# CNC 8060 8065

## Quick reference

(Ref: 1408)
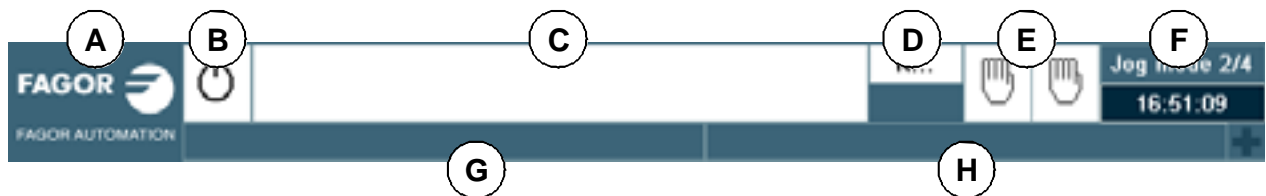
**FAGOR**

# I N D E X

# SCREEN DESCRIPTION.

## General description of the interface.
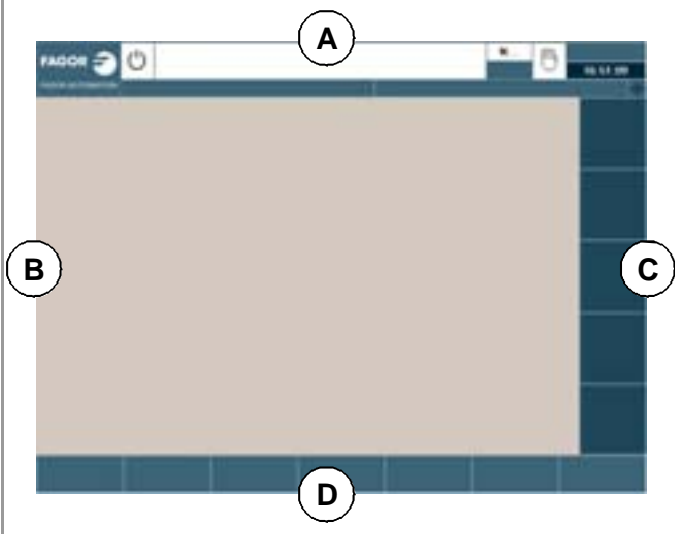


A General CNC-status bar.

B Screen for the active work mode.

C Vertical softkey menu.

D Horizontal softkey menu.

## General description of the interface.



A Icon (customizable) identifying the manufacturer. Clicking with the mouse or pressing on a touch-screen, the CNC shows the task window (same as pressing the keystroke sequence [CTRL]+[A]) that shows the list of the work modes and hotkeys of the CNC.

B Icon showing the status of the program of the active channel:

C Program selected in the active channel for execution. Clicking with the mouse or pressing a touch-screen has the same effect as the [Main-Menu] key, which shows the initial screen of the CNC.

D Number of the block in execution. The bottom icon indicates that the Single-block execution mode is active.

E Number of channels available and active channel (indicated in blue). Icons show which operating mode each channel is in. Clicking with the mouse or pressing a touch-screen to access the desired channel, doing it on the icon of the active channel, has the same effect as the [ESC] key.

F Active work mode (automatic, manual, etc.) selected screen number and total number of screens available. System clock. By clicking on the active work mode, the CNC shows the list of available pages and which ones are visible.

G Active CNC message.

H PLC messages.

## Turn the CNC off.

[ALT][F4]　　　　　　　Turn the CNC off.

# DESCRIPTION OF THE KEYS.

## MONITOR & KEYBOARD.

### Function keys.

| | |
|---|---|
| **F1** | **Softkeys.** Keys F1 through F12 select the options of the softkey menus. |

### Browsing keys.

| | |
|---|---|
| **NEXT** | **NEXT key.** OEM configurable table. |
| **FOCUS** | **FOCUS key.** It is used to switch between the different windows of the screen. |
| **BACK** | **BACK key.** On the horizontal softkey menu, it may be used to go from a softkey submenu up to the previous menu. |

### Help key.

| | |
|---|---|
| **HELP ?** | **HELP key.** Display CNC help. |

### Work modes.

| | |
|---|---|
| **AUTO** | Automatic mode. |
| **MANUAL** | Jog mode. |
| **EDIT** | EDISIMU mode. |
| **MDI** | MDI/MDA mode. |
| **TABLES** | User tables (zero offsets, fixtures and arithmetic parameters). |
| **TOOLS** | Tool and magazine table. |
| **UTILITIES** | Utilities mode. |
| **CUSTOM** | Configurable mode. OEM configurable table. |

### Browsing keys.

| | |
|---|---|
| **MAIN MENU** | Main menu. |
| | Changing the state of an icon. In the MC/TC mode, it toggles between the standard and the auxiliary screens. |

### Keys to move the cursor.

| | |
|---|---|
| **← →** **↑ ↓** | The arrow keys move the cursor one position to the left, right, up or down. |
| | The previous-page or next-page keys show the previous or next page at the part-program or PLC program editor. |
| **HOME END** | The home and end keys move the cursor the beginning or end of the line. |
| | The tab key moves the cursor to the next field of the active menu. |

### Editing keys.

| | |
|---|---|
| **←** | Delete. |
| **DEL** | Delete. |
| **INS** | Insert or overwrite. |
| **ESC** | Escape key, to cancel the current action without assuming the changes. |
| **ENTER** | Key to validate commands, data and program blocks of the editor. |
| **RECALL** | Recover data. |
| **CALC** | Calculator (*). |

(*) The calculator key is not available on all keyboards.

## JOG PANEL.

### Turn the CNC off.

| | |
|---|---|
| **CNC OFF** | Turn the CNC off. |

### Jog keyboard for jogging the axes.

| | | |
|---|---|---|
| **X+** | **7+** | Keys to select axes and jog them in the positive direction. |
| **X-** | **7-** | Keys to select axes and jog them in the negative direction. |
| **X** | **7** | Keys to select the axes and keys to select the jogging direction. Both keys (axis and direction) must be pressed to jog the axis. |
| **+** | **−** | |
| **∿** | | Rapid key. When pressing this key while moving an axis, the CNC applies the rapid feedrate. |

### Feed selectors.
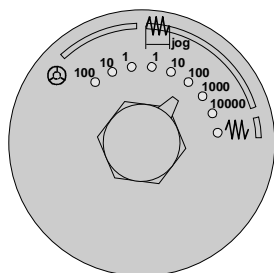
Sector for the type of jog; continuous / incremental jog or handwheels.

- In handwheel mode, it selects the multiplying factor for the handwheel pulses (x1, x10 o x100).
- In incremental mode, it selects the incremental value of the axis movements.
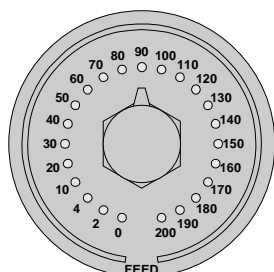
Selector of percentage of feedrate override, between 0% and 200%, for jog and automatic movements.

### Execution keys.

| | |
|---|---|
| **｜** | Cycle start key (START). Execute the selected program in automatic mode, a block in MDI/MDA mode, etc. |
| **○** | Cycle stop key (STOP). Interrupt the execution of the CNC. |
| **RESET ⫽** | Reset key. It initializes the system setting the initial conditions as defined by machine parameters. |
| **SINGLE ⇥** | Single-block execution mode. When selecting the "single block" execution mode, the program simulation will be interrupted at the end of each block. |
| **ZERO** | Home search. |

### Spindle control.

| | |
|---|---|
| **⤵** | Start the spindle clockwise. |
| **▽** | Stop the spindle. |
| **↺** | Start the spindle counteclockwise. |
| **+** **−** | To vary the spindle speed percentually. |
| **⟲** | Spindle orientation. |
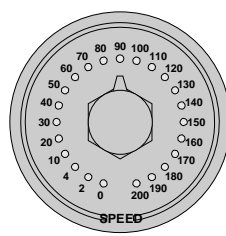
Selector of percentage of spindle speed override between 0% and 200%.

# KEYBOARD SHORTCUTS.
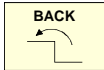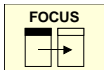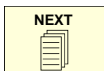
## Operations at the interface.

**[CTRL] + [W]**

Minimize/Maximize the CNC.

**[CTRL] + [J]**

Show / hide the virtual operator panel.

**[CTRL] + [M]**

Show / hide the PLC message list.

**[CTRL] + [O]**

Show / hide the CNC message list.

**[ALT] + [W]**

Show / hide the window for errors and warnings.

**[ALT] + [F4]**

Turn the CNC off.

## Work modes.

**[CTRL] + [A]**

To show the task window.

| | |
|---|---|
| **[CTRL] + [SHIFT] + [F1]** | **MAIN MENU** |
| Main menu. | |
| **[CTRL] + [F6]** | **AUTO** |
| Automatic mode. | |
| **[CTRL] + [F7]** | **MANUAL** |
| Jog mode. | |
| **[CTRL] + [F9]** | **EDIT** |
| EDISIMU mode. | |
| **[CTRL] + [F8]** | **MDI** |
| MDI/MDA mode. | |
| **[CTRL] + [F10]** | **TABLES** |
| User tables. | |
| **[CTRL] + [F11]** | **TOOLS** |
| Tool and magazine table. | |
| **[CTRL] + [F12]** | **UTILITIES** |
| Utilities mode. | |
| **[CTRL] + [K]** | **CALC** |
| Calculator. | |

## Browsing keys.

| | |
|---|---|
| **[CTRL] + [F1]** | **BACK** |
| Previous menu. | |
| **[CTRL] + [F2]** | **FOCUS** |
| Switch window. | |
| **[CTRL] + [F3]** | **NEXT** |
| Switch screens. | |
| **[ALT]+[B]** | |
| Two-colored key. | |

## Execution keys.

| | |
|---|---|
| **[CTRL]+[S]** | |
| Cycle start key (START). | |
| **[CTRL]+[P]** | |
| Cycle stop key (STOP). | |
| **[CTRL]+[R]** | **RESET** |
| Reset key. | |
| **[CTRL] + [B]** | **SINGLE** |
| Single-block execution mode. | |

The shortcuts for the [START] [STOP] and [RESET] keys are only available when the CNC is installed as simulator on a PC.
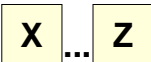
# JOG MODE.

| Softkey. | Description. |
|---|---|
|  | Change the units for data display (mm or inches). For programming, the CNC assumes the units defined with the active function G70 or G71, or, when not programmed, the units set by the machine manufacturer (INCHES parameter). |
|  | Setting and activating the zero offsets and the fixture offsets. This softkey shows the zero offsets and the fixture offsets of the system, either to store the active zero offset or to activate a new zero offset. |
|  | Tool calibration (·M· model). |
|  | Tool calibration (·T· model). |
|  | Part centering (·M· model). |

# HOME SEARCH.

**Manual home search (one axis at a time).**

The axis-by-axis home search cancels the zero offset, the fixture offset and the measuring offset. The CNC assumes the machine reference zero point (home) as the new part zero.

| Keyboard. | | Softkey menu. | |
|---|---|---|---|
| **X** ... **Z** | 1 Select the axis to be homed (on alphanumeric keyboard). The CNC will highlight the coordinate of that axis. | 1 | Press the home search softkey to show the list of axes of the channel. |
| ZERO | 2 Press the homing key [ZERO]. The CNC will display the symbol "1" in the numeric area. | 2 | Select the axis to be homed on the softkey menu. The CNC will highlight the coordinate of that axis and will show the symbol "1" in the numeric area. |
| \| | 3 Press [START] to go ahead with the home search or [ESC] to cancel the operation. | 3 | Press [START] to go ahead with the home search or [ESC] to cancel the operation. |
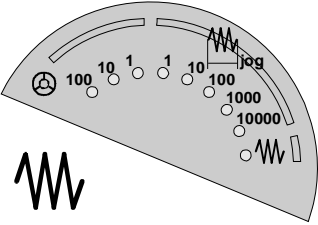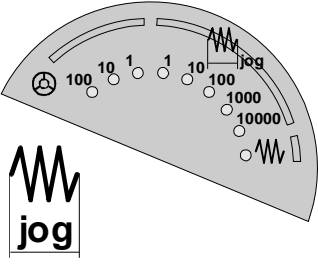
**Automatic home search (with subroutine).**

| Keyboard. | | Softkey menu. | |
|---|---|---|---|
| ZERO | 1 Press the homing key [ZERO]. The CNC will display the symbol "1" in the numeric area. | 1 | Press the home search softkey to show the list of axes of the channel. |
| \| | 2 Press [START] to go ahead with the home search or [ESC] to cancel the operation. | 2 | Select the "All" option on the softkey menu. |
| | | 3 | Press [START] to go ahead with the home search or [ESC] to cancel the operation. |
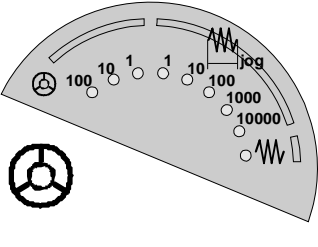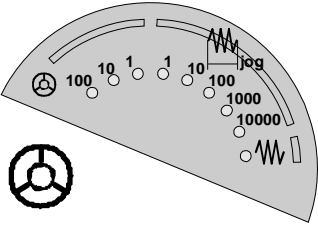
# MOVE THE AXES.

## JOG keypad.

| | | |
|---|---|---|
| X+ 7+ | Select an axis and move it in the positive direction. |
| X- 7- | Select an axis and move it in the negative direction. |
| X 7 + − | Keys to select the axes and keys to select the jogging direction. Both keys (axis and direction) must be pressed to jog the axis. |
| ∿ | Move the axis in rapid. |

## Jog

| | |
|---|---|
| | **Movement in continuous jog.** |
| | In continuous jog, the axes keep moving while the jog keyboard is acted upon. |
| | **1** Turn the jog selector switch to the continuous JOG position. |
| | **2** Jog the desired axis using the JOG panel (keypad). |
| | **Movement in incremental jog.** |
| | In incremental jog, the axis moves a specific distance every time the key is pressed. |
| | **1** Turn the jog selector switch to one of the incremental jog positions. |
| | **2** Jog the desired axis using the JOG panel (keypad). Every time the JOG panel is acted upon, the axis will move the distance indicated on the dial of the jog selector switch. |
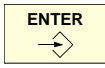
## Jogging the axes with handwheels.

| | |
|---|---|
| | **General handwheel (it may be used to jog any axis of the machine).** |
| | **1** Turn the jog selector switch to one of the handwheel positions. |
| | **2** Select the axis or axes to be jogged on the jog keyboard. The CNC will highlight the selected axes. |
| | **3** Once the axis has been selected, the CNC will move it as the handwheel is turned depending on the setting of the selector switch and on the turning direction of the handwheel. |
| | **Individual handwheel (it is associated with a particular axis).** |
| | **1** Turn the jog selector switch to one of the handwheel positions. |
| | **2** The CNC moves each axis as its relevant handwheel is turned depending on the setting of the selector switch and on the turning direction of the handwheel. |

## COORDINATE PRESET.

The [ESC] key may be used to cancel the operation at any time.

| Keyboard. | | Softkey menu. | |
|---|---|---|---|
| **X** ... **Z** | **1** Select the axis to be preset (on alphanumeric keyboard). The CNC will highlight the coordinate of that axis. | **1** Press the softkey for presetting axes to show the list of axes of the channel and select an axis. The CNC will highlight the coordinate of that axis. |
| | **2** Key in the desired preset value. | **2** Key in the desired preset value. |
| **ENTER** | **3** Press [ENTER] to assume the entered value. | **3** Press [ENTER] to assume the entered value. |

## MOVE AN AXIS TO A PARTICULAR POSITION.

The [ESC] key may be used to cancel the operation at any time.

| Keyboard. | | Softkey menu. | |
|---|---|---|---|
| **X** ... **Z** | **1** Select the axis to be moved (on alphanumeric keyboard). The CNC will highlight the coordinate of that axis. | **1** Press the softkey for presetting axes to show the list of axes of the channel and select an axis. The CNC will highlight the coordinate of that axis. |
| | **2** Enter the coordinate of the target point. | **2** Enter the coordinate of the target point. |
| ❘ | **3** Press [START] to make the move. | **3** Press [START] to make the move. |

## SET THE FEEDRATE, SPEED OR TOOL.
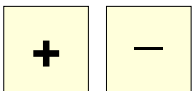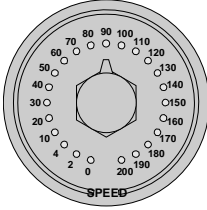
| **Feedrate.** | |
|---|---|
| **F** | **1** Press [F] at the alphanumeric keyboard.<br>**2** Enter the new feedrate. |
| ❘ | **3** Press [START] to assume the entered value or [ESC] to cancel the operation. |

| **Speed.** | |
|---|---|
| **S** | **1** Press [S] at the alphanumeric keyboard until selecting the desired spindle. When pressing this key for the first time, the CNC will highlight the relevant data indicating that it is selected.<br>**2** Enter the new spindle speed. |
| ❘ | **3** Press [START] to assume the entered value or [ESC] to cancel the operation. |

| **Tool.** | |
|---|---|
| **T** | **1** Press [T] on the alphanumeric keyboard.<br>**2** Enter the tool to be selected. |
| ❘ | **3** Press [START] to assume the entered value or [ESC] to cancel the operation. |

# MASTER SPINDLE CONTROL.

| | | | |
|---|---|---|---|
| (icon) | Start the spindle clockwise (same as M03 function) at the active speed. | (icon) | Stop the spindle (same as M05 function). |
| (icon) | Start the spindle counterclockwise (same as M04 function) at the active speed. | (icon) | Orient the spindle (same as M19 function). |

**Vary the speed override from the operator panel.**

With the operator panel, it is possible to change the percentage of spindle speed using the jog keyboard or a switch (depending on model).

| | |
|---|---|
| + − | **JOG keypad.** Increases or decreases the percentage of spindle speed. The maximum and minimum values as well as the incremental step are set by the OEM, the typical values being a variation between 50% and 120% with a 5% step. |
| (speed switch dial) | **Switch.** It sets the percentage of turning speed to be applied. The maximum and minimum values are set by the OEM, the typical values being a variation between 50% and 120%. |

# TOOL CALIBRATION.

| | | | |
|---|---|---|---|
| (icon) | Tool calibration in a mill model. | (icon) | If there is no tabletop probe, only manual calibration is available. All types of calibration are available when using a table-top probe. The different calibration methods may be selected from the vertical softkey menu. |
| (icon) | Tool calibration in a lathe model. | (icon) | |

- Manual calibration. (Calibration without probe).

In this mode, only the active tool can be calibrated. Since there is no probe, a reference part is required to calibrate the tool. All the movements are carried out manually.

- Semi-automatic calibration. (Calibration with probe).

The positioning movements are carried out manually and the CNC executes the probing movements.

- Automatic calibration. (Calibration with probe).

The CNC executes all the movements using the calibration canned cycle #PROBE.

**Probe selection.**

The CNC uses the active probe for calibration. The active probe may be changed via part-program or MDI using the instruction #SELECT PROBE.

#SELECT PROBE [1]          #SELECT PROBE [2]

## Manual calibration. Calibration without a probe.

All the movements are carried out manually. Since there is no probe, a reference part is required to calibrate the tool. The calibration consists in moving the tool manually until it touches the part and then validating the calibration on each axis. In this mode, only the active tool can be calibrated.

- Milling model.                 Calibrate the length of the endmills and the offsets of the lathe tools.
- Lathe model (plane).       Calibrate the offset of any tool.
- Lathe model (trihedron).  Calibrate the length or offsets of the endmills and the offsets of the lathe tools.

Tool calibration steps.

1 Define the dimensions of the reference part being used in the calibration.
2 Define the tool and the offset to be calibrated and press [START] to execute the tool change (if [ENTER] is pressed, the CNC will only show the tool data).
3 Calibrate the tool. Approach the tool manually until touching the part and then validate the calibration using the softkey menu. After validating the calibration, it updates the values and initializes the wear value to zero. Them, the new values are saved in the tool table.
4 Press [START] for the CNC to assume the new values of the offset.

## Semi-automatic calibration. Calibration with a probe.

The positioning movements are carried out manually and the CNC executes the probing movements. The CNC will move the tool on the selected axis until it touches the probe and validates the calibration only on that axis. In this mode, only the active tool can be calibrated.

- Milling model.                 Calibrate the length or radius of the endmills and the offsets of the lathe tools.
- Lathe model.                   Calibrate the offset of any tool.

Tool calibration steps.

1 Define the probing distance and feedrate. If the feedrate is not defined, the probing movement will be made at the feedrate set by the OEM.
2 Define the tool and the offset to be calibrated and press [START] to execute the tool change (if [ENTER] is pressed, the CNC will only show the tool data).
3 Manually approach the tool to the probe until it is placed on the path that will be used for probing. To calibrate the radius with a cylindrical probe, the path must coincide with the probe's center point; if not, the radius will be calculated wrong.
4 Calibrate the tool. Select the axis and the probing direction on the softkey menu and press [START]. The probe moves in parallel to the axis and in the selected direction until touching the probe. It updates the measured value and resets the wear value to zero. The data is stored in the tool table.
5 Once the tool has been calibrated, the CNC shows a message proposing to press [START] so the CNC assumes the new values of the offset. When pressing [START] while this message is displayed, the CNC assumes the new values of the offset; if the message is not displayed, pressing [START] executes the probing movement again.

**Automatic calibration. Calibration with a probe and a canned cycle.**

The calibration is done using a probing canned cycle. The CNC moves the tool until touching the probe and validates the calibration on each axis. This mode may be used to calibrate any tool.

- Milling model.      Calibrate the length, the radius or offsets of the endmills and the offsets of the lathe tools.

- Lathe model (plane).      Calibrate the offset of any tool.

- Lathe model (trihedron).      Calibrate the length, the radius or offsets of the endmills and the offsets of the lathe tools.

Tool calibration steps.

1   Select the tool and the offset to be calibrated.

2   Define the data defining the calibration.

3   Press the [CYCLE START] key to start the calibration. The CNC calibrates the tool making all the necessary movements; there is no need to manually approach the tool. If necessary, the CNC makes the tool change.

4   After the calibration It updates the tool table data. Also, the CNC assumes the new values.

# MDI/MDA MODE.

**Edit new blocks.**

- In MDI mode, the edit line is always visible.

- In MDA mode, one must select the "new block" option from the softkey menu.

**Modify a block from the block history.**

- In MDI mode, use the [⬆][⬇] keys to pop-up the history and scroll it. The [ENTER] key restores from the history the block selected with the cursor and insert it in the edit line.

- In MDA mode, use the [⬆][⬇] keys, select a block from the history and use the "modify" option from the softkey menu (or the [ENTER] key to copy it into the edit line.

**Execute blocks.**

- The [START] key executes the block currently displayed on the editing line. Once the block has been executed is saved in the block history.

- The [STOP] key interrupts the execution of the block. Press [START] again to resume execution from where it was interrupted.



     While the execution is interrupted, the "CANCEL" softkey cancels the execution of the block while keeping the programmed machining conditions (it does not do a general reset of the CNC).

- The [RESET] key cancels the execution of the block and resets the CNC to its initial conditions.

# AUTOMATIC MODE.

| Softkey. | Description. |
|---|---|
|  | Select a program for execution. |
|  | Begin tool inspection. Tool inspection is only available when program execution is interrupted. |
|  | End simulated execution and start executing the program |
|  | Select the program that is being edited. |

## PROGRAM EXECUTION.

**Select a program.**

Each channel executes the program selected in it. To select a program, press one of the following softkeys of the vertical menu.

| | |
|---|---|
|  | This softkey shows the list of available programs. |
|  | This softkey directly selects the program of the EDISIMU mode. |

**Execute a program.**

The name of the program selected in the channel for execution appears on the general status bar. If not indicated otherwise, the program execution will begin from the first block of the program to the execution of one of the end-of-program functions "M02" or "M30". As an option, it is possible to define the first and last blocks of the execution.

| | |
|---|---|
|  | To start the execution of the program, press [START] on the Operator Panel. |
|  | The [STOP] key interrupts the execution of the program. Press [START] again to resume execution from where it was interrupted. |
| RESET | The [RESET] key cancels the execution of the program and resets the CNC to its initial conditions. |
| SINGLE | Single-block execution mode. The program may be executed in –single block– or –automatic– mode; the mode may be selected even while executing the program. |

# EXECUTE BLOCKS SEPARATELY.

Press the "EXBLK" softkey of the horizontal menu. Being this option active, every time the [START] key is pressed, it only executes the block selected in the active program. Once that block is executed, another block may be executed by selecting it with the cursor and pressing [START] again and so on. Blocks executed like this change the history of the M and G functions.

# SIMULATED EXECUTION OF A PROGRAM.

With simulated execution, it is possible to simulate a program, interrupt it at a point and start execution from that point on. Depending on the type of simulation selected, it can involve movement of axes, spindle, etc.

|  | Tool path | Movement of the axes | Spindle control | Send to the PLC M-H-S-T | G04 | M00 M01 |
|---|---|---|---|---|---|---|
| Theoretical travel. | Programmed tool path | No | No | No | Yes | Yes |
| G functions. | Tool center | No | No | No | Yes | Yes |
| G M S T functions. | Tool center | No | No | Yes | Yes | Yes |
| Main plane. | Tool center | Yes (plane) | Yes | Yes | No | Yes |
| Rapid. | Tool center | Yes | Yes | Yes | No | Yes |
| Rapid [S=0]. | Tool center | Yes | No | Yes | No | Yes |

**Start the simulation of the program**

1 On the horizontal softkey menu, select the desired type of simulation.

2 If necessary, set the desired simulation conditions (first and last block)

3 Press the [START] key to start the simulation. The program may be simulated in "single block" or "continuous" mode; the mode may be selected even while simulating the program.

**End simulation and start executing the program.**

1 Press the [STOP] of the operator panel to interrupt the simulation. Once the program is interrupted, simulation may be resumed with the [START] key or switch to execution mode from the vertical softkey menu.

2 When switching to execution mode (after pressing the softkey), the CNC goes into tool inspection to reposition the axes, modify program conditions, etc. To complete the tool inspection and before starting the execution of the program, the spindle turning direction must be restored and the axes repositioned. The vertical softkey menu offers two options.

• Repositioning the axes at the interruption point.

• Repositioning the axes at the starting point of the interrupted block.

3 Press the [START] key to start the execution.

# BLOCK SEARCH.

With block search, it is possible to recover the program history up to a particular block in such way that if program execution is resumed at that block, it will do so with the same conditions as if it were executed from the beginning.

- The automatic block search may be used to recover the program history up to the block where the previous execution was canceled. The CNC remembers the block where interruption was canceled, thus not being necessary to set the stop block.

- The manual block search may be used to recover the program history up to a particular block of the program or of the subroutine, set by the operator.

### Executing block search.

1 Selecting the type of search: automatic or manual.

2 Selecting the stop block. In the automatic block search, there is no need to select the stop block; by default, the CNC runs the search up to the block where the program was interrupted.

3 Selecting the starting block for the search. If the first block is not selected, the block search starts at the beginning of the program.

4 Press the [CYCLE START] key to start the block search.

5 Depending on how the treatment of functions M, H, F, S is configured, it may be necessary to decide which ones are sent out to the PLC.

6 Reposition the axes to the point to resume execution.

7 Tool inspection may be accessed to change the machining conditions.

8 Press [START] to execute the program.

### Repositioning the axes.

Once the block search is finished, the CNC will show the axes that are out of position. The axes may be repositioned individually or several at the same time in one of the following ways:

- Manual repositioning of axes. Jog the axes with the handwheels or with the JOG keys. The movement is limited by the repositioning end point and the corresponding software limit.

- Automatic repositioning of axes. Select the axes with the relevant softkey and press [START]. Repositioning may be interrupted (using the [STOP] key) to select other axes.

### Changing the machining conditions.

After positioning the axes and before resuming execution, tool inspection may be accessed to change the machining conditions. In tool inspection, it is possible to change the feedrate and the spindle speed, execute blocks in MDI/MDA mode as well as activate M and H functions.

# TOOL INSPECTION.

**Begin tool inspection.**

Tool inspection may be accessed from the vertical softkey menu only when the execution of the program has been interrupted ([STOP] key). After activating tool inspection, it is possible to jog the axes using the jog keyboard, act upon the master spindle of the channel from the operator panel and execute blocks from the MDI/MDA mode.

**Execute blocks in MDI/MDA mode.**

Any program block may be executed in MDI/MDA mode. The conditions when entering the MDI/MDA mode will be those of the interruption point; i.e. the CNC maintains the history of active G and M functions, feedrate, spindle speed, tool and other commands that were programmed. However, the CNC treats certain functions and commands (type of movements, radius compensation, etc.) differently. Refer to the operating manual.

In general, all the changes made in MDI/MDA mode are kept active when resuming the program after tool inspection except the following functions that are restored at the time of interruption; type of interpolation (G00, G01, G02, G03, G33 or G63), G90/G91 function or #MCS function.

**Repositioning the axes and the spindle.**

To complete the tool inspection and before resuming the execution of the program, the spindle turning direction must be restored and the axes repositioned.

| | | | |
|---|---|---|---|
| | Repositioning the axes at the interruption point. | | Canceling repositioning. |
| | Repositioning the axes at the starting point of the interrupted block. | | |

**Repositioning the axes.**

The CNC allows repositioning the axes either one by one or in groups. Use the vertical softkeys to select the axes to be repositioned and press [START]. The CNC will reposition the axes at the selected point (according to the softkey selected earlier) at the feedrate set by the machine manufacturer. Once one axis has reached its position, it will no longer be available.

**Repositioning the master spindle.**

If the status of the master spindle has changed during the inspection, the softkeys will show the M3, M4, M5 or M19 function to restore. The spindle turning status may be restored either together with the repositioning of the axes or separately. If the spindle was interrupted in a positioning with M19, repositioning will complete that positioning.

**Resuming the execution of the program.**

Once all the axes are repositioned or after canceling repositioning, press [START] to resume program execution.

- If tool inspection has ended by repositioning all the axes, when pressing [START], the CNC completes the interrupted path and goes on with the rest of the program.
- If tool inspection has ended after canceling the repositioning of the axes, when pressing [START], the axes move from their current position to the end point of the interrupted path and then the CNC goes on with the rest of the program.

# EDISIMU MODE.

| Softkey. | Description. |
|---|---|
| | START (simulation). Starts program simulation or resume it if it was interrupted. |
| | STOP (simulation). Interrupt program simulation. Simulation will resume by pressing the START icon. |
| | RESET (simulation). Cancel program simulation. If an error occurs during simulation, reset eliminates the error status and returns the simulation mode to its initial conditions. |
| | Change the channel being displayed for editing and simulation. It does not affect the active channel at the CNC. (This icon will only be available when the CNC has channels). |
| | Select the "single block" or "continuous" execution mode. When "single block" mode is active (the icon will appear pressed), program simulation will be interrupted at the end of each block. |
| | Analyze the program looking for syntax errors. The syntax check is not available for programs written in 8055 CNC language. |
| | Offer an estimate of the total execution time at 100% of the programmed feedrate. The result will appear at the statistics window. |
| | Configuring simulation options. |

## PROGRAM SIMULATION.

**Select a program.**

The "Open program" softkey is used to select a program in EDISIMU mode and may be a new program or an existing one. A different program may be edited and executed in each channel. When selecting this option, the CNC shows a list of the available programs.

1. Select the folder that contains the program. If it is a new program, it will be saved in this folder.
2. Select the program from the list or write its name in the bottom window. To edit a new program, write the name of the program in the lower window and the CNC will open an empty program or a predefined template depending on how the editor is configured.
3. Press [ENTER] to accept the selection and open the program or [ESC] to cancel it and close the program listing.

**Simulating a program.**

1. Choose the type of graphic representation, its dimensions and the point of view. This data may also be modified during the simulation of the program.

2. Activate the desired simulation options using the icon menu.

3. Press the START icon to start the simulation. The simulation may be interrupted with the STOP icon or canceled with the RESET icon.

The simulation of the program starts at the first block of the program and ends after executing one of the end-of-program functions "M02" or "M30". As an option, it is possible to define the first and last blocks of the simulation. To simulate the program, the CNC assumes the real configuration of the spindles of the channel and the configuration of the machine parameters.

"Single block" simulation. The program may be simulated in –single block– or –automatic– mode; the mode may be selected even while simulating the program.

**Simulation options.**

| | | | |
|---|---|---|---|
| **Tool radius compensation.** Activate or cancel tool radius compensation to simulate the program. | | **Software limits.** Activate or deactivate the software limits for program simulation. | |
| **Block skip.** Option to simulate the external "block skip" switch. Being this option active, the CNC does not simulate the blocks containing the block-skip character "/". | | **Conditional stop during simulation.** Option to simulate the external "conditional stop" switch. Being this option active, the CNC interrupts the simulation in the blocks where the "M01" function is programmed. | |
| **Synchronizing spindles.** There is an icon for each spindle where to indicate the spindle to synchronize with. The ·0· value cancels the synchronization. | | **Cancel channel synchronization.** There is one icon per channel that cancels the wait for synchronism with the channel during simulation. | |
| **Assume the active origins for execution.** When starting the simulation or pressing the simulation reset, the CNC applies to the simulation the origins set in the execution environment (for example, the part zero set in jog mode). | | | |

## EXECUTE BLOCKS SEPARATELY.

Press the "EXBLK" softkey of the horizontal menu. Being this option active, every time the START icon key is pressed, it only simulates the block selected in the active program. Once that block is simulated, another block may be simulated by selecting it with the cursor and pressing [START] again and so on.

# USER TABLES.

| Softkey. | Description. |
|---|---|
| | Change the units for data display (mm or inches). For programming, the CNC assumes the units defined with the active function G70 or G71, or, when not programmed, the units set by the machine manufacturer (INCHES parameter). |
| | Initialize the table. Reset all the table data to "0". |
| | Search a text or a value in the table. |
| | Accessing the tables of other channels. With some tables, only the data of the active channel are displayed, this softkey is used to show the tables of the other channels. This softkey will only be available when using channels. |
| | Select the axes to be displayed in the tables. When using several channels, only those axes assigned to the active channel may be accessed. |
| | Save the values of the table into a file. |
| | Restore the values of the table previously saved into a file. |
| | Print the table in the pre-determined printer or save it as a file (prn format) at the CNC. |

**Zero offset tables.**

This table contains the absolute zero offsets and the PLC offset of all the axes and spindles that may be activated as C axis. The zero offsets associated with the possible C axes are always visible, even when the C axis is not active.

- PLC offset (PLC offset). The PLC offset may not be set directly in the table, its values are set from the PLC or via part-program using variables. The CNC always adds the PLC offset to the selected zero offset.
- Absolute zero offsets besides being set directly in the table may also be set from the PLC or via part-program using variables.

  The zero offsets are used to place the part zero at different positions of the machine. To apply an absolute zero offset, it must be activated via program using the relevant function.

**Fixture table.**

This table stores the clamp offsets for each axis.

The fixture offset besides being set directly in the table may also be set from the PLC or via part-program using variables.

The clamp offsets are used to set the position of the clamping system of the machine. To apply a clamp offset it must be activated from the program using the relevant variable.

**Arithmetic parameter tables.**

There are the following arithmetic parameter tables:

- Common parameters. The table is common to all the channels.
- Global parameters. There is a table for each channel.
- Local parameters. There are seven tables for each channel, one table per nesting level (7 levels).

The parameter values may be set directly in the table or from the PLC or via part-program. In this case, the table values are updated after carrying out the operations indicated in the block being executed.

# UTILITIES MODE.

| Softkey. | Description. |
| --- | --- |
| Cut | Cut the selected files onto the clipboard. With this option, when pasting the files to their new location, they are erased from the current folder. |
| Copy | Copy the selected files onto the clipboard. |
| Paste | Paste the files from the clipboard into the selected folder. If the files were placed using the "Cut" option, they will be removed from their original location. |
| Rename | Rename the selected folder or file. |
| Modify M | Change the "modifiable" attribute of the selected files. The attributes column shows the letter -M- indicating that the program may be modified. This attribute is used to protect the files so they can't be modified from the EDISIMU mode. |
| Hidden H | Change the "hidden" attribute of the selected files. The attributes column shows the letter -H- indicating that the program will be hidden (not visible). This attribute allows protecting the files so they are not displayed when selecting a program to be edited or executed. |
| | Encrypt files. Encrypting may be used to protect any file (part-program, subroutine, etc.) making it illegible so it cannot be used by anyone else. |
| Remove | Delete the selected folder or files. The folders can only be deleted if they are empty. |

# PROGRAMMING COMMANDS.

| Command. | Meaning. | Format. |
|---|---|---|
| / | Block skip condition. | |
| # | Programming instructions. | |
| $ | Flow controlling instructions. | |
| % | Program header. | 14 characters. (1) |
| ; (semi-colon) | Block comment . | |
| [   ] | Text type block label. | 14 characters. (1) |
| N | Number type block label. | 0 - 4294967295 |
| G | Preparatory functions. | 1 - 999 |
| X~C | Position of the axes. | ±99999.9999 mm ±9999.99999 inches |
| F | Feedrate of the axes. | |
| S | Spindle speed. | |
| T | Tool number. | 0 - 4294967295 |
| D | Tool offset number. | |
| M | Auxiliary functions. | 0 - 65535 |
| H | Auxiliary functions. | 0 - 65535 |
| NR | Number of block repetitions. | |
| (   ) | Block comment . | |

(1) The format admits upper and lower case letters and numbers (no blank spaces).

# TECHNOLOGICAL FUNCTIONS.

### Machining feedrate.

The machining feedrate may be selected by programmed using the "F" code which remains active until another value is programmed. The programming units depend on the active work mode (G93, G94 or G95) and the type of axis being moved (linear or rotary).

### Spindle speed.

The spindle speed is selected by program using the spindle name followed by the desired speed. The speeds of all the spindles of the channel may be programmed in the same block. The programmed speed stays active until another value is programmed. The programming units will be RPM unless selected otherwise. If G96 is active, the programming units will be m/min.

### Tool number.

The "T" code identifies the tool to be selected. The tools may be in a magazine managed by the CNC or in a manual magazine (referred to as ground tools).

### Tool offset number.

The tool offset contains the tool dimensions. Each tool may have several offsets associated with it. To activate an offset, it must be previously defined. To do that, the CNC offers a section of the tool table where several offsets may be defined.

# LIST OF AUXILIARY (MISCELLANEOUS) –M-FUNCTIONS.

| Function. | Meaning. |
|---|---|
| M00 | Program stop. |
| M01 | Conditional program stop. |
| M02/M30 | End of program. |
| M03 | Start the spindle clockwise. |
| M04 | Start the spindle counteclockwise. |
| M05 | Stop the spindle. |
| M06 | Tool change. |
| M17/M29 | End of a global or local subroutine. |
| M19 | Spindle orientation. |
| M41-M44 | Gear change. |

# LIST OF –G- FUNCTIONS.

·M· Modal function. Those cases indicated with "!", mean the function remains active even after an M02, M30 or a reset and after the CNC is powered off and back on.

·D· Default function. Those cases indicated with "?" mean that the default quality of the function depends on the settings of the CNC machine parameters.

·V· This function is displayed in the G-code history.

| Function | M | D | V | Meaning. |
|---|---|---|---|---|
| G00 | * | ? | * | Rapid traverse. |
| G01 | * | ? | * | Linear interpolation. |
| G02 | * |  | * | Clockwise circular (helical) interpolation. |
| G03 | * |  | * | Counterclockwise circular (helical) interpolation. |
| G04 |  |  | * | Dwell. |
| G05 | * | ? | * | Controlled corner rounding (modal). |
| G06 |  |  | * | Arc center in absolute coordinates (not modal). |
| G07 | * | ? | * | Square corner (modal). |
| G08 |  |  | * | Arc tangent to previous path. |
| G09 |  |  | * | Arc defined by three points. |
| G10 | * | * |  | Cancel mirror image on all the axes. |
| G11 | * |  | * | Mirror image on the abscissa axis. |
| G12 | * |  | * | Mirror image on the ordinate axis. |
| G13 | * |  | * | Mirror image on the axis perpendicular to the plane. |
| G14 | * |  | * | Activate or cancel mirror image on a axis. |
| G17 | * | ? | * | Main plane formed by the first axis (abscissa), second (ordinate) and third axis (perpendicular) of the channel. |

| Function | M | D | V | Meaning. |
|---|---|---|---|---|
| G18 | * | ? | * | Main plane formed by the third axis (abscissa), first axis (ordinate) and second axis (perpendicular) of the channel. |
| G19 | * | | * | Main plane formed by the second axis (abscissa), third axis (ordinate) and first axis (perpendicular) of the channel. |
| G20 | * | | * | Select any work plane formed by the first three axes of the channel. |
| G30 | | | * | Polar origin preset. |
| G31 | | | * | Temporary polar origin shift to the center of arc. |
| G33 | * | | * | Electronic threading with constant pitch. |
| G36 | | | * | Automatic radius blend. |
| G37 | | | * | Tangential entry. |
| G38 | | | * | Tangential exit. |
| G39 | | | * | Automatic chamfer blend. |
| G40 | * | * | | Cancel tool radius compensation. |
| G41 | * | | * | Left-hand tool radius compensation. |
| G42 | * | | * | Right-hand tool radius compensation. |
| G45 | | | | Turn tangential control on and off. |
| G50 | * | ? | | Semi-rounded corner. |
| G53 | * | | | Cancel zero offset. |
| G54 | ! | | * | Zero offset. Absolute zero offset 1. |
| G55 | ! | | * | Zero offset. Absolute zero offset 2. |
| G56 | ! | | * | Zero offset. Absolute zero offset 3. |
| G57 | ! | | * | Zero offset. Absolute zero offset 4. |
| G58 | ! | | * | Zero offset. Absolute zero offset 5. |
| G59 | ! | | * | Zero offset. Absolute zero offset 6. |
| G60 | | | * | Square corner (not modal). |
| G61 | | | * | Controlled corner rounding (not modal). |
| G63 | * | | * | Rigid tapping. |
| G66 | | | * | (·T· model). Pattern repeat canned cycle. |
| G68 | | | * | (·T· model). Stock removal cycle along X axis. |
| G69 | | | * | (·T· model). Stock removal canned cycle along Z axis. |
| G70 | * | ? | * | Programming in inches. |
| G71 | * | ? | | Programming in millimeters. |
| G72 | * | | * | Scaling factor. |
| G73 | * | | * | Coordinate system rotation. |
| G74 | | | * | Home search. |
| G80 | * | * | | (·M· model). Cancel the canned cycle. |
| G81 | * | | * | (·M· model). Drilling canned cycle. |
| G81 | | | * | (·T· model). Turning canned cycle with straight sections. |
| G82 | * | | * | (·M· model). Drilling canned cycle with a variable peck. |
| G82 | | | * | (·T· model). Facing canned cycle with straight sections. |
| G83 | * | | * | (·M· model). Deep-hole drilling canned cycle with constant peck. |
| G83 | | | * | (·T· model). Drilling / tapping canned cycle. |
| G84 | * | | * | (·M· model). Tapping canned cycle. |

| Function | M | D | V | Meaning. |
|----------|---|---|---|----------|
| G84 | | | * | (·T· model). Turning canned cycle with arcs. |
| G85 | * | | * | (·M· model). Reaming canned cycle. |
| G85 | | | * | (·T· model). Facing canned cycle with arcs. |
| G86 | * | | * | (·M· model). Boring canned cycle. |
| G86 | | | * | (·T· model). Longitudinal threading canned cycle. |
| G87 | * | | * | (·M· model). Rectangular pocket canned cycle. |
| G87 | | | * | (·T· model). Face threading canned cycle. |
| G88 | * | | * | (·M· model). Circular pocket canned cycle. |
| G88 | | | * | (·T· model). Grooving canned cycle along the X axis. |
| G89 | | | * | (·T· model). Grooving canned cycle along the Z axis. |
| G90 | * | ? | | Programming in absolute coordinates. |
| G91 | * | ? | * | Programming in incremental coordinates. |
| G92 | ! | | * | Zero offset. Coordinate preset. |
| G93 | * | | * | Machining time in seconds. |
| G94 | * | ? | | Feedrate in millimeters/minute (inches/minute). |
| G95 | * | ? | * | Feedrate in millimeters/revolution (inches/revolution). |
| G96 | * | | * | Constant surface speed. |
| G97 | * | * | | Constant turning speed. |
| G98 | * | * | | (·M· model). Withdrawal to the starting plane. |
| G99 | * | | * | (·M· model). Withdrawal to the reference plane. |
| G100 | | | * | Probing until making contact. |
| G101 | * | | | Include probe offset. |
| G102 | * | | | Exclude probe offset. |
| G103 | | | * | Probing until not making contact. |
| G104 | | | | Probe movement up to the programmed position. |
| G108 | * | * | | Blend the feedrate at the beginning of the block. |
| G109 | | | * | Blend the feedrate at the end of the block. |
| G112 | * | | | Changing of parameter range of an axis. |
| G130 | * | | * | Percentage of acceleration to be applied per axis or spindle. |
| G131 | * | | * | Percentage of acceleration to be applied to all the axes. |
| G132 | * | | * | Percentage of jerk to be applied per axis or spindle. |
| G133 | * | | * | Percentage of jerk to be applied to all the axes. |
| G134 | * | | * | Percentage of feed-forward to be applied. |
| G135 | * | | * | Percentage of AC-Forward to be applied. |
| G136 | * | | * | Circular transition between blocks. |
| G137 | * | * | | Linear transition between blocks. |
| G138 | * | | * | Direct activation/cancellation of tool compensation. |
| G139 | * | * | | Indirect activation/cancellation of tool compensation. |
| G145 | | | | Freeze tangential control. |
| G151 | * | * | * | Programming the face axis in diameter. |
| G152 | * | | | Programming the face axis in radius. |
| G157 | * | | * | Excluding axes in the absolute zero offset. |

| Function | M | D | V | Meaning. |
|---|---|---|---|---|
| G158 | * | | * | Zero offset. Incremental zero offset. |
| G159 | ! | | * | Zero offset. Absolute zero offset. |
| G160 | | | * | (·M· model). Multiple machining in straight line. |
| G160 | | | * | (·T· model). Drilling / tapping canned cycle on the face of the part. |
| G161 | | | * | (·M· model). Multiple machining in rectangular pattern. |
| G161 | | | * | (·T· model). Drilling / tapping canned cycle on the side of the part. |
| G162 | | | * | (·M· model). Multiple machining in grid pattern. |
| G162 | | | * | (·T· model). Slot milling canned cycle along the side of the part. |
| G163 | | | * | (·M· model). Multiple machining in a full circle. |
| G163 | | | * | (·T· model). Slot milling canned cycle along the face of the part. |
| G164 | | | * | (·M· model). Multiple machining in arc pattern. |
| G165 | | | * | (·M· model). Machining programmed with an arc-chord. |
| G170 | * | | | Cancel Hirth axes. |
| G171 | * | * | | Activate Hirth axes. |
| G174 | * | | | Set the machine coordinate. |
| G180-G189 | | | * | OEM subroutine execution. |
| G380-G399 | | | * | OEM subroutine execution. |
| G192 | * | | * | Turning speed limit. |
| G193 | | | * | Interpolate feedrate during the block. |
| G196 | * | | * | Constant tangential feedrate. |
| G197 | * | * | | Constant tool center feedrate. |
| G198 | | | | Set lower software travel limits. |
| G199 | | | | Set upper software travel limits. |
| G200 | | | | Exclusive manual intervention. |
| G201 | * | | | Activate additive manual intervention. |
| G202 | * | * | | Cancellation of additive manual intervention. |
| G210 | * | | * | (·M· model). Bore milling canned cycle. |
| G211 | * | | * | (·M· model). Inside thread milling cycle. |
| G212 | * | | * | (·M· model). Outside thread milling cycle. |
| G261 | * | | * | Arc center in absolute coordinates (modal). |
| G262 | * | * | | Arc center referred to starting point. |
| G263 | * | | * | Arc radius programming. |
| G264 | * | | * | Cancel arc center correction. |
| G265 | * | * | | Activate arc center correction. |
| G266 | | | * | Set feedrate percentage at 100% |
| G500-G599 | | | * | Generic user subroutines. |

# CANNED CYCLES (·M· MODEL).

As a general rule, the structure of a canned cycle defining block is the following. It is also possible to add the definition of the canned cycle (calling function and parameters) at the end of any block.

[G functions] G8x [Machining point] cycle parameters [F S T D M]

N10 G99 G1 G81 X60 Y0 Z2 I-20 F1000 S2000 M4

N10 G99 G1 X60 Y0 F1000 S2000 M4 G81 Z2 I-20

## G81. Drilling canned cycle.
### G81 Z I K A



| | |
|---|---|
| Z | Reference plane. |
| I | Drilling depth. |
| K | Dwell in seconds at the bottom of the hole. |
| A | Spindle behavior when going into the hole and coming out of it. |

## G82. Drilling canned cycle with a variable peck.
### G82 Z I D B H C J K R L A



| | |
|---|---|
| Z | Reference plane. |
| I | Drilling depth. |
| D | Distance between the reference plane and the part surface. |
| B | Drilling peck (step). |
| H | Distance or coordinate it returns to, in rapid (G0), after each drilling step. |
| C | Approach coordinate. |
| J | Number of drilling pecks to return in rapid (G0) to the reference plane (Z). |
| K | Dwell in seconds at the bottom of the hole. |
| R | Factor that increases or reduces the drilling peck (step) "B". |
| L | Minimum value allowed for the drilling peck. |
| A | Spindle behavior when going into the hole and coming out of it. |

## G83. Deep-hole drilling canned cycle with constant peck.

### G83 Z I J B K A



| | |
|---|---|
| Z | Reference plane. |
| I | Drilling peck (step). |
| J | Number of drilling pecks. |
| B | Rapid withdraw (G0) distance after each drilling step. |
| K | Dwell in seconds at the bottom of the hole. |

## G84. Tapping canned cycle.

### G84 Z I K R J



| | |
|---|---|
| Z | Reference plane. |
| I | Tap depth. |
| K | Dwell in seconds at the bottom of the hole. |
| R | Type of tapping (if R0, normal tapping; if R1, rigid tapping). |
| J | Withdrawal feedrate factor. |

## G85. Reaming canned cycle.

### G85 Z I K



| | |
|---|---|
| Z | Reference plane. |
| I | Reaming depth. |
| K | Dwell in seconds at the bottom of the hole. |

## G86. Boring canned cycle.
### G86 Z I K R A Q D E

**G90**  **G91**

| | |
|---|---|
| Z | Reference plane. |
| I | Boring depth. |
| K | Dwell in seconds at the bottom of the hole. |
| R | Type of withdrawal. |
| A | Spindle behavior when going into the hole and coming out of it. |
| Q | Spindle position, in degrees, to separate the cutter from the wall of the hole. |
| D | Distance to withdraw the cutter off the wall of the hole along the abscissa axis. |
| E | Distance to withdraw the cutter off the wall of the hole along the ordinate axis. |

## G87. Rectangular pocket canned cycle.
### G87 Z I D A J K M Q B C L H V

| | |
|---|---|
| Z | Reference plane. |
| I | Pocket depth. |
| D | Distance between the reference plane and the part surface. |
| A | Angle, in degrees, between the pocket and the abscissa axis. |
| J | Half length of the pocket. |
| K | Half width of the pocket. |
| M | Type of corner (straight, rounded or chamfer). |
| Q | Rounding radius or chamfer size. |
| B | Depth of pass. |
| C | Milling pass or width. |
| L | Finishing pass. |
| H | Feedrate for the finishing pass. |
| V | Tool penetrating feedrate. |

## G88. Circular pocket canned cycle.
### G88 Z I D J B C L H V

| | |
|---|---|
| Z | Reference plane. |
| I | Pocket depth. |
| D | Distance between the reference plane and the part surface. |
| J | Pocket radius. |
| B | Depth of pass. |
| C | Milling pass or width. |
| L | Finishing pass. |
| H | Feedrate for the finishing pass. |
| V | Tool penetrating feedrate. |

# MULTIPLE MACHINING (·M· MODEL).

Parameters P, Q, R, S, T, U and V are optional parameters that may be used in any type of multiple positioning. Thus, programming "P7" means that no machining operation takes place at point 7. Programming "Q10.013" means that no machining takes place at points 10, 11, 12 and 13. If these parameters are not programmed, the CNC executes the machining operation at all the points of the programmed path.

## G160. Multiple machining in straight line.

### G160 A X I K P Q R S T U V
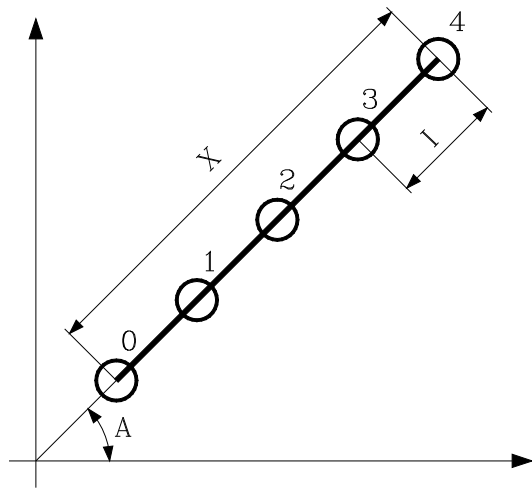


| | |
|---|---|
| A | Angle, in degrees of the machining path with respect to the abscissa axis. |
| X | Length of the machining path. |
| I | Step between machining operations. |
| K | Total number of machining operations in the section, including that of the machining definition point. |

When defining the machining operation, only two of parameters "X", "I" and "K" are required.

## G161. Multiple machining in rectangular pattern.

### G161 A B X I K Y J D P Q R S T U V
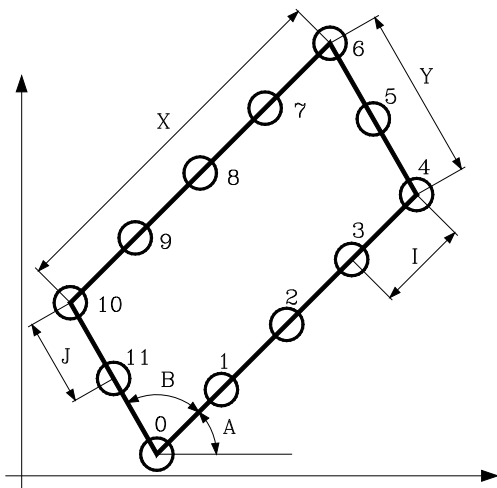


| | |
|---|---|
| A | Angle, in degrees of the machining path with respect to the abscissa axis. |
| B | Angle between both machining paths. |
| X | Length of the parallelogram. |
| I | Step between machining operations along the path. |
| K | Total number of machining operations along the path, including that of the machining definition point. |
| Y | Width of the parallelogram. |
| J | Step between machining operations along the path. |
| D | Total number of machining operations along the path, including that of the machining definition point. |

When defining the machining operation, only two of parameters "X", "I" and "K" are required.

When defining the machining operation, only two of parameters "X", "J" and "D" are required.

## G162. Multiple machining in grid pattern.

### G162 A B X I K Y J D P Q R S T U V
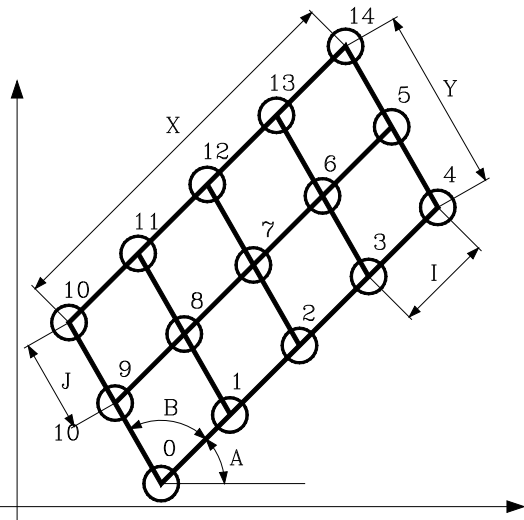


| | |
|---|---|
| A | Angle, in degrees of the machining path with respect to the abscissa axis. |
| B | Angle between both machining paths. |
| X | Length of the grid. |
| I | Step between machining operations along the path. |
| K | Total number of machining operations along the path, including that of the machining definition point. |
| Y | Width of the grid. |
| J | Step between machining operations along the path. |
| D | Total number of machining operations along the path, including that of the machining definition point. |

When defining the machining operation, only two of parameters "X", "I" and "K" are required.

When defining the machining operation, only two of parameters "X", "J" and "D" are required.

## G163. Multiple machining in a full circle.

### G163 X Y I K C F P Q R S T U V
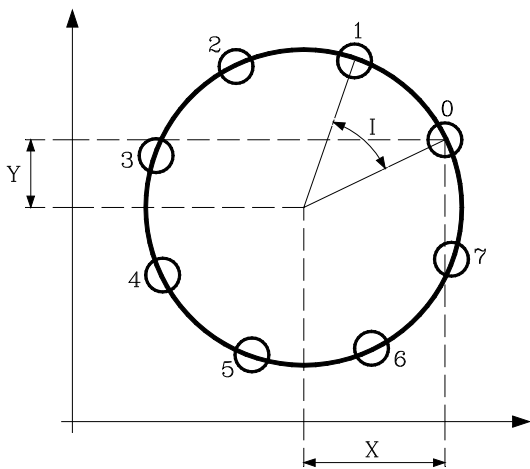


| | |
|---|---|
| X | Distance from the starting point to the center along the abscissa axis. |
| Y | Distance from the starting point to the center along the ordinate axis. |
| I | Angular step between machining operations. |
| K | Total number of machining operations including that of the machining definition point. |
| C | Type of movement between machining points. |
| F | Feedrate for the movement between points. |

When defining the machining operation, only one of parameters "I" and "K" is required.

## G164. Multiple machining in arc pattern.
### G164 X Y B I K C F P Q R S T U V
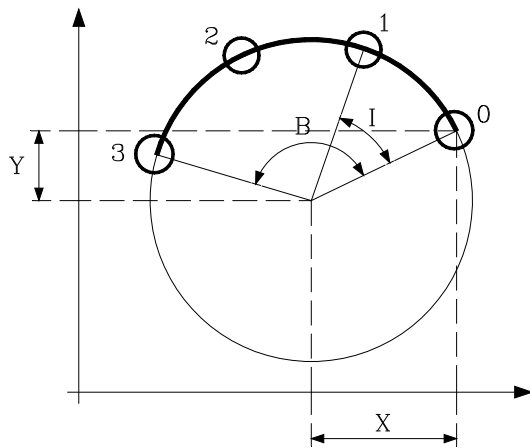


| | |
|---|---|
| X | Distance from the starting point to the center along the abscissa axis. |
| Y | Distance from the starting point to the center along the ordinate axis. |
| B | Angular distance in degrees of the machining path. |
| I | Angular step between machining operations. |
| K | Total number of machining operations including that of the machining definition point. |
| C | Type of movement between machining points. |
| F | Feedrate for the movement between points. |

When defining the machining operation, only one of parameters "I" and "K" is required.

## G165. Machining programmed with an arc-chord.
### G165 X Y A I C F
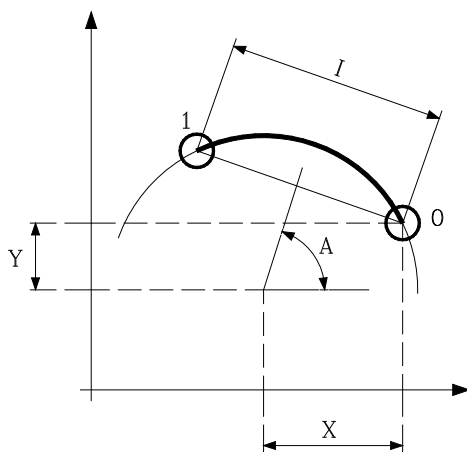


| | |
|---|---|
| X | Distance from the starting point to the center along the abscissa axis. |
| Y | Distance from the starting point to the center along the ordinate axis. |
| A | Angle, in degrees of the perpendicular bisector of the chord with respect to the abscissa axis. |
| I | Length of the chord. |
| K | Total number of machining operations including that of the machining definition point. |
| C | Type of movement between machining points. |
| F | Feedrate for the movement between points. |

When defining the machining operation, only one of parameters "A" and "I" is required.

# CANNED CYCLES (·T· MODEL).

## G66. Pattern repeat canned cycle.
### G66 X Z I C A L M H S E P Q
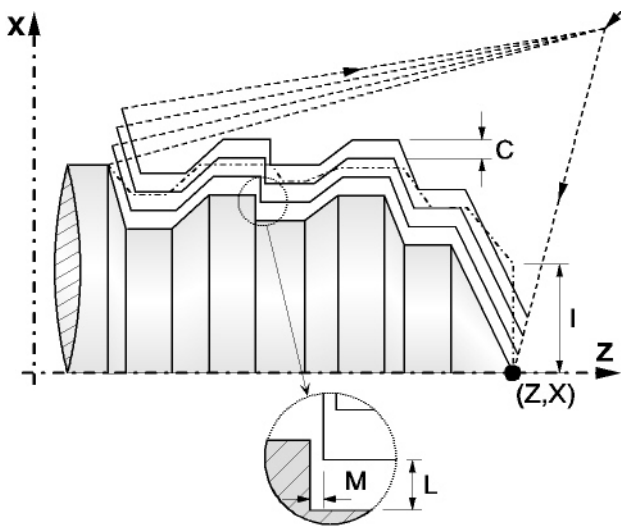


| | |
|---|---|
| X | X coordinate of the profile's starting point. |
| Z | Z coordinate of the profile's starting point. |
| I | Excess material (in radius). |
| C | Machining pass (in radius). |
| A | Machining main axis. |
| L | Finishing stock on the X axis (in radius). |
| M | Finishing stock on the Z axis. |
| H | Finishing feedrate. |
| S | Label number of the first block describing the geometry of the profile. |
| E | Label number of the last block describing the geometry of the profile. |
| P | Number of the subroutine where that defines the profile. |
| Q | Name of the global subroutine, that contains the definition of the profile (parameters "E" and "S") or of the program that contains the local subroutine that contains the profile (parameter "P"). |

## G68. Stock removal cycle along X axis.
### G68 X Z C D L M K F H S E P Q
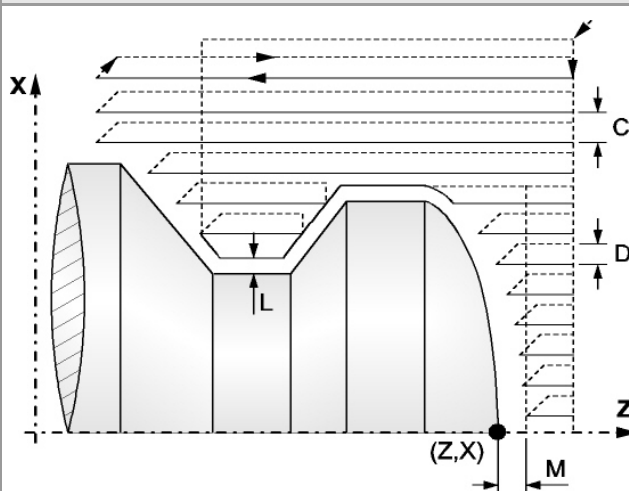


| | |
|---|---|
| X | X coordinate of the profile's starting point. |
| Z | Z coordinate of the profile's starting point. |
| C | Machining pass (in radius). |
| D | Withdrawal distance after each pass. |
| L | Finishing stock on the X axis (in radius). |
| M | Finishing stock on the Z axis. |
| K | Penetration feedrate in the roots. |
| F | Feedrate for the last roughing pass. |
| H | Finishing feedrate. |
| S | Label number of the first block describing the geometry of the profile. |
| E | Label number of the last block describing the geometry of the profile. |
| P | Number of the subroutine where that defines the profile. |
| Q | Name of the global subroutine, that contains the definition of the profile (parameters "E" and "S") or of the program that contains the local subroutine that contains the profile (parameter "P"). |

## G69. Stock removal canned cycle along Z axis.
### G69 X Z C D L M K F H S E P Q



| | |
|---|---|
| X | X coordinate of the profile's starting point. |
| Z | Z coordinate of the profile's starting point. |
| C | Machining pass. |
| D | Withdrawal distance after each pass. |
| L | Finishing stock on the X axis (in radius). |
| M | Finishing stock on the Z axis. |
| K | Penetration feedrate in the roots. |
| F | Feedrate for the last roughing pass. |
| H | Finishing feedrate. |
| S | Label number of the first block describing the geometry of the profile. |
| E | Label number of the last block describing the geometry of the profile. |
| P | Number of the subroutine where that defines the profile. |
| Q | Name of the global subroutine, that contains the definition of the profile (parameters "E" and "S") or of the program that contains the local subroutine that contains the profile (parameter "P"). |

## G81. Turning canned cycle with straight sections.
### G81 X Z Q R C D L M F H



| | |
|---|---|
| X | X coordinate of the profile's starting point. |
| Z | Z coordinate of the profile's starting point. |
| Q | X coordinate of the profile's last point. |
| R | Z coordinate of the profile's last point. |
| C | Machining pass. |
| D | Withdrawal distance after each pass. |
| L | Finishing stock on the X axis (in radius). |
| M | Finishing stock on the Z axis. |
| F | Feedrate for the last roughing pass. |
| H | Finishing feedrate. |

## G82. Facing canned cycle with straight sections.

### G82 X Z Q R C D L M F H
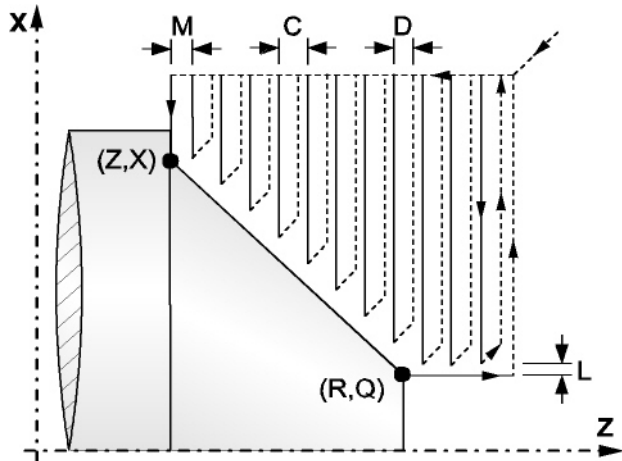


| | |
|---|---|
| X | X coordinate of the profile's starting point. |
| Z | Z coordinate of the profile's starting point. |
| Q | X coordinate of the profile's last point. |
| R | Z coordinate of the profile's last point. |
| C | Machining pass. |
| D | Withdrawal distance after each pass. |
| L | Finishing stock on the X axis (in radius). |
| M | Finishing stock on the Z axis. |
| F | Feedrate for the last roughing pass. |
| H | Finishing feedrate. |

## G83. Axial drilling and tapping canned cycle.

### G83 X Z I B D K H C R (axial drilling)
### G83 X Z I B0 D K R (tapping)
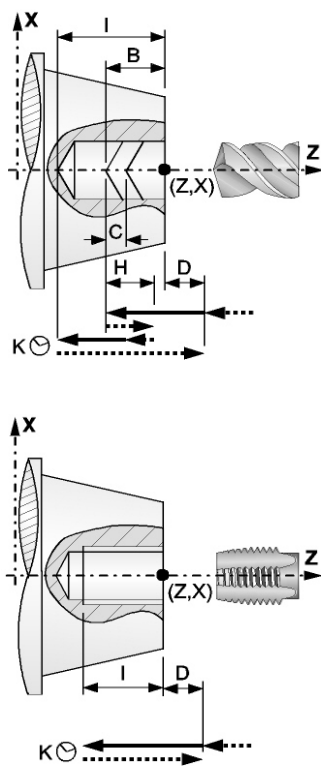


| | |
|---|---|
| X | X coordinate of the starting point. |
| Z | Z coordinate of the starting point. |
| I | Machining depth. |
| B | Machining type (if B0, tapping; if B>0; drilling). |
| D | Safety distance. |
| K | Dwell at the bottom. |
| H | Rapid withdrawal (G0) distance after each drilling step. |
| C | Approach distance to the previous drilling step in rapid (G00). |
| R | In the drilling cycle, factor that reduces the drilling peck "B". In the threading cycle, type of threading. |

## G84. Turning canned cycle with arcs.
### G84 X Z Q R C D L M F H I K



X   X coordinate of the profile's starting point.
Z   Z coordinate of the profile's starting point.
Q   X coordinate of the profile's last point.
R   Z coordinate of the profile's last point.
C   Machining pass.
D   Withdrawal distance after each pass.
L   Finishing stock on the X axis (in radius).
M   Finishing stock on the Z axis.
F   Feedrate for the last roughing pass.
H   Finishing feedrate.
I   Distance from the starting point to the arc center along the X axis.
K   Distance from the starting point to the arc center along the Z axis.

## G85. Facing canned cycle with arcs.
### G85 X Z Q R C D L M F H I K



X   X coordinate of the profile's starting point.
Z   Z coordinate of the profile's starting point.
Q   X coordinate of the profile's last point.
R   Z coordinate of the profile's last point.
C   Machining pass.
D   Withdrawal distance after each pass.
L   Finishing stock on the X axis (in radius).
M   Finishing stock on the Z axis.
F   Feedrate for the last roughing pass.
H   Finishing feedrate.
I   Distance from the starting point to the arc center along the X axis.
K   Distance from the starting point to the arc center along the Z axis.

## G86. Longitudinal threading or thread repair canned cycle.
## G86 X Z Q R K I B E D L C J A W
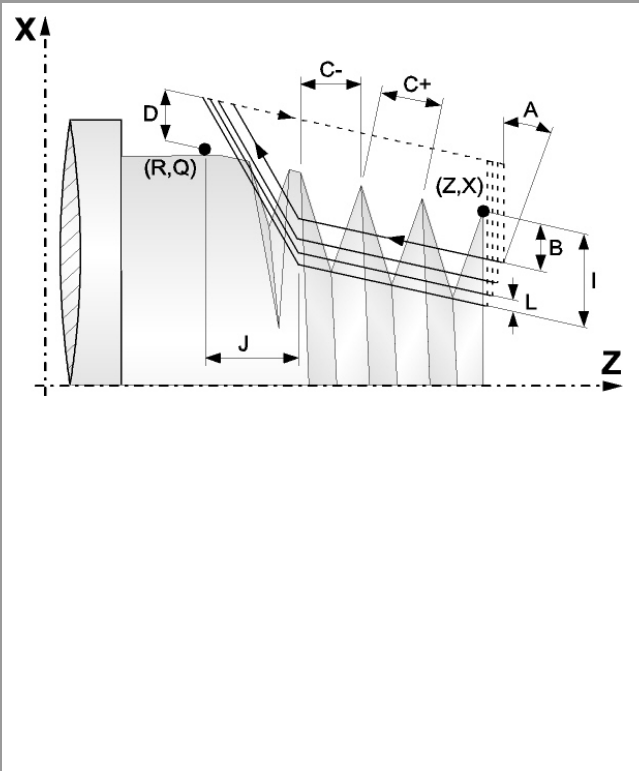


| | |
|---|---|
| X | X coordinate of the thread's starting point. |
| Z | Z coordinate of the thread's starting point. |
| Q | X coordinate of the thread's end point. |
| R | Z coordinate of the thread's end point. |
| K | Z coordinate of the point where the thread is measured (for thread repair). |
| I | Thread depth (in radius). |
| B | Depth of the passes. |
| E | Minimum value that the penetration pass can reach when "B">0. |
| D | X axis safety distance (in radius). |
| L | Finishing stock (in radius). |
| C | Thread pitch. |
| J | Z distance from the final point (R) to the point where the withdrawal from the thread begins. |
| A | Tool penetration angle referred to the ordinate axis. |
| W | Angular position of the thread starting point. In thread repair, angular position of the thread measuring point. |

## G87. Face threading or thread repair canned cycle.
## G87 X Z Q R K I B E D L C J A W
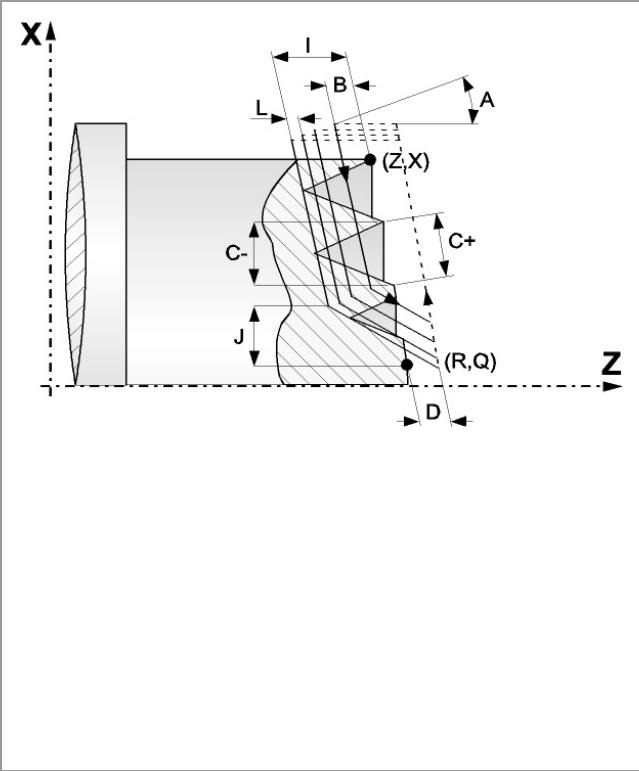


| | |
|---|---|
| X | X coordinate of the thread's starting point. |
| Z | Z coordinate of the thread's starting point. |
| Q | X coordinate of the thread's end point. |
| R | Z coordinate of the thread's end point. |
| K | X coordinate of the point where the thread is measured (for thread repair). |
| I | Thread depth (in radius). |
| B | Depth of the passes. |
| E | Minimum value that the penetration pass can reach when "B">0. |
| D | Safety distance along Z. |
| L | Finishing stock. |
| C | Thread pitch. |
| J | X distance from the final point (R) to the point where the withdrawal from the thread begins (in radius). |
| A | Tool penetration angle referred to the ordinate axis. |
| W | Angular position of the thread starting point. In thread repair, angular position of the thread measuring point. |

## G88. Grooving canned cycle along the X axis.

### G88 X Z Q R C D K
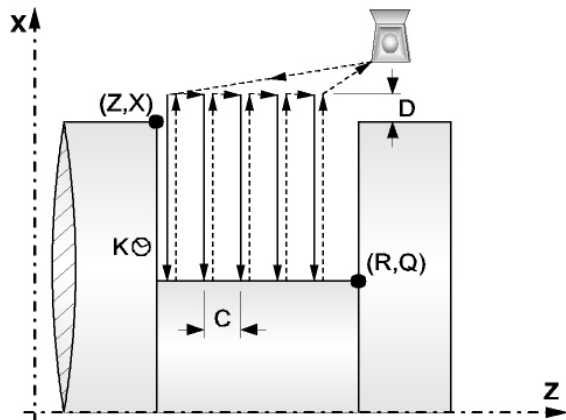
| | |
|---|---|
| X | X coordinate of the groove's starting point. |
| Z | Z coordinate of the groove's starting point. |
| Q | X coordinate of the groove's end point. |
| R | Z coordinate of the groove's end point. |
| C | Grooving pass. |
| D | X axis safety distance (in radius). |
| K | Dwell, in hundredths of a second, after each penetration until the withdrawal begins. |

## G89. Grooving canned cycle along the Z axis.

### G89 X Z Q R C D K
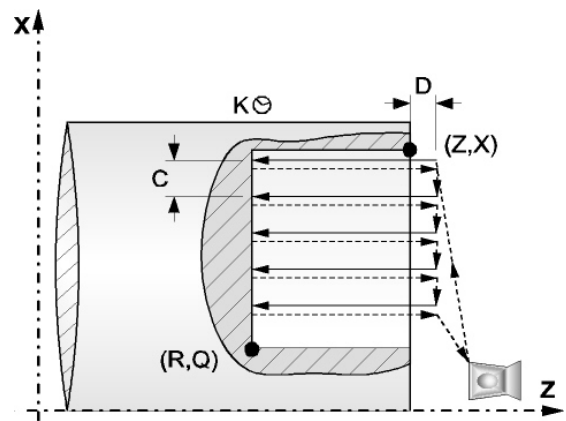
| | |
|---|---|
| X | X coordinate of the groove's starting point. |
| Z | Z coordinate of the groove's starting point. |
| Q | X coordinate of the groove's end point. |
| R | Z coordinate of the groove's end point. |
| C | Grooving pass (in radius). |
| D | Safety distance along Z. |
| K | Dwell, in hundredths of a second, after each penetration until the withdrawal begins. |

## G160. Drilling / tapping canned cycle on the face of the part.

### G160 X Z I B Q A J D K H C S R N (drilling)
### G160 X Z I B0 Q A J D S R N (tapping)



| | |
|---|---|
| X | X coordinate of the cycle starting point. |
| Z | Z coordinate of the cycle starting point. |
| I | Machining depth. |
| B | Machining type (if B0, tapping; if B>0; drilling). |
| Q | Angular position of the spindle for the first machining operation. |
| A | Angular step between machining operations. |
| J | Total number of machining operations. |
| D | Safety distance along Z. |
| K | Dwell at the bottom. |
| H | Rapid withdrawal (G0) distance after each drilling step. |
| C | Approach distance to the previous drilling step in rapid (G00). |
| R | In the drilling cycle, factor that reduces the drilling peck "B". In the threading cycle, type of threading. |
| S | Live tool turning speed and direction. |
| N | Number of the spindle for the live tool. |

## G161. Drilling / tapping canned cycle on the side of the part.

### G161 X Z I B Q A J D K H C S R N (drilling)
### G161 X Z I B0 Q A J D S R N (tapping)



| | |
|---|---|
| X | X coordinate of the cycle starting point. |
| Z | Z coordinate of the cycle starting point. |
| I | Machining depth. |
| B | Machining type (if B0, tapping; if B>0; drilling). |
| Q | Angular position of the spindle for the first machining operation. |
| A | Angular step between machining operations. |
| J | Total number of machining operations. |
| D | Safety distance along X. |
| K | Dwell at the bottom. |
| H | Rapid withdrawal (G0) distance after each drilling step. |
| C | Approach distance to the previous drilling step in rapid (G00). |
| R | In the drilling cycle, factor that reduces the drilling peck "B". In the threading cycle, type of threading. |
| S | Live tool turning speed and direction. |
| N | Number of the spindle for the live tool. |

## G162. Slot milling canned cycle along the side of the part.

### G162 X Z L I Q A J D F S N



| | |
|---|---|
| X | X coordinate of the cycle starting point. |
| Z | Z coordinate of the cycle starting point. |
| L | Length of the slot referred to the starting point. |
| I | Depth of the slot referred to the starting point. |
| Q | Angular position of the spindle for the first machining operation. |
| A | Angular step between machining operations. |
| J | Total number of machining operations. |
| D | Safety distance along X. |
| F | Machining feedrate. |
| S | Live tool turning speed and direction. |
| N | Number of the spindle for the live tool. |

## G163. Slot milling canned cycle along the face of the part.

### G163 X Z L I Q A J D F S N



| | |
|---|---|
| X | X coordinate of the cycle starting point. |
| Z | Z coordinate of the cycle starting point. |
| L | Length of the slot referred to the starting point. |
| I | Depth of the slot referred to the starting point. |
| Q | Angular position of the spindle for the first machining operation. |
| A | Angular step between machining operations. |
| J | Total number of machining operations. |
| D | Safety distance along Z. |
| F | Machining feedrate. |
| S | Live tool turning speed and direction. |
| N | Number of the spindle for the live tool. |

# HIGH LEVEL LANGUAGE.

| Instruction. | Meaning. |
|---|---|
| $GOTO | Block skip. |
| $IF   $ENDIF<br>$ELSEIF   $ELSE | Conditional execution. |
| $SWITCH   $CASE   $ENDSWITCH<br>$BREAK   $DEFAULT | Conditional execution. |
| $FOR   $ENDFOR<br>$BREAK   $CONTINUE | Block repetition. |
| $WHILE   $ENDWHILE<br>$BREAK   $CONTINUE | Conditional block repetition. |
| $DO   $ENDDO<br>$BREAK   $CONTINUE | Conditional block repetition. |

| Instruction. | Meaning. |
|---|---|
| L | Call to a global subroutine. |
| LL | Call to a local subroutine. |
| #ABORT | Abort the execution of the program and resume it in another block or program. |
| #ACS | Fixture coordinate system. |
| #ANGAX OFF | Turn angular transformation off. |
| #ANGAX ON | Turn angular transformation on. |
| #ANGAX SUSP | Freeze angular transformation. |
| #ASPLINE ENDTANG | Akima splines. Type of final tangent. |
| #ASPLINE MODE | Akima splines. Selection of tangent type. |
| #ASPLINE STARTTANG | Akima splines. Type of starting tangent. |
| #AXIS | Axis upon which the additive manual intervention is applied. |
| #CALL | Call to a global or local subroutine. |
| #CALL AX | Add a new axis to the configuration of the channel. |
| #CALL SP | Add a spindle to the configuration of the channel. |
| #CAM ON | Activate the electronic cam (real coordinates). |
| #CAM OFF | Cancel the electronic cam. |
| #CAX | "C" axis. Activating the spindle as ·C· axis. |
| #CD OFF | Cancel collision detection. |
| #CD ON | Activating collision detection. |
| #CLEAR | Channels. It clears the synchronism marks of the channel. |
| #CONTJOG | Manual intervention. Feedrate in continuous jog. |
| #COMMENT BEGIN | Beginning of comment. |
| #COMMENT END | End of comment. |
| #CS | Machining coordinate system. |
| #CSROT ON | Activate tool orientation in the part coordinate system. |
| #CSROT OFF | Cancel tool orientation in the part coordinate system. |
| #CYL | "C" axis. Machining of the turning side of the part. |
| #DEF | Macros. Define a macro. |

| Instruction. | Meaning. |
|---|---|
| #DEFROT | How to manage the discontinuities in the orientation of rotary axes. |
| #DELETE | It initializes the global user variables. |
| #DFHOLD | Disable the feed-hold signal. |
| #DGWZ | Define the size of the graphics area. |
| #DSBLK | Cancel the single-block treatment. |
| #DSTOP | Disable the cycle stop signal. |
| #EFHOLD | Disable the feed-hold signal. |
| #ERROR | Display an error on the screen. |
| #ESBLK | Activate the single-block treatment. |
| #ESTOP | Enable the cycle stop signal. |
| #EXBLK | It executes a block in the indicated channel. |
| #EXEC | It executes a program in the indicated channel. |
| #FACE | "C" axis. Machining of the face of the part. |
| #FLUSH | Interrupt block preparation. |
| #FOLLOW OFF | Independent axis. End the synchronization movement. |
| #FOLLOW ON | Independent axis. Begin the synchronization movement (real coordinates). |
| #FREE AX | Remove an axis from the configuration of the channel. |
| #FREE SP | Remove a spindle from the configuration of the channel. |
| #HSC OFF | Cancel the HSC mode. |
| #HSC ON | Activate the HSC mode. Optimizing the contouring error. |
| #HSC ON [FAST] | Activate the HSC mode. Optimizing the machining speed. |
| #INCJOG | Manual intervention. Feedrate in incremental jog. |
| #INIT MACROTAB | Macros. Initialize the table of macros. |
| #KIN ID | Select a kinematics. |
| #KINORG | Transform the current part zero considering the position of the table kinematics. |
| #LINK | Activate the electronic coupling (slaving) of axes. |
| #MASTER | Selecting the master spindle of the channel. |
| #MCALL | Modal call to a local or global subroutine initializing parameters. |
| #MCS | Program a movement referred to machine zero. |
| #MCS OFF | Cancel the machine coordinate system. |
| #MCS ON | Activate the machine coordinate system. |
| #MDOFF | Turning the subroutine into non-modal. |
| #MEET | Channels. Activate the mark in the indicated channel. |
| #MOVE | Independent axis. Positioning move. |
| #MPG | Manual intervention. Resolution of the handwheels. |
| #MSG | Display a message on the screen. |
| #PARK | Park an axis or spindle. |
| #PATH | Define the location of the global subroutines. |
| #PCALL | Call to a global or local subroutine initializing parameters. |
| #POLY | Polynomial interpolation. |
| #PROBE 1 | (·M· model). Tool calibration (dimensions and wear). |
| | (·T· model). Tool calibration. |

| Instruction. | Meaning. |
|---|---|
| #PROBE 2 | (·M· model). Probe calibration |
| | (·T· model). Tabletop probe calibration |
| #PROBE 3 | (·M· model). Surfacing measuring. |
| | (·T· model). Part measuring along the ordinate axis. |
| #PROBE 4 | (·M· model). Outside corner measuring. |
| | (·T· model). Part measuring along the abscissa axis. |
| #PROBE 5 | (·M· model). Inside corner measuring. |
| #PROBE 6 | (·M· model). Measuring the angle with the abscissa axis. |
| #PROBE 7 | (·M· model). Outside corner and angle measuring. |
| #PROBE 8 | (·M· model). Hole measuring. |
| #PROBE 9 | (·M· model). Circular boss measuring. |
| #PROBE 10 | (·M· model). Rectangular part centering. |
| #PROBE 11 | (·M· model). Circular part centering. |
| #PROBE 12 | (·M· model). Tabletop probe calibration |
| #RENAME AX | Rename the axes of the channel. |
| #RENAME SP | Rename the spindles of the channel. |
| #REPOS | Repositioning axes and spindles from an OEM subroutine. |
| #RET | End of a global or local subroutine. |
| #ROTATEMZ | Positioning a turret magazine. |
| #ROUNDPAR | Type of corner rounding. |
| #RPT | Repeat a group of blocks. |
| #RTCP | RTCP transformation. |
| #SCALE | Scaling factor. |
| #SELECT ORI | Select onto which rotary axes of the kinematics the tool orientation is calculated for a given direction on the work piece (part). |
| #SELECT PROBE | Probe selection. |
| #SERVO ON | Activate the closed loop mode. |
| #SERVO OFF | Activate the open loop mode. |
| #SET AX | Set a new axes configuration. |
| #SET OFFSET | Manual intervention. Moving limits. |
| #SET SP | Set a new spindle configuration. |
| #SIGNAL | Channels. Activate the mark in its own channel. |
| #SLOPE | Acceleration control. |
| #SPLINE OFF | Akima splines. Cancels spline adaptation. |
| #SPLINE ON | Akima splines. Activate spline adaptation. |
| #SWTOUT ON | Activate synchronized switching. |
| #SWTOUT OFF | Cancel synchronized switching. |
| #SYNC | Spindle synchronization based on the real (actual) coordinate. |
| #SYNC POS | Manual intervention. Coordinate synchronization. |
| #TANGCTRL OFF | Cancel tangential control. |
| #TANGCTRL ON | Activate tangential control. |
| #TANGCTRL SUSP | Freeze tangential control. |

| Instruction. | Meaning. |
|---|---|
| #TANGFEED RMIN | Minimum radius for applying constant tangential feedrate. |
| #TCAM ON | Activate the electronic cam (theoretical coordinates). |
| #TFOLLOW ON | Independent axis. Begin the synchronization movement (theoretical coordinates). |
| #TIME | Dwell. |
| #TLC | Correct the implicit tool length compensation of the program. |
| #TOOL AX | Select the longitudinal axis of the tool. |
| #TOOL ORI | Tool perpendicular to the inclined plane. |
| #TSYNC | Spindle synchronization based on the theoretical coordinate. |
| #UNLINK | Cancel the electronic coupling (slaving) of axes. |
| #UNPARK | Unpark an axis or spindle. |
| #UNSYNC | Cancel the spindle synchronization. |
| #VIRTAX ON | Activate the virtual tool axis. |
| #VIRTAX ON | Cancel the virtual tool axis. |
| #WAIT | Channels. It waits for a mark to be activated in the indicated channel. |
| #WAIT FOR | Wait for an event before resuming execution. |
| #WARNING | Display a warning on the screen. |
| #WARNINGSTOP | Display a warning on the screen and interrupt the program. |

# PROBING CANNED CYCLES (·M· MODEL).

## #PROBE 1. Tool calibration (dimensions and wear).
### #PROBE 1 B I J F K S N D E L M C X U Y V Z W
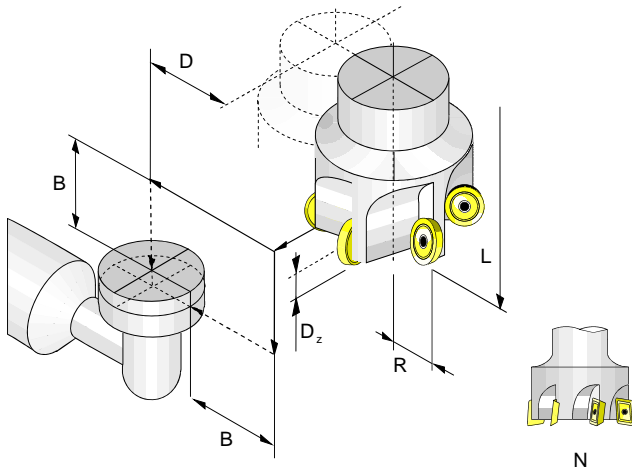
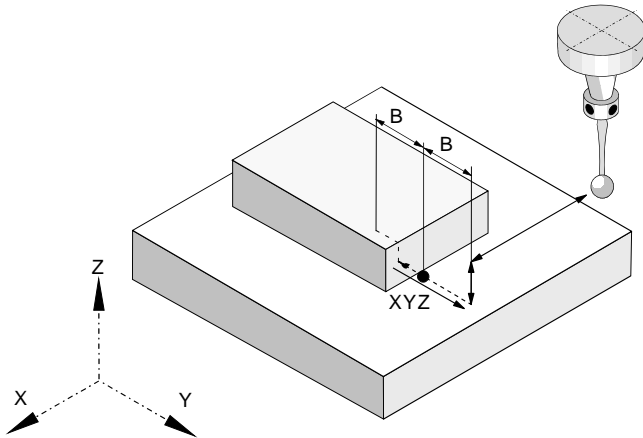| | |
|---|---|
| B | Safety distance. |
| I | Dimension of the tool to be calibrated (if I0, length on the axis; if I1, length at one end; if I2, radius; if I3, radius and length). |
| J | Type of operation (if J0, calibration; if J1, wear measurement). |
| F | Probing feedrate. |
| K | Side of the probe being used (if K0, X+ side; if K1, X- side; if K2, Y+ side; if K3, Y- side). |
| S | Speed and turning direction of the tool. |
| N | Number of edges to measure. |
| D | Distance from the probing point to the tool shaft. |
| E | Distance from the probing point to the tool base. |
| L | Maximum length wear allowed. |
| M | Maximum radius wear allowed. |
| C | Behavior when exceeding the maximum wear. |
| X..W | Tabletop probe position. |

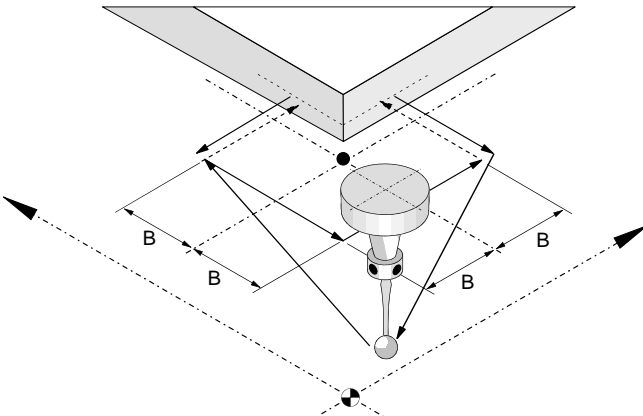## #PROBE 2. Probe calibration
### #PROBE 2 X Y Z B J E H F
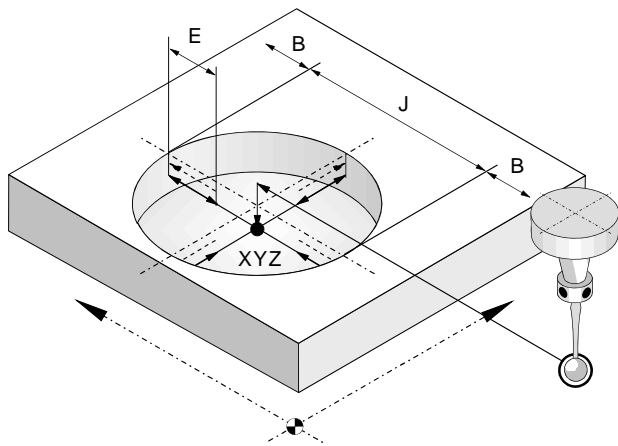
| | |
|---|---|
| X..Y | Real coordinates of the arc center. |
| Z | Position where probing takes place. |
| B | Safety distance. |
| J | Theoretical hole diameter. |
| E | Withdrawal distance after initial probing. |
| H | Feedrate for the first probing movement. |
| F | Feedrate for the second probing movement. |

## #PROBE 3. Surfacing measuring.

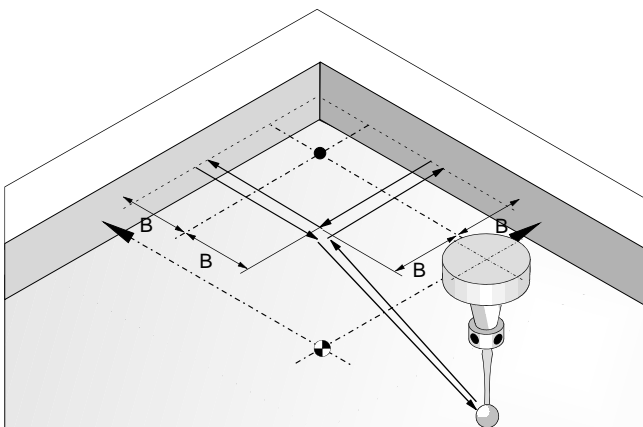### #PROBE 3 X Y Z B K F C L T D



| | |
|---|---|
| X..Z | Theoretical coordinates of the probing point. |
| B | Safety distance. |
| K | Probing axis (if K0, abscissa axis; if K1, ordinate axis; if K2, axis perpendicular to the plane). |
| F | Probing feedrate. |
| C | Cycle end point (if C0, at the calling point; if C1 at the measured point). |
| L | Tolerance for the measuring error. |
| T | Tool whose wear is to be corrected. |
| D | Offset whose wear is to be corrected. |

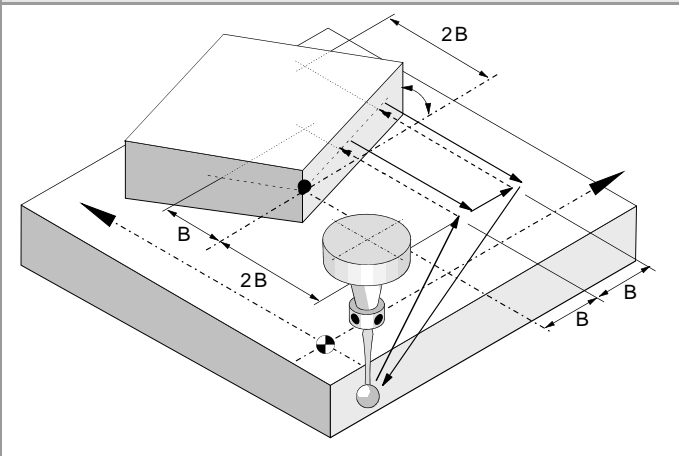## #PROBE 4. Outside corner measuring.

### #PROBE 4 X Y Z B F



| | |
|---|---|
| X..Z | Theoretical coordinates corner being measured. |
| B | Safety distance. |
| F | Probing feedrate. |

## #PROBE 5. Inside corner measuring.

### #PROBE 5 X Y Z B F
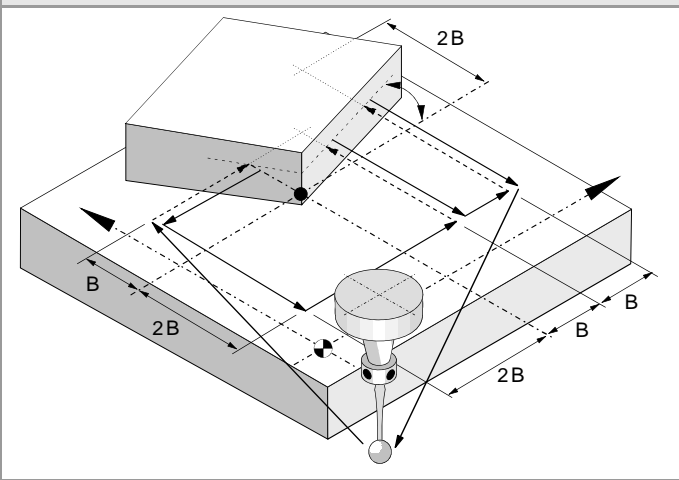


| | |
|---|---|
| X..Z | Theoretical coordinates corner being measured. |
| B | Safety distance. |
| F | Probing feedrate. |

## #PROBE 6. Measuring the angle with the abscissa axis.
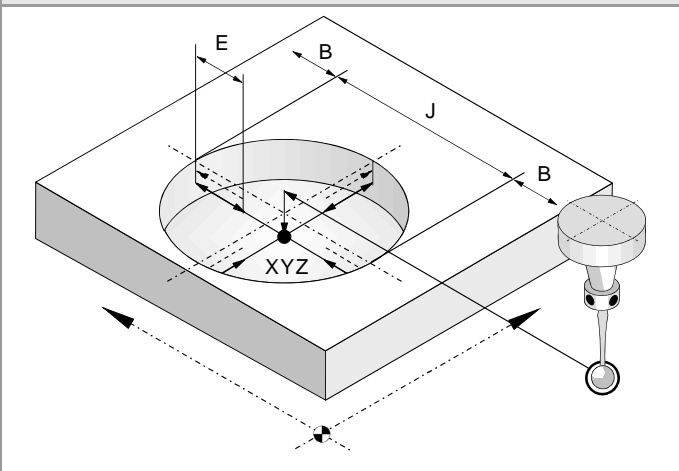### #PROBE 6 X Y Z B F

| | |
|---|---|
| X..Z | Theoretical coordinates corner being measured. |
| B | Safety distance. |
| F | Probing feedrate. |

## #PROBE 7. Outside corner and angle measuring.
### #PROBE 7 X Y Z B F

| | |
|---|---|
| X..Z | Theoretical coordinates corner being measured. |
| B | Safety distance. |
| F | Probing feedrate. |

## #PROBE 8. Hole measuring.
### #PROBE 8 X Y Z B J E C H F

| | |
|---|---|
| X..Z | Theoretical coordinates of the arc center. |
| B | Safety distance. |
| J | Theoretical hole diameter. |
| E | Withdrawal distance after initial probing. |
| C | Cycle end point (if C0, at the calling point; if C1 in the actual - real - center of the hole). |
| H | Feedrate for the first probing movement. |
| F | Feedrate for the second probing movement. |

## #PROBE 9. Circular boss measuring.
### #PROBE 9 X Y Z B J E C H F
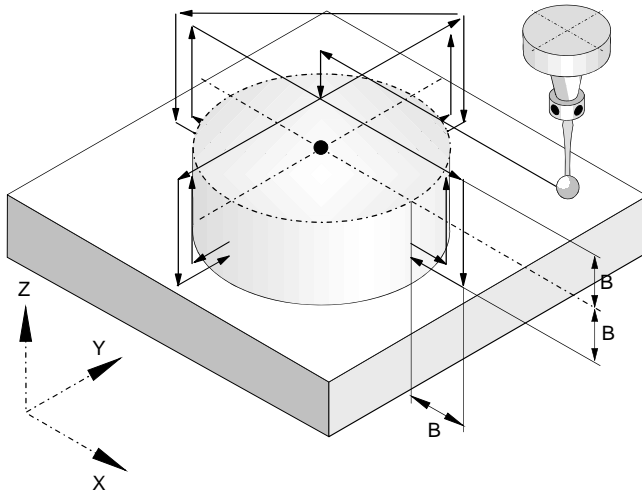


| X..Z | Coordinates of the boss center. |
|---|---|
| B | Safety distance. |
| J | Theoretical boss diameter. |
| E | Withdrawal distance after initial probing. |
| C | Cycle end point (if C0, at the calling point; if C1 in the actual - real - center of the hole). |
| H | Feedrate for the first probing movement. |
| F | Feedrate for the second probing movement. |

## #PROBE 10. Rectangular part centering.
### #PROBE 10 X Y Z I J K L B D E H F Q
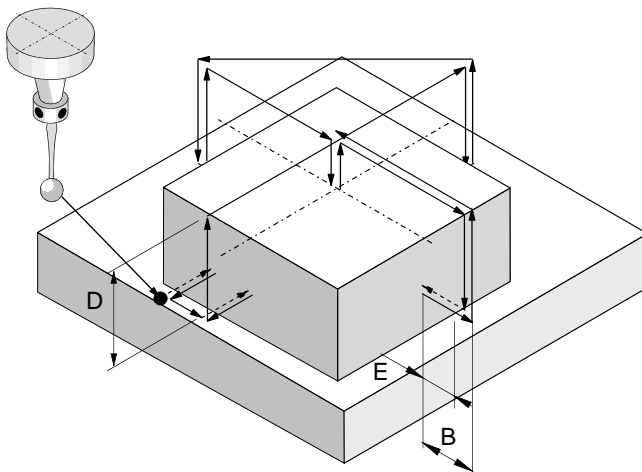


| X..Z | Probe position when calling the cycle. |
|---|---|
| I | Part length along the abscissa axis. |
| J | Part length along the ordinate axis. |
| K | Axis and direction of the first probing movement (if K0, in the positive X direction; if K1, in the negative X direction; if K2, in the positive Y direction; if K1, in the negative Y direction). |
| L | Surface coordinate measurement (if L0, do not measure the coordinate; if L1, measure the coordinate). |
| B | Safety distance. |
| D | Safety distance along Z. |
| E | Withdrawal distance after initial probing. |
| H | Feedrate for the first probing movement. |
| F | Feedrate for the second probing movement. |
| Q | Feedrate for approach movements. |

## #PROBE 11. Circular part centering.
### #PROBE 11 X Y Z J K L B D E H F Q

| | |
|---|---|
| X..Z | Probe position when calling the cycle. |
| J | Part diameter. |
| K | Axis and direction of the first probing movement (if K0, in the positive X direction; if K1, in the negative X direction; if K2, in the positive Y direction; if K1, in the negative Y direction). |
| L | Surface coordinate measurement (if L0, do not measure the coordinate; if L1, measure the coordinate). |
| B | Safety distance. |
| D | Safety distance along Z. |
| E | Withdrawal distance after initial probing. |
| H | Feedrate for the first probing movement. |
| F | Feedrate for the second probing movement. |
| Q | Feedrate for approach movements. |

## #PROBE 12. Tabletop probe calibration.
### #PROBE 12 B E H F I X U Y V Z W

| | |
|---|---|
| B | Safety distance. |
| E | Withdrawal distance after initial probing. |
| H | Feedrate for the first probing movement. |
| F | Feedrate for the second probing movement. |
| I | Type of calibration (if I0, single calibration; if I1 double calibration). |
| X..W | Theoretical position of the tabletop probe. |

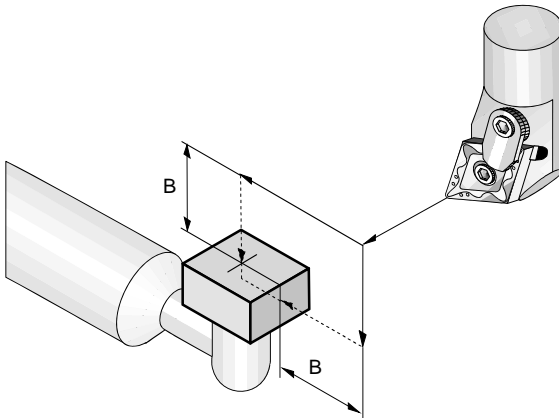# PROBING CANNED CYCLES (·T· MODEL).

## #PROBE 1. Tool calibration.
### #PROBE 1 B F K X U Y V Z W
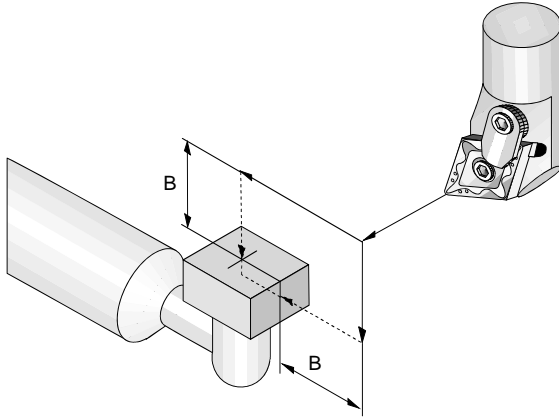


B    Safety distance (in radius).

F    Probing feedrate.

K    Sides of the probe being used (if K0, X Z sides; if K1, X Z Y+ sides; if K2, X Z Y- sides).

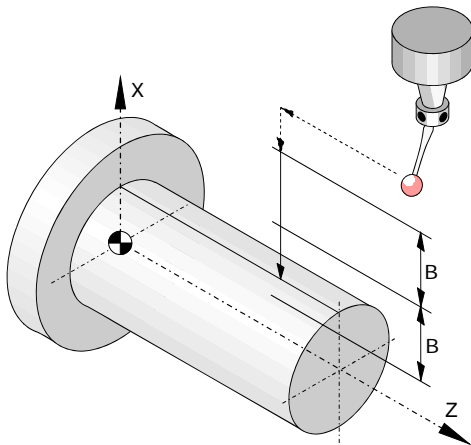X..W    Tabletop probe position.

## #PROBE 2. Tabletop probe calibration
### #PROBE 2 B F K X U Y V Z W



B    Safety distance (in radius).

F    Probing feedrate.

K    Sides of the probe being used (if K0, X Z sides; if K1, X Z Y+ sides; if K2, X Z Y- sides).

X..W    Tabletop probe position.
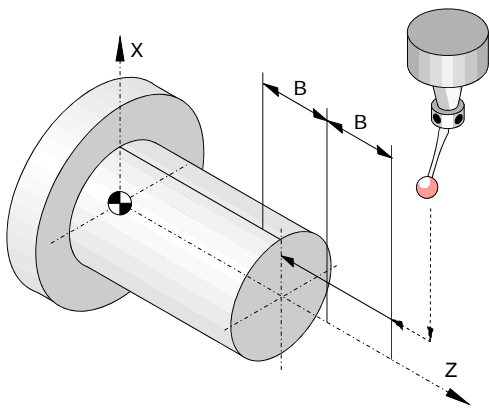
## #PROBE 3. Part measuring along the ordinate axis.
### #PROBE 3 X Z B F L T D



X    Theoretical coordinate of the probing point along the ordinate axis.

Z    Theoretical coordinate of the probing point along the abscissa axis.

B    Safety distance (in radius).

F    Probing feedrate.

L    Tolerance for the measuring error.

T    Tool whose wear is to be corrected.

D    Offset whose wear is to be corrected.

## #PROBE 4. Part measuring along the abscissa axis.
## #PROBE 4 X Z B F L T D



| | |
|---|---|
| X | Theoretical coordinate of the probing point along the ordinate axis. |
| Z | Theoretical coordinate of the probing point along the abscissa axis. |
| B | Safety distance. |
| F | Probing feedrate. |
| L | Tolerance for the measuring error. |
| T | Tool whose wear is to be corrected. |
| D | Offset whose wear is to be corrected. |

# OPERATORS AND FUNCTIONS.

| Arithmetic operators. | |
|---|---|
| + | Add |
| - | Subtract / change sign |
| * | Multiply |
| / | Divide |
| += | Compounded addition |
| -= | Compounded subtraction |
| *= | Compounded multiplication |
| /= | Compounded division |
| MOD | Remainder of the division |
| ** | Exponent |

| Relational operators. | |
|---|---|
| == | Equal to |
| != | Different |
| > | Greater than |
| < | Smaller than |
| >= | Greater than or equal to |
| <= | Smaller than or equal to |

| Relational operators. | |
|---|---|
| & | Binary AND |
| \| | Binary OR |
| ^ | Exclusive OR (XOR) |
| INV[...] | Inverse |

| Logic operators. | |
|---|---|
| * | Logic AND |
| + | Logic OR |

| Boolean constants. | |
|---|---|
| TRUE | True |
| FALSE | Not true |

| Trigonometric functions. | |
|---|---|
| SIN[...] | Sine |
| COS[...] | Cosine |
| TAN[...] | Tangent |
| ASIN[...] | Arc-sine |
| ACOS[...] | Arc-cosine |
| ATAN[...] | Arc-tangent (within ±90º). |
| ARG[...] | Arc-tangent (within 0-360º). |

| Mathematical functions. | |
|---|---|
| ABS[...] | Absolute value |
| SQR[...] | Square function |
| SQRT[...] | Square root |
| LOG[...] | Decimal logarithm |
| LN[...] | Neperian logarithm |
| EXP[...] | "e" function |
| DEXP[...] | Decimal exponent |

| Other functions. | |
|---|---|
| INT[...] | Returns the integer |
| FRACT[...] | Returns decimal portion |
| ROUND[...] | Rounds up or down to the nearest integer |
| FUP[...] | Returns the integer plus one. (If the number is an integer, it returns it) |
| EXIST[...] | It checks whether the selected variable or parameter exists or not |

**FAGOR**

FAGOR AUTOMATION