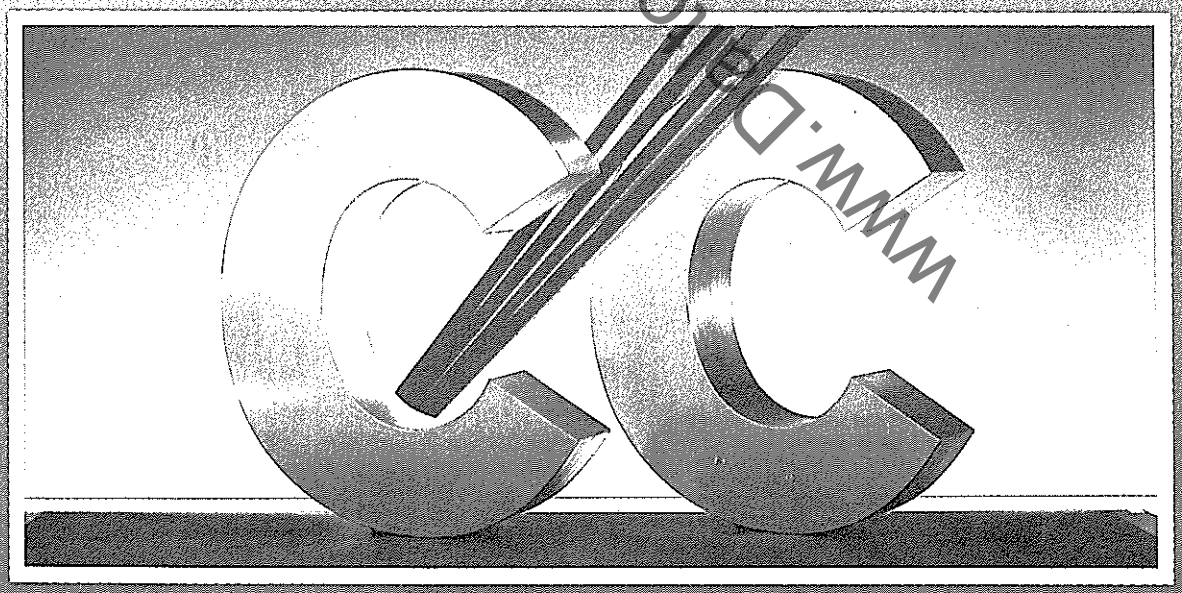
BOSCH

Flexible Automation

CC 100 M

User Manual
Stage 4

P.-Nr. 4110 / E2 – 05/90



Numerische Steuerungen
Computer Numerical Control

BOSCH

Schulungszentrum Flexible Automation

Kurse für:
- Bediener, Einrichter
- Projekteure, Programmierer
- Inbetriebnehmer, Instandhalter

Kurs-themen:
- Numerische Steuerungen (CNC) für Werkzeugmaschinen
- Speicherprogrammierbare Steuerungen (SPS)
- Elektrische Servoantriebe
- Robotersteuerungen für Roboter und Handhabungstechnik
- Widerstands-Schweißsteuerungen

Kurs-bezeichnung:
GL = Grundlagen
B = Bedienen
P = Programmieren
C = CPL/CPC
I = Inbetriebnahme Instandhalten

Ausbildungsvorschlag Numerische Steuerungen

Für Einsteiger ohne Erfahrung (alle Bereiche)

CC-GL
Basiskurs
mit CC 100M

Für Bediener und Einrichter mit allg. NC-Erfahrung

CC 100M-B
"Fräsen"
Bedienen

CC 100T-B
"Drehen"
Bedienen

CC 200T-B
"Drehen"
Bedienen

CC 300-B
Für CC 200M
CC 300M
Bedienen

Für Programmierer mit allg. NC-Erfahrung

Programmieren

Programmieren

grafisches Programmieren

CC 200M-P
grafisches
Programmieren

CC 300-P
für CC 200M
CC 300M
DIN-Programmieren

Für Programmierer: Aufbaukurs für parametrisches Programmieren

CC 100-C
für CC 100M
CC 100T
CPC-Programmieren

CC 300-C
für CC 200M
CC 300M
CPL-Programmieren

Für Inbetriebnehmer, Projekteure, Instandhalter mit produktspez. Kenntnissen

CC 100M-I
Instandhaltung
Inbetriebnahme

CC 100T-I
Instandhaltung
Inbetriebnahme

CC 200T-I
Instandhaltung
Inbetriebnahme

CC 300-I2
für CC 200M
CC 300M
Inbetriebnahme
Projektieren

CC 300-I1
für CC 200M
CC 300M
Inbetriebnahme
Instandhaltung

CC 100 M

User Manual
Stage 4

P.-Nr. 4110/E2-05/90

Description Stage 4

Contents

Description Stage 4 for the end user

Description Stage 4 for the machine tool builder

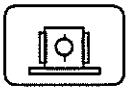
Conversion Stage 3 to Stage 4

1. Machine parameters

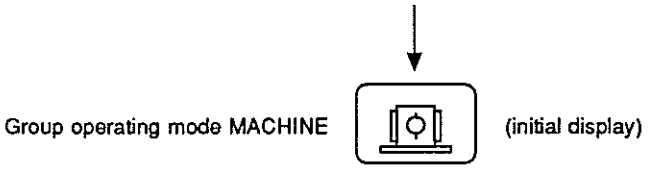
A new machine parameter MP 17 which permits optional assignment of the MTB keys on the manual panel with NC or PIC functions has been introduced in Stage 4.

The parameter can be assigned values between 0 and 15. The value specifies the number of NC keys. If compatibility with version ≤ Stage 3 is to be retained, the number 10 must be entered in these parameters. The upper 10 keys of the manual panel are then assigned NC functions, and the other keys set bits in the PIC.

2. Write protection

There are two write protection switches on the control. One is accessible on the front panel of the CP/MEM module (machine parameters). The second is mounted on the PIC board (PIC program). If one of the switches or both switches is or are not switched to write protection, the control signals this after power-up or after the Reset key on the power supply unit is pressed. One or both corresponding error messages are displayed, with softkey "PRESS TO CONTNUE" which must be pressed (see Figure). After acknowledgment by the softkeys, the control reverses to group operating mode Machine. If both switches are active, the control reverts directly to group operating mode MACHINE  after runup.

Mains switch ON



Write protection is active when the CP/MEM switch has been set to position 1 (write protection machine parameters) and the switch on PIC 200 supplementary board (I/O board) has been set to position 2 (write protection PIC program).

Simulation

Definition, function:

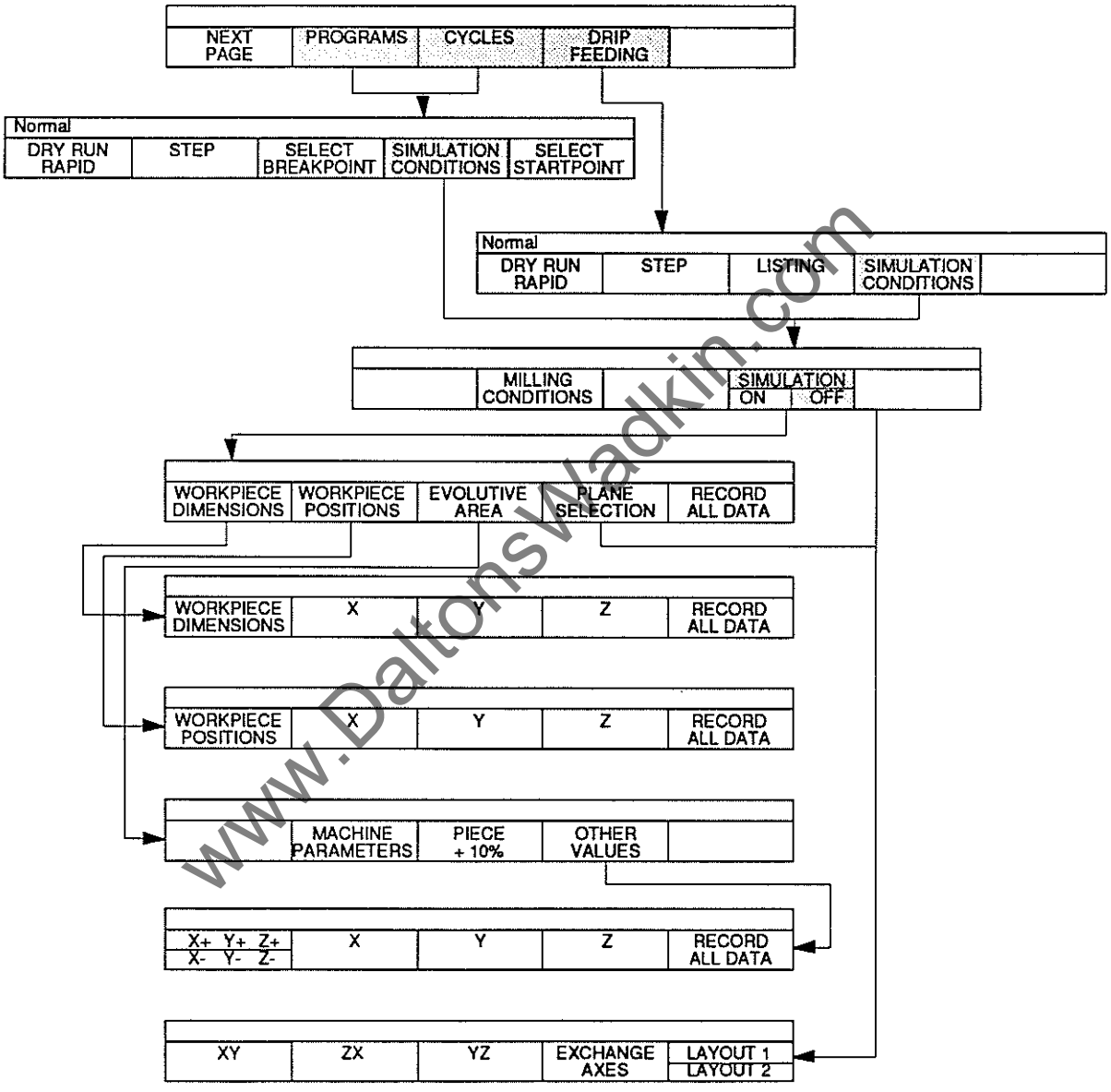
Simulation provides graphic support before execution. It offers the possibility of checking and optimizing the part program. The individual work steps of the part program are simulated in true-to-life manner. The user sees the tool traverse movements with the current speed (G0,G1). The speed can be varied via potentiometers.
The programmed contour becomes visible step by step from the workpiece which is displayed in the correct position (zero shift, actual values). Note that the contour is produced from the traverse paths of the tool center-point path. It is not a direct contour description. The control checks the program for operability, collision risks and technological boundary conditions. The function is available in group operating mode AUTOMATIC. The movements of the tool center point are visualized in a 2 1/2-D display. The axes of the selected working plane are represented in a 2-D image, whilst the third axis perpendicular to the working plane is represented on a vertical "scale" displayed next to the 2-D image.

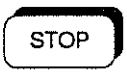
Simulation always starts with the current machine position; traversing of the machine is not possible.
The simulation speed can be switched over at any time between the programmed speed (status: NORMAL) and immediate display (without interpolation).

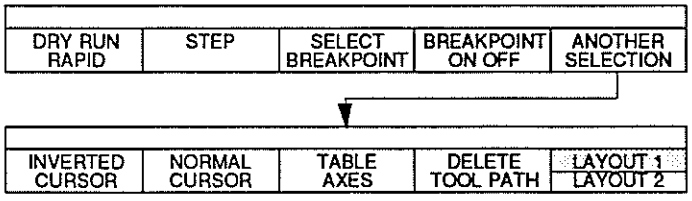
Operating overview:

The softkey architecture of group operating mode AUTOMATIC, extended with the simulation graphics, is displayed.

AUTOMATIC 



Interrupting simulation:  NC-STOP

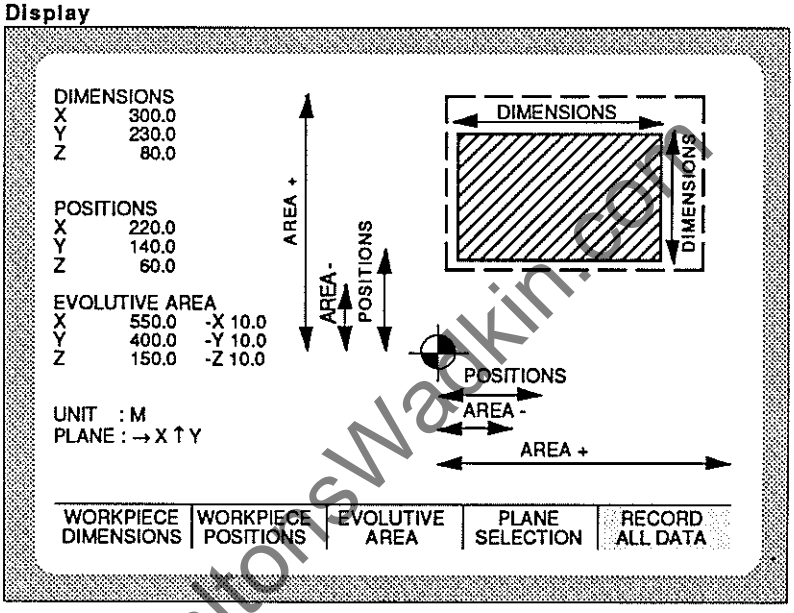


The simulation display:

	MILLING CONDITIONS		SIMULATION ON / OFF	

When key "SIMULATION ON" (group operating mode AUTOMATIC) is pressed, the control switches over to display of the simulation parameters.

Display



The following simulation parameters are available:

– Dimensions of the workpiece
– Position of the workpiece
– Determining the evolutive area
– Selection of the working plane

The parameter display offers the following softkey bottom panel:

WORKPIECE DIMENSIONS	WORKPIECE POSITIONS	EVOLUTIVE AREA	PLANE SELECTION	RECORD ALL DATA

Explanation of the individual softkey functions:

1. Softkeys WORKPIECE DIMENSIONS/WORKPIECE POSITIONS

Pressing one of the softkeys takes to the relevant softkey panel:

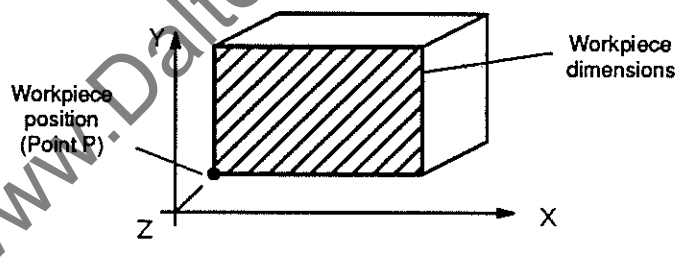
WORKPIECE DIMENSIONS	X	Y	Z	RECORD ALL DATA

or

WORKPIECE POSITIONS	X	Y	Z	RECORD ALL DATA

The softkey selected is displayed in reverse video mode. The dimensions or positions of the workpiece are programmed with softkeys X, Y and Z.

The axis value is accepted with ENTER. The sum of the axis entries must be confirmed with softkey "RECORD ALL DATA". When the axis values are accepted, the control switches back to the softkey bottom panel. The dimensions of a parallelepiped workpiece are transferred with softkey "WORKPIECE DIMENSIONS". The control stores the position of point P (workpiece datum) with softkey "WORKPIECE POSITIONS".

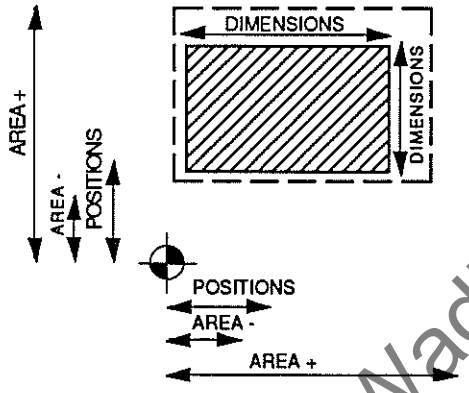


2. Softkey EVOLUTIVE AREA

This softkey takes you to the following softkey sub-level

	MACHINE PARAMETERS	PIECE + 10 %	OTHER VALUES	

The evolutive area (AREA +/-) is shown as a hatched rectangle on the general display. The size can be determined by the workpiece dimensions plus 10%, by the values of the software limit switches or by individual entries (softkey OTHER VALUES).



The machine parameters 38/39, 58/59, 78/79 (software limit switch range I) or 40/41, 60/61, 80/81 (software limit switch range II) are copied to the parameters for the evolutive area with softkey "MACHINE PARAMETERS". Which group of machine parameters is copied is dependent upon the interface signal A 1.0. With LOW level, the software limit switch range I is copied, and with HIGH level the software limit switch range II is copied.

Each side of the workpiece geometry is increased in size by 10 % with softkey "PIECE + 10 %", and the sum (110 %) is assigned to the "EVOLUTIVE AREA" parameters.

Example:

Workpiece position:	X = 0	Y = 0	Z = 0
Dimensions of the workpiece:	X = 10	Y = 20	Z = 30
"EVOLUTIVE AREA"	− X = 1	− Y = 2	− Z = 3
	+ X = 11	+ Y = 22	+ Z = 33

Individual "Evolutive area" parameters are defined or changed with softkey "OTHER VALUES". This leads you to a new softkey sub-level:

X+ Y+ Z+ X- Y- Z-	X	Y	Z	RECORD ALL DATA

The left-hand softkey defines whether positive (AREA +) or negative (AREA-) parameters are to be defined for the simulation area. The dimensions for each axis are entered with softkeys X, Y and Z and are transferred with ENTER.

After all axis values have been entered, this must be confirmed with softkey "RECORD ALL DATA". The control switches back to the softkey basic level.

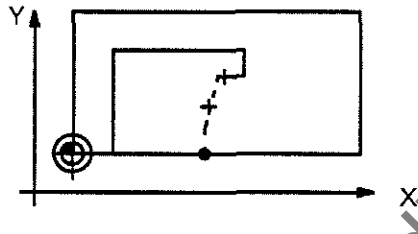
3. Softkey "PLANE SELECTION"

Pressing this key takes you to the following softkey sub-level:

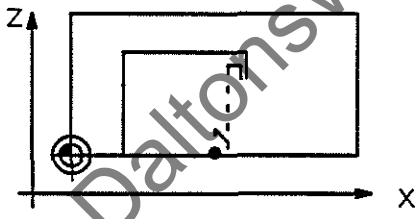
XY	ZX	YZ	EXCHANGE AXES	LAYOUT 1 LAYOUT 2

Softkeys XY, ZX or YZ select the two-dimensional part of the simulation. This axis combination is the simulation plane on the screen (the 3rd axis is recognizable as a column). The softkey of the active working plane is displayed in reverse video mode.

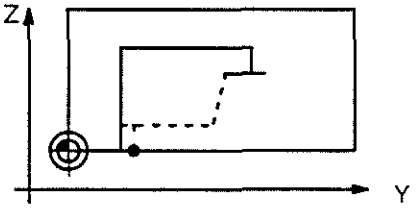
Working plane X/Y: Top view of the workpiece, finished part



Working plane Z/X: Front view of the workpiece, finished part

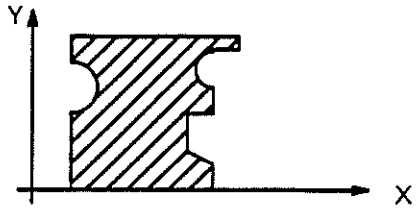


Working plane Y/Z: Side view of the workpiece, finished part



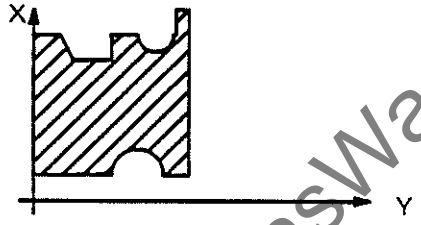
Softkey [EXCHANGE AXES] exchanges the axes of the active simulation plane

Example: Working plane X/Y - active:



Softkey [EXCHANGE AXES]

Exchange of the axes referred to the active simulation plane.

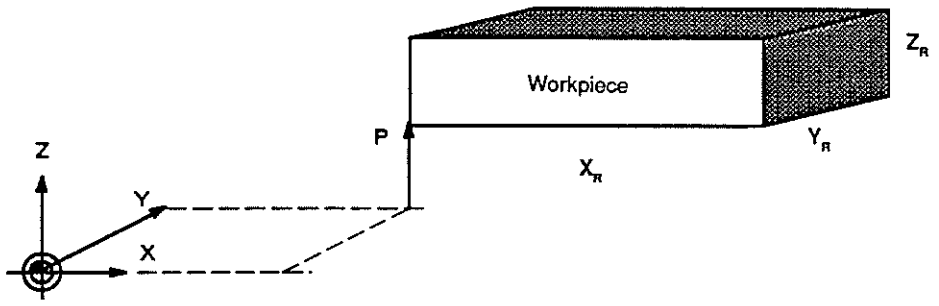


Softkeys "LAYOUT 1 / LAYOUT 2" changes the simulation display ("toggle" function). The active mode is displayed in reverse video mode (see example Page 8).

4. Softkey [RECORD ALL DATA]

This softkey transfers axis values (dimensions or positions of the workpiece) to the control. The control switches back to the softkey basic level. This softkey is displayed in reverse video mode in the basic level.
The softkey is omitted in the case of incorrect entries.
The simulation run must be started from this "level" (key "CYCLE START" on manual panel).

Illustration

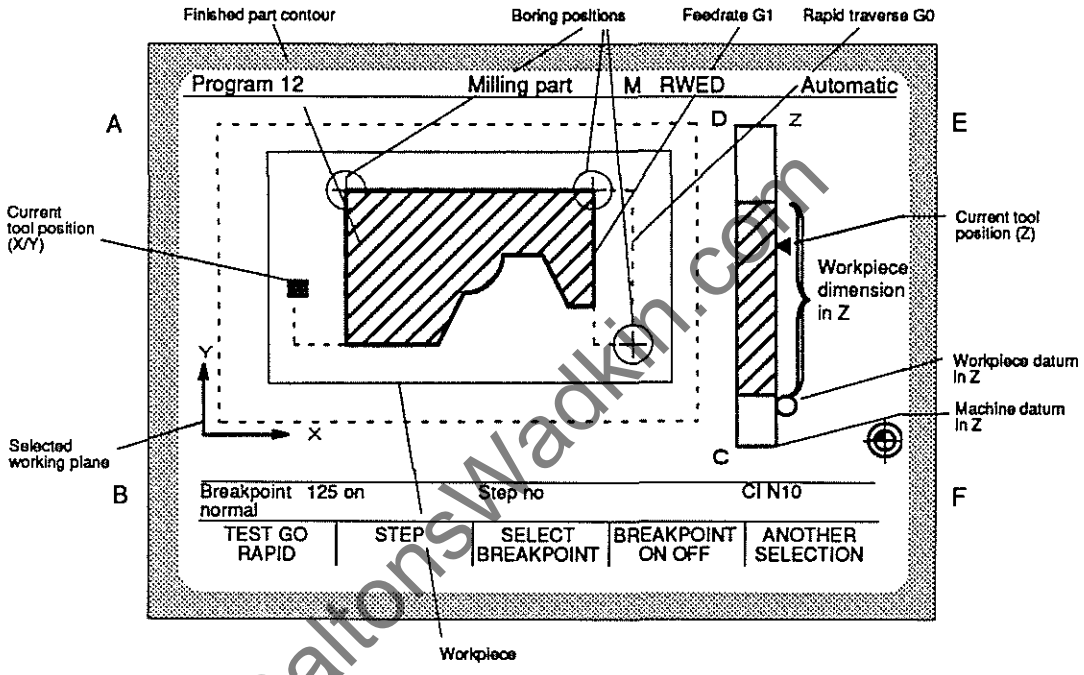


Machine datum

Simulation sequence:

After the parameters have been transferred with softkey RECORD ALL DATA , simulation starts with key "Cycle start" in 2 1/2-D graphics (in this case: working plane XY top view).

Illustration:



The window with the corner points A B C D shows:
– the tool in the selected working plane (XY, ZX or YZ)
– the workpiece and the programmed tool path.

The window with corner points C D E F shows:
– the tool position referred to the third axis perpendicular to the simulation plane
– the workpiece position referred to the third axis.

Tool position:
A small square, displayed in reverse video mode, shows the current tool position. The actual tool position is displayed as an arrow in the display window of the third axis with corner points C D E F.

Tool traverse movements:
– Tool rapid traverse movement (G0) as a dashed line
– Tool feed movement (G1, 2, 3, 5) as a continuous line.

Options during simulation:

START NC start key

Simulate break: Step N N 132 C67

NORMAL				
DRY RUN RAPID	STEP	SELECT BREAK = POINT	BREAKPOINT ON OFF	ANOTHER SELECTION

NORMAL: Interpolation with programmed feedrate; software limits and forbidden areas are allowed for.

RAPID: No axis interpolation; contours are computed using the graphic tools (straight line, circle).
Software limits and forbidden areas are ignored.

DRY RUN: Not possible in simulation

STOP NC stop key

After the simulation run has been interrupted with NC stop, there are the following setting and display options with softkey ANOTHER SELECTION

INVERTED CURSOR	NORMAL CURSOR	TABLE AXES	DELETE TOOL PATH	LAYOUT 1 / LAYOUT 2

1) Softkey DELETE TOOL PATH

Deleting the tool traverse paths
Paths of the tools which have already been traversed can be deleted to provide a clearer picture.

2) Softkey TABLE AXES

Besides the current axis positions and NC blocks, the display shows a section of the selected table.

Tables TOOLS | ZERO SHIFTS | VARIABLES can be selected.
Table contents **cannot be edited** during simulation.

3) Softkey INVERTED CURSOR , NORMAL CURSOR

ZOOM function (magnification)

After stopping the simulation, a window can be selected in order to permit better recognition of tool traverse paths, contour transitions or other details. The magnification can be changed as regards magnitude, direction and position via softkeys, regardless of the working plane selected in the part program.

- A "cursor" is displayed at the center of the workpiece when the key is pressed
- Positioning the cursor

ZOOM CURSOR	← ↓	SCROLL	→ ↑	RECORD

○ Determining the magnification:

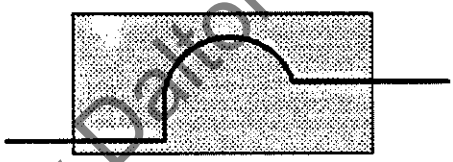
ZOOM CURSOR	ZOOM –		ZOOM +	RECORD

The larger the ZOOM area displayed in reverse video mode is, the lower is the magnification.

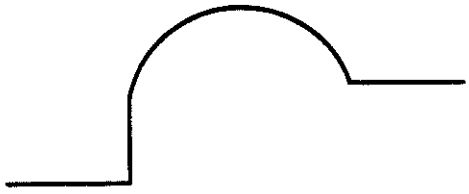
The selection must be confirmed with softkey [RECORD] . When the NC starts again, the control starts the program and displays the ZOOM cursor.

Example:
a. Determining the position of the "ZOOM cursor":

b. Selecting the magnification.



c. Displaying a section of the simulation area.



Difference between [ZOOM VERTICAL] and [ZOOM HORIZONTAL]

The cursor at the center of the workpiece (rectangle) is positioned horizontally (ZOOM HORIZONTAL) or positioned vertically (ZOOM VERTICAL).

4) The simulation graphics can be displayed in 2 different layouts.

Softkey LAYOUT 1 / LAYOUT 2

Layout 1 shows the tool traverse movements at feedrate (continuous line) or in rapid traverse (dashed line).

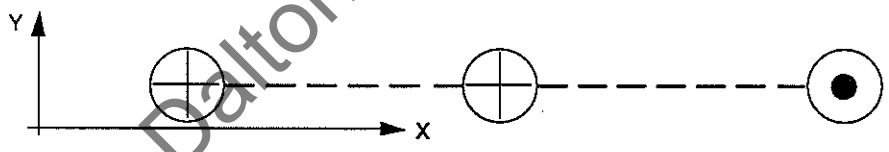
Softkey LAYOUT 1 / LAYOUT 2

Layout 2 also shows:

- The size of the tool radius, displayed as a circle at the end of a contour element. The circle illustrates the "cutter path".
- Numerical display of the end position of a contour element.
- Identification of boring cycles executed with symbol ⊕. The cross in the circle is inserted after execution of the boring cycle.

Boring cycles in simulation

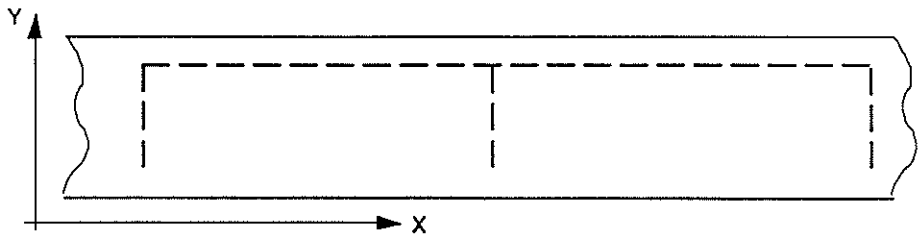
In the case of a bore which is vertical with respect to the selected simulation plane, a circle signals the position of the bore hole (start of cycle); symbol "+" identifies the end of the cycle.



Start of cycle and cycle execution of deep-hole boring, boring or internal thread cutting

End of cycle

If the bore hole is parallel to the selected simulation plane, a line shows the tool movements programmed in the cycle and the position of the bore hole in the workpiece.



Set E axis continuous (MODULO) G 30
Reset E axis continuous (MODULO) G 31

Definition:

The E axis can be rotated continuously. After one revolution (360°), the actual value is set to 0. Programmed traverse movements with angle > 360° are subjected to operation "modulo 360" and executed. The direction of rotation is determined by the sign of the axis values. Interpolating spindle mode of the E axis (1° - 360°); maximum 1 revolution. Continuous rotations are possible.

Preconditions:

- The E axis must be defined as a rotary axis. Machine parameter **1** must be set to **"N"** for this purpose (4th axis = rotary axis).
- Machine parameter **15** must be set to **"Y"**. The axis position display for the E axis is switched over to modulo display. The control automatically sets 0 in the case of 360°. The programmed values are offset in modulo.

Effect:

- Modal action
- Angle positions of the rotary axis or the angle of rotation are programmed in degrees
- G30 does not affect positions of the linear axes (X, Y, Z)
- G30 does not affect zero shifts (G54 - G60, G160)
- Zero shifts are active; they are subjected to the "modulo 360" operation before execution.
- The rotary axis processes positive values. The sign determines the direction of rotation.
- G31 resets G30 (consecutive degree display)

Programming:

G30 must be programmed separately in the NC block. NC addresses and other G functions are not permitted.

Example:

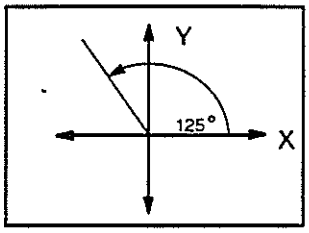
N..	G30		E axis modulo
N..	E0		E axis moves to 0
N..	G60	E=400	Additive NPV is activated; 400 modulo 360 = 40 Display in E: 320°
N..			

.
.
.

Examples:

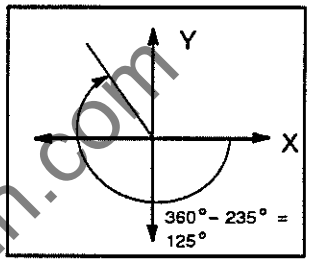
1. Rotation to 125° in mathematically positive direction

```
.
G90
G94   F2000
G30
E125
```

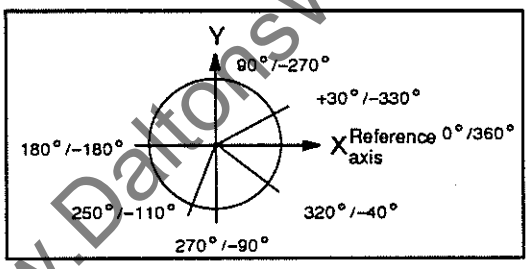


2. Rotation to 125° in mathematically negative direction

```
G90
G94   F2000
G30
E-125
```

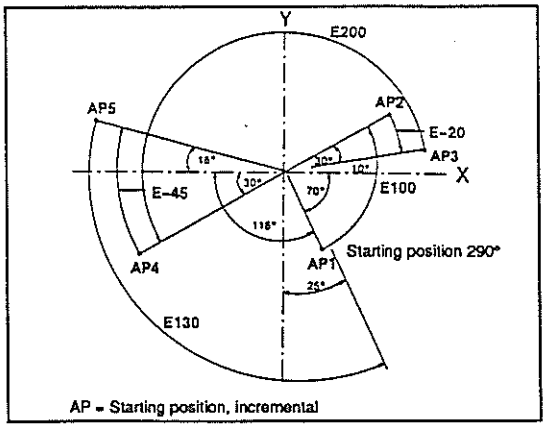


3. Absolute dimension programming (G90) is active.
The absolute angle positions are defined in the circle with radius 1. In this case, the absolute angle start position is always the X axis: Signs (+/-) define the direction of rotation.



4. Incremental dimension programming (G91) is active.
The incremental angle start position is always the preceding angle end position. A positive sign rotates the rotary axis counterclockwise. A negative sign rotates the rotary axis clockwise. The rotary axis is always rotated through the programmed value.

Preconditions:
- Check machine parameters
- G91/G30/F address for feedrate



Coordinate rotation G37 X_Y_Z_A_

Definition:

A programmed contour is rotated by a specific angle about a defined center of rotation.
The angle and center of rotation must be defined in the part program.

Programming/command syntax:

Coordinate rotation is programmed with function G37, with the coordinate specification for the center of rotation (X, Y, Z) and the rotation angle (A). Other addresses are not permitted.

Example: G37 X_Y_A_

The center of rotation is defined with two coordinates. Selection of the axes is dependent upon the active working plane:
- if **G17** is active, axis coordinates **X and Y** are selected
- if **G18** is active, axis coordinates **Z and X** are selected
- if **G19** is active, axis coordinates **Y and Z** are selected.

The axis entries are absolute values, referred to the active program datum.

G37 acts modally.
G37, programmed without axis statements, resets active coordinate rotation.

Input range:

The rotation angle (A) is programmed with a value between -360° and 360°.
Negative angle entries define the coordinate rotation clockwise; a **positive** sign results in coordinate rotation counterclockwise.

Interaction with other functions:

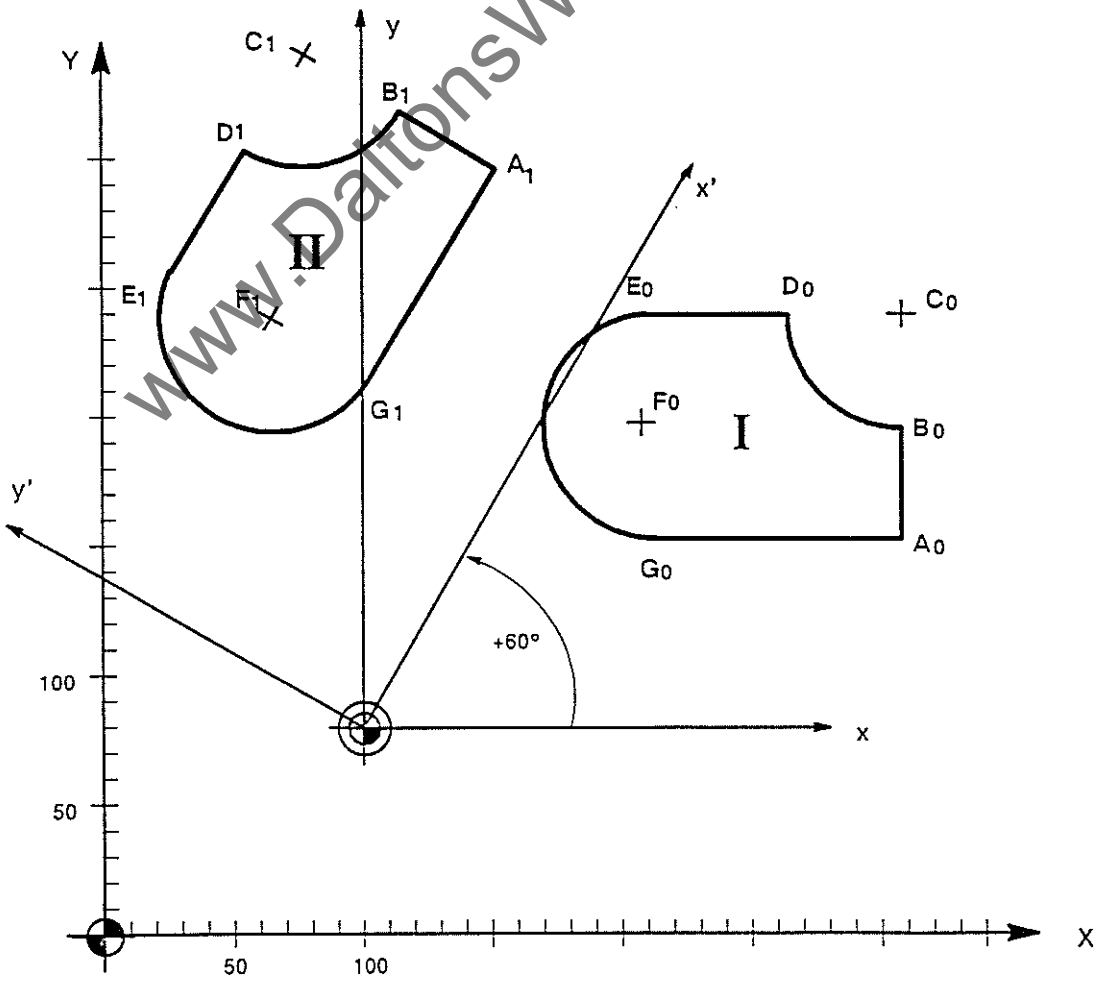
- Change of the group operating mode from AUTOMATIC to MACHINE or MACHINE to AUTOMATIC resets G37.
- G37 is not permitted in manual mode (group operating mode MACHINE). If a cycle in which G37 is programmed is called in manual mode, this block is executed.
- The following functions reset G37: RESET, changing the working plane (G17, G18, G19), switchover to polar coordinates (G20), G37 programmed separately, activate reference point approach cycle (G74), G92 programmed separately (reset to original machine coordinates). G37 is not active after power-up of the control.
- Coordinate rotation may not be programmed if radius compensation (G41, G42) is active. G41/G42 with active G37 are permitted.
- The minimum and maximum values of the work area limits (G25, G26) may not be modified when G37 is active.

Example:

A milling part stored in subroutine P1 is subjected to coordinate rotation G37 in the following part program:

```
N1   G17
N2   G67
N3   G39
N4   G22   P1                          (1st part)
N5   G37   X100   Y80    A+60          (rotation)
N6   G22   P1                          (2nd part)
N7   M30
     $1    (milling part coordinates and machining)
N8   G0    X220   Y140   Z60
N9   G1    Z10    F1000  M3    S150
N10  X230
N11  Y170
N12  G2    X200   Y200   R-30
N13  G1    X170
N14  G5    Y140
N15  G1    Z60
N16  G37
N17  G99
```

Illustration:

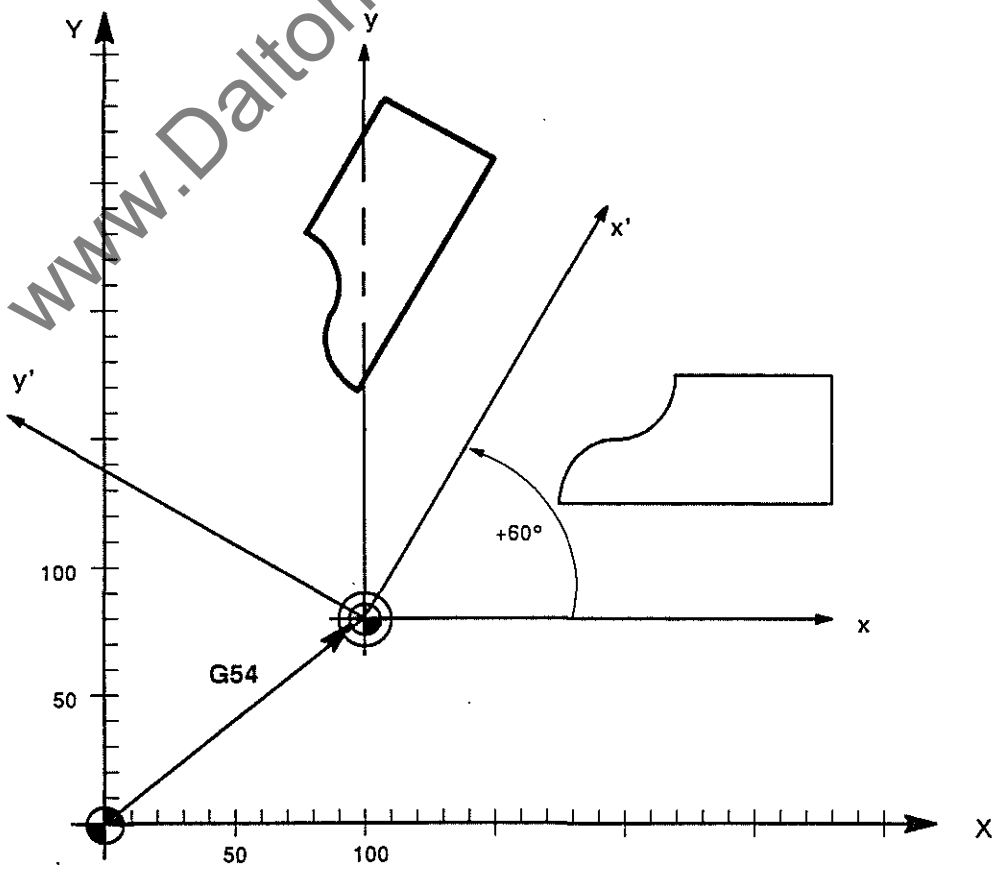


Interaction with other functions:
1. Additive zero shift before coordinate rotation

The coordinates are offset on the basis of additive zero shift and then rotated through the programmed rotation angle with coordinate rotation.

Part program:

```
N1  G17
N2  G39
N3  G60   X100  Y80
N4  G22   P1
N5  G37   X0    Y0    A60
N6  G22   P1
N7  G67
N8  G37
N9  M30
Contour (UP1)
N10 $1
N11 G0    Z50
N12 G1    X30   Y30   Z20   F1000
N13 X100
N14 Y70
N15 X70
N16 G2    X50   Y50   R-20
N17 G5    X30   Y30
N18 G0    Z50
N19 G99
```

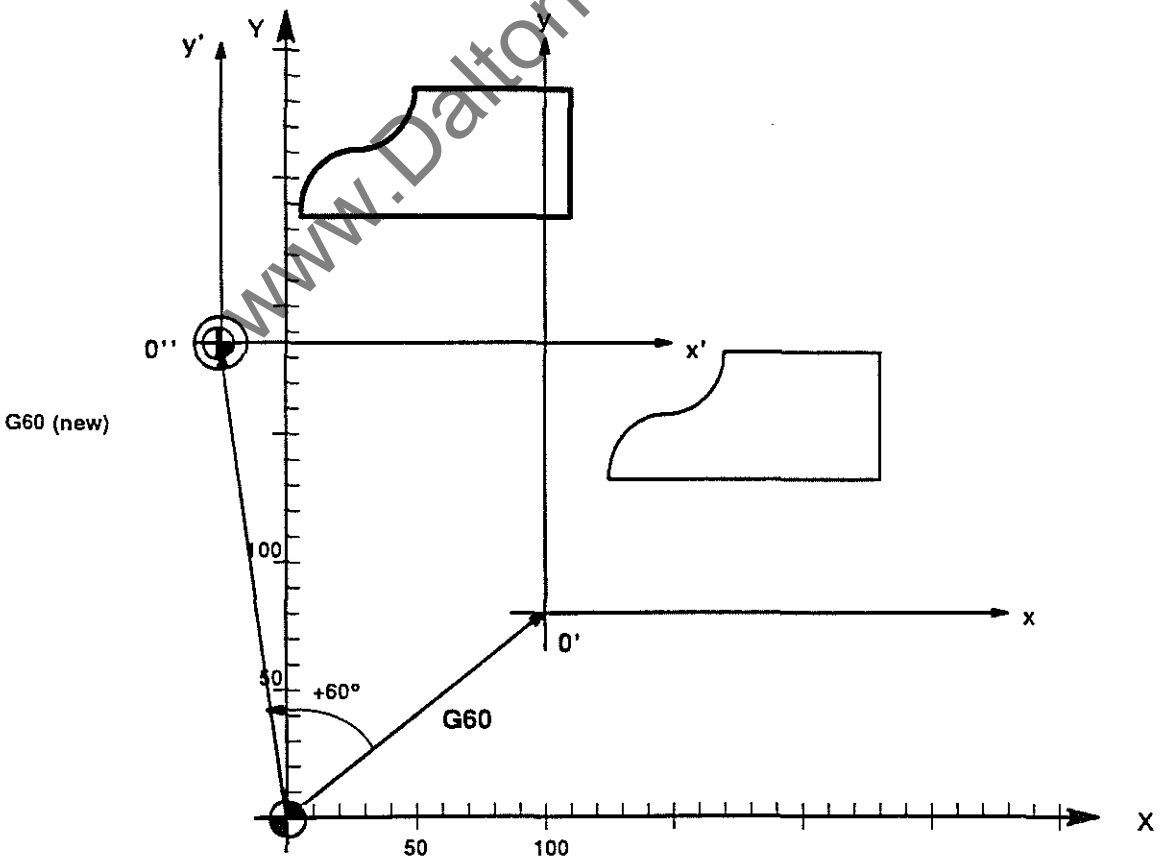


2. Coordinate rotation before additive zero shift without rotated contour

Coordination rotation is reset after the zero shift (G60) has been called.

Part program:

```
N1  G17
N2  G39
N3  G67
N4  G22   P1
N5  G37   X0    Y0    A60
N6  G60   X100  Y80
N7  G37
N8  G22   P1
N9  M30
Contour (UP1)
N10 $1
N11 G0    Z50
N12 G1    X30   Y30   Z20   F1000
N13 X100
N14 Y70
N15 X70
N16 G2    X50   Y50   R-20
N17 G5    X30   Y30
N18 G0    Z50
N19 G99
```

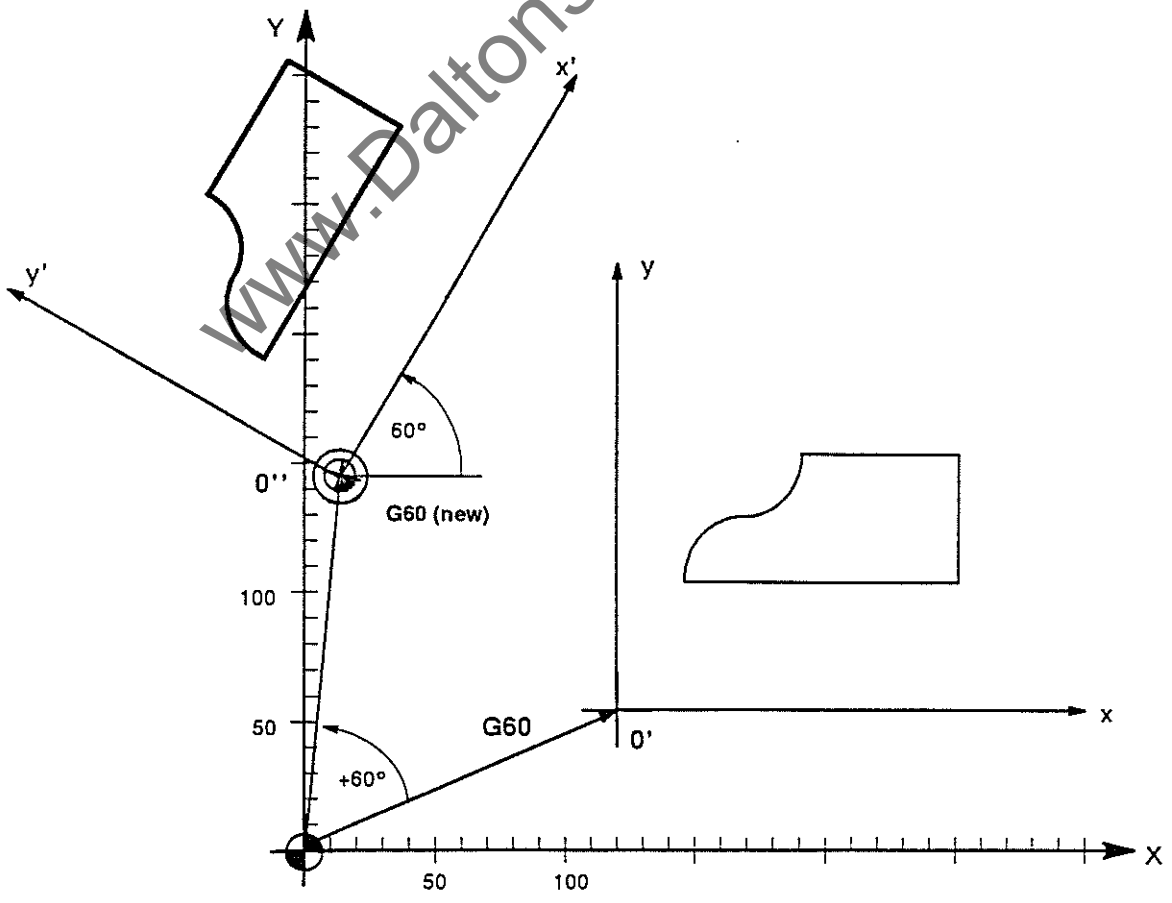


3. Coordinate rotation before additive zero shift and rotated contour

A coordinate rotation before and after a zero shift leads to offset of the milling part.

Part program:

```
N1  G17
N2  G67
N3  G39
N4  G22   P1
N5  G37   X0    Y0    A60
N6  G60   X120  Y60
N7  G22   P1
N8  G37
N9  M30
Contour (UP1)
N10 $1
N11 G0    Z50
N12 G1    X30   Y30   Z20   F1000
N13 X100
N14 Y70
N15 X70
N16 G2    X50   Y50   R-20
N17 G5    X30   Y30
N18 G0    Z50
N19 G99
```

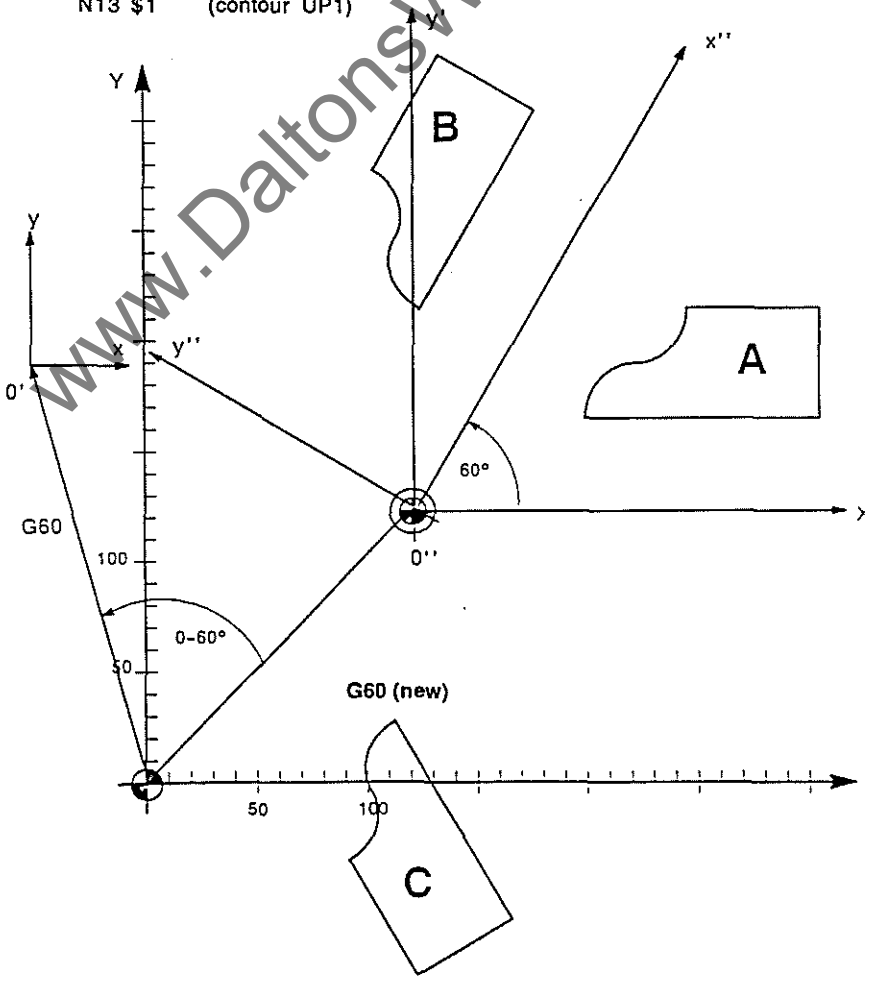


4. Coordinate rotation (G37), mirror image function (G38) and NPV (G60)

All functions in the part program influence the position of the contour.

Example

```
N1  G17
N2  G67
N3  G39
N4  G22   P1   (A)
N5  G37   X0    Y0    A60
N6  G22   P1   (B)
N7  G60   X0    Y0
N8  G38   Y
N9  G22   P1   (C)
N10 G39
N11 G37
N12 M30
N13 $1    (contour UP1)
```



Set TRA function (tangential rotary axis) G46
Reset TRA function (tangential rotary axis) G45

Definition:

The E axis can be operated optionally with tangential infeed. This function simplifies the use of rotationally asymmetric tools such as blades, punching dies and welding torches etc. In this operating mode, the E axis is rotated at each point of the contour to a fixed angle with respect to the resultant traverse direction. The axes which determine the direction of TRA result from the selected working plane (e.g. G17 ▸ X/Y plane). The angle between the contour and TRA can be preset in the part program. TRA rotates the tool at a fixed angle relative to the (feeding) speed vector in the selected working plane. TRA requires a continuously rotating axis with incremental measuring system. TRA uses the E axis as a rotary axis (not a HIRTH function); modulo-programmed.

Programming/effect:

- G46 Exx
  Range of values for Exx: -360 <= xx <= 360

- Setting the machine parameters for the TRA function:
  MP 1 is set to N; this defines the E axis as a rotary axis.
  MP 2 is set to N; a 2nd handwheel is not permitted.
  MP 5 is set to N; the E axis as a rotary axis is not a Hirth axis.

- If G46 E- without value is programmed, the previously programmed E value applies.

- With G46, G30 acts automatically; after each rotation (360°) the actual value of the rotary axis is set to 0 (modulo function).

- G46 acts modally.

- The sign of the E value defines the direction of rotation of the E axis:
  (–) sign - clockwise rotation
  (+) sign - counterclockwise rotation

The direction of rotation of the angle is not always the direction of rotation of the TRA function. The direction of rotation of the TRA function is dependent upon the shortest distance between the current position and the new position.

- The angle between contour and E axis can be programmed with Exx.

Example:

```
N10   G17
N11   G90
N12   G46 X.. Y.. E90   (the E axis is rotated counterclockwise to an angle of 90°, referred to
                         the resultant traverse direction in X and Y).
```

- The TRA function does not act on axis movements triggered by the handwheel if both handwheels are active.

Resetting the TRA function:
G45 resets G46.
G46 is also reset with power-up, power-down of the control, hardware reset (Reset switch on power supply), Control reset (triggered with IF signal or via softkey in group operating mode Information).

TRA and interpolation:

- TRA is permitted with the following interpolations: G0, G1, G2, G3, G5, G6, G75, thread milling with infeed in X, re-entry function.
- If TRA is active, the working plane (G17, G18, G19) cannot be changed.
- G20 (polar coordinate programming) is possible if the polar coordinates refer to the active working plane.

  Example:

N17	G17			(active working plane X, Y)
N12	G20	X..	Y..	(polar coordinate programming permitted)
N13	G20	X..	Z..	not permitted!

- It is not permitted to change the work area limits (G25/G26) for the E axis TRA.
- G31 (continuous E axis) is not permitted when TRA is active.
- Scaling (G36) is not possible for the E axis with TRA function if it is programmed before the TRA function in the part program.
- G37 (coordinate rotation) is possible with TRA.
- If mirror-image (G38) is programmed before the TRA function, axes X, Y and Z, at maximum, are mirror-imaged. The E axis is subject to the influence of the TRA function; G38 is not used here.
- Tool radius compensations (G41, G42) are permitted.
- With active G90 (absolute dimension programming), the axis value programmed with G46 Exx is the angle between the contour and E axis.

  Example:

N11	G90		
N12	G46	E 10	(angle between contour and TRA = 10°)
N13	E40		(angle betewen contour and TRA = 40°)

- With active G91 (incremental dimension programming), the value of the E axis programmed with G46 Exx is the angle by which the E value programmed beforehand is changed.

  Example: (programmed angle = 10° before block N11)

N11	G91		
N12	G46	E40	(after G91, the angle is 10° + 40° = 50°)

- Zero shifts (G54 - 59, G60, G160) do not act for the E axis (TRA). The shift remains stored and acts in the case of the next interpolation with the E axis if TRA is not active. Values can be stored in the zero shift (NPV) table. Zero shifts can be reset with G53, G67, G167.

- G92 Exx is not permitted for the E axis (TRA).
  G92 programmed separately does not reset the TRA function.

- G46 deletes the previously programmed actual value of the E axis.

  Example:

Ni	G45		TRA is reset
Ni+1	G92	E20	Set actual value for the E axis
Ni+2	G46	E90	Actual value for the E axis is reset

- G75 (measuring probe input) is not permitted for the E axis TRA.
  If G75 X.. Y.. Z.. is programmed, the TRA function may not be active.

- Approach reference point is possible for all axes including E axis (TRA). Whilst the G74 block is being executed, the control automatically resets the TRA function. After G74, G46 is active again.

- INCH/METRIC selection can be made for X, Y and Z. This does not affect the E axis. An active TRA function offsets E axis values in degrees since rotary axis parameters have been assigned to the E axis.

- Withdraw from contour and return to contour (REENTRY function) is possible with TRA.

- The maximum angle of rotation for the E axis TRA is 180° clockwise (positive angle entry) or 180° counterclockwise for negative angle entry.

- If EXACT POSITIONING ON (G61) is active, the control first approaches the tangential end position programmed with TRA before the next axis position is executed.
  The next axis position is executed immediately with EXACT POSITIONING OFF (G62) active (power-up state).

- TRA moves along the programmed finished part contour in the selected working plane asynchronously with respect to the other axes. In the case of significant circle contours, tangential infeed may deviate from the finished part contour. A warning indication is then issued.

Application of the parametric functions for G45/G46

Options:
Load NC addresses from variables
G=Vi E=Vj (Vi=45/46; -360 <= Vj <= 360)

CPC test function
TST G46

Copy the E value (TRA) to variable Vi
TRF=G46 Vi=E
TRF=Vj Vi=E (Vj=46)

Vi is the TRA angle between E axis and contour. The position of the E axis is not entered in the part program.

Restrictions:
CPC test function
TST G45

CPC test of the E axis value (angle)

Axis traverse with external command value for rotary axis E (POSE) is not possible if the TRA function is active.

The TRA function in group operating mode MACHINE:
G45/G46 are permitted in manual mode.

- Jog key and handwheel operation are possible with axes X, Y, Z and for E (TRA).
  If an angle is programmed with the E axis (Exx), this angle becomes valid with the next movement in the selected working plane.

- The reference point can be approached manually via softkey or via cycle G74.

- With 2 active handwheels, TRA does not respond to axis movements via the handwheel.

Machine parameters:

- The software limit switches must be set to infinity (MP 98 = MP 99 = MP 100 = MP 101 = 0) since rotary axis modulo parameters have been assigned to the E axis.
  If 0 is not entered:
  - the positive limit switches must be set to > 360° (MP98/MP100)
  - the negative limit switches must be set to < 0° (MP99/MP101).

- No SLOPE programming (MP135=0).

- The E axis TRA can be traversed in rapid traverse (G0).

Interface:

- The interface signal "E RUN" (PIC address E 0.3) is active when TRA is active.

- G0 (PIC address E 5.0) is not influenced by TRA.

- The interface signal "DRIVE ON E" (PIC address A 2.3) and "FEED ENABLE E" (PIC address A 3.0) must be set to "high".

Display:

- The curent (valid) value for the E axis is always displayed in group operating modes MACHINE and AUTOMATIC.

- G45/46 are displayed with the programmed angle in group operating mode INFORMATION.

Additive zero shift (NPV)

G60
G160

1. Function

A contour which has already been shifted with G54-59 can be shifted additionally with G60 and/ or with G160 with additive zero shift.

2. Programming

The additive NPV must be programmed with the G functions (preparatory functions) G60, G160. The function and mode of operation of both G codes are identical.

Activating additive NPV:

G function: G60 or G160
Parameters: Axes X, Y, Z, E
Options:
G60/160
G60/160 X...
G60/160 X... Y...
G60/160 X... Y... Z...
G60/160 X... Y... Z... E...
G60/160 X... Z... E...

G60 or G160 can be programmed separately or with the axes X, Y, Z, E. Axis coordinates programmed with G60 or G160 are stored in the NPV table corresponding to the active unit (INCH/METRIC).

Deactivating additive NPV:

G67/G167 reset additive NPV without changing the NPV table. Values stored with G60/G160 in the NPV table are not deleted.
G67 resets only G60.
G167 resets only G160.
G67/167 can be programmed with the addresses X, Y, Z, E, M, S, H in one NC block.
Options:
G67/167
G67/167 X100 E20
G67/167 M3 S100
G67/167 M3 S100 X100 E20

Mode of operation, programming principles:

- G60/160 act modally
- G60/160 programmed separately activates additive NPV with the axis values from the NPV table
- Besides axis information, no other addresses are permitted with G60/160
- G60/160, G67/167 do not lead to axis movements
- Additive NPV does not become operative until the subsequent traverse movement
- M2, M30, change of group operating mode from AUTOMATIC to MANUAL mode (MDI) in group operating mode MACHINE or vice versa resets additive NPV (G60/160).
- G60/160 or G67/167 cannot be activated or deactivated when tool radius compensation (G41/42) or circular interpolation (G2/3/5) are active.
- G74, G92, mains switch ON/OFF or software/hardware reset reset additive NPV.

3. Interaction with other functions

Zero shift (G54 - G59)

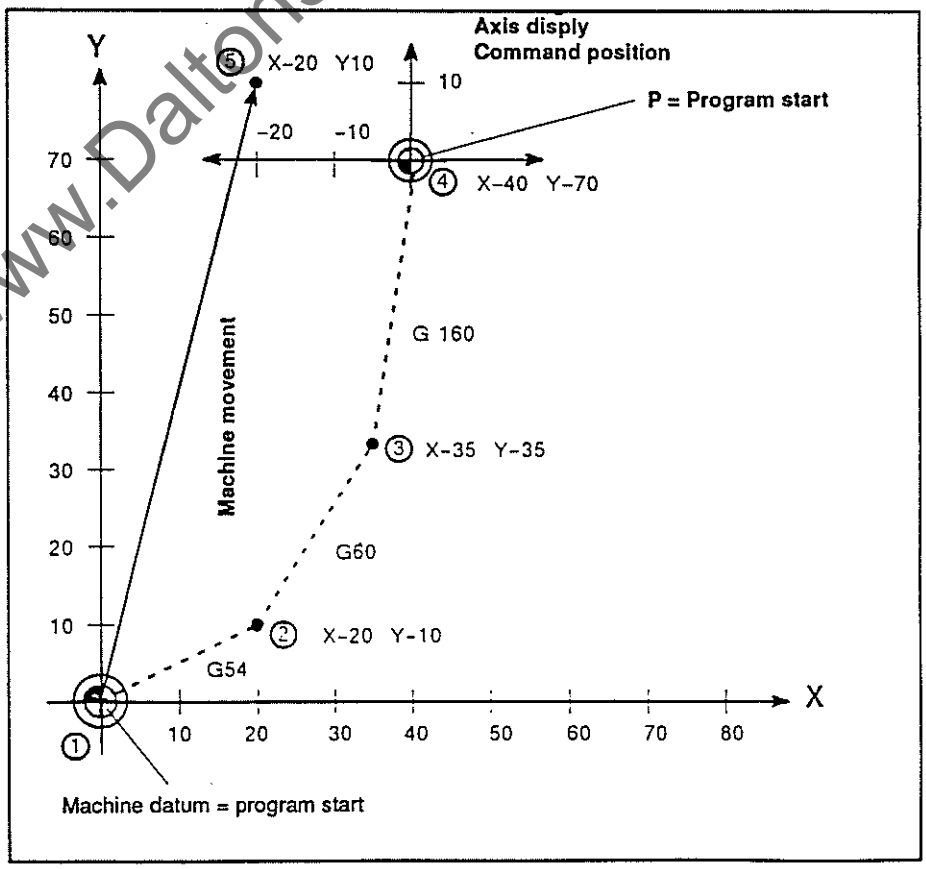
If NPV G54 - G59 is active, the values of additive NPV are added to the already programmed values in the NPV table.

Example:

	Program extract				NPV table		
	N ..	G53					
	N..	G67					
	N..	G167					
	N	G90					
①	N..	G0	X0	Y0			
②	N..	G54			G54	X20	Y10
③	N..	G60	X15	Y25	G60	X15	Y25
④	N..	G160			G160	X5	Y35
⑤	N..	G0	X-20	Y10			

If necessary, the display can be switched over from COMMAND to MACHINE position.

Axis movements and axis display in COMMAND position:



Coordinate rotation (G37)

G37 acts on additive NPV (G60, G160) but not on the NPV (G54-59).

- Coordinate rotation G37 does not act upon G60/G160 if G60/160 are programmed in the part program **before** G37 X... Y... A... .
- Coordinate rotation G37 acts on G60/160 if G60/160 are programmed in the part program **after** G37 X... Y... A... .
- The axis values of G60/160 are written into the additive NPV table.

Scale factor change (G36)

Values of the additive NPV can be subjected to scale factor change. The additive NPV is then executed with scaled values. If the factor is reset to 1 whilst additive NPV is active, this remains active with the scaled values. Stored axis values in the additive NPV table are not scaled.

Mirror image (G38)

G60/G160 are executed mirror-imaged if G38 is already active.
The original axis values are stored in the additive NPV table; G38 does not influence the NPV table.

Activate measuring probe (G75)

Programmed axis values with G75 are shifted by additive NPV.

Work area limits (G25, G26)

No effects on the work area limits if G25/26 is active **before** additive NPV G60/160.
Effects on the work area limits if these are programmed in the part program **after** G60/160.

4. Additive NPV and CPC functions

Basically, functions G60, G160, G67, G167 are permitted in the CPC program;

Options:

☛ **Load G functions - G=V1**

a) V1=60 V2=100 V3=40
G=V1 X=V2 Z=V3

b) V1=160
G=V1

c) V3=67
G=V3

d) V3=67 V4=80 V5=40
G=V3 X=V4 Y=V5 (V4/V5 lead to axis movements)

e) V3=167
G=V3

☛ **CPC function TRF (load additive NPV table with values)**
The TRF function can be used only for additive NPV G60.

a) TRF=G60 VA=Z VB=E

b) Not permitted: * TRF=G67
* VK=67
TRF VK

c) Not permitted: * TRF=G160
* TRF=G167

☛ **CPC function TST (checking the modality of G functions)**
TST G60 The condition register is set to 0 if G60 is active;
the condition register is set to 1 if G67 is active.
TST G160 The modality check of this additive NPV is not permitted.

☛ **POS function (axis traverse with external command value)**
The POS function is permitted if additive NPV (G60, G160) is active. The axis position of the axis traversed with external command value is changed by the programmed axis value of additive NPV.

Extending the tool table to 99 tools (option)

Definition:

The tool table can be extended optionally to up to 99 tools. The table lists at least 12 tools (lower limit).

Input range:

The maximum number of tools in the tool table is programmed via machine parameter 25 (maximum number of tools in the tool memory). A number of tools between 12 and 48 can be programmed without the option.
A number of tools between 12 and 99 can be programmed with the option.

Display:

The selected number of tools is displayed in the tool table even if the tools are not used.

Example:

Setting machine parameter 25 to the numerical value 40:
Display of the tools from T1 to T40 in the tool table.

Machine parameter 25:
MP 25 can be set only to greater than 48 to 99 tools if option "99 tools" is activated.

Variable extension

The variable table is extended to a total of 619 variables. The additional variables are addressed with combinations of letters and digits.

The following variable ranges are available:

- V1 to V99 99 variables
- V0A to V0Z 260 variables
  .
  .
  V9A to V97
- VAA to VAZ 260 variables
  .
  .
  VJA to VJZ

The variables are global variables.

Note on distinguishing between system variables and free user variables:

The following variables are used internally by the system for executing the machining and contour cycles:

- V1 to V17
  V90 to V99
  V0A to V0Z

These system variables are used internally if machining and contour cycles are active. They are thus not unrestricted free user variables.

The user should not use these variables if machining and contour cycles are active.

External program selection (via BCD bus)

Definition/function:

A part program can be selected for execution via interface signal. The address and data signal is transmitted via the BCD input bus from the PLC to the NC. The signal acts both in ON-LINE-PANEL mode and in ON-LINE-PC mode.

The BCD signal:

	BCD address							BCD data							
Strobe	0	0	0	0	0	0	1	pp	pp	pp	pp	pp	pp	pp	pp
	40	20	10	8	4	2	1	80	40	20	10	8	4	2	1
	Address (external program selection)							Program number							

Conditions in the individual group operating modes:

1. Group operating mode AUTOMATIC

Program selection is possible via panel (on-line-panel mode) or externally via interface (on-line-PC mode).
External program selection is not possible during part program execution

2. Group operating mode INFORMATION

No external program selection is possible in softkey branch SERVICE or MTB SERVICE. The control sends an error message.
In the other softkey branches, an external program selection causes a change in the display (INFORMATION ▸ AUTOMATIC). The control waits for Cycle start.

3. Group operating mode MACHINE

If MDI, approach reference point or teach-in are selected, no external program selection is possible. The control generates an error message. An external program selection can be carried out only in the basic display image MACHINE.

4. Group operating mode MEMORY

External program selection is always possible. The control aborts procedures which have been started, closes any open files and automatically changes to group operating mode AUTOMATIC. There, the control awaits signal "Cycle start".

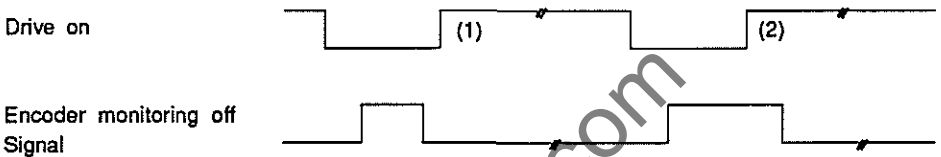
Deactivatable encoder monitoring

Function:

The interface signal "Encoder monitoring off" stops monitoring of the encoder if signal "Drive on" is not active for the relevant axis.

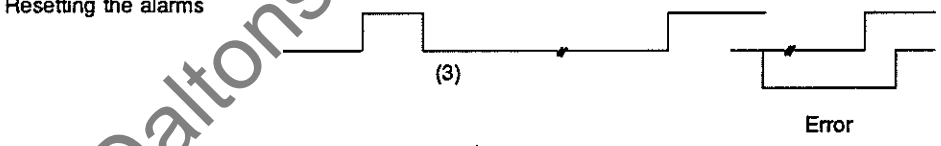
Activating the signal:

If address (A) 4.7 is programmed in the PIC, the signal is set after the PIC cycle.
Signal "Encoder monitoring off" acts only for the axes whose "Drive on" (A2.0 - A2.4) interface signal is switched to "Low" (1).



If the "Drive on" interface signal changes to "High" when "Encoder monitoring off" is also active, the monitoring process is continued (2).

If the "Encoder monitoring off" signal is set to "Low", the pending alarm messages of the monitoring logic are reset (3).



Preconditions

"Encoder monitoring off" is permitted for each axis if the IF signal A 4.7 is set and if the "Drive on" signal of the relevant axis is set to "Low".

Alarm messages

An alarm message is issued if

- an encoder is not connected without signals "Encoder monitoring off" and "Drive on" being inactive.
- "Drive on" is set before "Encoder monitoring off" has been reset.

Approach reference point

It is possible to approach the reference point after the encoder is switched off. The following combinations result in a warning or error message:

- If the software limits are not active whilst axes are traversed in group operating mode AUTOMATIC, jog mode or MDI in group operating mode MACHINE, the following warning is issued:
  X, Y, Z, E - Measuring system interrupt

- With active software limits and if the axes are used in jog mode, the following warning is also issued:
  X, Y, Z, E - Measuring system interrupt

- With active software limits and programmed traverse movements (group operating mode AUTOMATIC) or in MDI mode (group operating mode MACHINE), reference point approach is required. Otherwise, the following error message is issued:

ERROR	235 FUNCTION DISABLED, APPROACH REFERENCE POINT

The message refers to the corresponding axis.

Display

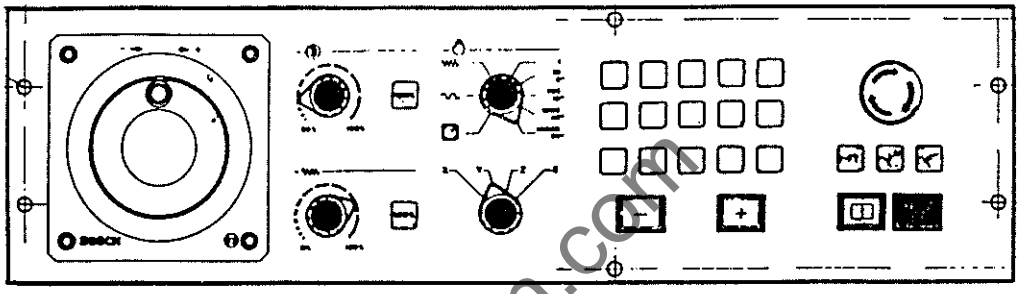
The status of the IF signal can be read off in group operating mode Information, Machine status, I/O status.
The last signal of the INPUT column shows **Encoder monitoring off L/H.**

Inputs on the manual panel

Definition:

There are three terminals which have not been used to date on the rear of the manual panel. The signals at these terminals are now evaluated in the PIC 200.

Illustration (manual panel and arrangement of the PIC inputs E1 - E3):



X12 PIN 1 - E2.5
PIN 2 - E2.6
PIN 3 - E2.7

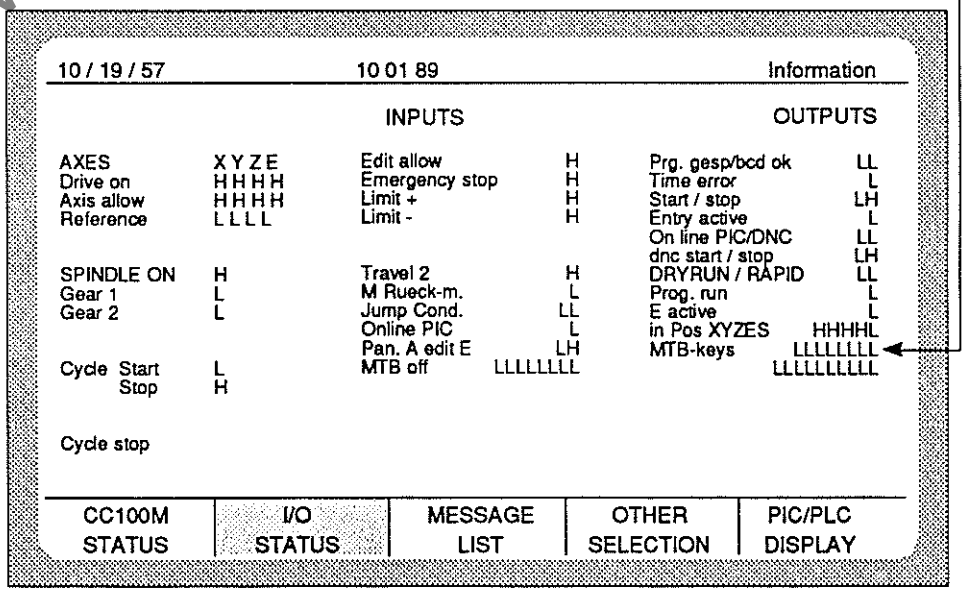
NC outputs ⇾ PIC inputs:
The statuses of the signals E1 - E3 are stored in the PIC input bits 2.5 to 2.7.

Display:

The statuses of the PIC input signals 2.5 - 2.7 are displayed in the I/O status overview (group operating mode INFORMATION).
The PIC input signals (E1 - E3) are listed with the PIC keys (F1 - F15) in the following order under criterion "MTB keys" in the list of the outputs:

	F11	F12	F13	F14	F15	E1	E2	E3
F1	F2	F3	F4	F5	F6	F7	F8	F9	F10

Illustration (CC 100 M - I/O status):



System disk PIC 200 for PG4

The system disk contains the symbol tables which, in turn, contain all defined interface signals between CC 100M and PIC 200, in German and English.
Standard abbreviated designations are defined within the system (e.g. CC 100M) for these input and output signals and markers.
This saves the user time when creating and documenting a PIC program. The symbol tables can be modified or supplemented at any time.

The symbol tables have the following significance:

CC100MDT	Symbol table PIC for CC 100= M in German
CC100MEN	Symbol table PIC for CC 100= M in English
CC100TDT	Symbol table PIC for CC 100= T in German
CC100TEN	Symbol table PIC for CC 100= T in English

The system disk for the CC 100= M has the
disk number CCPIC1MA and the
drawing number 062614

The symbol table CC100MDT is enclosed as an annex in this software description.

Symbol tables (PIC 200 ⏵ PG4)

Address	Symbol	Comment	Type
E 0.0	CCRESETO	Control Reset output	
E 0.1	CCIPOSAL	Axis INPOS	
E 0.2	CCSPIPOS	Spindle INPOS	
E 0.3	CCELAUFT	E axis running	
E 0.4	CCPROGAK	Programing running	
E 0.5	CCDRYRUN	Test without movement (dry run)	
E 0.6	CCOPCEIN	Online PIC/PC	
E 0.7	CCDNCAKT	DNC_Mode	
E 1.0	CCSTART0	Start key (NO contact)	
E 1.1	CCSTOP0Q	Stop key (NC contact)	
E 1.2	CCBCDQIT	BCD_Acknowledgment	
E 1.3	CCPROMEM	Program stored	
E 1.4	CCEINAKT	Re-entry cycle active	
E 1.5	CCDNCSTR	DNC start	
E 1.6	CCDNCRST	DNC RESET	
E 1.7	CCDNCSTP	DNC STOP transverse	
E 2.0	CCWMHTS1	MTB_key 1	
E 2.1	CCWMHTS2	MTB_key 2	
E 2.2	CCWMHTS3	MTB_key 3	
E 2.3	CCWMHTS4	MTB_key 4	
E 2.4	CCWMHTS5	MTB_key 5	
E 2.5	CCWMHTS6	MTB_key 6	
E 2.6	CCWMHTS7	MTB_key 7	
E 2.7	CCWMHTS8	MTB_key 8	
E 3.0	CCBCAA01	BCD–output bus address 1	
E 3.1	CCBCAA02	BCD–output bus address 2	
E 3.2	CCBCAA04	BCD–output bus address 4	
E 3.3	CCBCAA08	BCD–output bus address 8	
E 3.4	CCBCAA10	BCD–output bus address 10	
E 3.5	CCBCAA20	BCD–output bus address 20	
E 3.6	CCBCAA40	BCD–output bus address 40	
E 3.7	CCBCASTB	Data valid (Strobe)	
E 4.0	CCBCAD01	BCD–output bus data 1	
E 4.1	CCBCAD02	BCD–output bus data 2	
E 4.2	CCBCAD04	BCD–output bus data 4	
E 4.3	CCBCAD08	BCD–output bus data 8	
E 4.4	CCBCAD10	BCD–output bus data 10	
E 4.5	CCBCAD20	BCD–output bus data 20	
E 4.6	CCBCAD40	BCD–output bus data 40	
E 4.7	CCBCAD80	BCD–output bus data 80	
E 5.0	CCEILJOG	G0 or jog in rapid traverse	
E 5.1	CCINPOSX	X axis in pos	
E 5.2	CCINPOSY	Y axis in pos	
E 5.3	CCINPOSZ	Z axis in pos	
E 5.4	CCINPOSE	E axis in pos	

Address	Symbol	Comment	Type
A 0.0	CCENDSQP	Limit switch + transverse	
A 0.1	CCENDSQN	Limit switch - transverse	
A 0.2	CCREFPXQ	Reference point X transverse	
A 0.3	CCREFPYQ	Reference point Y transverse	
A 0.4	CCREFPZQ	Reference point Z transverse	

Address	Symbol	Comment	Type
A 0.5	CCREFPEQ	Reference point E transverse	
A 0.6	CCGETHA1	Gear identifier 1 main spindle	
A 0.7	CCGETHA2	Gear identifier 2 main spindle	
A 1.0	CCSWEND2	2nd software limit switch range	
A 1.1	CCBEDSPR	Conditional jump (G23)	
A 1.2	CCBEDUPR	Conditional subroutine call (G21)	
A 1.3	CCMQUITT	M acknowledgment	
A 1.4	CCZYSTPQ	Cycle stop transverse	
A 1.5	CCZYSTRT	Cycle start	
A 1.6	CCEINLFG	Read-in enable	
A 1.7	CCRESETI	Control reset	
A 2.0	CCANEINX	Drive on X	
A 2.1	CCANEINY	Drive on Y	
A 2.2	CCANEINZ	Drive on Z	
A 2.3	CCANEINE	Drive on E	
A 2.4	CCANEINS	Drive on spindle	
A 2.5	CCVSFRGX	Feed enable X	
A 2.6	CCVSFRGY	Feed enable Y	
A 2.7	CCVSFRGZ	Feed enable Z	
A 3.0	CCVSFRGE	Feed enable E	
A 3.1	CCNHQUER	Emergency stop transverse	
A 3.2	CCPANAUS	Panel off	
A 3.3	CCEDITFR	Edit enable	
A 3.4	CCONPCST	Activate online PC	
A 3.5	CCTCHISP	Teach-in store (every 100 ms)	
A 4.6	CCDNCEIN	Activate DNC remote	
A 4.7	CCBCDFEL	Error message via BCD–output bus	
A 5.0	CCSTPLMP	Cycle stop lamp or output X13.1 on manual panel	
A 5.1	CCSTRLMP	Cycle stop lamp or output X13.2 on manual panel	
A 5.2	CCWMHEB2	Output X13.3 on manual panel	
A 5.3	CCWMHEB3	Output X13.4 on manual panel	
A 5.4	CCWMHEB4	Output X13.5 on manual panel	
A 5.5	CCWMHEB5	Output X13.6 on manual panel	
A 5.6	CCWMHEB6	Output X13.7 on manual panel	
A 5.7	CCWMHEB7	Output X13.8 on manual panel	
A 6.0	CCADRE01	BCD–input bus address 1	
A 6.1	CCADRE02	BCD–input bus address 2	
A 6.2	CCADRE04	BCD–input bus address 4	
A 6.3	CCADRE08	BCD–input bus address 8	
A 6.4	CCADRE10	BCD–input bus address 10	
A 6.5	CCADRE20	BCD–input bus address 20	
A 6.6	CCADRE40	BCD–input bus address 40	
A 6.7	CCSTROBE	Strobe (data valid)	
A 7.0	CCDATE01	BCD–input bus data 1	
A 7.1	CCDATE02	BCD–input bus data 2	
A 7.2	CCDATE04	BCD–input bus data 4	
A 7.3	CCDATE08	BCD–input bus data 8	
A 7.4	CCDATE10	BCD–input bus data 10	
A 7.5	CCDATE20	BCD–input bus data 20	
A 7.6	CCDATE40	BCD–input bus data 40	
A 7.7	CCDATE80	BCD–input bus data 80	

Address	Symbol	Comment	Type	
M 55.0	INC	Z0:=Z0+1 IF Z0=ZE+1 SET Z0=0		
M 55.1	DEC	Z0:=Z0-1 IF Z0=-1 SET Z0=ZE		
M 55.2	CRL	Z0:=0 DELETE ACCUMULATOR		
M 55.3	LD	Z0:=ZL LOAD ACCUMULATOR WITH ZL		
M 55.4	ADD	Z0:=Z0+ZL		
M 55.5	SUB	Z0:=Z0-ZL		
M 55.6	LDE	Z0:=ZE LOAD ACCUMULATOR WITH ZE		
M 55.7	EINIT	ZE:=0 LOAD ZE REGISTER WITH Z0		
M 56.3	EQE	Z0=ZE IF YES, THEN MARKER = H		
M 56.4	EQ0	Z0=0 IF YES, THEN MARKER = H		
M 56.5	LTL	Z0<ZL IF YES, THEN MARKER = H		
M 56.6	EQL	Z0=ZL IF YES, THEN MARKER = H		
M 56.7	GTL	Z0>ZL IF YES, THEN MARKER = H		
M 57.0	ZL1	LOAD REGISTER 1		
M 57.1	ZL2	LOAD REGISTER 2		
M 57.2	ZL3	LOAD REGISTER 3		
M 57.3	ZL4	LOAD REGISTER 4		
M 57.4	ZL5	LOAD REGISTER 5		
M 57.5	ZL6	LOAD REGISTER 6		
M 57.6	ZL7	LOAD REGISTER 7		
M 57.7	ZL8	LOAD REGISTER 8		
M 58.0	Z01	ACCUMULATOR 1		
M 58.1	Z02	ACCUMULATOR 2		
M 58.2	Z03	ACCUMULATOR 3		
M 58.3	Z04	ACCUMULATOR 4		
M 58.4	Z05	ACCUMULATOR 5		
M 58.5	Z06	ACCUMULATOR 6		
M 58.6	Z07	ACCUMULATOR 7		
M 58.7	Z08	ACCUMULATOR 8		
M 59.0	CLK1	COUNTER 1 DECREMENT		
M 59.1	CLK2	COUNTER 2 DECREMENT		
M 59.2	CLK3	COUNTER 3 DECREMENT		
M 59.3	CLK4	COUNTER 4 DECREMENT		
M 59.4	CLK5	COUNTER 5 DECREMENT		
M 59.5	CLK6	COUNTER 6 DECREMENT		
M 59.6	CLK7	COUNTER 7 DECREMENT		
M 59.7	CLK8	COUNTER 8 DECREMENT		
M 60.0	ZE1	COUNTER 1 :=0		
M 60.1	ZE2	COUNTER 2 :=0		
M 60.2	ZE3	COUNTER 3 :=0		
M 60.3	ZE4	COUNTER 4 :=0		
M 60.4	ZE5	COUNTER 5 :=0		
M 60.5	ZE6	COUNTER 6 :=0		
M 60.6	ZE7	COUNTER 7 :=0		
M 60.7	ZE8	COUNTER 8 :=0		
M 61.0	ST1	START TIME 1		
M 61.1	ST2	START TIME 2		
M 61.2	ST3	START TIME 3		
M 61.3	ST4	START TIME 4		
M 61.4	ST5	START TIME 5		
M 61.5	ST6	START TIME 6		
M 61.6	ST7	START TIME 7		
M 61.7	ST8	START TIME 8		
M 62.0	TE1	TIME END 1		

Address	Symbol	Comment	Type
M 62.1	TE2	TIME END 2	
M 62.2	TE3	TIME END 3	
M 62.3	TE4	TIME END 4	
M 62.4	TE5	TIME END 5	
M 62.5	TE6	TIME END 6	
M 62.6	TE7	TIME END 7	
M 62.7	TE8	TIME END 8	
M 63.0	TA1	TIME ACTIVE 1	
M 63.1	TA2	TIME ACTIVE 2	
M 63.2	TA3	TIME ACTIVE 3	
M 63.3	TA4	TIME ACTIVE 4	
M 63.4	TA5	TIME ACTIVE 5	
M 63.5	TA6	TIME ACTIVE 6	
M 63.6	TA7	TIME ACTIVE 7	
M 63.7	TA8	TIME ACTIVE 8	
M 64.0	AAAAAAAA	Markers M64.0 to M127.7 are assigned by Bosch!	

New milling and Noring cycles

The package comprises the cycles with input displays and separate documentation (universal milling cycles K No. 016/A2 - 05/89).

Required storage capacity approx. 27.5 KB:
The cycles are available on disk (PG4, CC disk) or cassette (Minicass).

The package covers the following cycles:

*** Workshop-oriented milling cycles for mechanical engineering and production**

– Bore/circular pocket
– Inside frame/rectangular pocket
– Outside frame
– Slot
– Journal

*** Drilling patterns**

– Row of holes
– Grid of holes
– Hole circle

The display texts are available optionally in German or English.

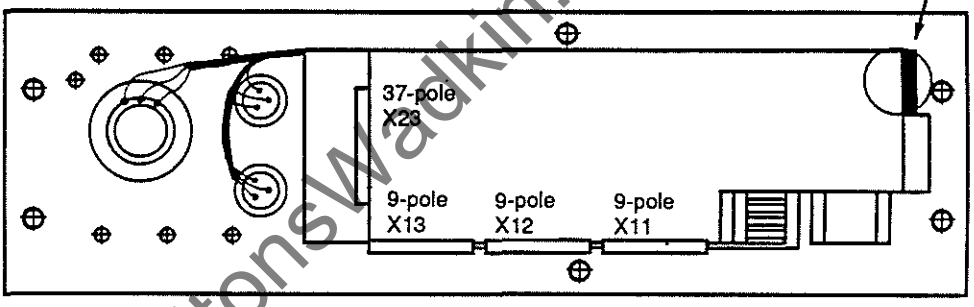
2nd Handwheel (option)

Function:

A 2nd electronic handwheel can be activated in the CC 100M. The second handwheel acts only on the X axis, whilst the first handwheel can be switched to the Y, Z or E axis via the axis selector switch. This means that two axes can be moved mutually independently with two handwheels. The function 2nd handwheel is an option and must be activated in the control.

Commissioning, interaction:
The new manual panel (EBF) is equipped with an additional Weidmüller terminal (see marked position in the figure below). This terminal is provided for the 2nd handwheel.
The machine parameter 2 (2nd handwheel only for X axis) must be set to Y. The function of the 2nd handwheel is evaluated when the machine parameter is changed accordingly. If the 2nd handwheel is active with the axis selector switch set to X, an axis movement in X via the 1st handwheel is not possible. The feedrate potentiometer acts simultaneously for both handwheels.

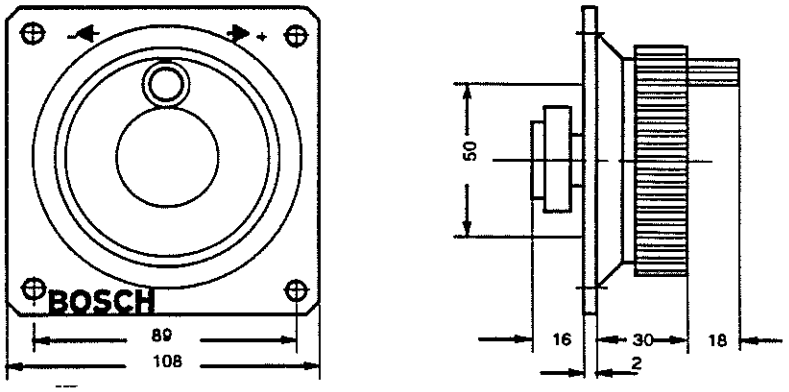
Illustration (connection on manual panel (EBF) for 2nd handwheel):



Dimensions:

The dimensions of the 2nd handwheel correspond to those of the 1st handwheel (see drawing below and dimension specifications). The 2nd handwheel is designed for installation in a machine operating panel or in a separate housing (mobile use).

Illustration:

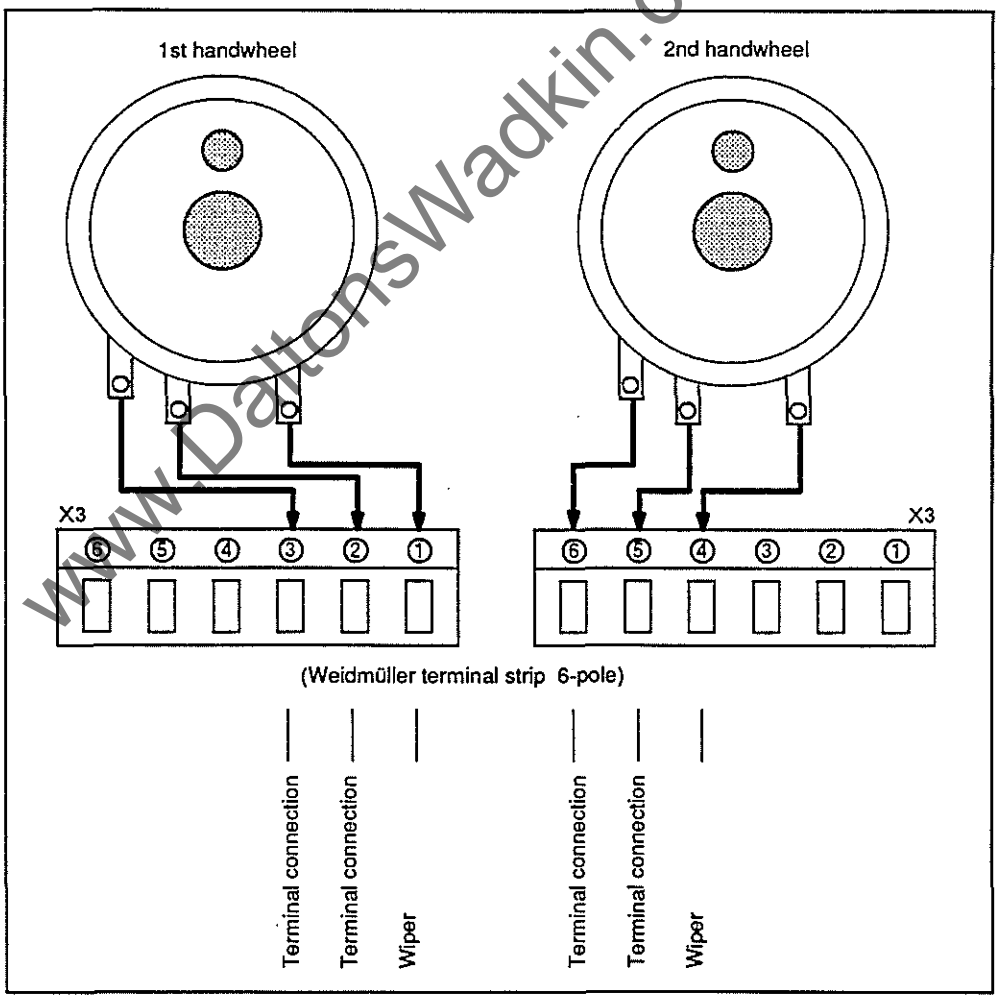


Cable lengths:

The maximum cable length between the manual panel and the second handwheel is 5 m if the BOSCH cable connections approved for this are used. Screened cables (3-core) with a mininum cross-sectional area of 0.14 mm² are approved.
Overall, the maximum cable length between CP/MEM and second handwheel adds up to 60 m.

Subdivision:

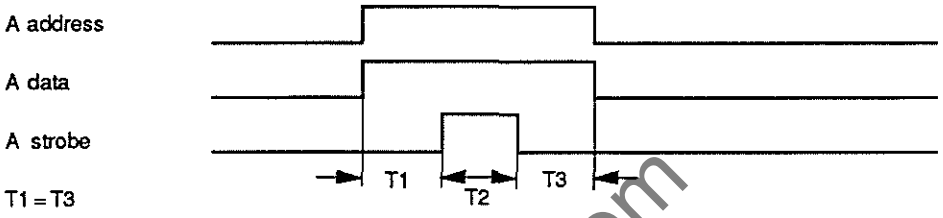
CP/MEM ⇔) operating panel	max. 25 m
Operating panel⇔) manual panel (EBF)	max. 30 m
Manual panel (EBF) ⇔) 2nd handwheel	max. 5 m
Total:	**max. 60 m**



Machine parameters for timing of auxiliary functions

Definition:

Auxiliary functions and other data can be output via the BCD bus. The timing of this output can now be selected. If machine parameter 21 is not entered, the timing is fixed and compatible with previous software versions (see Interface conditions, Pages 3-27/3-28).
The diagram below shows the times whose values can be entered:



Entering the times:

Times T1 and T2 can be selected via machine parameter 21 between 10 ms and 127 ms (1.27 sec.). The minimum permitted time unit is 10 ms. The input range of the machine parameter is two times 0 - 127 time units. The two times may differ. The first time unit for T1 (T3) can be specified with 2 digits. The second time unit for (T2) must be programmed with three digits (e.g. T2 = 50 ms ⏵ time unit 5 ⏵ entry for T2 = 005).

Machine parameters:

P21 : xxx yyy

T2	yyy	ε	[0.127]
T1	xxx	ε	[0.127]

Example entry:

Time T1 (T3) is to be programmed with 330 ms, and time T2 is to be programmed with 1.2 sec.
Entry for T1 (T3): 330 ms/10 ms = 33
Entry for T2 : 1200 ms/10 ms= 120

Entry syntax in MP 21: 033120 ENTER

For reasons related to compatibility with software versions < S40, a machine parameter entry of 0 must be accepted.
The following times are set fixed in this case:

P21: xxx yyy

0 or	T1 = 20 ms with PIC 200 T1 = 10 ms with PC coupling
0 or	T2 = 50 ms with PIC 200 T2 = 20 ms with PC coupling

Advantage:

Output of H functions to the PLC can be speeded up substantially.

Machine parameters for PIC user bits

Application:

During each run-up phase, the control sets marker bits dependent upon the entries in the machine parameters P141- P144.
This permits the MTB to determine what machine functions gäted in the PIC are to be available to the user. The MTB thus activates functions in the machine, which are to be controlled by the PIC, via machine parameters.

Machine parameters/entry syntax:

PIC user bits can be set with the following machine parameters:

P 141	PIC user bits M50.0 - M50.7
P 142	PIC user bits M51.0 - M51.7
P 143	PIC user bits M52.0 - M52.7
P 144	PIC user bits M53.0 - M53.7

Numbers between 0 and 255 are accepted. Number 0 sets all PIC user bits of the byte called with the machine parameter to 0. Number 255 is formed from the sum of 2^0 to 2^7 and sets all PIC user bits to 1.

Assignment of the PIC user bits:

The number programmed in the machine parameters is assigned to the relevant PIC marker byte, binary-coded.

Example:

The number 178 is programmed in MP 141. The assignment of the PIC marker byte 50 is shown below.

50.7	50.6	50.5	50.4	50.3	50.2	50.1	50.0	
128	64	32	16	8	4	2	0	
1	0	1	1	0	0	1	0	➧ internal assignment

Explanation:

If the MTB wishes to activate functions of the machine which are interlocked via marker bits 50.1, 50.4, 50.5 and 50.7, the MTB must enter number 178 in MP 141, thus releasing these functions.

More simple output of machine messages

Application:

Defined logic operations in the PIC 200 can set the non-retentive markers 46.0 - 49.7. Dependent upon the marker bits, the NC selects the machine message from the first 32 messages of the error list (cycle 1 in group operating mode "Information" Text + Graphics).

Marker areas:

46.0 to 46.7	output of the machine messages	1 to 8
47.0 to 47.7	output of the machine messages	9 to 16
48.0 to 48.7	output of the machine messages	17 to 24
49.0 to 49.7	output of the machine messages	25 to 32

These markers are non-retentive.

Activation:

Machine parameter 20 activates this simplified output function. MP 20 must be assigned parameter 4.
To date, machine parameter 20 only defined the PIC/PC coupling.

P20 = 1 PIC coupling
P20 = 2 PC 400/CL 300 coupling
P20 = 3 PC 600 coupling
P20 = 4 PIC coupling with 32 markers for error messages

Note:

This function is possible only in conjunction with PIC 200 coupling. The simplified output function does not operate in conjunction with PC coupling (PC 400, CL 300, PC 600).

Number of NC keys on the manual panel (MP 17)

Definition:

There are 15 function keys on the manual panel, 10 of which have been able to be assigned NC functions to date and 5 of which have been able to be assigned PIC functions. With machine parameter 17, it is possible to define how many of the 15 keys are to be assigned with fixed NC functions. The ratio of the keys to NC and PIC functions is freely selectable.

Input range:

The input range of machine parameter 17 is restricted from 0 to 15;
Example:

Entry 0 ➧ the keys are to be assigned exclusively with PIC functions.

Entry 15 ➧ the keys are to be assigned exclusively with NC functions.

Entry 8 ➧ 8 keys can be assigned with NC functions, and 7 keys can be assigned withPIC functions.

Compatibility:

If machine parameters of older software versions are loaded on version S 40, machine parameter 17 must be assigned the fixed value 10. Any change in this machine parameter is accepted after Reset.

Customized cycles

Purpose, function:

The contour and boring cycles and the related graphic menus are stored in the EPROM.
The contour and boring cycles can be deactivated with this function. The free G codes (81-89) are assigned customized cycles and stored in the RAM. This means that the cycles and graphic menus can also be edited by the customer at any time.

Selection:

The machine tool builder, in group operating mode INFORMATION, MTB SERVICE, selects whether contour and boring cycles and graphic menus are fixed or whether they can be replaced by customized cycles and menus.

Operating sequence

MACHINE STATUS	SERVICE	MTB SERVICE	LINES SERVICE	RESET + DELETE

Enter the pasword: _____ ENTER

TEXT + GRAPHICS	INTERNAL CYCLES	MTB CYCLES	MACHINE PARAMETERS	PIC/PLC SERVICE

CYCLES IN RAM		BORING			CONTOUR	
		ON	OFF		ON	OFF

The default setting (ON) is displayed in reverse video mode in the fields of softkeys BORING and CONTOUR.

Significance:

BORING ON: The menu of the boring cycles is fixed and cannot be replaced.

CONTOUR ON: The menu of the contour cycles is fixed and cannot be replaced.

The cycles are deactivated by pressing softkey BORING and/or CONTOUR; they can no longer be called by the user in group operating mode MEMORY [>]. Status "fixed" is cancelled only with softkey [CYCLES IN RAM]. The user can now program his own cycles and graphic menus which are stored in the RAM. Softkey [CYCLES IN RAM] is also displayed in reverse video mode.

Creating the cycles and graphic menus in the RAM

In group operating mode INFORMATION [◁] softkeys [MTB SERVICE] and [TEXT+ GRAPHICS] permit access to the editing level in which specific cycles can be created for text files, menus and graphic files.

Graphic menu	Availability in group operating modes MEMORY and MACHINE	Defined by:
Menu 1 Menu 2 Menu 3 Menu 4 Menu 4 Menu 5 Menu 5	always always always if BORING ON if BORING OFF if CONTOUR ON if CONTOUR OFF	File number 3 in RAM File number 4 in RAM File number 5 in RAM Fixed in EPROM File number 40 in RAM Fixed in EPROM File number 41 in RAM
10 display images from menu 1	always	File number 10-19 in RAM
10 display images from menu 2	always	File number 20-29 in RAM
10 display images from menu 3	always	File number 30-39 in RAM
10 display images from menu 4	if BORING ON if BORING OFF	fixed in EPROM File number 70-79 in RAM
10 display images from menu 5	if CONTOUR ON if CONTOUR OFF	fixed in EPROM File number 80-89 in RAM

Conditions for editing the files:

File number 3, 4, 5, 10 - 39 no special conditions

File number 40, 41, 70 - 89 softkey [CYCLES IN RAM] is active.

Operation and programming of the graphic display images are identical to handling the user cycles. The customized cycles can be edited using the cycle editor in group operating mode MEMORY [▷].

The boring cycles are stored under numbers 81 - 89;
the contour cycles are stored under numbers 90 - 99.

Function:

1. Activating the customized cycles

The menus and graphic files can be used as a programming aid in group operating mode MACHINE (MDI mode) or in group operating mode MEMORY (editing programs/cycles).

Menu 1	Menu 2	Menu 3	Menu 4	Menu 5
Graphic 1.1	Graphic 2.1	Graphic 3.1	Graphic 4.1	Graphic 5.1
.	.	.	.	.
.	.	.	.	.
Graphic 1.10	Graphic 2.10	Graphic 3.10	Graphic 4.10	Graphic 5.10

Each menu permits access to 10 graphic display images which each contain the variables and information of one cycle.

2. Explanation of the softkey panel

CYCLES IN RAM		BORING		CONTOUR	
		ON	OFF	ON	OFF

Softkey **"Cycles in RAM"**

Softkey displayed in reverse video mode: The user has no access to the fixed cycles stored in the EPROM if softkey BORING and softkey CONTOUR are set to "OFF". He has access to the customized cycles and graphic menus in the RAM.

Softkey not displayed in reverse video mode: The user has access to the contour and boring cycles and graphic menus stored in the EPROM, provided softkeys BORING and CONTOUR are set to "ON". The cycles are fixed and cannot be changed.

Softkey **"BORING/CONTOUR ON/OFF"**
Boring cycles, contour cycles ON: The contour and boring cycles and graphic menus are stored fixed in the EPROM. They cannot be changed.

Boring and contour cycles OFF: The contour and boring cycles and graphic menus are not stored in the EPROM. No access.

3. Programming the cycles

The cycles can be programmed with or without graphic assistance. The user cycles, stored under G881 to G889, operate modally. Programming does not automatically lead to traverse movements. Modally acting cycles are not executed until the axes have been traversed to the reference point. G80 deactivates the cycles.
Contour cycles 90 - 99 do not act modally.

Revolution-coded, incremental rotary encoder (Messrs. Stegmann)

Basic mode of operation:

The difference with respect to a conventional incremental encoder lies in the additional absolute disks which are subordinate to the incremental disk. The counted pulses of the disks are supplied to a control logic within the encoder.
This control logic, at a defined count, now inserts an additional pulse (marker) per revolution, in addition to the reference pulse, dependent upon the number of revolutions.
This point is offset by one more division period each revolution. On the basis of the distance between the two markers, it is possible to determine the absolute position, referred to the encoder zero point. The absolute position, referred to a displacement, is determined by including an offset which must be determined during commissioning. Once the absolute position has been determined, the encoder is handled in the same way as an incremental rotary encoder.
No special position is required when installing the rotary encoder. There is no need for mechanical adjustment, as on conventional encoders. Different encoders can be connected unrestrictedly to one system: e.g. incremental encoder for the X axis, revolution-coded, rotary encoder for the Z axis.

Limitation by the encoder:

The traverse path is limited by the encoder. The maximum path in pulses results from the maximum number of countable revolutions of the encoder and the numbers of lines on the encoder x factor 4 (see also Machine parameters).

Determining the absolute position:

By contrast with conventional incremental rotary encoders, traversing to a "reference point cam", followed by synchronization, is not required when determining the absolute position.
The operating software of the CC 100 determines the absolute position within one encoder revolution.

Advantages of the revolution-coded incremental encoder:

- Simplification and shortening of the reference point approach procedure since the absolute position can be determined accurate to the increment after a maximum of 1 encoder revolution.
- Reference point switches are not required, and neither are wiring, adjustment or matching etc.
- Extended measuring system monitoring thanks to plausibility check of the revolutions signalled cyclically by the encoder.
- The revolution-coded incremental encoder can take the place of any standard incremental encoder (same interface, cables etc.).

Machine parameters:

- The signal lines for direction reversal may not be confused, owing to decoding of the zero pulse.
  Counting direction reversal can be implemented with machine parameters.
  Axis parameters 44, 64, 84, 104 must be set to 4-7, dependent upon the situation.

- The distance between machine datum and encoder datum must be corrected with the preset (MP 42, 62, 82, 102).

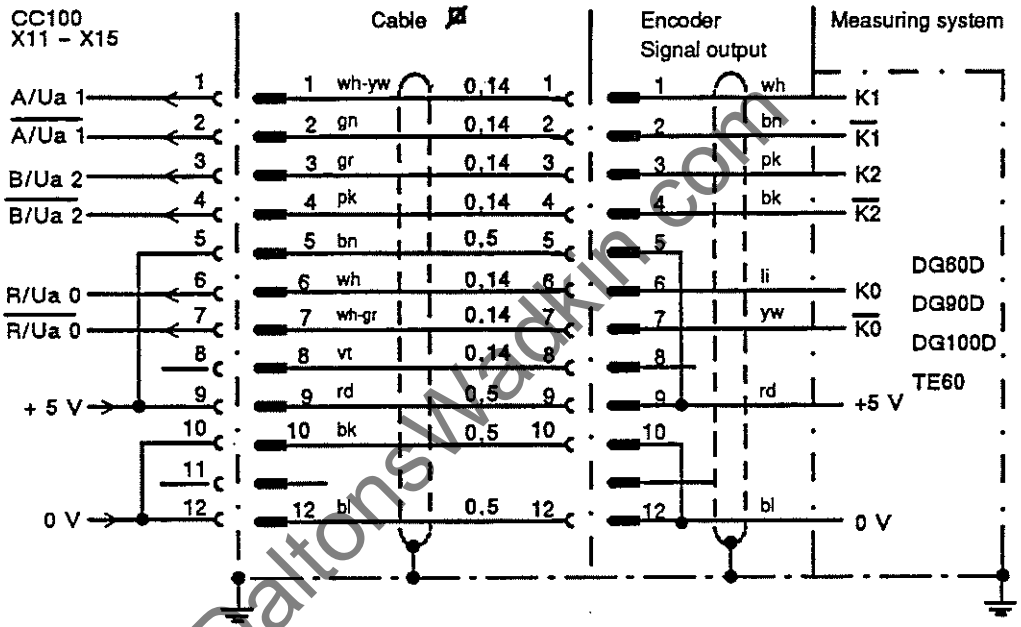
- The axis parameters 46, 66, 86, 106 determine the pulses between 2 zero pulses. The number of lines on the encoder must be programmed, and this is specified on the rating plate on the encoder's housing (entering 0 is not permitted).

- Axis parameters 136, 137, 138, 139, 140 determine the maximum number of revolutions to be coded by the revolution-coded incremental encoder. The required entry is also specified on the rating plate on the encoder's housing.

Interface conditions:

The interface conditions correspond to the conventional rotary encoder. The encoder cable of the CC 100 control to the measuring system is available from BOSCH. It contains a 15-pole subminiature-D connector at the NC end and a 12-pole round socket, type Souriau, at the encoder end.

Illustration



The pin assignment applies to cables supplied by BOSCH.
If the cable with housing form "K", available from Messrs. Stegmann, is used instead of the cable supplied by BOSCH, the cable length is max. 25 m.
If the Bosch cable is used, only housing form "1" with a 0.5 m cable stub is permitted for the encoder. The user must then use a 12-pole round connector, type Souriau, on the encoder.

Notes:

Points "Pin assignment" and "Available numbers of lines and revolutions, in addition to item numbers" have been taken from the Stegmann data sheets for the TE 60 rotary encoder.
Please refer to the Stegmann data sheet No. 8967, Edition 01/89, for further information.

Available numbers of lines and revolutions, in addition to item numbers.

5 T 4 X □ 0 □ 1 F 0 □

Enclosure:

Item number	Version
0	= with shaft seal
1	= without shaft seal

Item number		Number of lines	Number of coded revolutions
15	=	100	32
T6	=	150	64
26	=	200	64
36	=	250	64
46	=	256	64
R7	=	300	128
S7	=	314	128
57	=	360	128
67	=	400	128
77	=	500	128
87	=	512	128
N8	=	570	256
98	=	600	256
B8	=	720	256
P8	=	750	256
C8	=	800	256
D8	=	900	256
E8	=	1000	256
F8	=	1024	256
Q9	=	1200	512
G9	=	1250	512
H9	=	1500	512
I9	=	1800	512
J9	=	2000	512
K9	=	2048	512
LA	=	2500	1024
VA	=	5000	1024

Item number		Housing type
1	=	Housing 1
B	=	Housing B
D	=	Housing D
K	=	Housing K

BOSCH

Flexible Automation

Numerische Steuerungen · Computer Numerical Control
Speicherprogrammierbare Steuerungen · Programmable Logic Controllers
Antriebstechnik · Drives
Robotersteuerungen · Robot Controls
Schweißsteuerungen · Welding Controls
Automatisierungstechnik · Automation Technology

Bosch Elektronische Steuerungen · Verkaufsorganisation Inland
Bosch Electronic Controls · German Sales Organisation

Robert Bosch GmbH · Geschäftsbereich Industrieausrüstung · Produktbereich Industrielle Steuerungselektronik
Berliner Straße 25 · Postfach 1162 · 64701 Erbach/Odw. · Telefon (06062) 78-0 · Telex 41916-60 rbd · Telefax (06062) 78-428

Robert Bosch GmbH
Abteilung IA/VHR
Postfach 110152
30856 Laatzen
Tel. (0511) 8606-263
Telex 921163
Telefax (0511) 8606-419

Robert Bosch GmbH
Abteilung IA/VKÖ
Postfach 100543
50445 Köln
Tel. (0221) 4905-297
Telefax (0221) 4905-228

Robert Bosch GmbH
Abteilung IA/VSW
Postfach 300268
70442 Stuttgart
Tel. (0711) 8934-615
Telefax (0711) 8934-632

Robert Bosch GmbH
Abteilung IA/VMÜ
Postfach 200362
80003 München
Tel. (089) 5128-284
Telex 522151
Telefax (089) 5128-205

Bosch Elektronische Steuerungen · Verkaufsorganisation Ausland
Bosch Electronic Controls · Foreign Sales Organisation

Robert Bosch AG
Geiereckstraße 6
A-1110 Wien
Tel. (43-1) 797 22-0
Telex (047) 131638
Telefax (43-1) 797 22 75 99

EMTS PTY Ltd.
Factory 1, 40 Strong Avenue
Thomastown Victoria 3074
Australia
Tel. (61-3) 460 69 00
Telefax (61-3) 462 12 14

N.V. Robert Bosch S.A.
Henri Genessestr. 1
B-1070 Bruxelles
Tel. (32-2) 525 52 86
Telex (046) 21582
Telefax (32-2) 525 52 56

WEG Automaçao Ltda
Rua Joinville, 3000
BR 89256-900 Jaraguá do Sul, SC
Tel. (55) (473) 72-20 20
Telex 475-247
Telefax (55) (473) 72-40 20

Robert Bosch AG
Hohlstraße 188
CH-8021 Zürich
Tel. (41-1) 247 62 11
Telex (045) 812255
Telefax (41-1) 247 64 21

Representaciones Tecnicas
Ripoll
Apartado 163
E-Zarauz-Guipúzcoa
Tel. (34-43) 83 10 51
Telex (052) 38762
Telefax (34-43) 13 35 94

Robert Bosch France
Département VCI
32, Av. Michelet
B.P. 170
F-93404 Saint-Ouen Cedex
Tel. (33-1) 40 10 72 34
Telex (042) 234132
Telefax (33-1) 40 10 20 43

Robert Bosch Ltd.
Meridian South
Meridian Business Park
GB-Braunstone/Leicester
Tel. (44-533) 89 27 23
Telex (051) 341 254
Telefax (44-533) 89 28 78

Novotech Company Ltd.
Novosibirskaja St. 2
GUS 450059 UFA
Tel. (34 72) 32 74 55
Telex 162125 ptb su
Telefax (34 72) 331677

Robert Bosch
Industriale Comerciale S.P.A.
Casella Postale 15049
Via M.A. Colonna, 35
I-20149 Milano
Tel. (39-2) 3 69 61
Telex (043) 321506
Telefax (39-2) 3 69 64 42

Electronica
Elektra Chambers
44 Mukand Nagar
IND Pune 411037
Tel. 42 2013
Telex 145-7432 EDM IN
Telefax 91-212-448532

Robert Bosch B.V.
Industrietechniek
Postbus 8061
NL-1005 AB Amsterdam
Tel. (31-20) 5 80 02 47
Telex (044) 12282
Telefax (31-20) 5 80 08 82

PIAP
Industrial Research Institute
for Automation and
Measurements
Al. Jenozolimskie 202
PL 02-222 Warsaw
Tel. (48) (22) 238-366
Telex 813-726 PI
Telefax (48) (22) 238-864

Automationshuset
Robcona AB
Elektrodgatan 1
S-72137 Vaesteraas
Tel. (46-21) 81 08 80
Telex (054) 40442
Telefax (46-21) 81 08 88

Flexible Electronic Systemes
(PTY.) Ltd.
P.O. Box 3881
Kempton Park 1620
27, Forge Road
SA-Republic of South Africa
Tel. (27-11) 9 75 70 00
Telefax (27-11) 3 94 20 85

Robmatic OY
P.O. Box 59
Kamreerintie 6
SF-02771 Espoo
Tel. (35-80) 85 91 31
Telex (057) 125316
Telefax (35 80) 8 59 13 05

Robert Bosch
(South East Asia)
PTE. Ltd
P.O. Box 4
(Thomson Road)
38c-38d Jalan Pemimpin
SGP-2057 Singapore 20
Tel. (65) 2 58 55 11
Telex (087) 24062
Telefax (65) 2 58 46 71

Robert Bosch Corp.
Canton Business Park
97 River Road
Collinsville, Ct. 06022-1231
USA
Tel. (1-203) 6 93 63 21
Telex (023) 643830
Telefax (1-203) 6 93 17 39

BOSCH

Flexible Automation

CNC Steuerungen für Werkzeugmaschinen	**CNC** controls for machine tools
RC Robotersteuerungen	**RC** robot controls
SPS Speicherprogrammierbare Steuerungen	**PLC** programmable logic controllers
Antriebstechnik Bürstenlose Servo- und Hauptspindelantriebe	**Drives** brushless servo and main spindle drives
Schweißtechnik Steuerungen für Widerstandsschweißanlagen	**Welding** controls for resistance welding
Automatisierungstechnik DNC – BDE/MDE	**Automation Technology** DNC – MDP/ODP

Berliner Straße 25, Postfach 1162
D-64701 Erbach/Odenwald
Telefon 06062/78-0, Telefax 06062/78-428