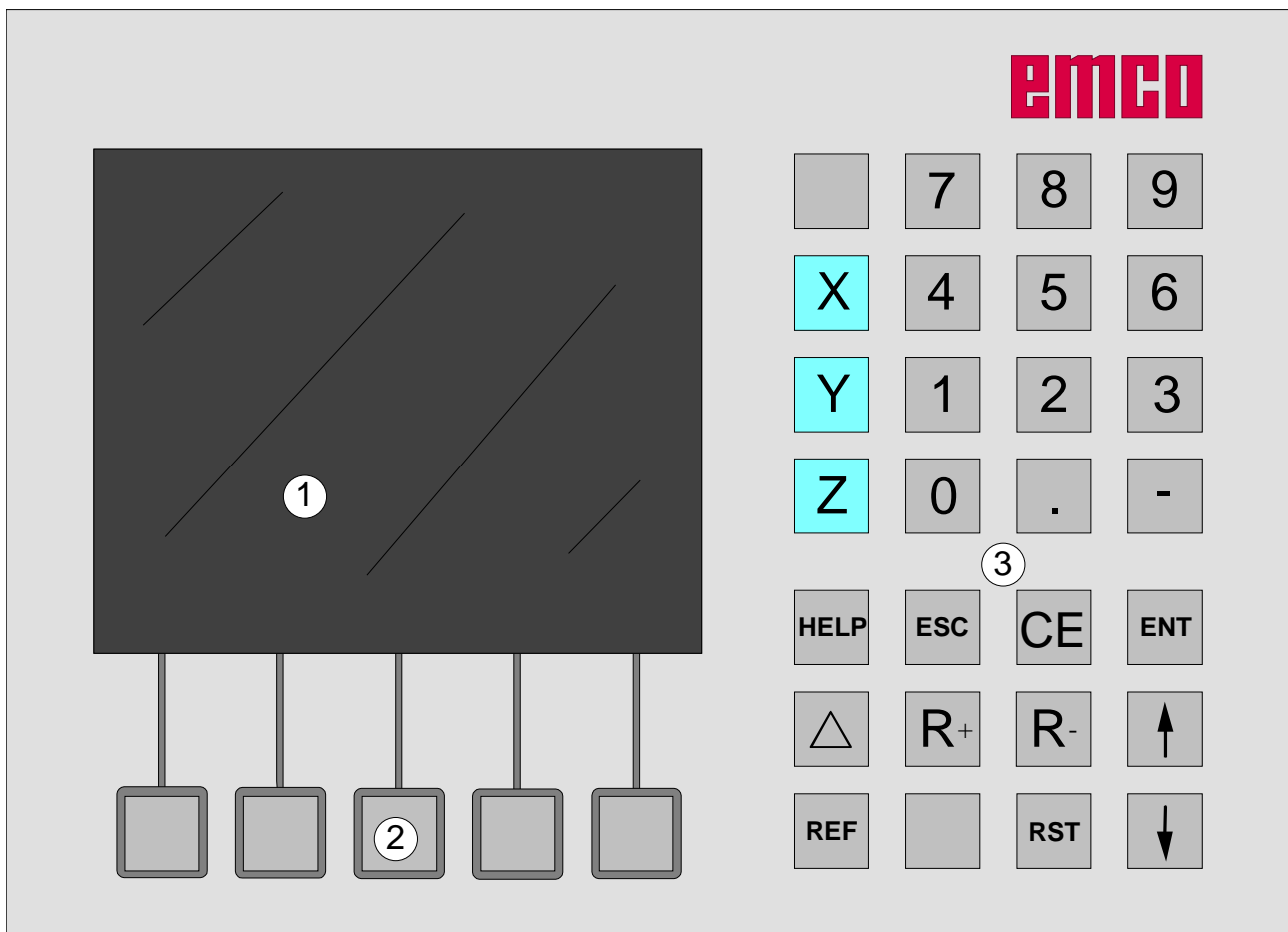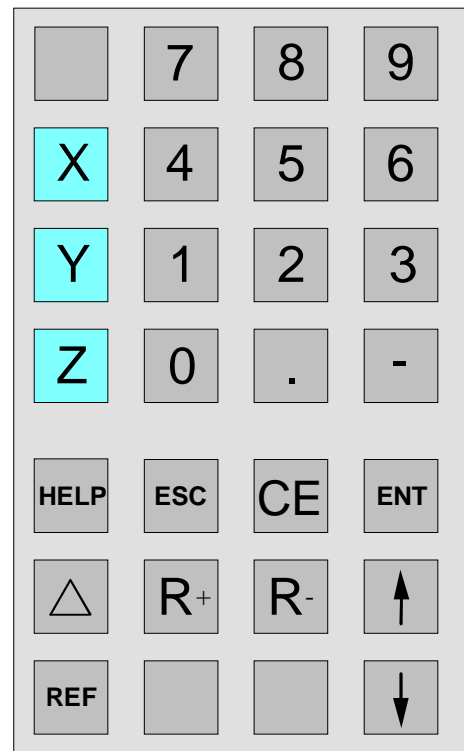# Digital display
for EMCO milling machines

Input box

1 .... Screen (working window, displays)
2 .... 5 soft keys (function depends on the respective text shown on the screen)
3 .... Key block

D2009-12

Key block

	7	8	9
X	4	5	6
Y	1	2	3
Z	0	.	-
HELP	ESC	CE	ENT
△	R+	R-	↑
REF			↓

Coordinate keys

X Y Z

By pressing one of the coordinate keys the position display of the axis is selected (path display shown with red frame). This axis is to be set via the key block (e.g. set to zero, enter a certain value, path to go - display).

Reference key

REF

You enter the reference mode by pressing the reference key.
In this mode the reference labels are traversed.
In addition, tools and contour points can be entered. In the "system" area, diagnosis function and data import/export can be selected. When pressing the key again, the reference mode will be terminated.
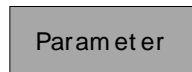
0 to 9   **Numeric key block**

.   **Decimal key**

-   **Sign key**

Path to go - key

By pressing this key you switch over to the path to go - display (the symbol "△" appears).
The path to go - display is switched off by pressing the path to go - key twice (the symbol "△" disappears).

Soft key Parameter

With this key change-over into parameter input mode is carried out. In this mode standard selections such as display in [mm] or in [inch] or tool zero point active or inactive can be preselected. After having entered the parameters, you leave the parameter input mode by pressing the soft key "Back".
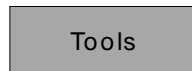
Soft key Tools

With this soft key you call the tool table. Tools can be measured (see "Tool measurement").

Soft key Contour

The contour table is called with this soft key. You can define contour points (see "Saving contour points").

Soft key Pattern

With this soft key you call the circular hole pattern and the linear hole pattern.

Soft key Info

The information page is called by means of this soft key. Data on software version, AC version, PLC-Version, electrical documentation number and power supply are displayed.

Radius compensation

When you press this key, the radius compensation is activated.
The radius compensation is only possible, after a tool has been selected.

Correction key

With the correction key incorrect inputs, that have not been stored with the enter-key yet, can be corrected.

emco

Soft key Select

Select

Tools and contour points can be selected by means of this key. A selected tool or a selected contour point is displayed in the top left-hand screen (see screen layout).

Soft key Deselect

Deselect

This soft key enables you to deselect already selected tools and contour points. In addition, you can leave the contour point mode with this key.

Input key

ENT

With this key changed data or data entered new via the key block are taken over and stored.

Escape key

ESC

This key enables you to reject changed or newly entered data via the key block.
An error message will be deleted.

Reset key

RST

- Alarms are cleared.
- The control is in bascic setting.
- working off of a movement will be stopped.

Screen layout

| T ?① P ?② | ③ | ④ 06/21/07 |

X ⑤   **0,00** mm
R+

⑥
△Y   **-20,00** mm

Z   **0,00** mm

⑦ Permissible value range: -10000 to 10000

| Parameter | Tools | | Contour ⑧ | Info |

Note:
The symbol "?" means that no tool or contour point has been selected.

1 Number of the selected tool
2 Number of the selected contour point
3 Spindle speed display (Option)
4 Feed display / date
5 Path display
6 "Δ" path to go - display
7 Alarm and message line
8 Soft-key row; soft keys are selected by means of the keys being positioned below.

Note:
Nominal values are displayed on a yellow background.
Actual values are displayed on a cyan background.

emco
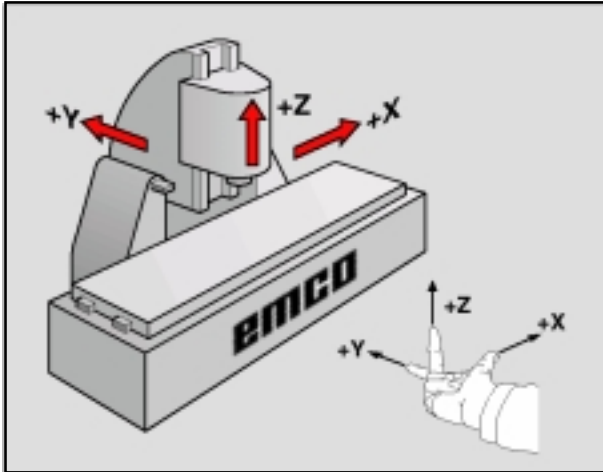
Reference system for milling machines

A reference system is required to define positions in a plane or in space. The position data are always referenced to a predetermined point and are described through coordinates.

The Cartesian coordinate system (a rectangular coordinate system) is based on three coordinate axes X, Y and Z. The axes are mutually perpendicular and intersect at one point called the datum. A coordinate indicates the distance from the datum in one of these directions. A position in a plane is thus described through two coordinates, and a position in space through three coordinates.

Coordinates that are referenced to the datum are called **absolute coordinates**. Relative coordinates are referenced to any other known position (reference point) within the coordinate system. Relative coordinate values are also referred to as **incremental coordinate** values.

When working a workpiece on a milling machine you generally orient tool movement to the Cartesian coordinate system. The illustration on the left shows how the Cartesian coordinate system describes the machine axes. The "right-hand-rule" helps to remember the three axes directions: the middle finger is pointing in the positive direction of the tool axis from the workpiece towards the tool (the Z axis), the thumb is pointing in the positive X direction, and the index finger in the positive Y direction.
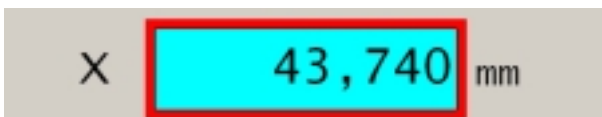
Path display

In the path displays the position of the tool or of a certain point on the upper slide with respect to a reference point can be read.

A red framing of the path display shows which of the three axes is currently active. Active means which position display of the respective axis can be set (e.g. setting to zero, enter a certain measurement, path to go - display).

Parameters of the path-measuring system

Caution:
All protected parameters marked with "Key" should only be changed with the greatest care, so as to avoid errors within the measurement system.

In parameter input mode certain default settings can be carried out which are permanently effective after machine switch-on. 3 groups of parameters are distinguished:
• settings for initial operation
• settings for operating surfaces
• machine parameters
There are protected parameters as well as ones being not protected. In case you want to change a protected parameter, you have to enter a password for the parameter change.

Start-Up   Control   Settings

Changing parameters

Parameter

| T ? | P ? | Parameter | 14:17:39 |

No.	Description:	Value:	Edit:
8	Datum point/zero offset	0	Yes
9	Display	2	Yes
10	Resolution spindle speed	4	Yes
11	X-Display active	1	Yes
12	Z0-Display active	1	Yes
13	Z-Display active	1	Yes
14	Digital readout only	0	Yes
15	Reference point X	0.00 mm	Key
16	Reference point Z0	0.00 mm	Key

1...limit to digital readout, 0...no limit

Go to   Edit   Back

| T ? | P ? | Parameter | 14:25:51 |

Key entry:

3141

Enter key for parameter change

Apply   Back

Back

1. Call the parameter table (press the soft key "Parameter")

2. Select the parameter number by means of the cursor keys or the soft key "Go to". The function of the parameter is explained in the alarm and message line.

3. Press the soft key "Key". In case only the soft key "Edit" appears, press this key and continue with point 5.

4. Enter the password for the parameter change. The key reads: 3141! Confirm with the soft key "Apply" or the enter-key. Incorrect entries can be corrected by means of the "CE" key, before pressing the enter-key.

5. The selection possibilities for alterations are displayed in the alarm and message line. Save changes with the soft key "Apply". Incorrect entries can be corrected by means of the "CE" key, before pressing the enter-key.

6. After having changed a parameter, the machine has to be started again.

7. Quit the parameter table by pressing the soft key "Back" or the ESC key.

emco

Position display and approach

Path to go - display
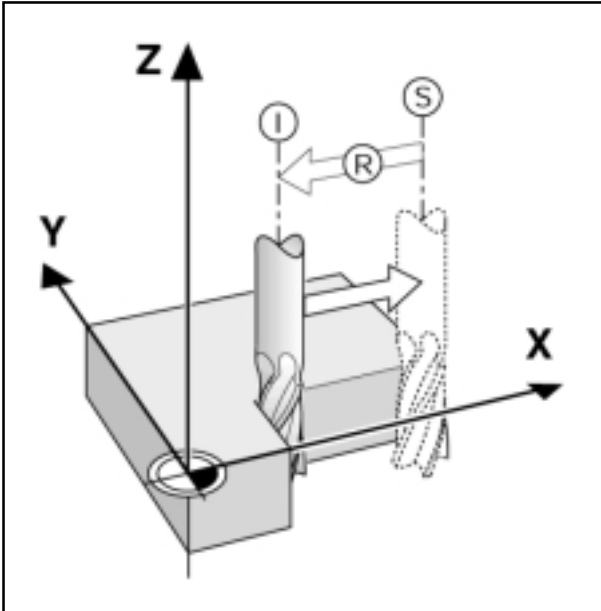
The target position is the position where the tool has to be moved to; the actual position is the position where the tool currently is.
The path between the target and the actual position is the path to go.
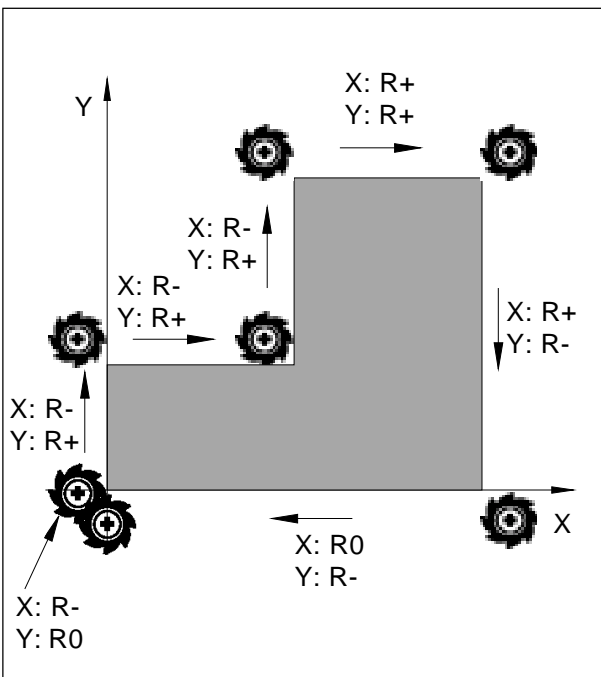
Often, it is sufficient when the coordinates of the actual position of the tool are displayed. Most of the time it is better to display the path to go. You may easily position by moving to the display value zero.

Arithmetic sign of the path to go
The path to go has a positive arithmetic sign if you move from the actual to the target position in negative axis direction.
The path to go has a negative arithmetic sign if you move from the actual to the target position in positive axis direction.
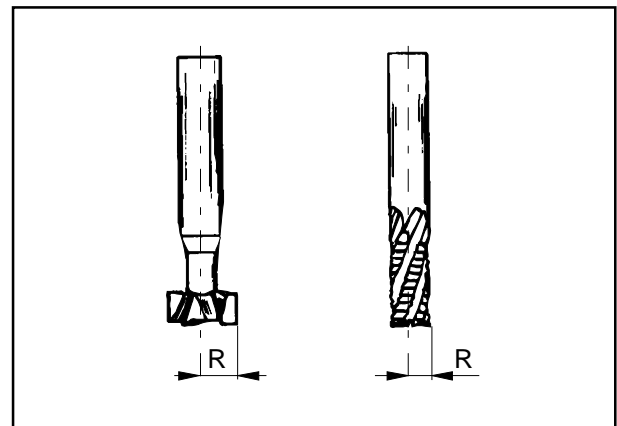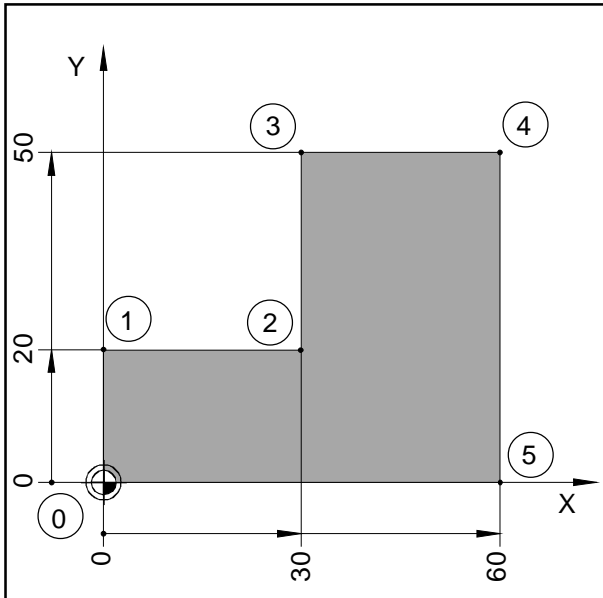
Tool radius compensation

The digital display has a tool radius compensation that makes it possible to enter drawing dimensions directly. During machining the digital display automatically shows a travel path that is lengthened (R+) or shortened (R−) by the tool radius.
You have to select a tool, so that the digital display is enabled to activate the tool radius compensation.
An active radius compensation (R+ respectively R-) is deactivated by first pressing the axis key and afterwards pressing the respective R key.

Tool radius related to traversing movement

Example: Milling a step by "Traversing to zero"

The coordinates are entered as absolute dimensions, the reference point is the workpiece zero point.

corner point **0**: X = 0 mm / Y = 0 mm
corner point **1**: X = 0 mm / Y = 20 mm
corner point **2**: X = 30 mm / Y = 20 mm
corner point **3**: X = 30 mm / Y = 50 mm
corner point **4**: X = 60 mm / Y = 50 mm
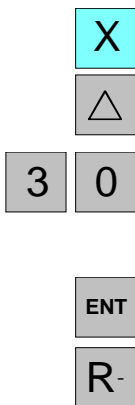corner point **5**: X = 60 mm / Y = 0 mm

Preparation:
Select tool 1.
Position the tool in way that makes sense (e.g. X = Y = − 20 mm).
Traverse the tool to the milling depth.

Traversing the corner point 0-1:

1. Press the Y-axis key Y (the selected active axis gets a red frame).

2. Press the path-to-go key **Δ**.

3. Enter the position nominal value for corner point **1**: Y = 20 mm
   Corrections can be carried out by means of the "CE" key.

4. Confirm the input.

5. The tool radius compensation R+ must be active. The axis displays show the paths-to-go that still have to be traversed with a yellow background and the symbol **Δ**. The path displays where no target positions have been entered, show the current actual position with a blue background.

6. Traverse the Y axis to the display value zero.

| Y |
| △ |
| 2 | 0 |
| ENT |
| R+ |

T1 P ?		06/21/07
X	-12,50	mm
△Y R+	7,50	mm
Z	0,00	mm
Parameter	Tools	Datum point	Contour	Info

EMCO

Traversing from corner point 1-2:

X

△

3 0

ENT

R-

7. Press the X-axis key (the selected active axis gets a red frame).

8. Press the path-to-go key Δ.

9. Enter the position nominal value for corner point **2**: X = 30 mm
Corrections can be carried out by means of the "CE" key.

10. Confirm the input.

11. The tool radius compensation R- must be active.

12. Traverse the X axis to the display value zero.
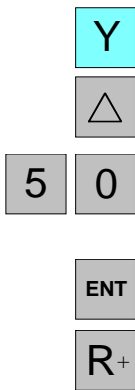
Traversing from corner point 2-3:

Y

△

5 0

ENT

R+

13. Press the Y-axis key (the selected active axis gets a red frame).

14. Press the path-to-go key Δ.

15. Enter the position nominal value for corner point **3**: Y = 50 mm
Corrections can be carried out by means of the "CE" key.

16. Confirm the input.

17. The tool radius compensation R+ must be active.

18. Traverse the Y axis to the display value zero.
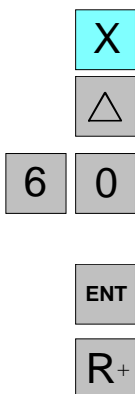
Traversing from corner point 3-4:

X

△

6 0

ENT

R+

19. Press the X-axis key (the selected active axis gets a red frame).

20. Press the path-to-go key Δ.

21. Enter the position nominal value for corner point **4** : X = 60 mm
Corrections can be carried out by means of the "CE" key.

22. Confirm the input.

23. The tool radius compensation R+ must be active.

24. Traverse the X axis to the display value zero.

EMCO

Traversing from corner point 4-5:

Y

25. Press the Y-axis key (the selected active axis gets a red frame).

△

26. Press the path-to-go key **Δ**.

0

27. Enter the position nominal value for corner point **5** : Y = 0 mm
Corrections can be carried out by means of the key "CE".

ENT

28. Confirm the input.

R-

29. The tool radius compensation R- must be active.

30. Traverse the Y axis to the display value zero.

Traversing from corner point 5-0:

X

31. Press the X-axis key (the selected active axis gets a red frame).

△

32. Press the path-to-go key **Δ**.

0

33. Enter the position nominal value for corner point **0** : X = 0 mm
Corrections can be carried out by means of the "CE" key.
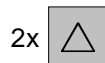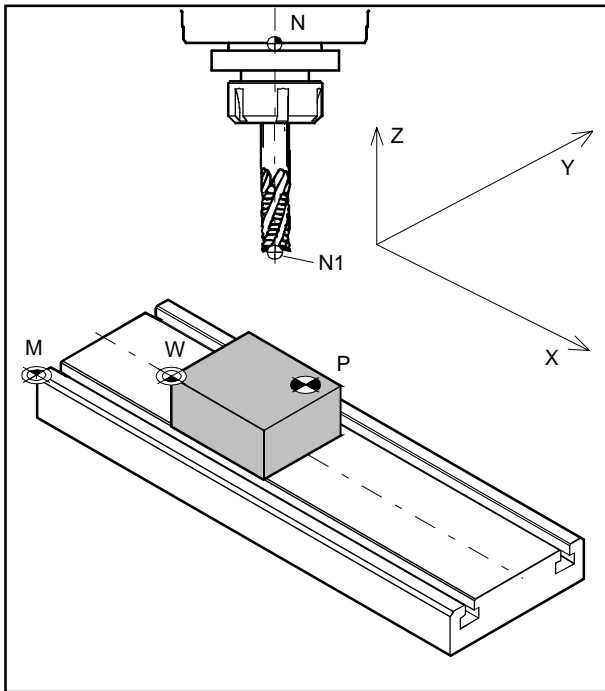
ENT

34. Confirm the input.

X R+

35. There must be no tool radius compensation (R0) active. The radius compensation (R+, respectively R-) is deselected by first pressing the axis key and afterwards the respective R key.

36. Traverse the X axis to the display value zero.

2x △

37. The path-to-go display is switched off by pressing the path-to-go key two times.

EMCO

Machine points

Machine zero point M ✛
The machine zero point M is a fixed point on the machine.
In this point the origin of the coordinate system is to be found.
The machine zero point is exactly positioned on the left front corner of the milling table.

Workpiece zero point W ✛

The workpiece zero point can be defined by the operator at will.
The workpiece zero point W should usefully be positioned on a workpiece edge where most of the workpiece dimensions refer to.
Through the zero offset, the origin of the coordinate system is shifted from the machine zero point M to the workpiece zero point W.

Tool holder reference point N ✛

The tool fixture reference point N is a fixed point on the machine. It is positioned in the axis of the vertical milling spindle and on the front side of the sleeve. After activation of the reference marks, the path display continually shows the position (X, Y, Z) of the tool fixture reference point N towards the machine zero point M.

Tool reference point N1 ✛

The tool reference point N1 is determined by the tool measurement. After the measurement and the calling of the tool, the path display continually shows the position (X, Y, Z) of the tool reference point N1 (tool tip) towards the machine zero point M.
The tool reference point N1 can only be saved in the reference mode (see tool calibration).
A maximum of 999 tools can be saved (standard setting 99 tools; can be set with the parameter "tools").

Contour point P ✛

The contour point K is a point which can be freely selected and stored. It is used to be able to find   certain points of a workpiece, e.g. when manufacturing a number of similar workpieces.
A maximum of 999 contour points can be stored (standard setting 99 contour points; can be set with the parameter "contour points").

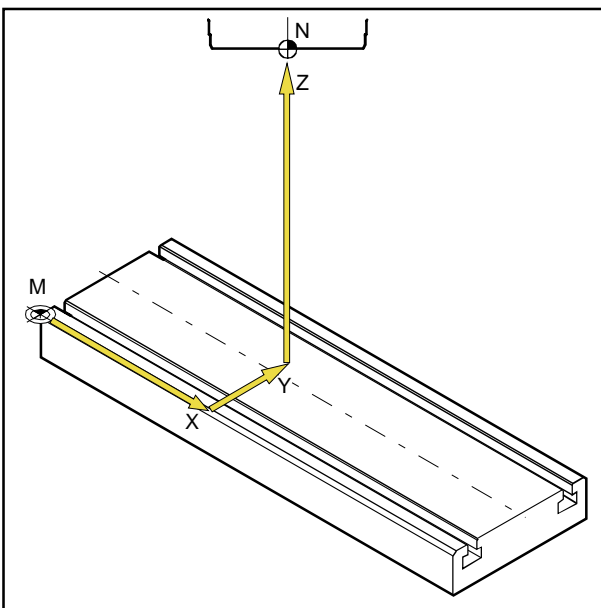*Machine zero point .... M*
Workpiece zero point .... W
Tool holder reference point .... N
Tool point .... N 1
Contour point .... P

T ? P ?		06/21/07
X	**0,00**	mm
Y	**0,00**	mm
Z	**0,00**	mm
Parameter	Tools	Contour	Info

Switch-on state - path displays flash RED

Switch-on state

When switching the machine on, all path displays of the axes are flashing in red colour.
The flashing indicates that the measuring electronics have not been referenced to the machine yet. The displays show slide positions that have not been defined yet (the reference labels have not been activated yet).

T ? P ?		06/21/07
X	**35,00**	mm
Y	**21,87**	mm
Z	**21,87**	mm
Parameter	Tools	Contour	Info

P ath displays referenced

REF

Activation of reference labels

Through activation of the reference labels the measuring electronics is informed about the exact position of the tool holder reference point N to the machine zero point M.
After each switch-off and switch-on of the machine thus exactly the same points can be approached and the path displays always show the same positions (repeating accuracy).

Procedure:

1. Switch the machine on.

2. Activate the reference mode (press the reference key).

3. Traverse all three slides one after the other by pressing the axis keys X, Y and Z until the single displays stop flashing. The flashing of the displays stops when the reference point in the respective axis has been traversed. Simultaneously, the current absolute position of the tool holder reference point N towards the machine zero point M is displayed.
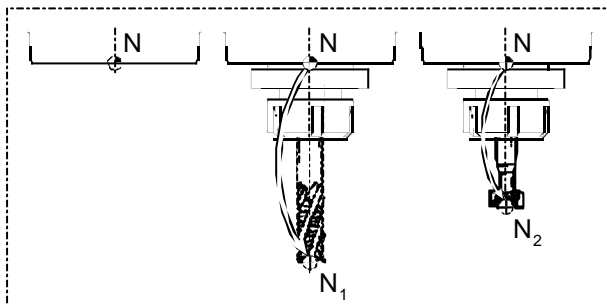
The digital readout is configured in accordance with the machine.
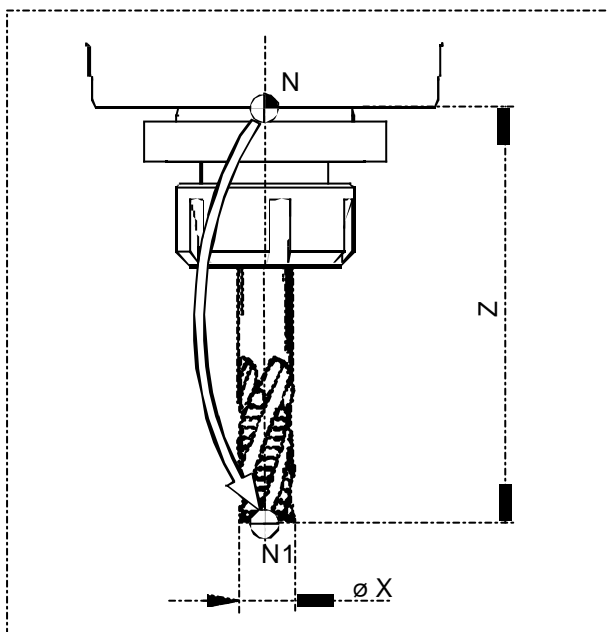
emco

CE

It is not necessary to activate the reference point, when you press the correction key after having switched the machine on.

However, the disadvantage is that the reproducibility of stored tool points and contour points is lost since the reference points refer to the absolute values and these absolute values are changed with each machine switch-on without activation of reference labels.
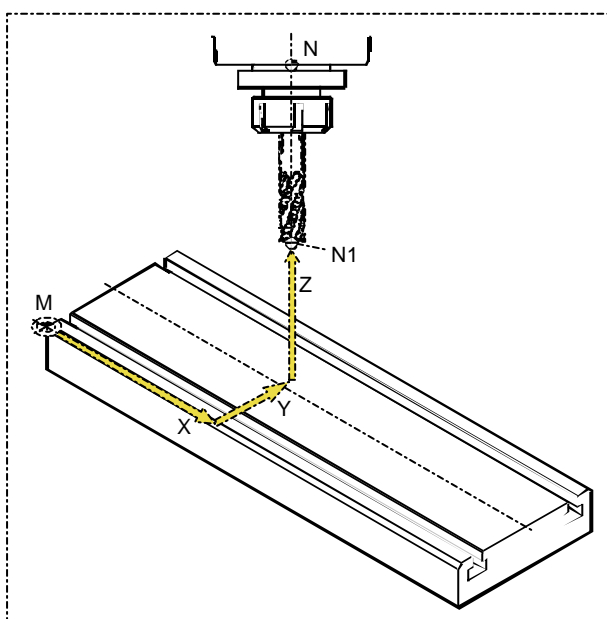
Also the backlash-settlement will not work.

Tool reference points

Tool measurement

Tool length
The tool length is the distance between the tool holder reference point (N) and the tool reference point (N$_1$, N$_2$, etc.).
For a tool system (tool with toolholder) this tool length (Z) remains the same after every new clamping procedure.

Tool data (length Z and øX)

Purpose of a tool measurement
For a tool that is used again and again, the tool data (length Z and øX) are measured once and are saved as "tools". By calling this tool, the calculation point will be shifted from point N to point N$_1$ for all traversing movements. This means, that when you clamp and call a measured tool again, the screen will constantly display the actual value (X, Y, Z) of this tool reference point N$_1$ in relation to the machine zero point (M).
Without tool measurement, the actual value (X, Y, Z) of the tool holder reference point (N) is displayed.

Position displayed after tool call

Note:
Working with measured tools makes sense only if the reference point was activated previously.
If this is not the case, the repeating accuracy is lost from switch-off to the next machine switch-on.

For tool measurement either the method "Tool measurement with touching" OR "Manual tool measurement" can be used.

EMCO

N

N

Z

W N₁ N₂ measuring plane
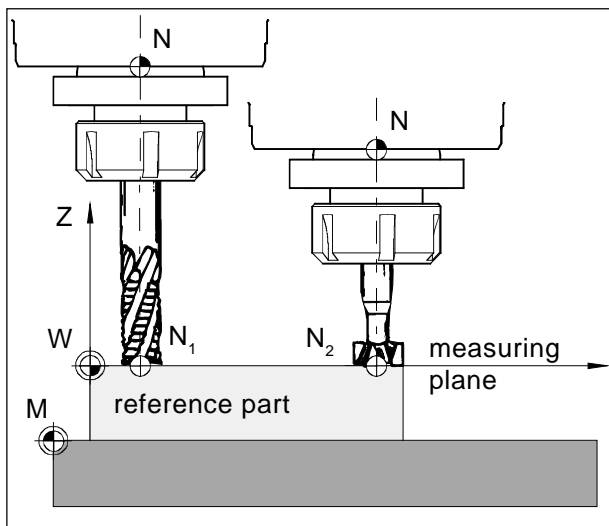
M reference part

Saving the tool data

For tool measurement any workpiece (reference part) can be used. The surface of the workpiece is defined as measuring plane. The tools that are going to be measured touch one ofter the other on the measuring plane. The Z value at the time of touching (related to the tool) is defined with the value Z=0).
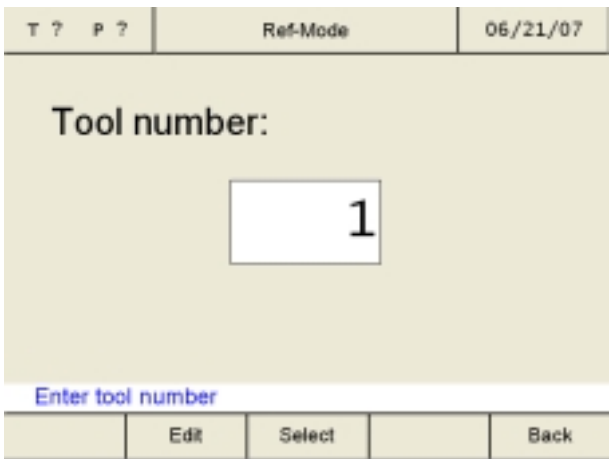
Purpose: When an already measured tool (tool with toolholder) traverses to Z=0 after being clamped again, the tool is exactly positioned on the user-defined measuring plane.

Tool measurement by means of touching:
1. Clamp tool 1.

REF

2. Enter the reference mode (press REF key).

Tool

3. Press the soft key "Tool".

4. Enter the number of the tool and confirm with "Edit". For tool measurement, the tool itsself must not have been selected.

T ? P ?	Ref-Mode	06/21/07

Tool number:

1

Enter tool number

	Edit	Select		Back

Edit X

5. Enter the X value of the tool (diameter) and confirm the value with the enter key.

T ? P ?	Ref-Mode	06/21/07

tooldiameter 1:

∅ X 20,00 mm

Toollengh Z:

Z 0,00 mm

calibrate tool 1

Edit X	Touching in Z	Select	Apply	Back

emco

Touching in Z

Reference part

0 | ENT

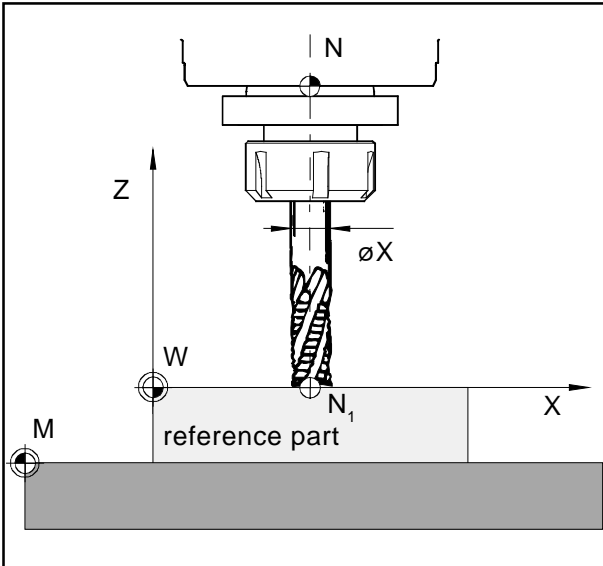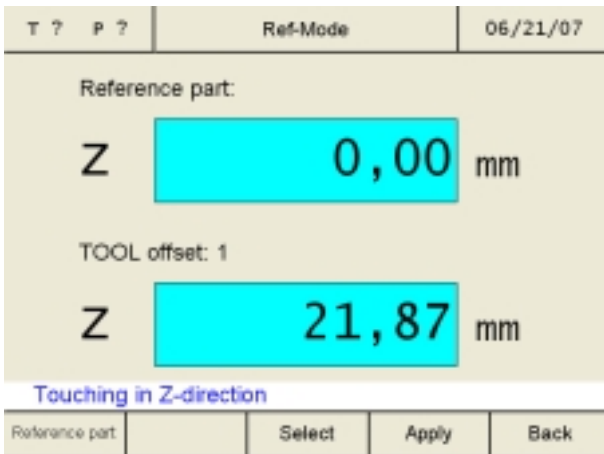*Touching in Z*

Apply

6. Press the soft key "Touching in Z".
   For the touching process, a workpiece is required as reference part.

7. Define the Z value of the reference part with "0" (Z coordinate related to the X-Y plane (measuring plane)) and confirm the value with the enter key.

8. Touch the reference part (workpiece) in the Z axis. Save the values with the soft key "Apply".

9. Tool 1 is now measured. The values are saved in the tool table under the respective tool number.

10. Leave the reference mode by pressing the soft key "Back" or by pressing the ESC key twice.

emco

Variants of tool measurement with the touching method:
The measuring plane (Z=0) can also be defined at any other position within the working area.

Example 1:
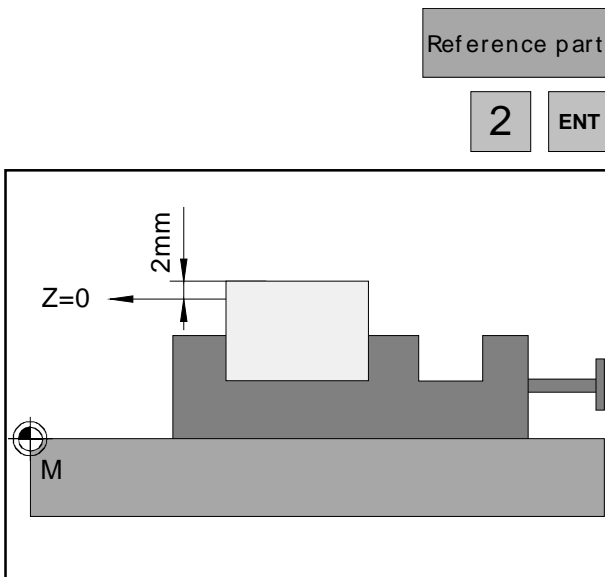The reference part (workpiece) has an exactly defined height (e.g.: 20mm).
If during tool measurement with touching, the Z value of the reference part is defined "20" instead of "0", the position Z=0 is situated on the machine table.

Reference part

| 2 | 0 | **ENT** |

20mm

reference part

Z=0

M
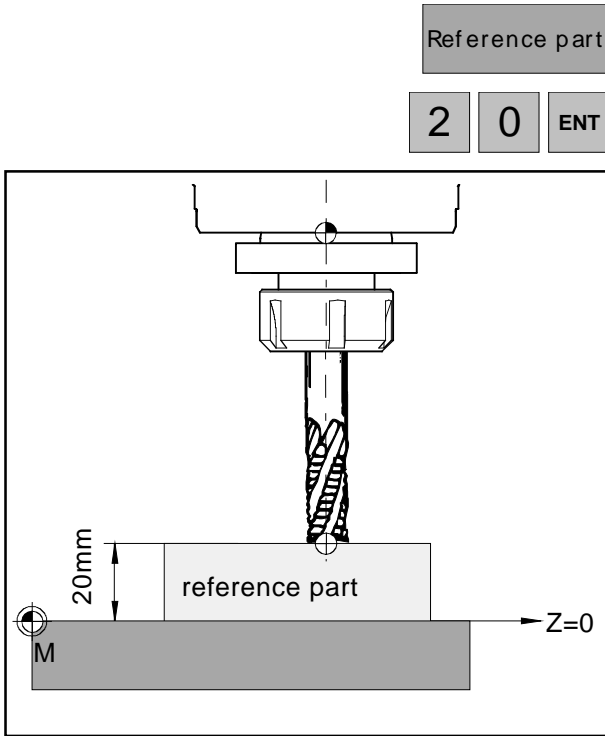
Example 2:
The reference part (workpiece) is a still unmachined workpiece with 2mm allowance.
If during tool measurement with touching, the Z value of the reference part is saved with "2", the position Z=0 is situated on the surface of the ready-machined workpiece.

Reference part

| 2 | **ENT** |

2mm

Z=0

M

Manual tool measurement:
The tool dimensions can also be measured by means of a measuring machine. Then the received values can be entered manually into the tool table.
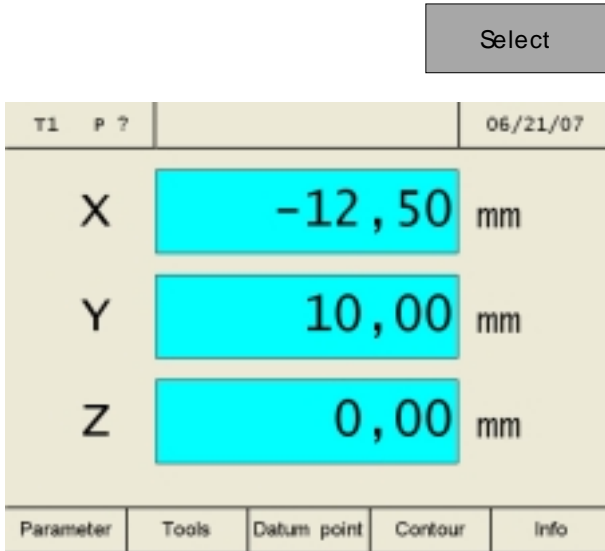
	Tools	1. Call the tool table.

T ?	P ?		Tools		06/21/07

No.	Diameter:	Z position
1	20.00	0.00
2	8.00	0.00
3	20.00	4.38
4	90.00	45.00
5	12.00	0.00
6	46.00	0.00
7	0.00	0.00
8	0.00	0.00
9	0.00	0.00

Select tool

Go to	Edit	Select	Deselect	Back

2. Select the tool number with the cursor keys or with the soft key "Go to".

Edit

3. Press the soft key "Edit".

Edit X	Edit Z

Apply

4. Use the slide gauge to measure the tool dimensions (see illustration) and enter the values with the soft keys "Edit X" and "Edit Z".
Save with "Apply".
Prior to pressing the enter key, wrong inputs can be corrected by means of the "CE" key.

T ?	P ?		Edit tool no. 1		06/21/07

Tool number: 1

∅ X   20,00 mm

Z   0,00 mm

Enter tool dimensions

Edit X	Edit Z	Select	Apply	Back

emco

Definition of the reference point / zero offset

Select

T1 P ?		06/21/07
X	$-12,50$	mm
Y	$10,00$	mm
Z	$0,00$	mm

| Parameter | Tools | Datum point | Contour | Info |

The soft keys "Datum point" or "Zero Offset" only appear when a tool has been selected.

Use the parameter "Datum point / zero offset" to switch between the definition of the reference point and the zero offset.

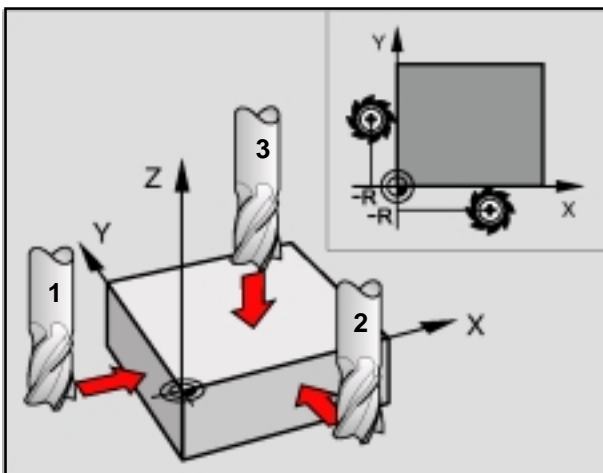After having changed a parameter, the machine has to be started again (see changing parameters).

M
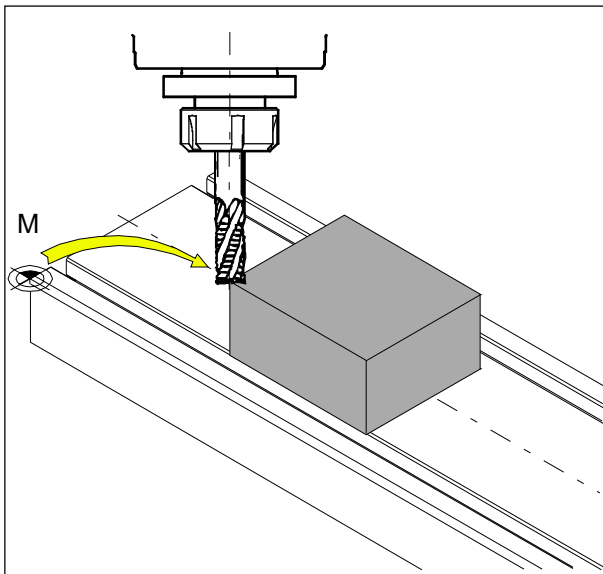
Definition of the reference point:
Move the tool to the required reference point (e.g. workpiece edge) and define the X, Y and Z value with "0".

Purpose: The position display shows the position of the tool in reference to the workpiece edge and not, as before, in reference to the machine zero point (M).
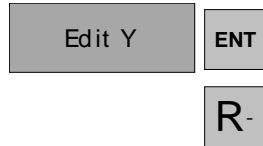
Setting the reference point:

1. Clamp the measured tool and select it from the tool table.

Datum point

2. Press the soft key "Datum point".

T1 P ?	Set datum point	06/21/07

X −12,50 mm

Y 10,00 mm

Z 0,00 mm

Edit X	Edit Y	Edit Z	Reset	Back

Step 1 to 3

In step 1 the reference point shall be set in X-axis direction:

3. Use care when traversing the tool in X-axis direction until it touches the workpiece.

Edit X ENT

4. Press the soft key "Edit X" and confirm the input.

R-

5. Here, the tool radius has to be considered. Select the tool radius compension R-.

emco

In step 2 the reference point shall be set in Y-axis direction:

6. Use care when traversing the tool in Y-axis direction until it touches the workpiece.

Edit Y | ENT

7. Press the soft key "Edit Y" and confirm the input.

R-

8. Here, the tool radius has to be considered. Select the tool radius compensation R-.

In step 3 the reference point shall be set in Z-axis direction:

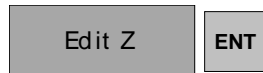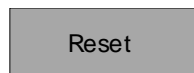9. Use care when traversing the tool in Z-axis direction until it touches the workpiece.

10. Press the soft key "Edit Z" and confirm the input.

Edit Z | ENT

11. Here, the tool radius must not be considered (see chapter "Tool radius compensation").

Reset

Use the soft key "Reset" to delete a reference point already defined.

Zero Offset

| Edit X | Edit Y | Edit Z |

| T1   P ? | Set zero offset | 06/21/07 |

X $\quad$ 0,00 mm

Y $\quad$ 0,00 mm

Z $\quad$ 0,00 mm

Enter values for zero offset

| Edit X | Edit Y | Edit Z | Reset | Back |

Zero offset:
Use the soft keys "Edit X", "Edit Y" and "Edit Z" to enter the values of the zero offset.
Purpose: The position display shows the position of the tool (N_1) in reference to the workpiece zero point (W).
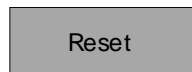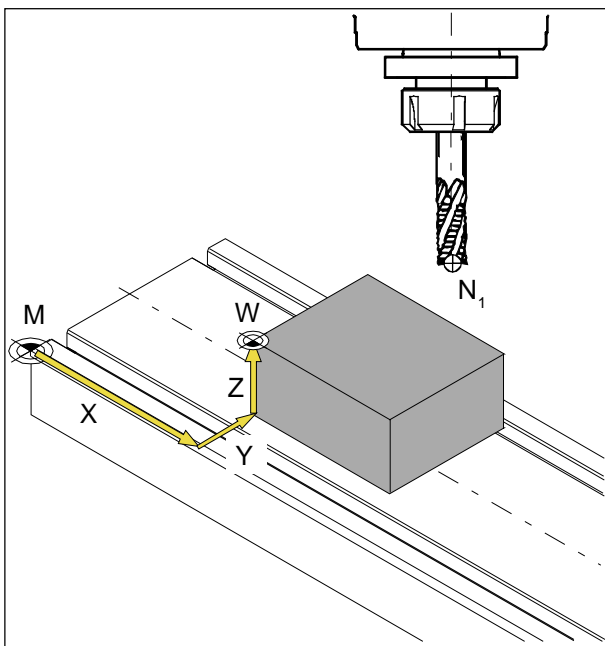
Reset

Use the soft key "Reset" to cancel the zero offset.

EMCO

Setting axes to zero

Without activated tool
All axes of the path display can be defined with any other value (e.g. 0).

The "Setting axes to zero" without activated tool can be cancelled by:
• overwriting the path displays with another value
• **selection** of a tool
• referencing the axes again
• switching the machine off

Note:
The "Setting axes to zero" does not influence any reference point that has been set, since a tool has to be selected before. When a new tool is selected, the "Setting axes to zero" is cancelled.

Storing of contour point

The contour points P serve as ancillary points with workpieces which always remain equal.
The prototype has to be machined only once by means of the path displays.
The remaining workpieces are machined simply by approaching the stored contour points.

Note:
A maximum of 999 contour points can be saved (standard setting 99 contour points; can be set with parameter "Contour points").

General:

Note:
In case the soft key "Contour" is not displayed in the soft-key row, there is still a working pattern (circular hole pattern or linear hole pattern) selected.

Contour

Activation of the soft key "Contour" in the soft-key row:

REF

1. Enter the reference mode (press the REF key).

Contour

2. Press the soft key "Contour".

T ?	P ?	Ref-Mode	06/21/07

Contour point:

Enter number of contour point

| | Edit | | Activate | Back |

Activate

3. Press the soft key "Activate".

Back

4. Exit the reference mode by pressing the soft key "Back" or the ESC key once.

emco

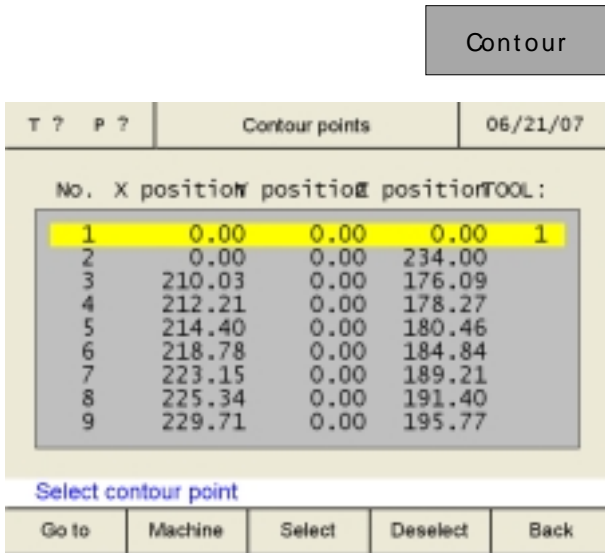## Contour point definition with workpiece sketch

Prior to milling, you can enter all contour points in the contour point table by means of a workpiece sketch.
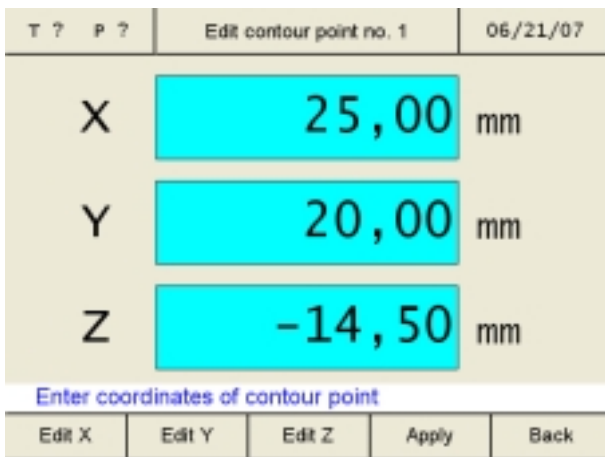
	Contour

1. Call the contour point table.

T ?	P ?	Contour points	06/21/07

No. X position Y position Z position TOOL:

No.	X position	Y position	Z position	TOOL:
1	0.00	0.00	0.00	1
2	0.00	0.00	234.00	
3	210.03	0.00	176.09	
4	212.21	0.00	178.27	
5	214.40	0.00	180.46	
6	218.78	0.00	184.84	
7	223.15	0.00	189.21	
8	225.34	0.00	191.40	
9	229.71	0.00	195.77	

Select contour point

Go to	Machine	Select	Deselect	Back

2. Use the cursor keys or the soft key "Go to" to select the contour point.

	Machine

3. Press the soft key "Machine".

	Edit

4. Press the soft key "Edit".

5. Enter the contour values from the workpiece sketch by means of the soft keys "Edit X", "Edit Y" and "Edit Z".

T ?	P ?	Edit contour point no. 1	06/21/07

X   25,00 mm

Y   20,00 mm

Z   -14,50 mm

Enter coordinates of contour point

Edit X	Edit Y	Edit Z	Apply	Back

Save by means of "Apply".
Before pressing the enter key, incorrect inputs can be corrected with the "CE" key.

Edit Tool

| T ? P ? | Contour points | 06/21/07 |

```
No. X position position position TOOL:

   1     25.00    20.00   -14.50     1
   2      0.00     0.00   234.00
   3    210.03     0.00   176.09
   4    212.21     0.00   178.27
   5    214.40     0.00   180.46
   6    218.78     0.00   184.84
   7    223.15     0.00   189.21
   8    225.34     0.00   191.40
   9    229.71     0.00   195.77
```

Select contour point

| Edit | Edit Tool | Insert | Remove | Back |

6. In addition to the axis coordinates of the contour point, you can also enter the corresponding tool. Press the soft key "Edit Tool" and define the tool number with which the contour point should be approached.
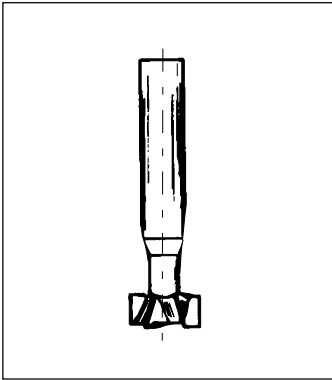
EMCO

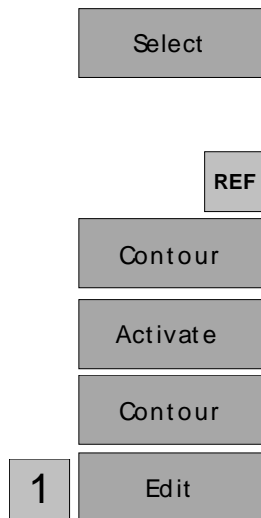# Contour point definition with prototype production

Procedure:
The first workpiece (prototype) is manufactured with the help of the path displays.
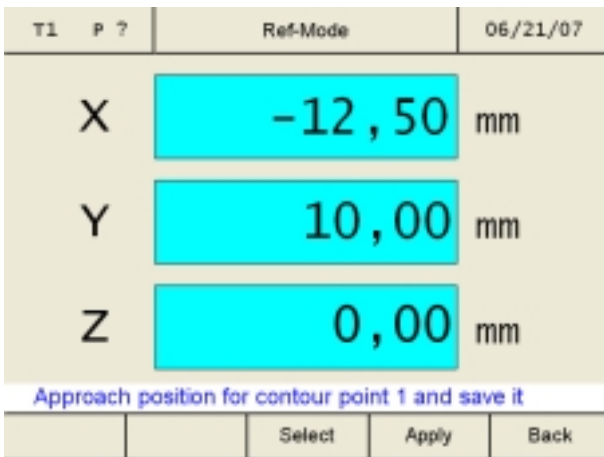The characteristic contour points of the workpiece are saved.
For all further workpieces only the saved contour points will be activated and approached.


Clamp tool 1

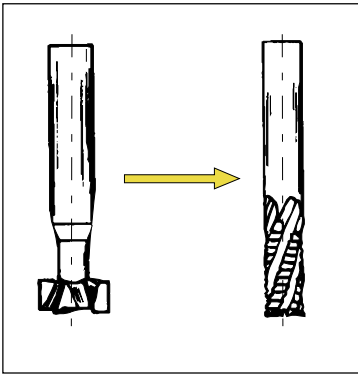|	
	Select
	REF
	Contour
	Activate
	Contour
1	Edit

1. Clamp tool 1 and select a tool in the tool table. The path display indicates the position of the tool tip towards the workpiece zero point.

2. Enter the reference mode (press the REF key).

3. Press the soft key "Contour".

4. Press the soft key "Activate".

5. Press the soft key "Contour".

6. Enter the number of the contour point (here start number 1) and confirm with "Edit".

T1 P ?	Ref-Mode	06/21/07
X	$-12,50$	mm
Y	$10,00$	mm
Z	$0,00$	mm

Approach position for contour point 1 and save it

		Select	Apply	Back

Mill the workpiece until the roughing tolerance is reached.

Apply

Back

7. Mill the 1st contour section until you reach the required dimension. The milling cutter should remain exactly in the end point of the machining operation.
This point should be saved as contour point P_1.
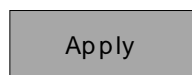
Press "Apply" in order to save the contour point P_1. After having saved it, the next contour point will be automatically called for machining.
In this example for the contour point 2 the tool should be changed to number 2.

8. Exit the reference mode by pressing the soft key "Back" or the ESC key three times.

Tool change

| Select |
| REF |
| Contour |

2 | Edit

9. Clamp tool 2 and select the tool in the tool table. The path display indicates the position of the tool tip towards the workpiece zero point.

10. Enter the reference mode (press the REF key).

11. Press the soft key "Contour".

12. Enter the number of the contour point and confirm with "Edit".

T2	P ?	Ref-Mode	06/21/07

X	21,87	mm
Y	10,00	mm
Z	-14,50	mm

Approach position for contour point 2 and save it

| | | Select | Apply | Back |

| Apply |

| Back |

13. Mill the 2nd contour section until you reach the required dimension. The milling cutter should remain exactly in the end point of the machining operation.
   This point should be saved as contour point P_2.

   Press "Apply" in order to save the contour point P_2. After having saved it, the next contour point will be automatically called for machining.

14. Exit the reference mode by pressing the soft key "Back" or the ESC key three times.

emco

Activating and approaching contour points

Activating and approaching contour points facilitates machining of series of equal work-pieces.
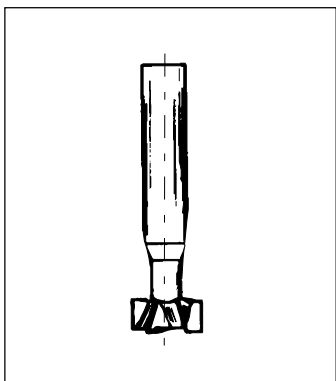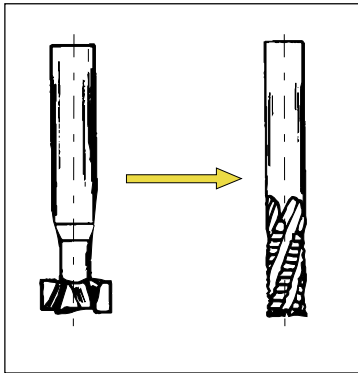

Clamp tool 1

Note:
Make sure that together with the activated (called) contour point P there is always the tool clamped, that was clamped during saving the contour point. Also take care that the corresponding tool is selected.

Procedure:
1. Clamp tool 1 and select the tool in the tool table. The path display indicates the position of the tool tip towards the workpiece zero point.

Contour

2. Call the contour table.

3. Select the contour point by means of the cursor keys or the soft key "Go to".

Select

4. Press the soft key "Select".

Back

5. Press the soft key "Back".

| T1 | P1 | | 06/21/07 |

$\triangle X$ 0,00 mm

$\triangle Y$ 0,00 mm

$\triangle Z$ 0,00 mm

Contour point reached. Select next point with ENTER.

| Parameter | Tools | Zero Offset | Contour | Info |

Approach saved values for P₁

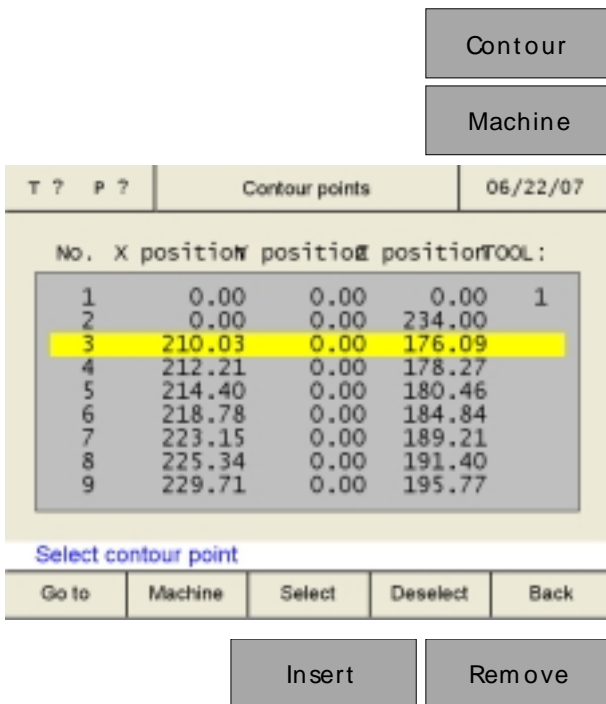6. The pathes to go that still have to be traversed, are displayed in the axis displays on a yellow background with the symbol Δ (see path-to-go display).
Mill the workpiece until the axis displays will show the value "0".
Now the tool is exactly positioned in the saved contour point P$_1$.
In the alarm and message line the message, when the contour point will be reached in X, Y and Z direction, will appear on time.
The area where the message appears can be defined by means of the parameters "Target range X", "Target range Y" and "Target range Z" (see parameter change).

Tool change

ENT

Select

7. Select the next contour point with the "Enter" key.

8. Clamp tool 2 and select the tool in the tool table.

Approach the saved values for P₂

9. The pathes to go that still have to be traversed, are displayed in the axis displays on a yellow background with the symbol **Δ**.
    Mill the workpiece until the axis displays will show the value "0".
    Now the tool is exactly positioned in the saved contour point P₂.
    In the alarm and message line the message, when the contour point will be reached in X, Y, respectively Z direction, will appear on time. The area where the message appears can be defined by means of the parameters "Target range X", "Target range Y" and "Target range Z" (see parameter change).

emco

	Contour		
	Machine		

T ? P ?	Contour points	06/22/07

No.	X position	position	position	TOOL:
1	0.00	0.00	0.00	1
2	0.00	0.00	234.00	
3	210.03	0.00	176.09	
4	212.21	0.00	178.27	
5	214.40	0.00	180.46	
6	218.78	0.00	184.84	
7	223.15	0.00	189.21	
8	225.34	0.00	191.40	
9	229.71	0.00	195.77	

Select contour point

Go to	Machine	Select	Deselect	Back

| Insert	Remove

Insert / remove contour point

1. Press the soft key "Contour".

2. Press the soft key "Machine" .

3. Select the contour point with the cursor keys.

Use the cursor keys to traverse to the respective position. With Insert / Remove contour points can be inserted or removed.

4. Blank lines can be added with the soft key "Insert". Exisiting lines can be deleted with the soft key "Remove".

Quit the contour point mode

Contour
Deselect
Back

1. Press the soft key "Contour".

2. Press the soft key "Deselect".

3. Press the soft key "Back".
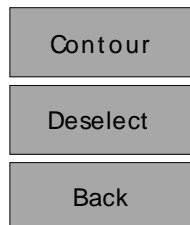
Working patterns

In this chapter the working pattern-functions circular hole pattern and linear hole pattern are described. In the reference mode you select the required hole pattern function by means of the soft key "Pattern" and enter the necessary data. Normally, these data can be taken from the workpiece drawing without any problems (e.g. drilling depth, number of holes). For hole patterns the digital display calculates the position of all holes and displays the input graphic on the screen.

The path-to-go display supports the positioning process during machining: You position easily by "Moving to zero".

REF

Pattern

Circular hole pattern

Circle

1. Press the soft key "Circle".

2. Use the cursor keys to select the input boxes. Enter the values with the soft key "Edit".

   Save with "Apply".
   Before pressing the enter key, incorrect inputs can be corrected by means of the "CE" key.

T ? P ?	Circle Pattern	11/26/09

Centre X [XPOS]	20.000
Centre Y [YPOS]	20.000
Radius [RADIUS]	5.000
Holes [COUNT_P]	6
Start Angle [ANGLE]	0.000
Step Angle [ANGLE_P]	60.000
Use depth [USE]	1
Depth [DEPTH]	8.000

Depth in Z-direction

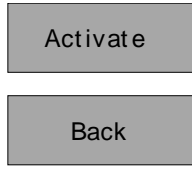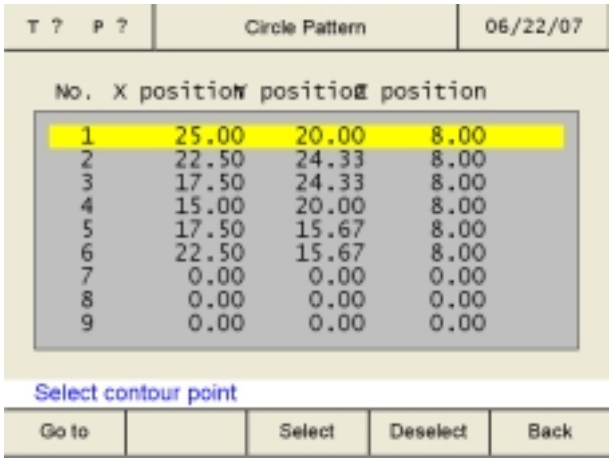| Drawing	Edit	Preview	Activate	Back

Note:

In case the parameter "Use depth [USE] = 0" is selected, the digital display does no longer check the Z-axis position in the point table and the path-to-go display.

T ? P ?	Circle Pattern	11/26/09

				Back

3. Press the soft key "Drawing" to switch over to the input graphics. You return to the input screen by means of the soft key "Back".
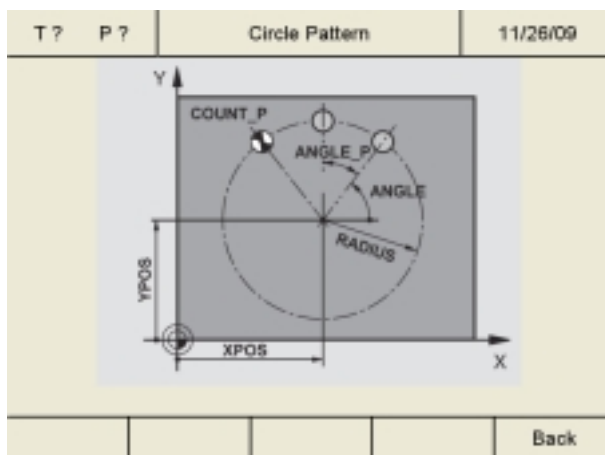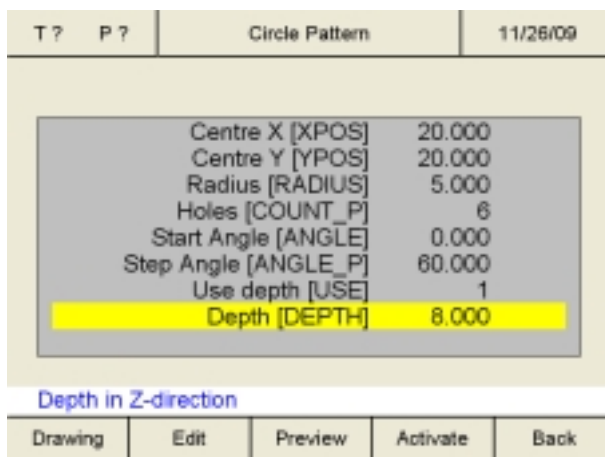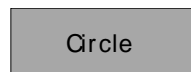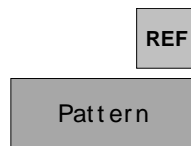
33

emco
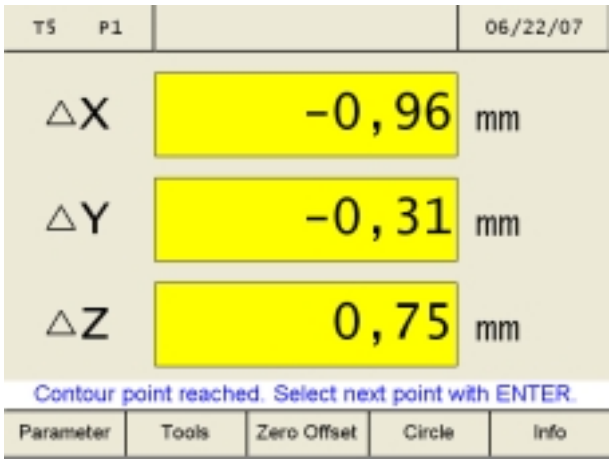
Activate	4. Press the soft key "Activate".
Back	5. Exit the reference mode by pressing the soft key "Back" or the ESC key twice.

T ? P ?	Circle Pattern	06/22/07

No. X position Y position Z position

No.	X position	Y position	Z position
1	25.00	20.00	8.00
2	22.50	24.33	8.00
3	17.50	24.33	8.00
4	15.00	20.00	8.00
5	17.50	15.67	8.00
6	22.50	15.67	8.00
7	0.00	0.00	0.00
8	0.00	0.00	0.00
9	0.00	0.00	0.00

Select contour point

Go to		Select	Deselect	Back

6. Press the soft key "Circle".
The digital display calculates all point positions for the values that have been entered before and enters them into the point table.
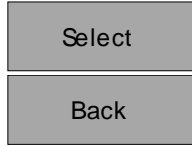
7. Select the contour point by means of the cursor keys or the soft key "Go to".

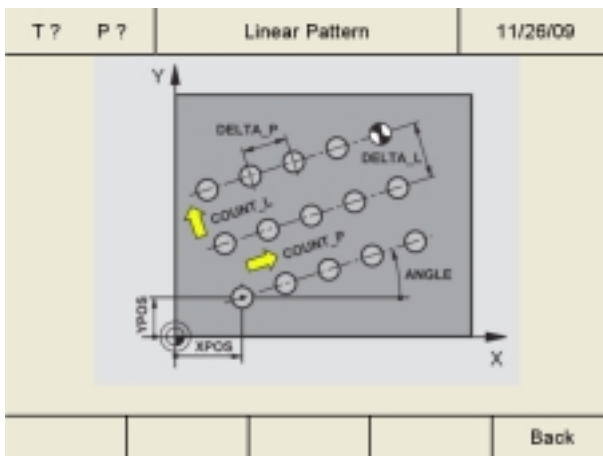| Select	8. Press the soft key "Select".
Back	9. Press the soft key "Back".

10. Clamp a drill and select the tool in the tool table. The path display indicates the position of the tool tip towards the workpiece zero point.

T5 P1		06/22/07

$\triangle X$ **-0,96** mm

$\triangle Y$ **-0,31** mm

$\triangle Z$ **0,75** mm

Contour point reached. Select next point with ENTER.

Parameter	Tools	Zero Offset	Circle	Info

Approach the calculated hole pattern position

11. The pathes to go that still have to be traversed, are displayed in the axis displays on a yellow background with the symbol Δ (see path-to-go display).
Mill the workpiece until the axis displays will show the value "0".
Now the tool is exactly positioned in the first calculated contour point of the hole pattern.
In the alarm and message line the message, when the contour point will be reached in X, Y, respectively Z direction, will appear on time. The area where the message appears can be defined by means of the parameters "Target range X", "Target range Y" and "Target range Z" (see parameter change).

ENT

12. Select the next contour point by means of the "Enter" key.

Linear

Linear hole pattern

1. Press the soft key "Linear".

2. Use the cursor keys to select the input boxes. Enter the values with the soft key "Edit".

   Save with "Apply".
   Before pressing the enter key, incorrect inputs can be corrected with the "CE" key.

T ? P ?	Linear Pattern	11/26/09
	First hole X [XPOS]	5.000
	First hole Y [YPOS]	5.000
	Holes per line [COUNT_P]	10
	Hole spacing [DELTA_P]	5.000
	Angle [ANGLE]	30.000
	Use depth [USE]	0
	Depth [DEPTH]	0.000
	Number of lines [COUNT_L]	2
	Line spacing [DELTA_L]	5.000

X-position of first point

Drawing	Edit	Preview	Activate	Back

Note:
In case the parameter "Use depth [USE] = 0" is selected, the digital display does no longer check the Z-axis position in the point table and path-to-go display.

T ? P ?	Linear Pattern	11/26/09

				Back

3. Press the soft key "Drawing" to switch over to the input graphics. With the soft key "Back" you return to the input screen.

All further working steps are identical to the working steps 4 to 12 of the circular hole pattern.

Exit the working pattern mode

| Circle	Linear

1. Press the soft key "Circle" or "Linear".

Deselect

2. Press the soft key "Deselect".

Back

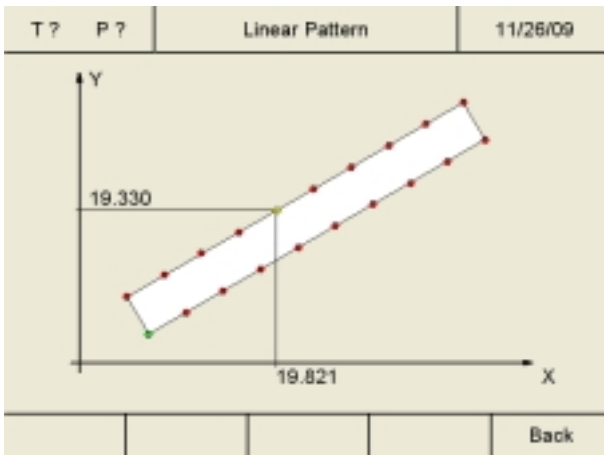3. Press the soft key "Back".

EMCO

Preview for processing sample

The position of all boreholes for the Circular hole pattern and Linear hole pattern can be controlled using the Preview function.
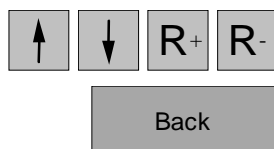
1. Define the desired processing sample ("Circle" or "Linear"), as described above.

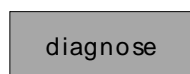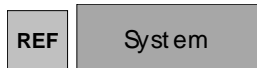2. To change to preview graphics, press the "Preview" softkey.
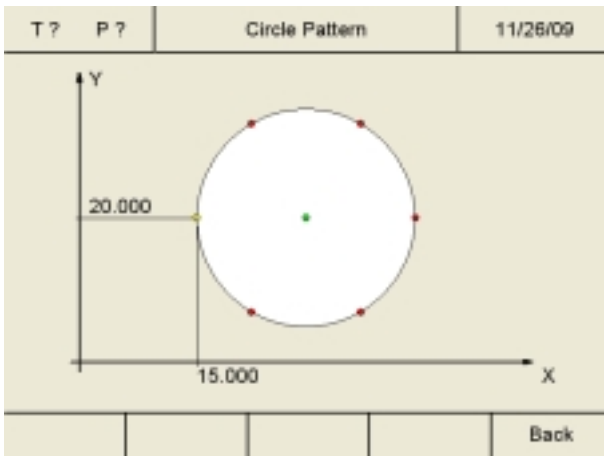
Preview

Example: Borehole sample preview of Circular hole pattern

Example: Borehole sample preview of Linear hole pattern

3. You can check the coordinates of all boreholes of the respective row using the cursor keys and the R+ and R- keys. Press the "Back" softkey to return to the input screen.

REF   System

System

In the system (activate the reference mode) you can export important system data.

T ? P ?	Ref-Mode	07/30/08

X **0,000** mm

Y **0,000** mm

Z **0,000** mm

diagnose	import	export		Back

Diagnosis function

diagnose

In the diagnosis area, the inputs and outputs of the PLC are displayed. A SPS diagnosis can be carried out via the USB interface.
Active alarms and messages can also be displayed.

Data import and export

import   export

With these soft keys it is possible to export machine data, tool data and contour data on an USB stick or to import the data from the USB stick into the digital readout. Alterations due to an import of machine data are only captured after a new start of the machine.

Caution:
Existing data are overwritten during a data import!

37

emco