2. Setting Up/Manual Operation
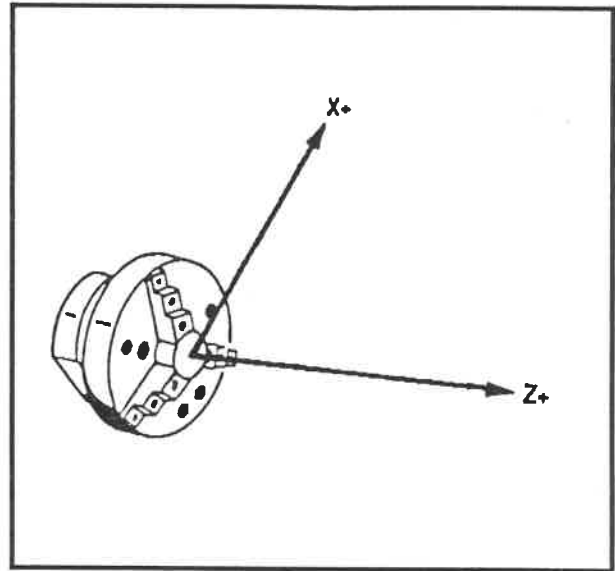## Geometrical Principles

System of coordinates
In order to control a lathe with a
CNC-control system, the possible
directions of movements are given
names.

Transversal: X – axis

Longitudinal: Z – axis

These axes form the system of
coordinates of the machine.

Coordinates
The point of intersection of the two
axes is called the zero point.
If number values are indicated on the
axes, then all target and actual positions
of the workpiece or the tool can be described
using the X and Z values.

When coordinates are used for dimensioning
the parts to be turned, the diameter is
indicated on the **X-axis**, and the **length of
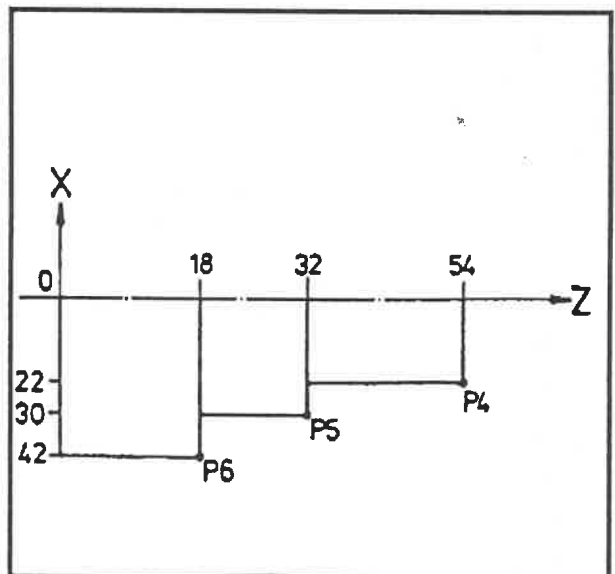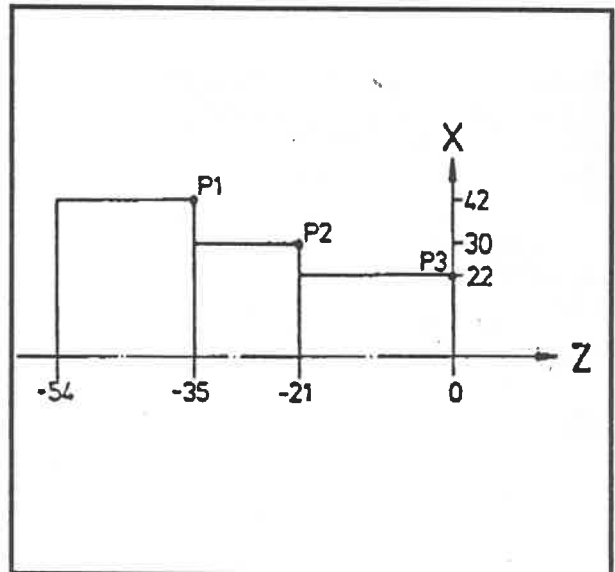the workpiece** on the **Z-axis**.
The sign is used to indicate on which side
from the point of intersection of the axes
the described point is located.

> \+ in direction of the axis
> – against the direction of the axis

Example
The coordinates of the described points
are as follows:

|		
P1:	X42	Z-35
P2:	X30	Z-21
P3:	X22	Z0
P4:	X-22	Z54
P5:	X-30	Z32
P6:	X-42	Z18

2. Setting Up/Manual Operation
Geometrical Principles

Reference points

NC-zero point, workpiece zero point (W, symbol ⊕)
The coordinates of the individual workpiece depend, given absolute dimensioning, upon the position of the zero point.
The zero point can be redetermined by the programmer for each program and at any position within the work range. It is then called the NC zero point or workpiece zero point (W). The NC zero point is usually set to the turning center at the beginning of the workpiece.
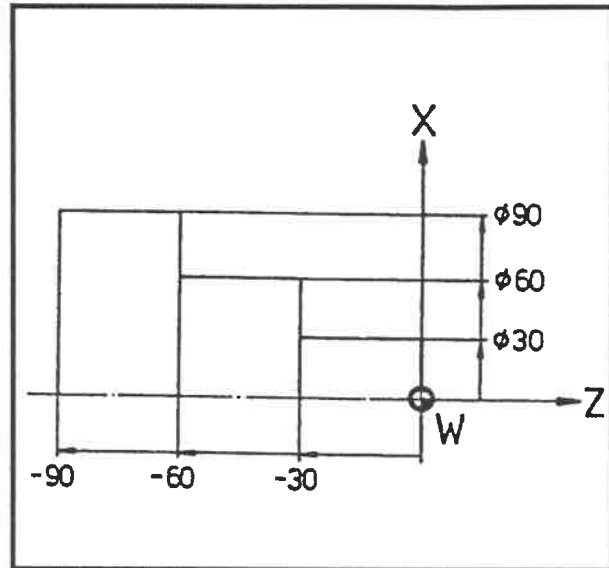
Machine zero point (M, symbol ⊕)
As well as the work-dependent coordinate system with workpiece zero point, there is also a fixed machine coordinate system with machine zero. It is located on the turning axis on the spindle nose.
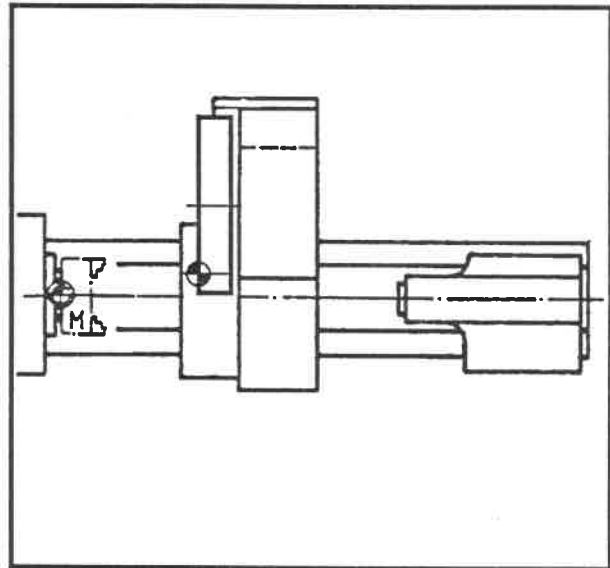
Reference point (R, symbol ⊕)
After switching on, the machine does not know where the machine slides are located. The measuring systems deliver signals only concerning the distance to be traversed (incremental position measuring system). This is why a reference point must be engaged (see REFERENCE operating mode).
Each slide axis has a reference mark. The control system can compute the position of the mark relative to machine zero using parameters (see sect. 6).
When the slide crosses the reference mark, the control system starts counting and the relation between display and slide position is established.

Absolute dimensioning, incremental dimensioning
Drawings can be dimensioned by two methods:

1. All dimensions relate to one point.
This dimensioning method is called **absolute dimensioning.**

The X values are diameters.
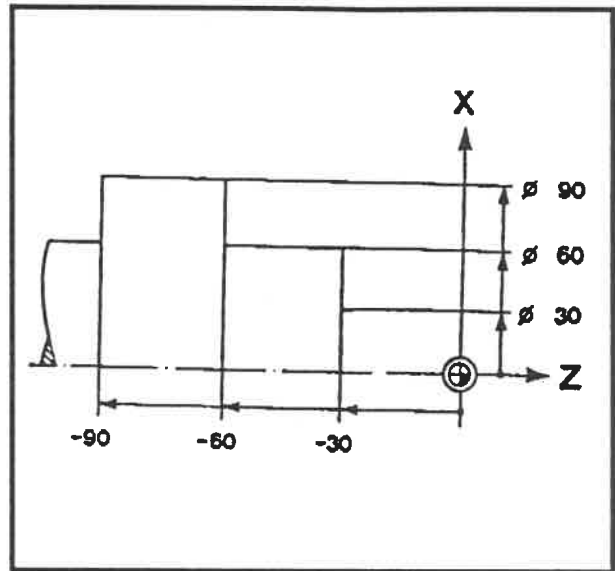
2. The dimensions indicate the distance from the previous point. This dimensioning method is called **incremental dimensioning.**

The X values are radii.

Depending on dimension data, target
positions can be described to the
control system by two methods:

1. To which point, relative to work-
piece zero point, must the traverse
be made?
This information applies to absolute
dimensioning. It is designated by
the command G90 in the program.
Display on screen G90.
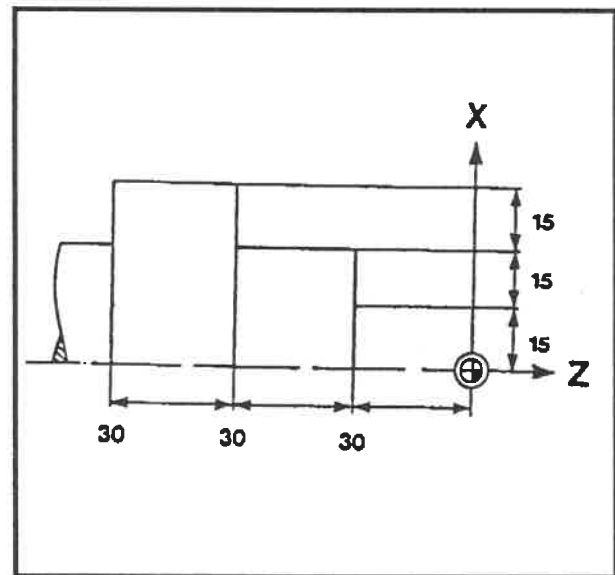G90 takes effect automatically after
switching on the control system
and need not be programmed.

Example

N... G90 G1 X100 Z-100

The tool runs to point X100
Z-100 relative to NC zero point,
independent of the actual current
position.
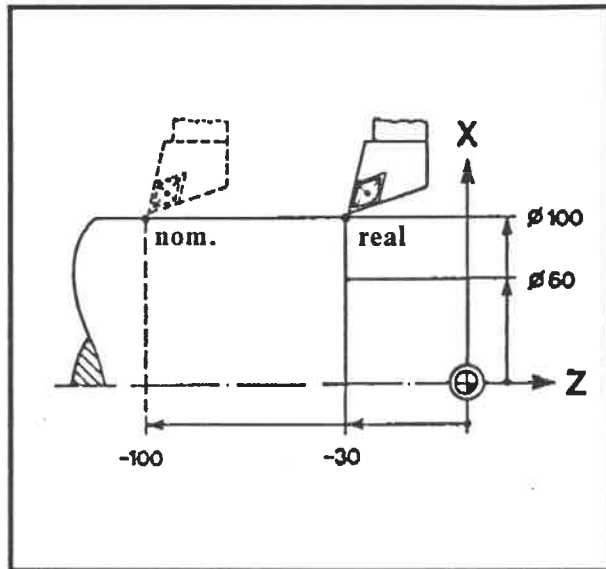
2. By which measure should the tool
traverse?
This information corresponds to
incremental dimensioning.
It is indicated in the program
by the function G91.
Display on screen G91.
The location of the workpiece
zero point is irrelevant.
Example

N... G91 G1 Z-70

The tool traverses in negative
Z direction by 70 mm opposed to the
direction of the Z axis.
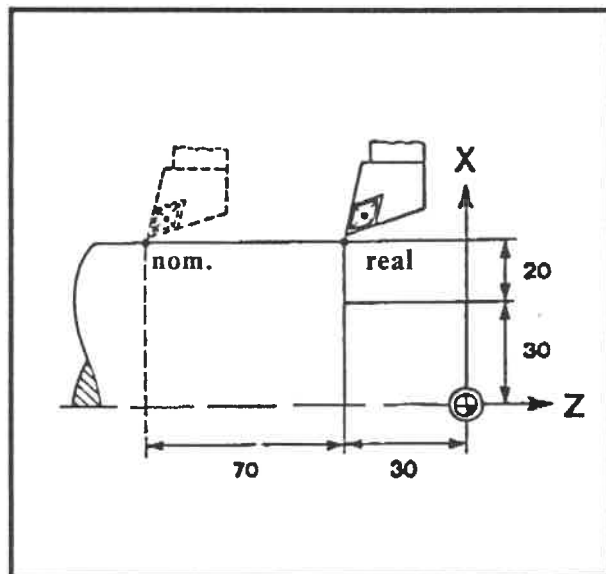
Detailed programming examples can be
found in section 3.2 and 10.

Dimensioning system
Coordinate dimensioning (DIN 406) is
used for the drawings in this hand-
book.

The zero point of the coordinate system
is set to coincide with the workpiece
zero point. The coordinates then
correspond to the absolute dimensioning
of the workpiece.
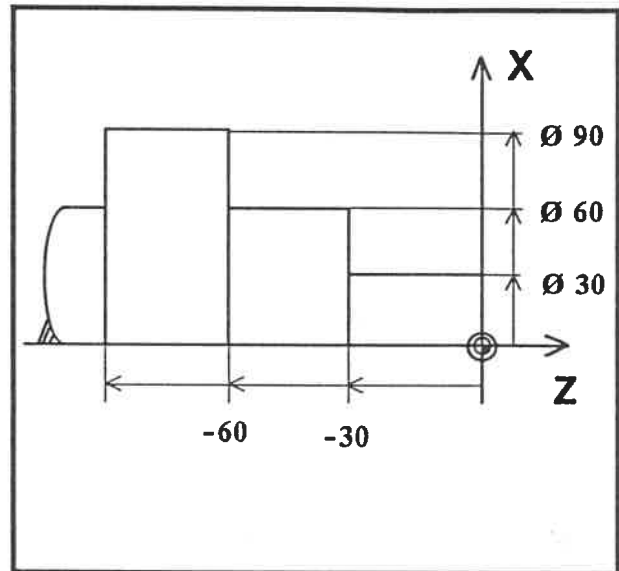
REFERENCE operating mode

Reaching reference point
In order for the slide to correlate with the measuring
system, the slides must be traversed to the reference
point. This is done in the REFERENCE operating mode.
First the transversal slide (X), then the longitudinal slide (Z)
reaches the reference point.

To reach the reference point, proceed
as follows:

Press the operating mode key.	The operating modes menu is displayed on the screen.
Press the REFERENCE softkey	The reference menu is displayed on the screen as an empty menu

Press the CYCLE START key.	The reference point is now reached automatically by the control system.
	Then the screen display shows:

When the reference point has been
reached, then the display is active
as determined by parameter N6.

END REFERENCE
SELECT OPERATING MODE

[⊡]   Press the operating mode key   The operating modes menu is displayed on
                                                 the screen.

Note
If reaching the reference point is impossible,
(e.g. due to a fixed roller steady, a clamped
workpiece or because the reference was cancelled
in parameter N3 by "1" instead of "0"), then the
slides can be synchronized by the control system when
"setting zero points" is selected in the MANUAL
CONTROL operating mode.

MANUAL CONTROL Operating Mode

In MANUAL CONTROL operating mode,
the slides can be traversed using the
manual direction keys or handwheel.
The machine is also set up in this
operating mode.

Press operating mode key.   The operating modes menu is displayed.

Select operating mode

Press the MANUAL CONTROL
softkey.

FEEDRATE MM/REV.	SPEED REV./MIN	V-CONST. M/MIN
TOOL SETTING MODE	T TOOL	SPINDLE SELECT. >
H-WHEEL X-AXIS 0.01	H-WHEEL Z-AXIS 0.01	MACHINE FUNC- TIONS

Press they "continue" key.   The following menu on the same level
is displayed.

H-WHEEL X-AXIS 0.001	H-WHEEL X-AXIS 0.1	CLAMPING DEVICE
H-WHEEL Z-AXIS 0.001	H-WHEEL Z-AXIS 0.1	SPINDLE SELECT. >
FEED % ON/OFF	¨SPEED REV. % ON/OFF	CONTIN. FEED

> Press the "continue" key

The second following menu on the same level is displayed:

		MANUAL PROGRAM SELECTION
FINISH TOOL INSPECTION	SPS G-FUNCTION	SPINDLE SELECT. >
DISPLAY SELECT.	DIAGNOS-TIC MODE	

Slide movements using the manual direction keys

Enter feed (see second next page).

Slide can be traversed using the manual direction keys.

If the reference point has been reached beforehand, the display of the actual value shows the position of the tool nose according to the value programmed under parameter N6.

Slide movement by handwheel
Handwheel divisions are always metric.
Actuating a manual direction key or changing the
operating mode cancels the preset handwheel setting.

The functions
HANDWHEEL X-AXIS 0.1
HANDWHEEL X-AXIS 0.01
HANDWHEEL X-AXIS 0.001
have the same practical significance.

	Press HANDWHEEL X-AXIS 0.1 softkey	HANDWHEEL X 0.1
	Slide can be run in transversal direction, using the handwheel.	The cross slide moves 0.05 mm radius per handwheel division. The diameter display changes by 0.1 mm.
	Press HANDWHEEL Z-AXIS 0.1 softkey.	HANDWHEEL Z 0.1
	The slide can traverse in longitudinal direction using the handwheel.	The longitudinal slide moves 0.1 mm per handwheel division.

Selection of a spindle
The ELTROPILOT L2 control system is designed for driving
3 spindles (auxiliary spindle for driven tool included).
The number of spindles which are present depends, however,
on the type of the machine which is realized.
When the control system is switched on, the inputs and values
displayed for feedrate, speed of rotation and cutting speed refer
to the main spindle (M).

If another spindle is to be selected, then proceed as follows:		Status display SPINDLE M
	Press the SELECT SPINDLE softkey.	SPINDLE 1 is displayed in the status line.

The renewed pressing of this softkey
effects the switchover to spindle 2
and subsequently back to the main spindle.

spindle M = main spindle,
spindle 1 = driven tool,
spindle 2 = mechanically movable auxiliary spindle

Feed input

Press FEED MM/REV.   G95 FEED MM (INCH)/REV.   F:

Enter digits   F ... 100%
                                            F ... NOM

Confirm   Control system uses set
                                            feed value.

If no entry is made at this point,
the feed value entered in the
parameters is active.

Feed override

Press the FEED %   HANDWHEEL
ON/OFF softkey   F % % % %

Move handwheel.   HANDWHEEL
                                            F % – 150 %
                                            F ... ... %
                                            F ... NOM

                                            The feed override percentage set
                                            with the handwheel is active.

Press the FEED %   Handwheel feed override is switched
ON/OFF softkey.   off, the value last selected is
                                            retained. Changing the operating
                                            mode also switches off the existing
                                            handwheel allocation.

The current override can be
altered again by renewed
pressing of this softkey.

Constant cutting speed
[m/min or feet/min]

Press V.CONST M/MIN   G96 V-CONSTANT S:

Enter digits   1S ... 100%
                                             V ... NOM

Confirm   The main spindle uses the set
                                             cutting speed.

Note:
If spindle 2 is selected, the input
refers to the auxiliary spindle and is
executed with G296.

Speed input [Rev./min]

Press the   G97 SPEED REV/MIN S:
SPEED REV/MIN
softkey.

Enter digits   1 S ... 100%
                                             S ... NOM

Confirm   The control system works
                                             with the set speed.

Note:
If spindle 1 is selected, the input
refers to the driven tool and is
executed with function G98.
The spindle speed of the auxiliary
spindle 2 is set with function G297.

Speed override

⊞	Press the SPEED % ON/OFF softkey	HANDWHEEL S % % % %
○	Move handwheel	HANDWHEEL S 50 % – 150 % S ... ... % S ... NOM The speed override percentage set with the handwheel is active.
⊞	Press the SPEED % ON/OFF softkey.	Handwheel override is switched off, the value last selected is retained. Changing the operating mode also switches off the existing handwheel allocation.

Note
If SPINDLE 1 or 2 is selected, the
input refers to the corresponding
auxiliary spindle.

Machine functions

⊞	Press the MACHINE FUNCTIONS softkey	The following sub-menu is displayed on the screen:

COOL. 1 ON	COOL. 2 ON	COOLANT 1+2+3+4 OFF
COOL. 3 ON	COOL. 4 ON	M-FUNCTION MENU
SET UP MACHINE DIMENS.	APPROACH MACHINE DIMENS.	M-FUNCTION DIRECT

⊞	Press the COOL. 1 (2, 3, 4) on or COOLANT 1+2+3+4 OFF softkey.	By means of these softkeys, the coolant circuits can be activated or deactivated (if existing).

2. Setting Up / Manual Operation

Selecting machine functions

Press the softkey
M-FUNCTION MENU

All existing M-functions can be activated
using the following menues and the
corresponding softkeys.

If the corresponding command of the
M-function is known, then it may also
be entered directly.

Press the softkey M-FUNCTION
DIRECT

INPUT M-FUNCTION:

Enter the desired M-function in
digits (for the meaning of the
M-functions, see section 3.6)

Confirm

Corresponding machine function
activated.

Indexing a spindle dividing device
If the spindle has a spindle dividing device (position-
controlled M19), the selected spindle can be indexed with a
rack which engages in an indexing gear. The control (SPS)
retracts the spindle release in this process to avoid reversed
rotation forces being exerted on the spindle dividing device
due to the integral portion of the spindle control

Press the SPINDLES
softkey.

The following submenu will appear.

MAIN SPINDLE ON CW	MAIN SPINDLE ON CCW	MAIN SPINDLE OFF
POINT STOP	INDEX ON	INDEX OFF

Press the INDEX ON
softkey.

Main spindle is indexed.

Press the INDEX OFF
softkey.

Rack indexing is disengaged.

Setting and approaching machine dimensions
Certain dimensions within the workspace can be
stored as machine dimension 1 to 9 in the parameters
N751 to N759 by entering the corresponding position
or by approaching these positions by means of the
manual direction keys.
These dimensions can be approached again, either auto-
matically in the MANUAL CONTROL operating mode
or via the parts program.

	Press the SET UP MACHINE DIMENS. softkey.	ENTER NO. MACH. DIMENS.:
	Enter digits (e.g.: 1)	
	Confirm	REACH MEAS. POINT +X AND/ OR MENU KEY

>N 751 DIM. 1   -X 0.000   +X 0.000   -Z 0.000   +Z 0.000

Approach the position using
manual direction keys.

	Press the ACCEPT NEW VALUE softkey.	CONFIRM OR NEW INPUT:
	Enter digits	
	Confirm	Entered value is accepted Note: The values refer to the zero point of the machine or the coordinate system of the machine, that is, the values for X are not regarded as diameter dimensions.

or

	Confirm	The approached value is transferred to the selected parameter memory.

If the other dimensions (-X,
+Z, -Z) are to be switched
over to:

Press the MAINTAIN VALUE
softkey.

Same procedure as for value X+

Press the SET DIMENS.
INACTIVE softkey.

The selected dimension is set to
±9999.999.

Press the HANDWHEEL
X 0.01 or Z 0.01
softkey.

The particular positions can be approached
by traversing the slides by means of the
handwheel.

Press the TERMINATE
INPUT softkey.

Return to the main level of the
setting menu.

If those dimensions stored under para-
meters N751 to N759 are to be approached,
then proceed as follows:

Press the APPROACH MACH.
DIMENS. softkey.

ENTER NO. MACHINE DIMENSION:

Enter digits

Confirm

ACTUATE   MANUAL   DIRECTION

Keep manual
direction keys
pressed down.

The corresponding slides are moved in the
direction of the programmed dimension
and stop there.

Press the END softkey.

Return to the main level of the setting
menu.

Extended actual value display
If in addition to the "large" actual value display the spindle values etc.
are to te displayed, the operator has to switch over to the extended
actual value display (see also display configuration in chapter 0)

>	Press continue key if necessary.	The second follow-up menu of the MANUAL CONTROL operating mode appears.
▦	Press softkey DISPLAY SELECT.	Switch-over to the extended actual value display. The mode of representation corresponds to the values programmed values under parameter N0006. If no reference run has yet taken place, an --- R --- appears instead of the relevant value.
▦	Press again the softkey DISPLAY SELECT.	Switch-back to the normal actual value display.

**Turning in longitudinal and transversal
direction with constant feed**
If a workpiece or a blank part is to be turned
in longitudinal or transversal direction, the
direction can be chosen using the following menu.

Press the CONSTANT
FEED softkey.

SELECT DIRECTION FOR CONSTANT
    FEED:

	FACING +X	
TURNING LONGIT. -Z		TURNING LONGIT. +Z
	FACING -X	

e.g. press the softkey FACING
-X.

ACTIVATE CONSTANT FEED BY
"CYCLE START"

Press CYCLE START.

TERMINATE CONTINUOUS FEED
    ONLY BY "CYCLE STOP"

The facing process is performed until the
button Cycle Stop is pressed.

Press CYCLE STOP.

End of processing and return to the menu
"continuous feed".

Press the HIGHER key.

Return to the main level of the manual
control operating mode.

Note
It is necessary to define the protection zones
for the manual control operating mode before
working with the continuous feed mode in order
to prevent any collisions which might
possibly occur.
During the turning process the tool movements should
be watched closely, if necessary a low feed rate should
be defined.

D-correction via handwheel
By means of this function the D-corrections derived "by hand" can be transferred into the corresponding parameter memory. D-corrections for the milling cutter diameter (DD) cannot be altered with the handwheel. They are only shown on the dialogue line together with corrections DX and DZ.

	Press the HANDWHEEL ACTIVE D-CORRECT. softkey.	The currently active D-correction of the tool used at that moment is displayed in the dialog line: D... DX +0.000 DZ +0.000 DD +0.000
	e.g. press the HANDWHEEL X-AXIS 0.01 softkey.	HANDWHEEL 0.01
	Moving the handwheel	The D-correction is displayed in the dialog line.
	e.g. press the HANDWHEEL Z-AXIS 0.01 softkey.	HANDWHEEL 0.01
	Moving the handwheel	The D-correction is displayed in the dialog line.
	Press the TERMINATE HANDWHEEL CALCULAT. softkey.	The D-correction is accepted by the control system and transferred into the corresponding parameter memory.

Setting mode

Press the TOOL SETTING MODE softkey.

The menu for setting up is displayed.

TOOL GAUGING PROBE	TOOL CHANGE POINT	SET ZERO POINT
PROTEC- TIVE ZONES	T TOOL	ZERO POINT SHIFT
BACK-UP TOOL	SERVICE LIFE	M FUNCTION

Defining protective zones
The ELTROPILOT L2-control system
provides the possibility in MANUAL
CONTROL mode of defining protective
zones outside of which the tool tip can
no longer be moved: the "manually"
established values are automatically
transferred into parameter N0201.
To set the individual limit values
for the protective zones, proceed
as follows:

Press the PROTECTIVE
ZONES softkey.

Press the MAINTAIN VALUE   Displayed limit value is retained.
softkey.

Press the PROTEC. Z.   The protective zone is set; the value
INACTIVE softkey.   9999.999 appears.

If the value is to be altered:

Run slide to desired position
with control keys.

Press the ACCEPT NEW VALUE   CONFIRMATION OR NEW INPUT
softkey.

Confirm   Desired value is accepted.

Enter digits.

Confirm.   Entered value is accepted.

Proceed in the same way for limit
values +Z, +X, -Z

If not all position values are to
be altered:

Press the TERMINATE INPUT   Return to the SETTING MODE menu.
softkey.

Tool change

   Press the T TOOL
softkey.

INPUT TOOL NUMBER:

T5112

D51
DX   0.000
DZ   0.000

   Enter digits.

Either 2-digit (Tpp):
tool file number =
position number = pp
With the 2-digit input of the tool
number an additional offset can be
programmed under parameter N12
(see section 6).

or 4-digit (Tnnpp):
Tool file number = nn
position number = pp

For example: Tool no. 51 is in the
turret at position 12, so enter
5112.

   Confirm.

Turret position 12 is moved into
machining position. The tool data
(-lengths) were obtained from tool
file N1051.
Tool compensation **nn** is active.

or as a 6-digit tool identity number:
TnnnOpp / iiiiii
tool file number = nn
tool position number = Opp
tool identity number = iiiiii

(For this purpose, parameter N1600 must be set to 1 and the tool reference list must be programmed appropriately; see description of parameters N1600 to N1664.)

For example: Enter 280455 if parameter N1651 is programmed as follows (N1651 51 ID 280455 T012).

Confirm   Turret position 12 is moved into machining position. The tool data (lengths) was/were obtained from tool file N1051.
Tool compensation **nn** is active.

T   51012 / 280455

D51
DX   0.000
DZ   0.000

If the tool control was switched on using   T   51012
parameter N1200, and tool life/batch size   01:54:12.0   73%
of the selected tool was determined under   D51
parameters N1201 to N1264, then the value   DX   0.000
for the **current tool life/batch size** is   DZ   0.000
displayed between the T- and D-display.
The display in percentage values repre-
sents the current tool life/batch size relative to
the programmed tool life/batch size (100%).

Note: In MANUAL CONTROL operating
           mode the control system does not
           count time/batch size.

Setting NC zero point
The function SET ZERO POINT
determines the workpiece zero
point. Usually this position is
set on the turning axis in X
and on the work face in Z.

Press the SET ZERO POINT softkey.	INPUT TOOL NUMBER:

Select setting tool

Enter tool number (e.g. of the setting tool). (Number can also be in 4 or 6 digits)	ACTUATE MENU KEY:

ACCEPT NEW VALUE	MAINTAIN VALUE	TERMI- NATE INPUT
WITHOUT PROTECTION ZONES		TOOL TYPE MENU
H-WHEEL X-AXIS 0.01	H-WHEEL Z-AXIS 0.01	

Confirm	Tool type is not altered.

If a different tool type is to
be selected

Press the TOOL TYPE MENU softkey.	The tool type menu appears with a following menu on the same level.
Softkey for desired tool type selection.	Graphic support facility displays tool type.
Switch to left or right tool position if required.	

⊞	Press the FURTHER TOOL POSITION softkey	Sets different tool position. (I and K signs of the file are automatically allocated to the selected tool position.)
⊞	Press the INPUT TOOL TYPE FINISHED softkey.	Selected tool type is defined; the cursor jumps to the next address parameter (FC). The selected tool type continues to be displayed on the monitor.

Color code for graphic simulation (option)
The color code determines the color of the tool path in graphic simulation.

⊞	Confirm color code or enter new code.	

Scratching

↑ ↓ ◯	Move in X-direction to scratch position, using manual direction keys or handwheel.	REACH MEASURING POINT IN X AND/OR MENU KEY: Tool tip approaches scratch point in X.
→	Move away from part in Z direction only.	Slide moves away from workpiece.
⊞	Press the ACCEPT NEW VALUE softkey.	CONFIRMATION OR NEW INPUT:
▬	Confirm.	Value is transferred to control system.

If workpiece zero is to be located on the turning axis, then enter the turned diameter.

Enter digits.

Entered value is transferred
to the control system.

Confirm

REACH MEASURING POINT IN Z
AND/OR MENU KEY.

Move in Z-direction to scratch
position using manual direction
keys or handwheel.

Tool tip approaches scratch point
in Z on workpiece.

Move away from part in X-
direction only.

Slide moves away from workpiece.

Press the ACCEPT NEW VALUE
softkey.

CONFIRMATION OR NEW INPUT:

Confirm.

Value is transferred to the control
system.

Enter zero if the workpiece zero
point is to be located on the
workpiece face.

Enter digits.

Confirm.

Entered value is transferred to
the control system.

Workpiece zero is set with the setting
tool.
If the value zero was entered for Z
and the diameter for X, actual value
display will be X0 and Z0 when the tip
of the setting tool is located
at workpiece zero.
The relevant tool offset is zeroed.

Enter tool data
The control system requests the
data required for the desired
tool type:
For the setting tool, enter zero
for X and Z, for subsequent tools
see next page.

ACTUATE MENU KEY:

Press the MAINTAIN
VALUE softkey.

Displayed value is transferred.

If the value is to be altered:

Press the ACCEPT NEW
VALUE softkey.

INPUT TOOL PARAMETER
OR CONFIRM:

Enter digits

Confirm

Entered value is transferred
to the control system.

After input has been completed, the
actual value display contains the
scratch value for X and Z, the tool
data line contains the dimension
value related to the setting tool.
Thus, X0 and Z0 are given for the
setting tool itself.
If input is to be terminated at
a certain point:

Press the TERMINATE INPUT
softkey.

Entering back-up tool
In order to be able to run a
program with various different
tools, the programmed workpiece
contour may not be dependent on
tool geometry.
If the zero point is set with the
roughing tool, the value zero is
allocated when the momentary slide
position is scratched.
The finishing tool, for example,
will not scratch at the same
position. If the finishing tool is
to be located at the workpiece zero
point, the slides must be in a
different position.
This correction of the slide
position must be made for all
tools.
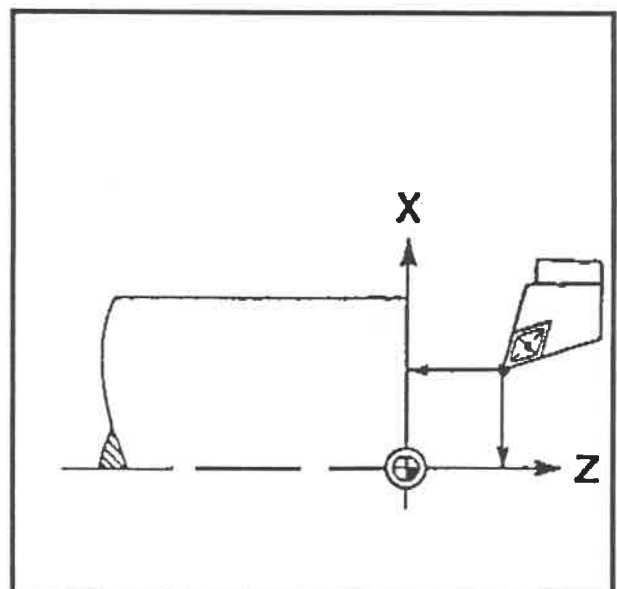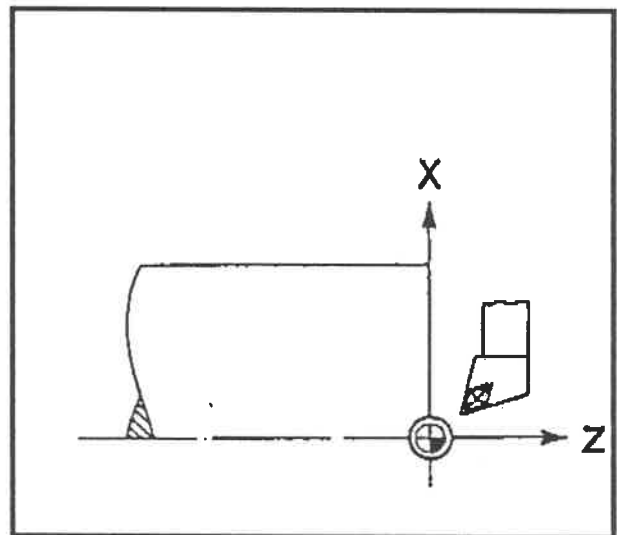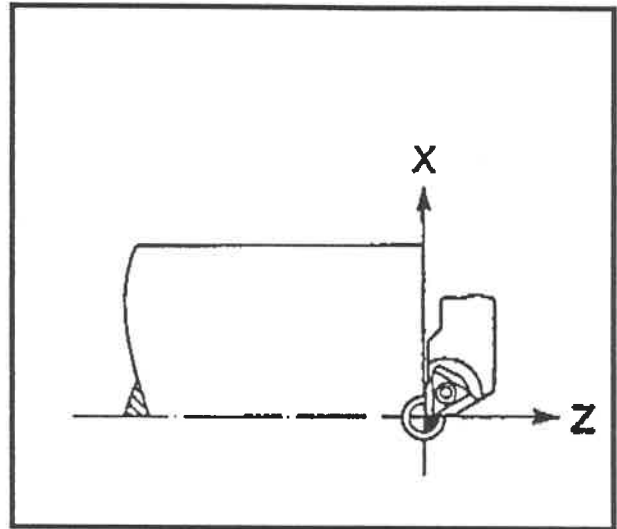
To do this, the control system is
equipped with a standard **tool
compensation facility.**
The length difference in Z-
direction and the offset in X-
direction are called compensation
values.
The sign determines the direction
of compensation.

Example
Compensation in X: tool 2 must
be corrected in the direction of
the X-axis, i.e. +.
Compensation in Z: tool 2 must
be corrected in the direction of
the Z-axis, i.e. -.

For the back-up tool, two cases
are differentiated:

1. Compensation values of the
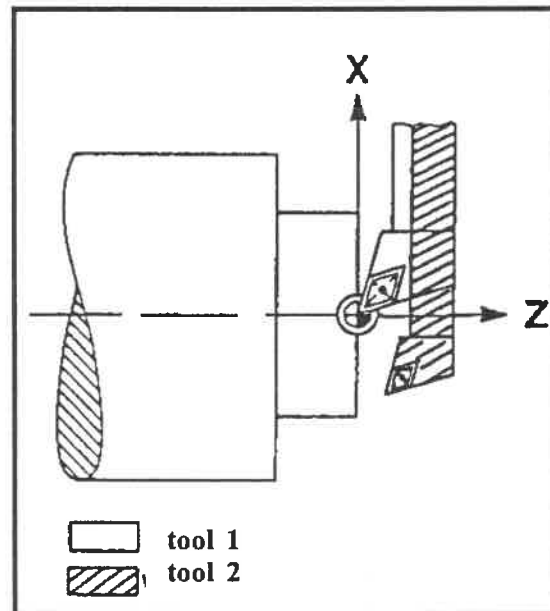back-up tool to the setting tool
are **unknown**:

Press the BACK-UP TOOL
softkey.

Scratch and transfer values as
for setting tool.

Note: By pressing the softkey WITHOUT
PROTECTION ZONES it is possible to
deactivate the protection zone monitor for
the tool nose (N201) during the scratching
procedure by the back-up tool, for
example so that very long tools such as
boring bars can also be scratched. The
message WITHOUT PROTECTION
ZONES then appears on the control
screen above the dialogue line. In this
case the operator must be extremely
careful when moving the tool.
After exiting the tool set-up mode
(softkey TERMINATE ENTRY), the
control automatically reactivates the
protective effect of parameter N201.

tool 1
tool 2

2. Compensation values of the back-
up tool to the setting tool are
known; e.g. for preset tools:

Press the BACK-UP TOOL
softkey.

Enter the known compensation values
for X and Z as parameters.

Tool change point (for machines with tool turret)
To change a tool, a slide
position can be set within the
working area.
This position is stored under
parameter N202 TOOL CHANGE POINT.
This position can be engaged in the
NC program with G14.
The numeric values relate to the
machine zero point.

Note

The tool change point should be selected
with particular care (consider all tools
of the turret).

Danger of collision!

Press the TOOL CHANGE
POINT softkey.

REACH MEASURING POINT IN X
AND / OR MENU KEY

N202 TOOL CHANGE POINT
X...   Z...

ACCEPT NEW VALUE	MAINTAIN VALUE	TERMI-NATE INPUT
H-WHEEL X-AXIS 0.01	H-WHEEL Z-AXIS 0.01	

If the value in X is to be
retained:

Press the MAINTAIN
VALUE softkey.

Value is maintained.

Traverse to X-value of the
tool change point using manual
direction keys and handwheel.

Press the ACCEPT NEW   CONFIRMATION OR NEW INPUT
VALUE softkey

Confirm.   Engaged X-value for tool change
                                        point is transferred.

Enter digits.

Confirm.   Entered X-value for tool change
                                        point is transferred.

                                        REACH MEASURING POINT IN Z
                                        AND/OR MENU KEY

If the value in Z is to
be retained:

Press the MAINTAIN   Value is maintained.
VALUE softkey

Traverse to Z-value of tool
change point using manual
direction keys or handwheel.

Press the ACCEPT NEW   CONFIRMATION OR NEW INPUT:
VALUE softkey.

Confirm   Engaged Z-value for tool change
                                        point is transferred.

Enter digits.

Confirm.

Entered Z-value for tool change
point is transferred.

Press the TERMINATE INPUT
softkey.

Return to setup menu.

Zero offset
Once tool file data have
been correlated, and if
the original setting tool
is no longer in the tool
file, a new workpiece zero
cannot be set without
rendering the set tool
compensation invalid.
It is then simply a case of
offsetting the zero point

Press the ZERO POINT
SHIFT softkey.

REACH MEASURING POINT IN X
AND/OR MENU KEY.

N50 NC-ZERO POINT X...   Z...

ACCEPT NEW VALUE	MAINTAIN VALUE	TERMI-NATE INPUT
H-WHEEL X-AXIS 0.01	H-WHEEL Z-AXIS 0.01	

If the displayed X direction
value is to be maintained:

Press the MAINTAIN
VALUE softkey.

Original X-value is maintained.

Traverse to new zero point
using manual direction key
or handwheel.

Press the ACCEPT NEW
VALUE softkey.

CONFIRMATION OR NEW INPUT

Confirm.

Engaged value is transferred to
the control system.

X   0.000
Z   .....

Enter digits.

Confirm.

Entered value is transferred to
the control system.

Proceed accordingly for the Z-axis

Press the TERMINATE INPUT
softkey.

Return to setup menu.

MANUAL CONTROL Operating Mode/Tool Measuring

Measuring in the manual control operating mode
If the machine is provided with the corresponding equipment, then the
ELTROPILOT L2 control system allows for the measuring of tools within
the working area of the machine in order to be able to determine the
absolute lengths of a tool (presetter dimensions L and Q). This is done
in the manual control mode, and the received values are transferred to the
corresponding tool files.

There are three possibilities for measuring tool lengths in the manual control
mode. These depend on the equipment of the machine:

- Measuring with an optical system

- Measuring with a measuring probe

- Scratching with respect to an input measure entered by
  the operator

Measuring with an optical system
The following conditions must be fulfilled:

- The crossed wires of the optical system are placed in a defined
  position in the working area of the machine.
  These measures must be entered as machine dimension 9 under
  Parameter N759 +X +Z.

- Under parameter N693 must be selected the type of measuring 2.

Principle of functioning
By means of the manual direction keys the tip of the tool to be measured
is positioned near the crossed wires.
With the help of the handwheel and the optical system, the tool tip and
the crossed wires are positioned congruently.
Then the control calculates the presetter dimensions L and Q of the tool
using the difference between the measured value of the tool reference
point N and the measure N759 of the crossed wires. The presetter
dimensions are then transferred into the address parameters X and Z of
the corresponding tool file.

X

N = R
(222;100)

L

Q

optical system with crossed wires

N759   +X
(22)

M

Z

N759   +Z
(10)
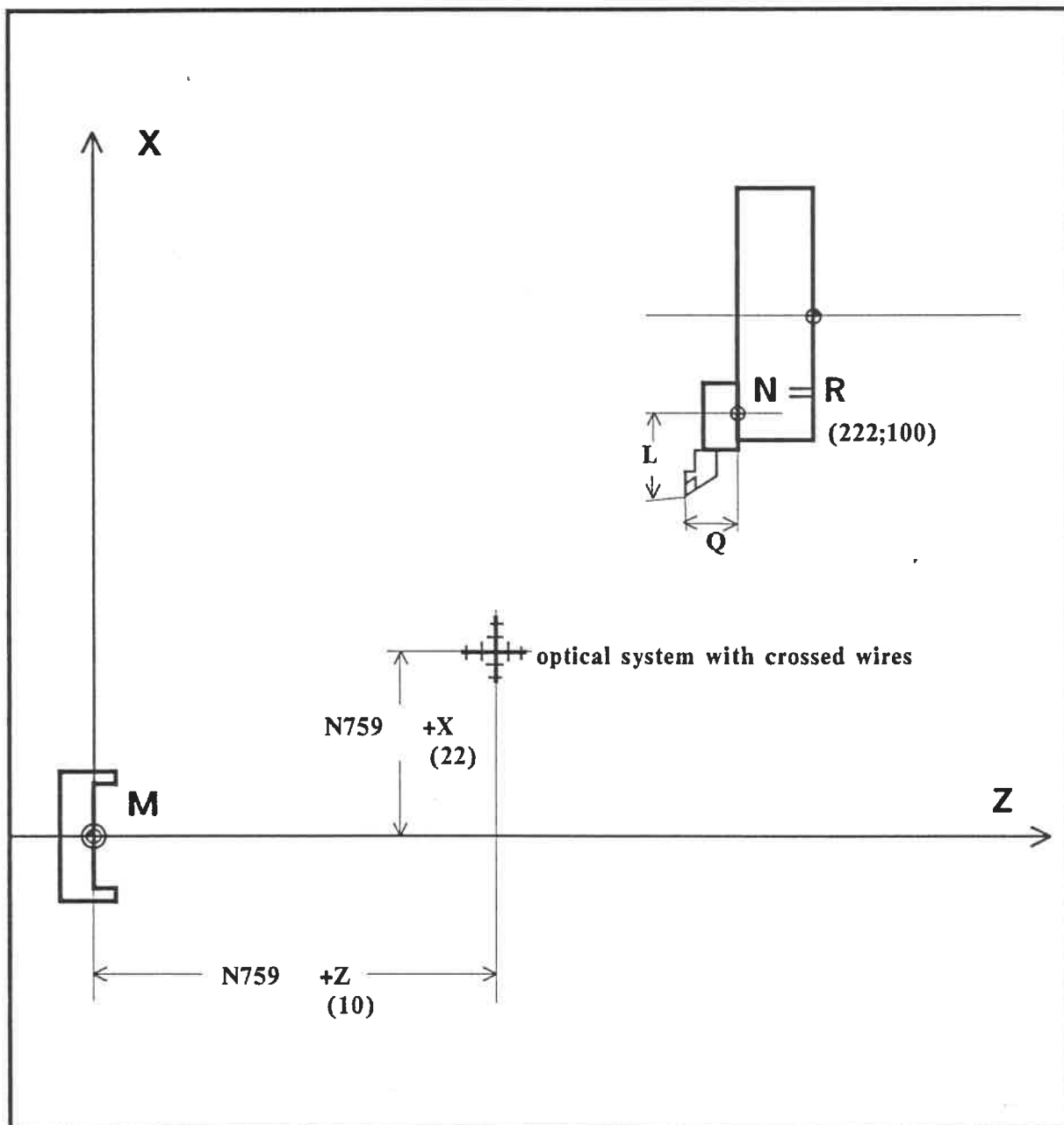
Performing of measurement
Proceed as follows:

	Select the manual control operating mode	
	Position the tool tip in front of the crossed wires using the manual direction keys	
	Press the TOOL SET-UP MODE softkey.	The set-up mode menu is displayed on the screen.
	Press the BACK-UP TOOL softkey.	INPUT TOOL NUMBER
	Enter digits.	
	Confirm.	ACTUATE MENU KEY N10.. T.. WT FC X Z I K > A B C D L
	If necessary, enter tool type and color code.	REACH MEASURING POINT IN X AND/OR MENU KEY:
	Press the HANDWHEEL X-AXIS 0.01 softkey.	The handwheel is activated.
	Movement of the handwheel	Positioning of the tool tip in X-direction into the center of the crossed wires.
	Press the ACCEPT NEW VALUE softkey.	X-value is transferred into the tool file. REACH MEASURING POINT IN Z AND/OR MENU KEY:
	Press the HANDWHEEL Z-AXIS 0.01 softkey.	The handwheel is activated.

◯	Movement of the handwheel.	Positioning the tool tip in Z-direction into the center of the crossed wires.
▦	Press the ACCEPT NEW VALUE softkey..	The value for Z is transferred into the tool file. ACTUATE MENU KEY

If necessary, enter the addresses I, K, A, B, C, D and L.

▦	Press the TERMINATE INPUT softkey	Return to the set-up menu.

Note
If the tool to be measured is a boring bar, then this is measured only in Z-direction. The control system takes the X-value from parameter N57 and enters it into the tool file.

Measuring with a measuring probe

For this, the following conditions must be fulfilled:

- The measuring probe is placed in a defined position within the workspace of the machine.
  The value of this position must be entered as machine dimension **8** under parameter N758 +X +Z .

- Parameter N690 is used to switch on the in-process-measuring that is, the measuring circuit number 1 or 2 is selected.

- Under parameter N692 must be entered a measuring feed rate in [mm/min].

Principle
With the help of the manual direction keys the tool tip of the tool to be measured is positioned near the measuring probe.
By means of the manual direction keys and the measuring feed programmed in parameter N692 the measuring probe is started up and deflected in X-direction.

After a deceleration of a maximum of 20 milliseconds the axis drive is
stopped.
In order for the NC to be able to monitor the measuring probe and for the
probe to be deflected in Z-direction, too, the tool must be withdrawn by
at least 5 mm against the direction of the starting movement.
Taking the difference between the measure of the tool reference point
N and the measure of the measuring probe N758 the control system
calculates the presetter dimensions L and Q of the tool and transfers them
into the address parameters X and Z of the corresponding tool file.

switching measuring probe
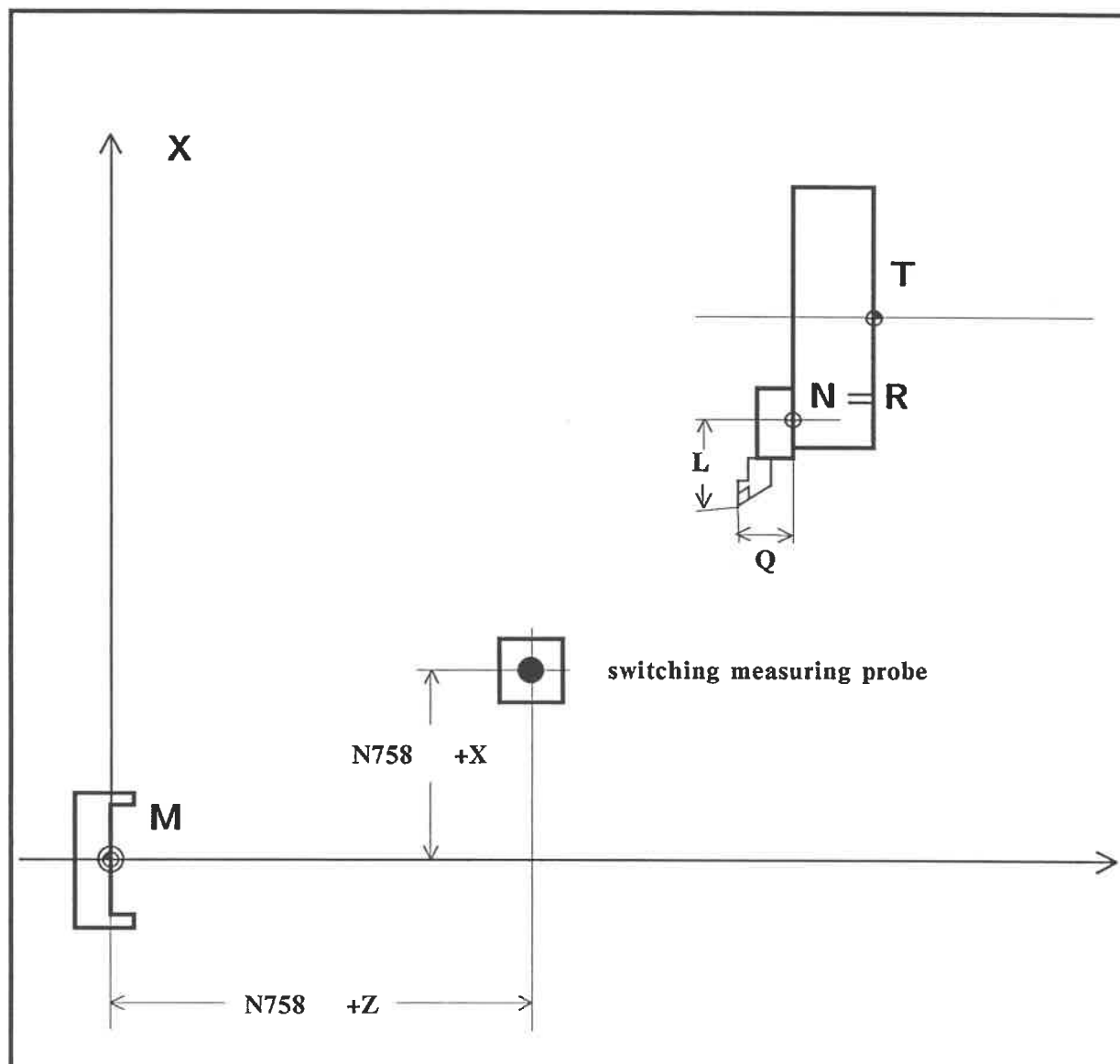
Program execution
Proceed as follows:

Select the manual control
operating mode.

Position the tool tip in front ot
the measuring probe by using the
manual direction keys.

Press the TOOL SET-UP MODE SOFTKEY .	The set-up mode is displayed on the screen.
Press the TOOL GAUGING PROBE softkey	INPUT TOOL NUMBER
Enter digits	

	Confirm	ACTUATE MENU KEY N10.. T.. WT FC X Z I K > A B C D L

If necessary, enter tool type or
color code or press twice
softkey MAINTAIN VALUE.

REACH MEASURE POINT IN X
ˈAND/OR MENU KEY:

Start-up the measuring probe in
X-direction using the manual
direction keys.

The tool tip approaches the probe in X-
direction at a measuring feed rate
defined under parameter N692.
Then the tool tip is stopped there.
MEASURE VALUE X= ...
RETURN FROM MEASURING
PROBE

Withdraw in opposite direction of
the starting movement by using the
manual direction keys.

The probe is again monitored by the
NC.
REACH MEASURE POINT IN Z/
AND/OR MENU KEY:

When even the reference gauge ist to be
calculated in direction Z, the following is to be effected:

Press softkey
RETAIN VALUE

REACH MEASURE POINT IN Z/
AND/OR MENU KEY:

Start-up the probe in Z-direction by using the manual direction keys.

The tool tip approaches the probe in Z-direction at the measuring feed pro- grammed under parameter N692 Then it is stopped there. MEASURE VALUE X= ... RETURN FROM MEASURING PROBE

Withdraw in opposite direction of the starting movement by using the manual direction keys.

The probe is again monitored by the NC. REACH MEASURE POINT OR MENU KEY

Note:
Thereby the reference gauge can not to be fixed by input of numbers.

Using the manual direction keys, withdraw in opposite direction to the starting movement.

The probe is again monitored by the NC. ACTUATE MENU KEY:

If necessary, enter I, K, A, B, C, D and L

Press the TERMINATE INPUT softkey.

Return to the set-up menu.

Note
If a milling tool is to be measured, then this is done only in Z-direction. The value for X is taken from parameter N57 and transferred to the tool file.

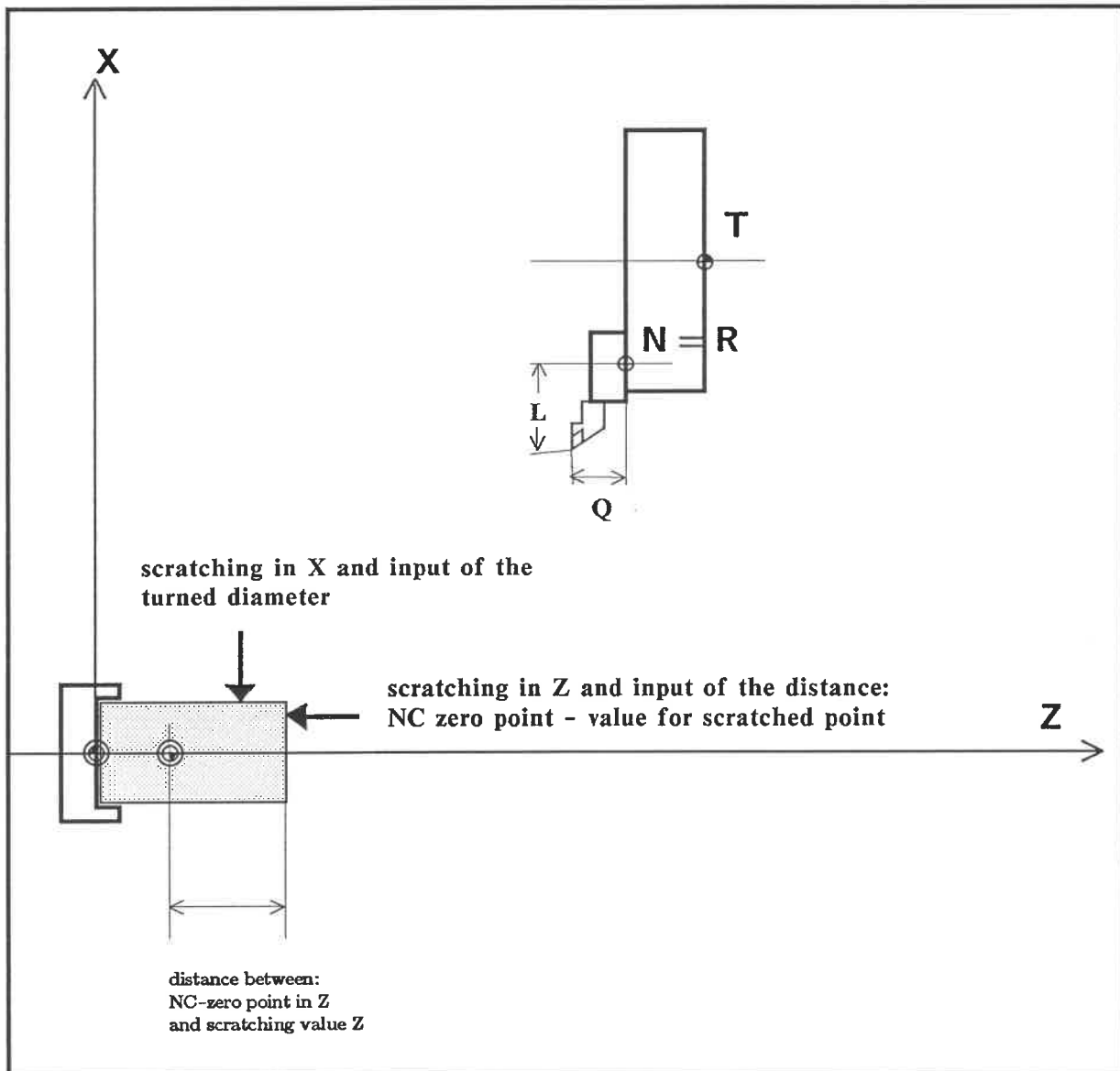## Scratching with respect to an input measure entered by the operator

On the previous pages (tool set-up mode) was described the scratching and fixing of the NC zero point by using the setting tool, to which then all follow-up tools refer (for this, parameter N693 must be set to zero) . However, it is also possible to determine the absolute lengths of the tools by scratching without reference to a setting tool. The values determined are the presetter dimensions L and Q. This technique of determining the absolute tool lengths is required when a tool is to be measured, but access to the crossed wires of the measuring optical system or the measuring probe is hindered by a clamped workpiece.

The following conditions must be fulfilled when using this technique:

-   The NC zero point (parameter N50) must be determined and
    the X-value must **absolutely be zero.**

-   Measuring type 1 must be selected under parameter N693.

Principle and program execution

scratching in X and input of the
turned diameter

scratching in Z and input of the distance:
NC zero point - value for scratched point

distance between:
NC-zero point in Z
and scratching value Z

After the scratching process, the diameter reached by this is transferred to the control by actuating the ACCEPT NEW VALUE softkey and by entering the value of the diameter in digits. Since the NC zero point is identical with the machine zero point as far as the X direction is concerned, the value reached by scratching refers directly to the machine zero. Into the tool file is entered the difference between tool reference point N and the scratching value as presetter dimension L under address X.

Then scratching is done in the Z-direction. The NC zero point in Z-direction is known, and also the distance between NC zero in Z and the scratching value in Z must be known. After scratching this distance is entered into the control system by actuating the softkey ACCEPT NEW VALUE and by entering this value in digits.

Thus the difference between tool reference point N and the scratching value is entered as presetter dimension Q under address Z into the control system. The actual execution of this scratching process happens in the same way as the scratching process described on the previous pages and is therefore not described in detail (see scratching with setting tool and follow-up tools referring to it).

MANUAL CONTROL Operating Mode

Entering tool service life
Entering the tool service life/batch size is possible both in
the PARAMETER operating mode and in the MANUAL
CONTROL operating mode where this is possible
when in the setting up mode (see parameters N1200 to N1264).
Furthermore the expired current tool life/batch size can be
returned to the value of the originally programmed service life/
batch size by actuating the **ACCEPT PROGR. TIME** softkey.
Thus the tool diagnosis variables N1401 to N1464 are updated
as well.

⊞	Press the TOOL SET-UP MODE softkey.	TOOL SETTING UP
⊞	Press the SERVICE LIFE softkey.	INPUT TOOL NUMBER:

If the service life/batch size of the current tool, i.e. the one that is connected,
is to be updated, then proceed as follows:

▬	Confirm.	N12.. T.. S AZ PS AS >

If the service life of another tool
is to be updated:

⊞	Enter digits. (e.g.: 2)	
▬	Confirm.	N12.. T.. S PZ AZ PS AS >

The following submenu is displayed

ACCEPT NEW VALUE	MAINTAIN VALUE	TERMI- NATE INPUT
ACCEPT PROG. TIME		

Press the ACCEPT NEW VALUE

TYPE OF MONITOR. (TIME = 0, PIECE = 1) or
INPUT PROGRAMMED SERVICE LIFE OR CONFIRM:

Confirm.

Value is retained

Enter digits

Confirm.

Entered value is accepted

Press the RETAIN VALUE softkey.

Cursor jumps to the next value

If all service life data have been updated, then also
the **D-compensations** for the tool concerned are offered
in dialog by the control system.

N1102 D2 X 0.000   Z 0.000 D 0.000

D-compensation values can now be altered by actuating the corresponding softkeys.

If the expired current service life is to
be reset to the value of the programmed
service life, then select the corresponding
tool and proceed as follows:

Press the ACCEPT PROGR. TIME softkey.

Service life is reset to the programmed value.

Press the TERMINATE INPUT softkey.

Data input for service life is terminated and input of D compensation is offered.

Press the TERMINATE INPUT softkey again.

Return to the **tool setup menu**

Setting-up the chucking equipment
The chucking equipment can be set up by means of the handwheel. In the process, the C-axis can first be pre-positioned in the jog control mode to an angular value freely definable by the operator. The travelling speed desired at this time can also be defined by the operator. Subsequently the spindle can be moved to the exact target position with the handwheel. Furthermore this position can be transferred to the parameter memory N127 as the zero point of the C-axis (additive zero point shift). As an alternative, it is possible to transfer to the parameter memory the value travelled with the handwheel without moving the spindle (absolute zero point shift referred to the reference dimension of C-axis).
When the control system is equipped with a position regulation unit, then the point stopping of the spindle can be adjusted using the handwheel as well. The values determined in that way can also be transferred to the parameter memory as zero point shift.

Press the CLAMPING
DEVICE SET-UP
MODE softkey.

The following menu is displayed on the monitor:

C-AXIS 1 SET-UP MODE		
		SPINDLE SET-UP MODE
		SPINDLE SELECT.

Positioning the C-axis in the jog control mode
Proceed as follows to position the C-axis in the jog control mode:

Press the C-AXIS
SET-UP MODE
softkey.

The following menu is displayed:

ACCEPT NEW	JOG	FINISH INPUT
ON	H-WHEEL C-AXIS	OFF
H-WHEEL C-AXIS 0.1	H-WHEEL C-AXIS 0.01	H-WHEEL C-AXIS 0.001

	Press the ON softkey.	The C-axis is swivelled in and executes a reference run 1 C 0.000
	Press the JOG softkey.	ANGULAR VELOCITY C - 1:
	Enter the digits in [degrees/min], e.g.: 80	ANGULAR VELOCITY C - 1: 80

Information: If a higher angular velocity is entered in the manual
control mode than the one defined under the parameters,
positioning is executed at the maximum speed defined in
the parameters. The following note appears in the input line:
"MAXIMUM VELOCITY FROM PARAMETER".

	Confirm	ANGULAR VALUE C - 1:
	Enter the digits, e.g.: 20	ANGULAR VALUE C - 1: 20.000

	Confirm.		1/MIN	DEGREES
		CA1:	80	20.000

	Jog	The C-axis rotates at the defined angular velocity in jog control mode and stops at the programmed position.

Moving the C-axis with the handwheel and setting the zero point
Proceed as follows to the exact position further with the handwheel:

	e.g. press the H-WHEEL C-AXIS 1.0 softkey.	POSITION THE SPINDLE AND/OR PRESS MENU KEY
	Turning the handwheel (e.g. by 10 dividing lines)	**1C +10.000**

Press softkey
TRANSFER NEW VALUE

Just confirm

IC 0.000

The actual-value display indicates
the new position to which the C-
axis was moved with the
handwheel as the zero point.
In parameter N127 +10.000 is
stored as the zero point shift. A
return branch then follows to the
previous menu level.

Note: This type of zero point shift
has an additive effect, i.e. when
executed repeatedly, each of the
individual shifts in parameter
N127 is added to the existing
value.

or

Digit entry (e.g. 30°)

Confirm

IC 30.000

The actual-value display indicates
the new position in response to the
digit entry, however the C-axis is
not moved. Only the reading of
the actual-value display shifts.
This zero point shift is
compensated in parameter N127,
i.e. parameter N127 stands at
minus 30.

Note: This zero point shift is not
additive in effect; it is always
absolutely referred to the
reference dimension of the C-axis
(0°).

or

Press softkey TERMINATE
ENTRY

The reading remains at IC +10,000
in the actual-value display. The
spindle was moved 10°, however
no zero point shift was carried
out. Return to the next high menu
level.

Turning the spindle and setting the zero point (position-controlled M19)
If the control system is equipped with a position control (position-controlled M19), a point stop
adjustment of the spindle can be executed. After the spindle has been moved to position control, proceed as follows:

Press softkey SPINDLE
SET-UP MODE.

The following menu is displayed:

ACCEPT NEW VALUE.		FINISH INPUT
	H-WHEEL SPINDLE 1.0	
H-WHEEL SPINDLE 0.1		

e.g. press softkey HANDWHEEL
SPINDLE 1.0.

POSITION THE SPINDLE AND/
OR PRESS MENU KEY

Turning the handwheel
(e.g. by 25 dividing lines)

DEGREE
25.000

Press softkey ACCEPT
NEW VALUE.

The actual-value-display of the spindle position displays the new positon as zero point.
In parameter N515 the value +25.000 is stored as zero point shift.
Then the system returns to the previous menu level.

or

Press softkey FINISH INPUT.
BEEENDEN drücken.

The actual value display continuously shows **DEGREES +25.000** .

Diagnosis as side-line operating mode
If the dignostic mode is to be selected as side line operation starting from this mode, then proceed as follows:

Press the DIAGNOSTIC MODE
softkey.

The main menu of the diagnosis operating mode is displayed (see also section 7.1 "diagnosis operating mode in side-line operation")

G-functions evaluated by the SPS
If the G-functions (G600 - G699)
evaluated by the SPS are to be
executed in the manual control
operating mode, proceed as follows:

Press softkey SPS G-FUNCTION.   ENTER G-FUNCTION:

Digit entry of the desired SPS   G SPS RECORD WORD X:
function (if supported by the
SPS)

Digit entry

Confirm   G SPS RECORD WORD Z:

Proceed in a like manner when entering
the other address parameters.
If the last address parameter has been
entered and/or confirmed, the function
is evaluated by the SPS and the machine
interface is configured accordingly

Auxiliary programs for extensive movement sequences
For extensive sequences of movement which are to be repeatedly
executed in the MANUAL CONTROL operating mode the
ELTROPILOT L2 control offers the option of filing these sequences in
the control's program memory as auxiliary programs. 99 auxiliary
programs can be stored; 1 to 9 are safeguarded with a password and
stored in the control's program memory as MANUAL 1 to MANUAL 9.
These programs have to be entered via punched tape or DNC operation
and cannot be altered by the operator at the control in the EDITOR
operating mode.
The operator can do as he wishes with programs 10 to 99 since they are
not protected. Consequently they have only the flag %10 to %99 in the
program memory and correspond to a normal parts program. Proceed as
follows when calling an auxiliary program in the MANUAL CONTROL
operating mode:

Press softkey MANUAL
PROGRAM SELECTION   ENTER NO. FOR PROGRAM "MANUAL":

Digit entry (from 1 to 99)

Confirm.   The selected program is indicated.

Press CYCLE START.   The selected program is started at
                                     the first record and processed under
                                     the protection conditions of the
                                     operating mode AUTOMATIC.

Note: It is imperative to adhere to the operating steps shown here
           for starting a manual program. If a feed override, speed
           override or the like is initiated after selecting the manual
           program for example, instead of pressing the button CYCLE
           START directly, the program in question is not started and
           has to be activated again although it is still on the screen.

           While the program is being executed, the handwheel can be
           used to override the speed and feed settings; the feed can be

interrupted by pressing the button FEED STOP.

Press the button CYCLE STOP,   The processing of the particular
if applicable.   auxiliary program is terminated
                                     immediately.