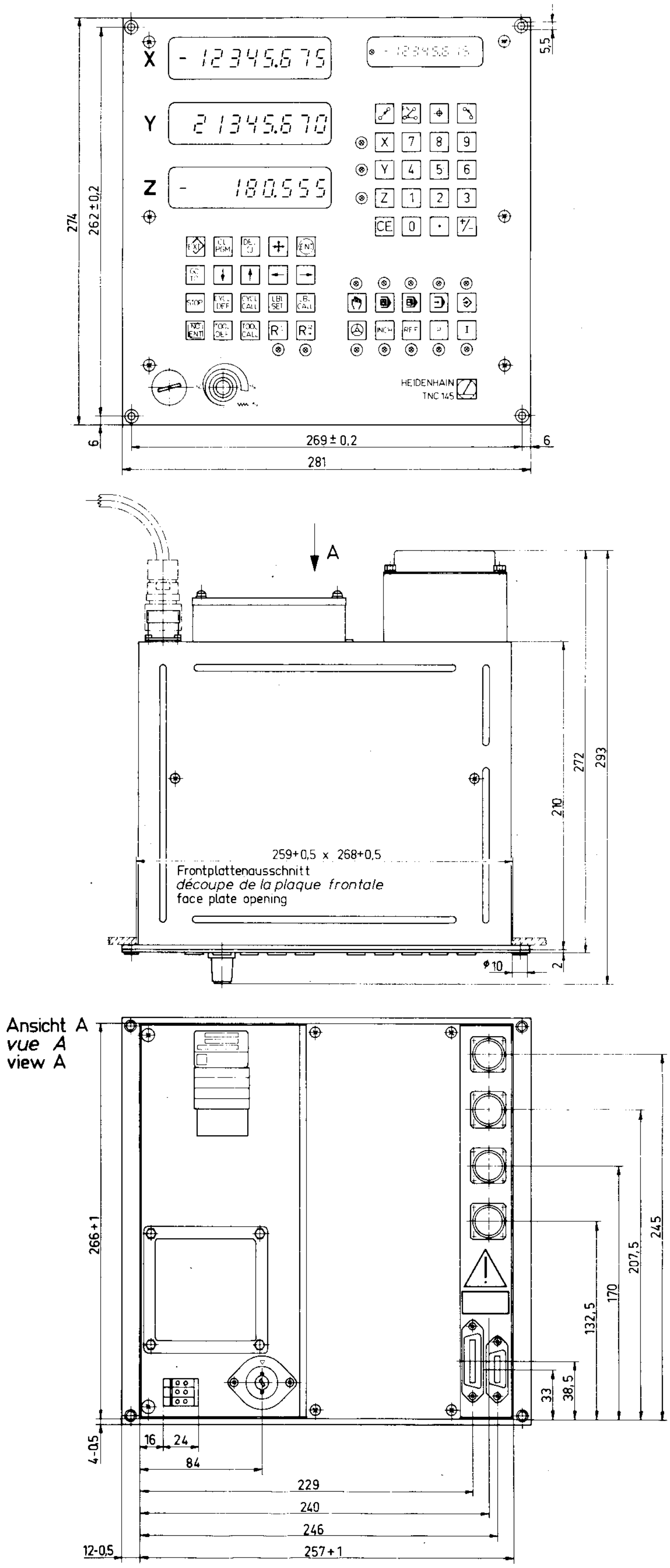
The max. permissible cable length between the transducer and the EXE-unit is 20 m. The max. permissible length between the EXE and the TNC 145 CR is 50 m. Therefore, the max. total cable length is 70 m.

For direct measurement the LIDA 325 (measuring length approx. 30 m, grating pitch 100 µm) in conjunction with an EXE 829 (3 axes and 25x subdivision) may be used.
The following EXE 829 versions are available:

Type	**Subdivision**
EXE 828	Axes 1 and 2 : 25 x Axes 3 : 5 x
EXE 827	Axis 1 : 25 x Axes 2 and 3 : 5 x
EXE 822	Axes 1, 2 and 3 : 5 x

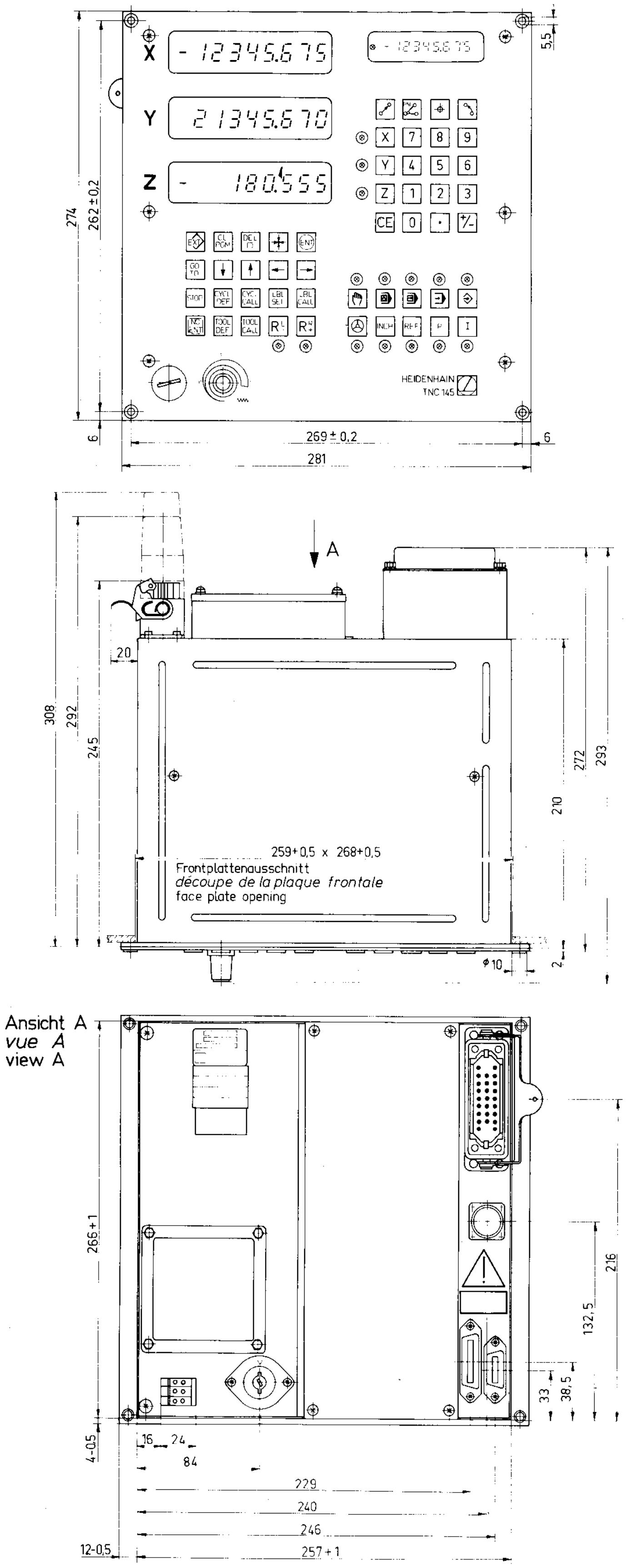
For angle positioning, a rotary encoder can be used as with the TNC 145 CS. The encoder signal is subdivided by e.g. 5 x in the EXE. Therefore, with 4x evaluation of the TNC rotary encoders with 18 000 lines per rev., e.g. ROD 250, are to be used.

TNC 145/CS



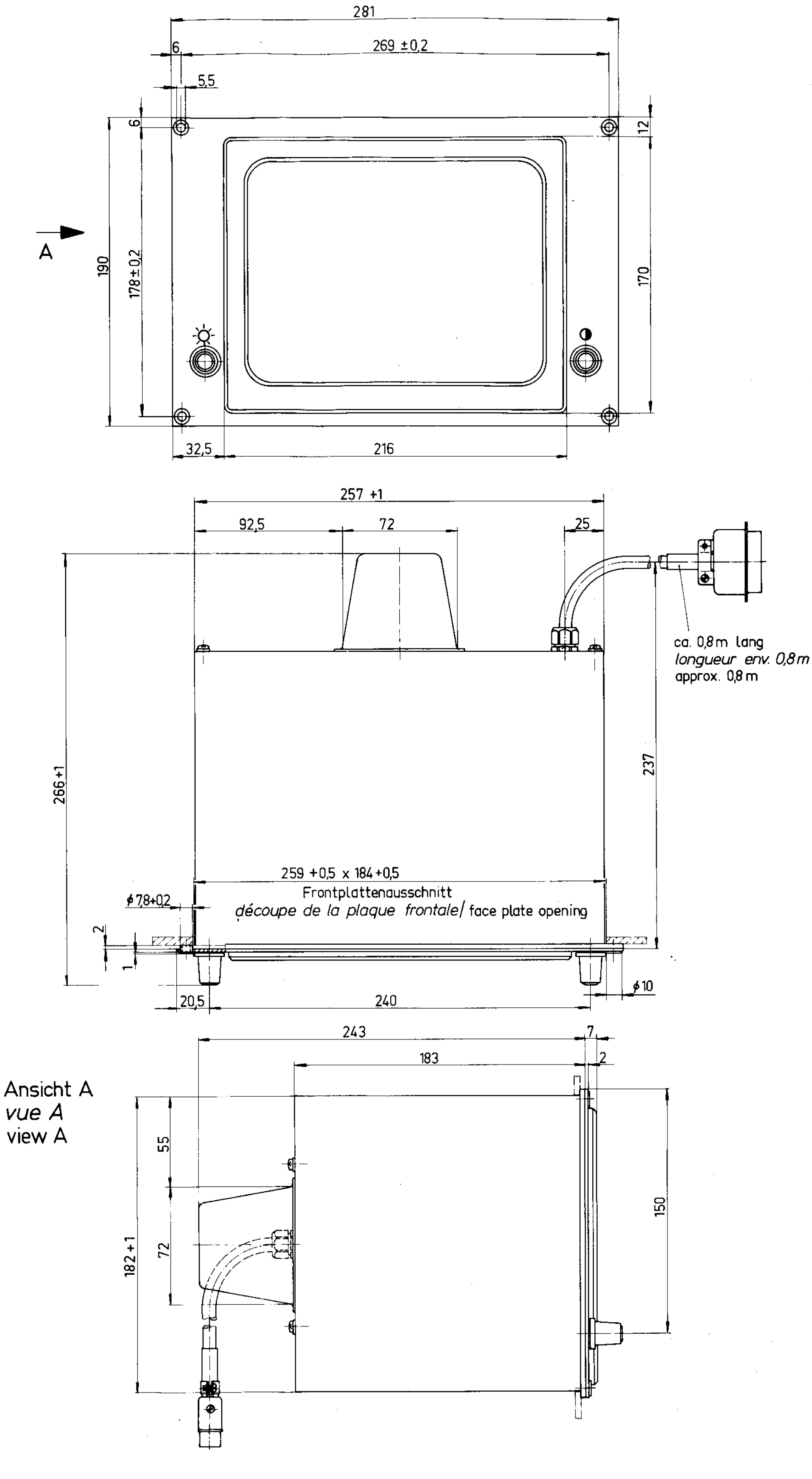
5,5
X - 12345.675
Y 21345.670
Z - 180.555
274
262 ± 0,2
HEIDENHAIN
TNC 145
6
269 ± 0,2
6
281
A
272
293
210
259+0,5 x 268+0,5
Frontplattenausschnitt
découpe de la plaque frontale
face plate opening
φ10
2
Ansicht A
vue A
view A
266+1
245
207,5
170
132,5
38,5
33
4-0,5
16
24
84
229
240
246
12-0,5
257+1

TNC 145/CR

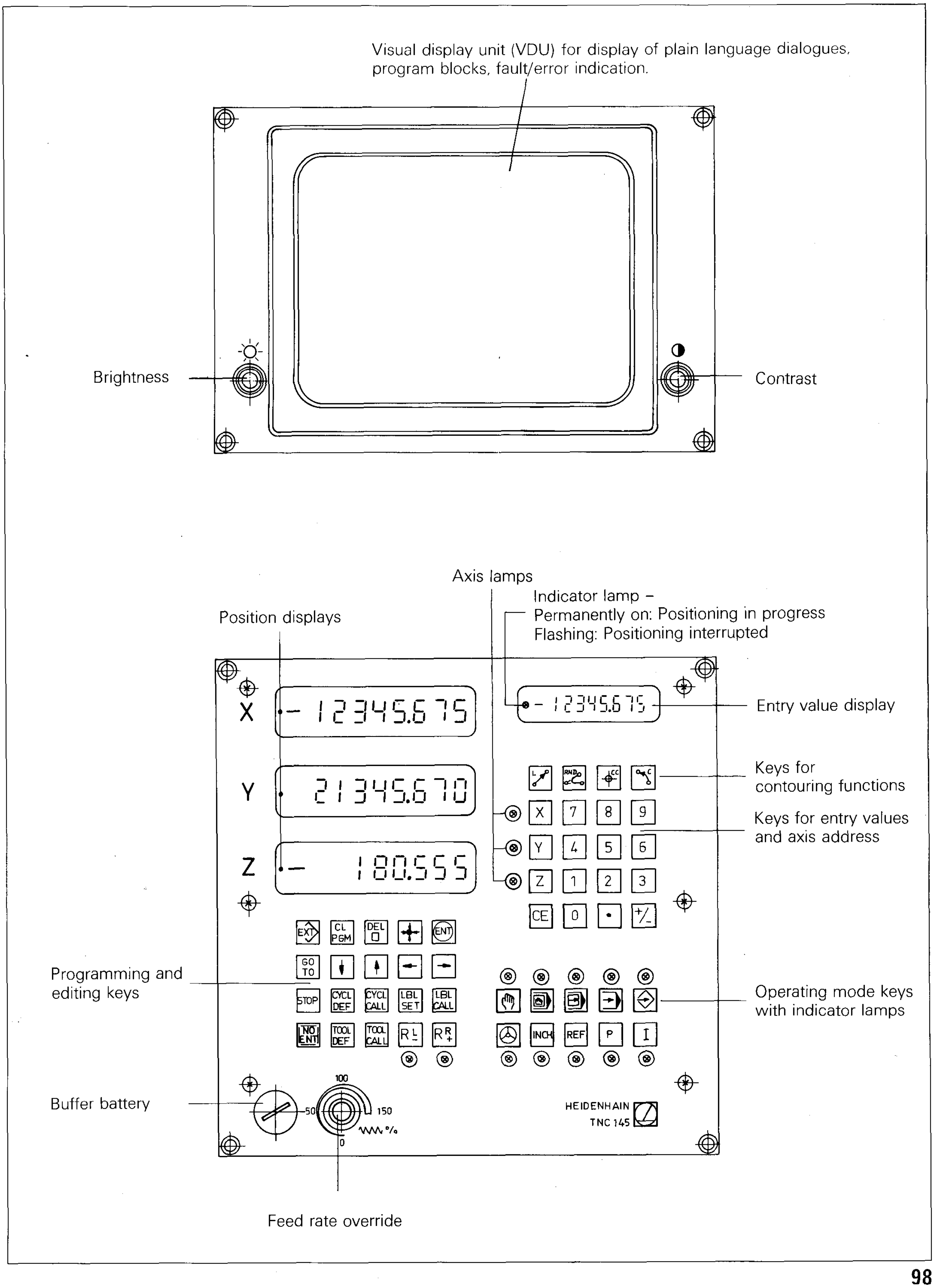


X
Y
Z
- 12345.675
21345.670
- 180.555
HEIDENHAIN
TNC 145
274
262 ± 0,2
5,5
6
269 ± 0,2
281
6
A
20
308
292
245
272
293
210
259+0,5 x 268+0,5
Frontplattenausschnitt
découpe de la plaque frontale
face plate opening
φ10
2
Ansicht A
vue A
view A
266+1
216
132,5
33
38,5
4-0,5
16
24
84
229
240
246
12-0,5
257+1

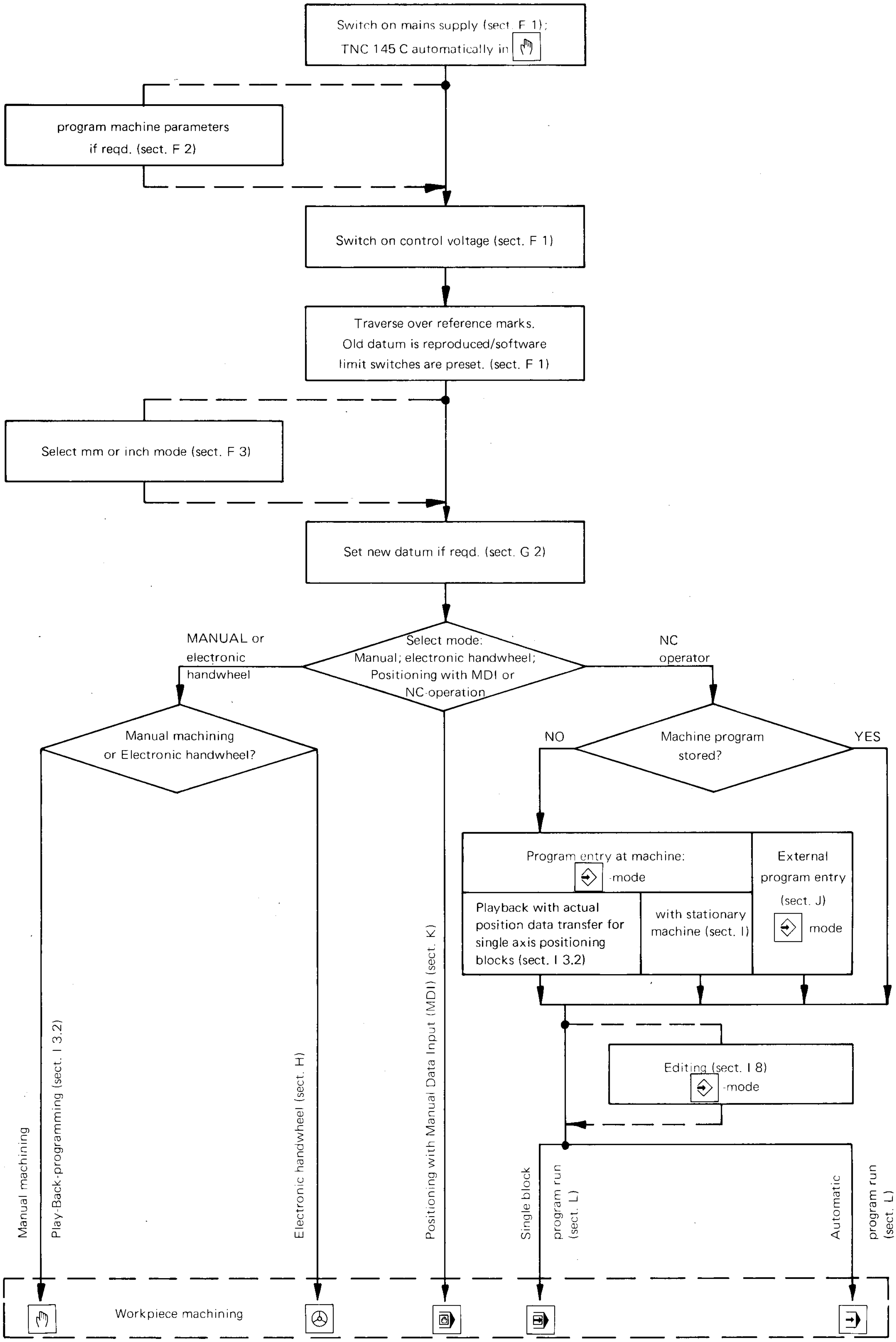
BE-Visual display unit (VDU)



Operating panel



O) Diagram for TNC 145 C-operation



HEIDENHAIN
DR. JOHANNES HEIDENHAIN GmbH
D-8225 Traunreut · Tel. (08669) 31-0

222 256 25 · 2 · 4/86 · H · Printed in West Germany · Subject to technical modifications