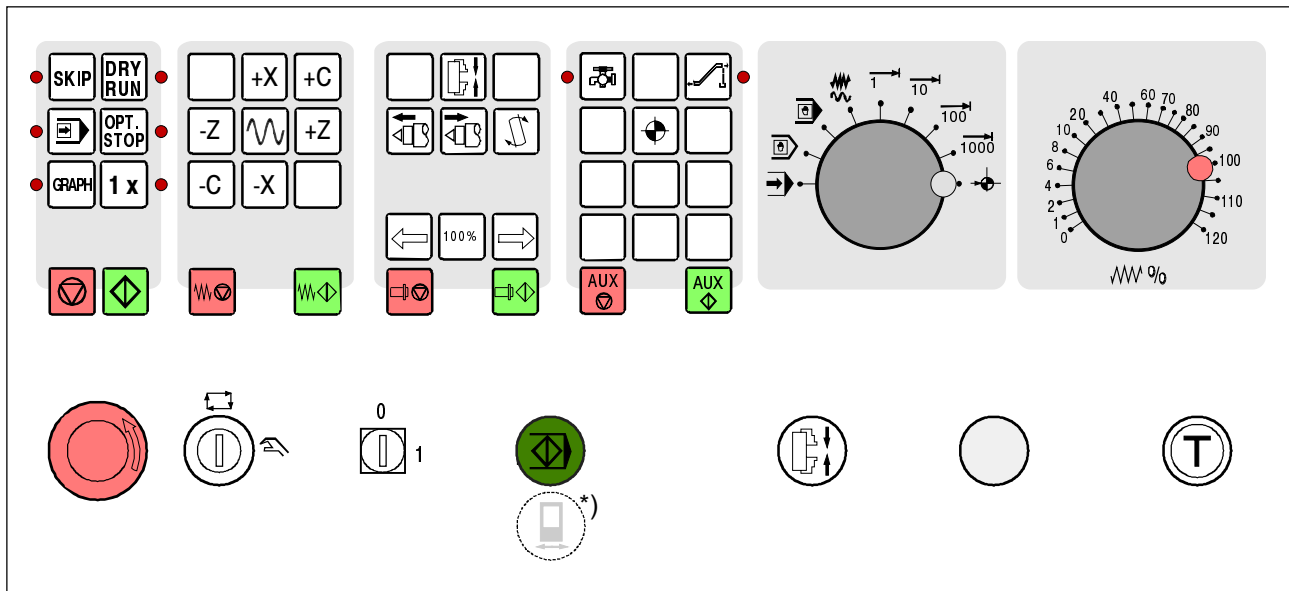# D Programming and Operation
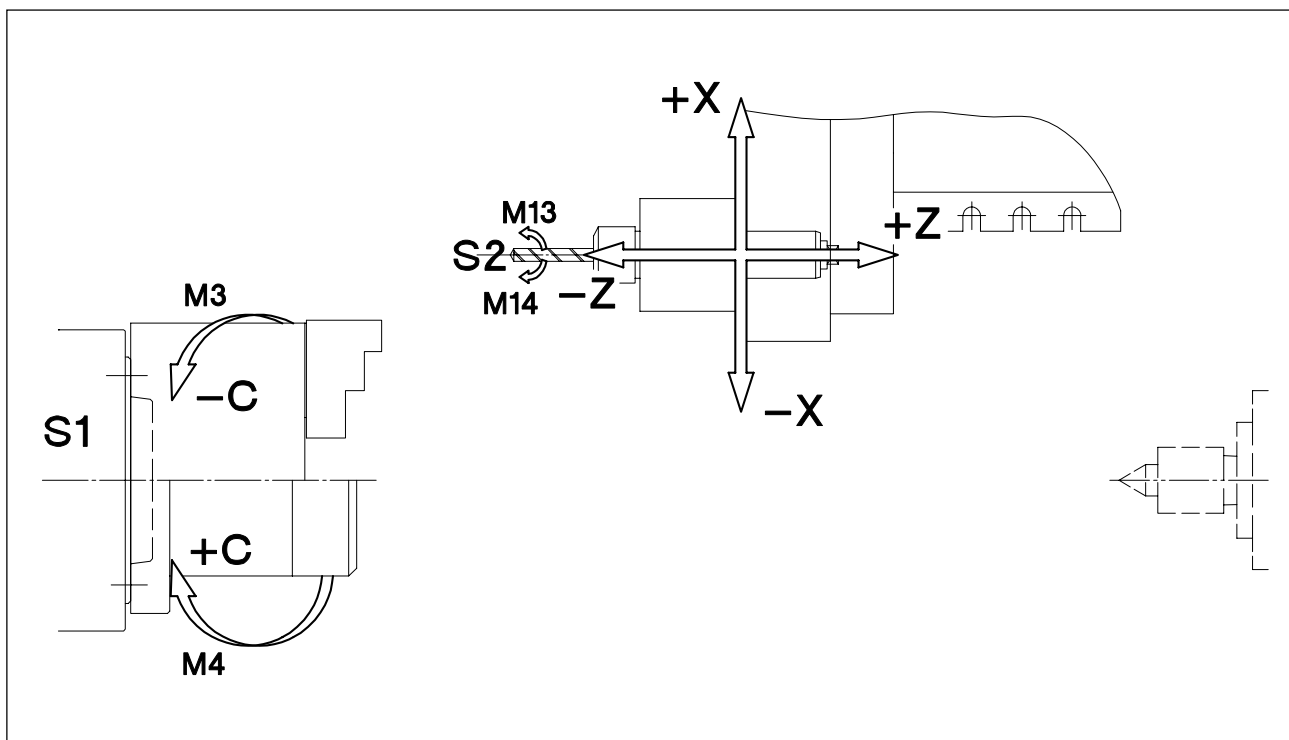# EMCO-specific - Fanuc 21i-T

Machine Control Panel

*)   By machines with door automatic instead of the external cycle-start-key

Axes Description

Axes description of machine

EMCO

Description of Keys

Reset Key

Actuating Reset causes:
- Stop of working off the actual part program.
- Alarms and messages are cleared with the exception of Power On or Recall alarms.
- The channel will be set to the reset status, that means:
  - The NC control stays synchronous to the machine.
  - All intermediate and working memory is cleared (the contents of the part program memory stays resident).
  - The control is in basic setting and ready for program run.

Skip (skip block)

In skip operation program blocks marked before the block number with a slash "/", are skipped during program run (e.g.: /N100).
Active with illuminated LED.

Dryrun (Test run feed)

In dryrun operation traversing motions are executed with the feed value preset in the setting date "test run-feed".
The test run feed is effective instead of the programmed motion commands.
Spindle commands are not executed.
Active with illuminated LED.

> **Caution:**
> The test run feed normally is higher than the programmed feedrate.
> Be sure, that no workpiece is clamped before starting the Dry Run.
> By machining parts pay attention, that the Dry Run mode is switched off (LED of the key is off), before starting the machine.

Single Piece Operation

With this key the single piece operation or continuous operation in connection with automatic loading facilities are available.
The switch-on state is single piece operation.
The active single piece operation is indicated by the illumination of the respective LED at the machine control panel.

Graphic Simulation

GRAPH

With this key you can execute a program test run without axis movement. The programmed tool path is displayed on the screen. This test run serves for testing and recognizing program errors. Active with illuminated LED.

After leaving the graphic simulation the reference point is to be approached again.

> **Remark:**
> To display the programmed path during the
> program test in Graphic Mode press GRAPH and
> softkey "GRAPH".
> The workpiece dimensions (Length=W and Diameter=D) are to be set in [µm] in the screen "G.PRM".

Single Block

This function allows to work off a part program block by block.
The function single block can be activated in AUTOMATIC mode.

Active single block mode causes:
- The actual block of the part program will be worked off after pressing NC Start.
- Working off will be stopped after each block.
- The following block will be worked off after pressing NC Start.

Deselect the function by pressing again on the single block key.
The Mode is active, when the LED is lightening.

Optional STOP

OPT. STOP

At active function (key was pressed) the program run will be stopped at blocks that contain the command M01.
Continue the program with the key NC Start.
When the function is not active, M01 (in the part program) will be ignored.
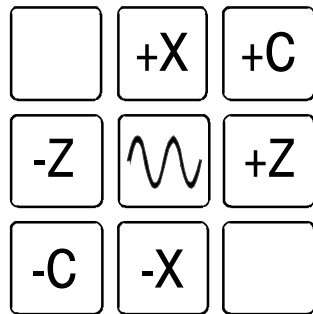
NC STOP

After pressing the NC Stop key and takeover of the function by the control, working off of the running part program will be stopped.
You can continue working off by pressing NC Start.

NC START

After pressing the NC Start key the selected part program will be started with the actual block.

EMCO

Direction Keys

With these keys the NC axes can be traversed in operating mode JOG.

Unless the machine is referenced, you must simultaneously press the key AUX ON to move the axes (release).

Rapid

If this key is pressed simultaneous to a direction key the responding axis traverses in rapid feed.

Feed STOP

This key stops slide movements in AUTOMATIC mode (not for threads).

Feed START

This key continues programmed, interrupted slide movements.
When spindle run was interrupted additionally first the spindle run must be continued.

Clamping Device

The clamping device is operated using this key. Clamping direction see M90-M92.

Via NC program
M25 .... open clamping device
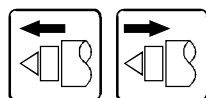M26 .... close clamping device

Tailstock forward, tailstock backward

With these keys the tailstock is moved forward or backward.

Via NC program
M20 .... tailstock backward (rear final position)
M21 .... tailstock forward (clamping position)

Tool Turret

In JOG mode this key swivels the tool turret for one position.

Spindle Speed Override

The set spindle speed value S can be changed manually.

Setting range: 50 - 120 % of the pro-
                              grammed spindle speed
Step width: 10% per key pressure
100% spindle speed: 100% key

Spindle STOP

This key stops the running of the main and counter spindle and driven tools. Before stopping the spindle you must stop the slides.

Spindle START

This key continues the programmed run of the main and counter spindle and driven tools.

Programming
see "M-functions" forwards in this chapter

Rotation directions
see "Axes description" forwards in this chapter

Coolant

With this key the coolant device is switched on and/or off.
The LED indicates the running of the coolant pump.
With open chip guard door the coolant pump runs only as long as the key is pressed.

If this key is pressed in operating mode AUTO-MATIC after the coolant has been switched on in the program with M8, the coolant pump is switched off and the LED flashes. Switch on again by pressing the key again.

Programming:
M8   Coolant on
M9   Coolant off

EMCO

Chip Conveyor (option)
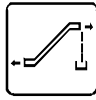
Switch on chip conveyor:
Forward: press key less than 1 second, forward run for about 35 s (LED illuminates).
Backward: press key less than 1 second, backward run as long as key is pressed (LED flashes slowly), then forward run for about 35s.
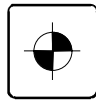
The running time of the chip conveyor is set in the counter C8 (by the manufacturer, approx. 35 seconds).
The intermission time is set in the counter C12 (by the manufacturer 300 seconds).

Interrupt chip conveyor:
For emptying etc. simultaneously press AUX ON and chip conveyor key (LED flashes quickly). Switch on again with chip conveyor key.

Reference Point

By pressing this key in operating mode REFERENCE approaching the reference points in all axes is carried out.

Auxiliary OFF

This key switches off the auxiliary drives of the machine. Effective only at spindle and program stop.

Auxiliary ON

With this key the auxiliary units of the machine are made ready for operation (hydraulics, feed drives, spindle drives, lubrication chip conveyor, coolant).
The key must be pressed for about 1 second.
Short pressing of the AUX ON key is a quit function and effects a lubricating impulse of the central lubrication.
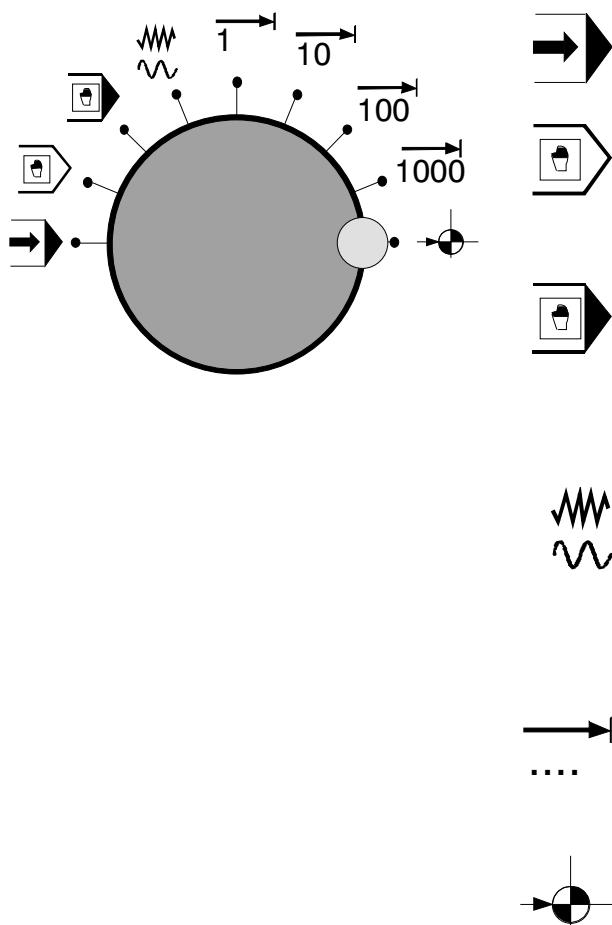
Release prior to referencing

If the slide must be released prior to referencing

(e.g. from a collision exposed position), press [AUX] and the respective direction key simultaneously.

Swivelling free the tool turret

If, in case of an accruing alarm, the tool turret must be swivelled free, press the keys [AUX] and

[ ] simultaneously.

Mode Selection Switch

Automatic
Control of the machine by automatic working off of programs

Edit
Edit part programs

MDI
Manual Data Input
Control of the machine by working off of blocks or sequences of blocks. Input of blocks via operation panel.
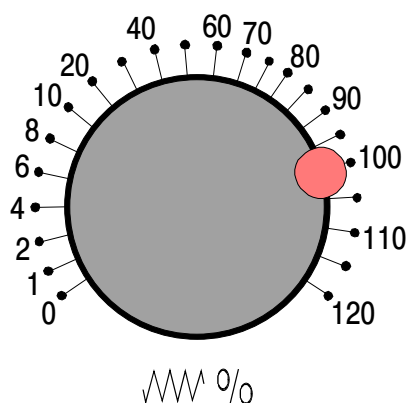
Manual operation
Conventional traversing of the machine by continuous movement of the axes via direction keys or incremental movement of the axes via direction keys or handwheel.

Incremental feed 1 - 1000
See manual operation, traverse increments with fix step width from 1 to 1000 increments (µm or inch/10000).

Ref
Manual approaching the reference point (Ref)

Feed Override Switch

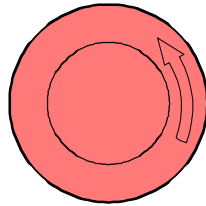The rotary switch with 20 positions allows to alter the programmed feed F (meets 100 %).
The selected feed value F in % is displayed at the screen.

Setting range:
0 % to 120 % of the programmed feed.
In rapid feed 100 % will not be exceeded.

EMCO

EMERGENCY OFF

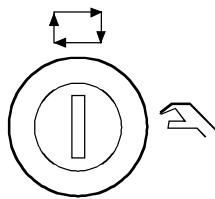Actuate the red button only in emergency situations.

Effects:
Normally all drives are stopped with maximum deceleration by EMERGENCY OFF.

Unlock: Turn button

To continue press the following keys:
RESET, AUX ON, OPEN and CLOSE door.

Key Switch Special Operation

The key switch can be set to "AUTOMATIC" or "SETUP" (manual) position.
Only by pressing the consent key the respective function of the key switch will be released (see consent key).
This key switch allows some dangerous movements with open door in tipping operation (with consent key).

Danger:

Active special operation increases the danger of accidents.
The key must be used only by authorized persons, who have the knowledge about the dangers and are careful in operation.
Keep the chip guard door closed also during setup operation.

Take off key always after working in special operation (danger of accidents).

Observe the local safety regulations (e.g.: SUVA, BG, UVV ....).
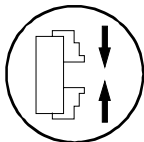
Key Switch Data Protection
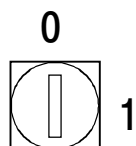
Position 0
- Part program input is locked
- Tool wear can be compensated

Position 1
- Part program input is released
- Possible further inputs:
  Zero offsets, tool geometry, setting data
- With active single piece key the limit proximity switches of the clamping device are not active (for test run without workpiece)!
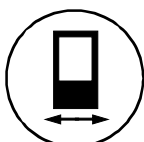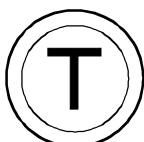
Additional NC START Key

Additional Key Clamping Device

The additional keys have the same function as on the machine control panel (Siemens).
(Double equipment for better comfort).

Key "Close Door " with Door Automatic

Notes see "Open Machine door"

Consent Key

Axis movements via direction keys and tool turret movements with open door are admitted by pressing the consent key (prerequisite key switch in position SET UP).
To open the chip guard door the consent key must be pressed, too.
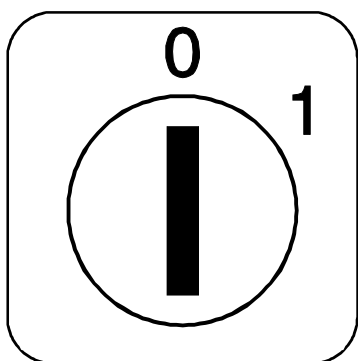
Main Switch

Function:

0 - OFF
1 - ON

The main switch has no emergency off function, that means the drives run out (unbreaked).
The main switch can be locked (unauthorized operation of the machine).
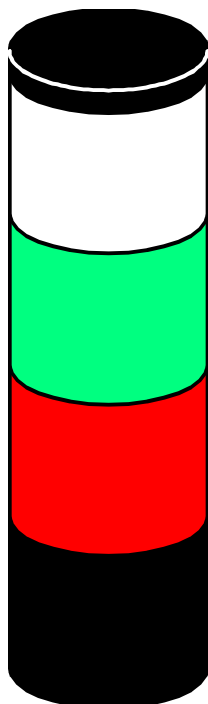
Signal Lamp (Option)

The signal lamp is divided in three indication fields (white, green, red) which indicate the operating status of the machine.

Operating statii:

WHITE
Flashing .... lack of raw parts
Permanent light .... loading active

GREEN
Flashing .... automatic operation selected
Permanent light .... automatic operation active

RED
Flashing .... Alarm / EMERGENCY OFF

EMCO

Switch On/Off Sequence

Switch ON the machine

Switch on the main switch at the electrical cabinet.

Open Chip guard door (with consent key) and close again; this sequence checks the function of the door safety switch.

Press AUX ON (auxiliary drives) for at least 1 second to activate the auxiliary drives (Hydraulic, axes, coolant).

After a longer period of standstill press the key multiple (every press a lubrication pulse).

REF mode is active after switching on the machine and the auxiliary drives (independent from the position of the mode selection switch)
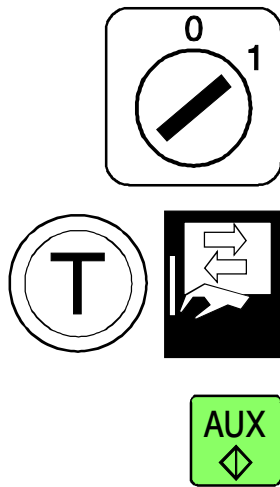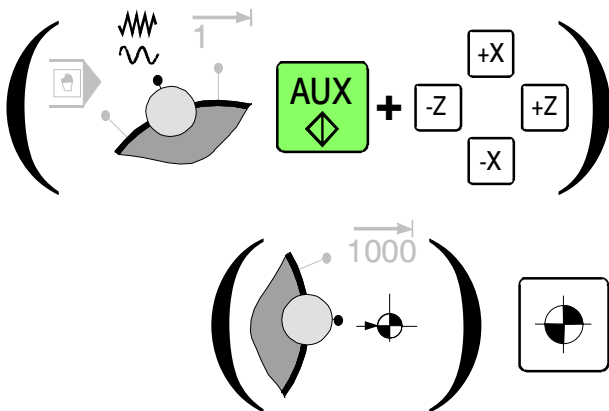
Approach reference point

When the slide is near X+ or Z+ end position or in an area with the danger of a crash, select JOG mode and traverse free the slide by simultanous pressing the AUX ON key and the corresponding direction key.

Press reference key.
The slides approach the reference point first in X and afterwards in Z.

Select desired operation mode.

Notes:
- After reaching the reference point the software limit switches are active.
- With an active tool turret alarm the key AUX ON must be pressed simultanous with the reference key, to approach te reference point.
- When an overtravel message is shown always with approaching the reference point, switch on the machine and press simultanuously the keys CAN and P_Q until the screen display appears.
  This error can occur e.g. if the axis position was altered when the machine was switched off (motor shaft turned).

Switch OFF the Machine

Press AUX OFF.
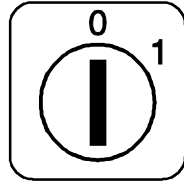
Switch off main switch

Notes:
- Switching off the machine occurs with the main switch.
  We recommend to switch off the machine only at ready position of the tool turret.

- Interruptions of operating are done with the RESET key.
  RESET stops all running machine functions.

Open Machine Door

Danger:
Do not remove or bypass control or safety devices of the machine.
These devices are installed for your own safety and in case of elimination will increase the risk of accidents.

For reasons of safety door safety switches are used for supervising.

Note:
After the machine has been switched on the door must be opened and closed once as otherwise message "door open" is emitted.
This measure serves to check whether the door safety switch is operating.
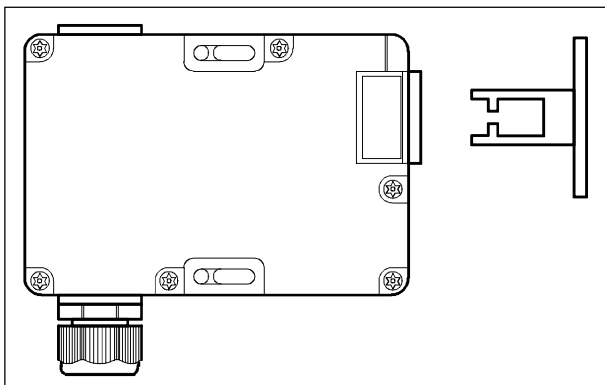
Door safety switch
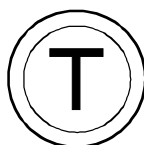
If NC START-key is pressed with open door, message "door open" appears. The NC START command will not be executed.

To open the chip guard door the consent key must be pressed.
Thus the door lock bolt will be released and the door can be opened.

Therefore the following conditions must be met:

1. All spindle drives must stand still.
2. No part program must be active.

EMCO

Door Automatic (Option)

Open door automatically
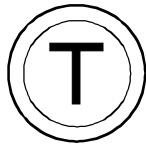
On machines that are equipped with the acessory "automatic door", the door will be opened automatically when releasing the locking bolt (with consent key T).

In automatic mode the door will be opened automatically with programmed stop or program end (M0, M1, M2, M30).

Close automatic door
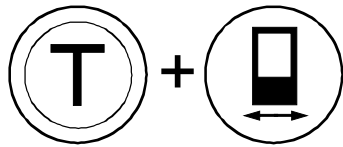
Machines with automatic door have a "close door" key instead of the additional "NC start" key.

When this key is pressed simultaneous with the consent key the door will be closed automatically.

Automatic program start

With active M96 the program will be started automatically after closing the door.

M96 is self-holding until switching off the machine, for safety reasons it must be activated again after switching on the machine.

M96 is disactivated by M97.

M96/M97 also can be used without automatic door.

Clamping Device Control

Purpose

With automatic workpiece loading for safety reasons prior to switching on the main spindle, it is automatically asked if the clamping device is within the permitted clamping area.
For this purpose the control must know certain data of each clamping device.

Sensors for the clamping device control

Function
(Example jaw chuck)

With opened jaw chuck (1) the clamping cylinder with bore is in front (right) position.
The sensor (2) measures the distance (b1) and thus the condition "chuck open" is defined.

When closing the jaw chuck the piston (1) moves backwards with the tensioning tube (traversing movement a), thus the measurable distance changes (b).
The new distance (b2) defines the condition "chuck closed and workpiece clamped".

If the chuck is closed without workpiece, the piston (1) moves until stop to the left. The distance b is bigger than b2 and thus the control knows that no workpiece is clamped and emits an alarm.

Condition displays of the clamping device

The condition of the clamping device is indicated at the additional clamping device keys:
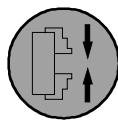
Clamping device key dark
Clamping device open
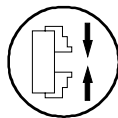
Clamping device key clear
Clamping device closed, workpiece clamped; the clamping device pressure is achieved, the final position of the clamping device has not been reached.
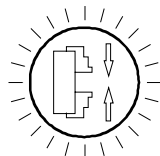
Clamping device key blinks
Clamping device closed, no workpiece clamped; the clamping pressure is achieved, the final position of the clamping device has been reached.

EMCO

Adjustment of the clamping device control

During this procedure the final positions of the clamping device for the clamping condition open and closed are adjusted (see "clamping device key dark" and/or "clamping device key blinks").

Furthermore, the permitted clamping area is defined (see "clamping device key lights").

Note:
The adjustment of the clamping device control must be carried out:
- after each clamping device change
- in case of the emission of the alarm "clamping device open missing"

Clamping and safety areas

Procedure

Note:
Prior to the adjustment check the machine configuration for the clamping device (chuck/collet)!
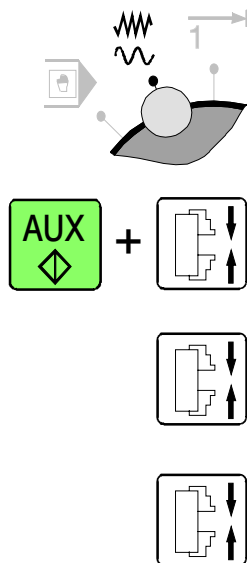
- Set operating mode selector to "JOG".

- Delete actual datas
  Delete the actually stored values by actuating simultaneously the key "AUX ON" and the clamping device key.

- Open clamping device
  Release key "AUX ON" and actuate clamping device key once; the clamping device will be opened.

- Close clamping device
  Actuate clamping device key once; the clamping device will be closed.

- The control computes and stores the permitted clamping area with the help of the identified final positions.

Bridging the final position clamping device detectors

0

1
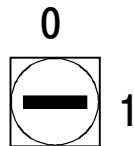
With "Key switch data protection"

Key switch - position "1"
With active single piece key the final position sensors of the clamping devices are not active.

Application
test run without workpiece

Programming with H-word

If clamping is to be carried out in an area exceeding the limits of the permitted clamping area, the final position control must be switched off (clamping in the safety area).

maximum clamping area

permitted clamping area
(clamping device closed, workpiece clamped)

Safety area

required clamping area
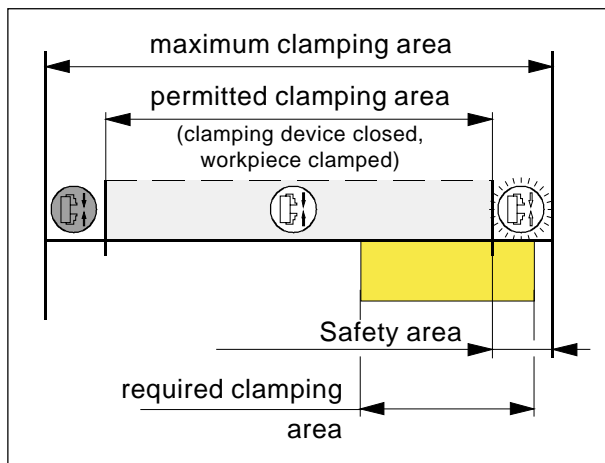
Clamping area outside of the permitted area

⚠

Caution:
Consider that with this possibility the survey function is permanently switched off.
For this reason it may only be used if this clamping state (clamping in the safety area) cannot be changed by other measures (e.g. collet change, chuck change, etc.).

Switching off the final position detector

M-code   M93

Activate the final position detector

M-code   M91/M92

EMCO

Standard Tailstock

Programming

M20   Tailstock quill backward (back position)

M21   Tailstock quill forward
The tailstock quill moves to the adjusted clamping position.

If the tailstock quill overtravels the adjusted position (JOG), alarm occurs.

Adjusting the detection of the tailstock quill

The position of the tailstock quill is monitored by the proximity switches (1) and (3).
The monitoring of the forward position of the tailstock quill can be adjusted by traversing the proximity switch (1).

• Chose the JOG-mode.

• Set the deliberated pressure for the tailstock quill (see "Hydraulic unit" in chapter "B Description").

• Clamp workpiece in clamping device, traverse tailstock into deliberated position and clamp it.

• Move Tailstock quill forward by pressing the key.

• Loosen screw (2) for the proximity switch (1) with an allen wrench, key size 5.

• Traverse proximity switch into the right direction, until message "Tailstock in intermediate position" occurs (arrow 1).

• Retraverse proximity switch (1) in left direction, until the message just disappears (arrow 2).

• Clamp proximity switch in this position.

Control of the adjusting
• Move tailstock quill backwards by pressing the key.

• Press key   to move tailstock quill forwards.
There may no message appear on the screen.
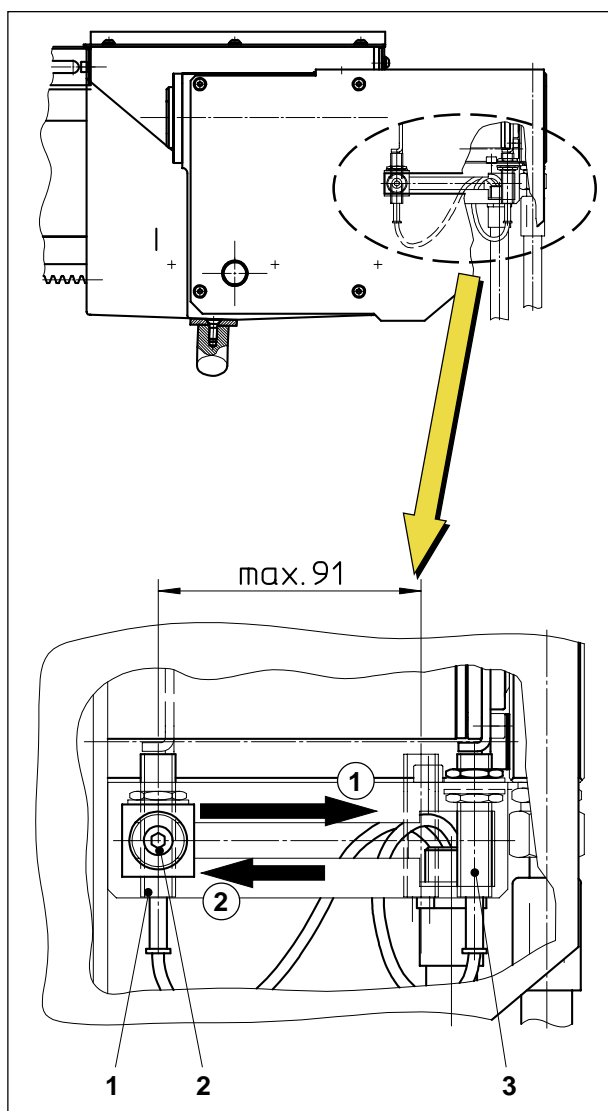
Adjusting the proximiy switch

Para-meter-Nr.	Address	Description
0030	D0060	Counter (momentary position)
0031	D0062	Clamping position
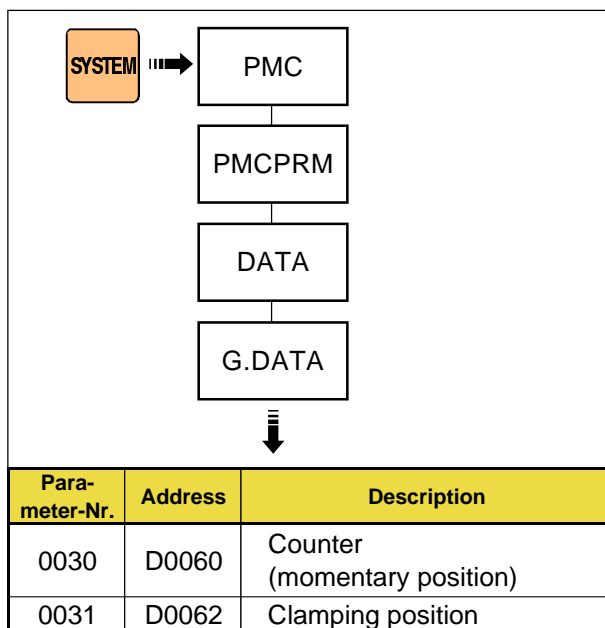
Call-up of parameters "D0060 / D0062" on the screen
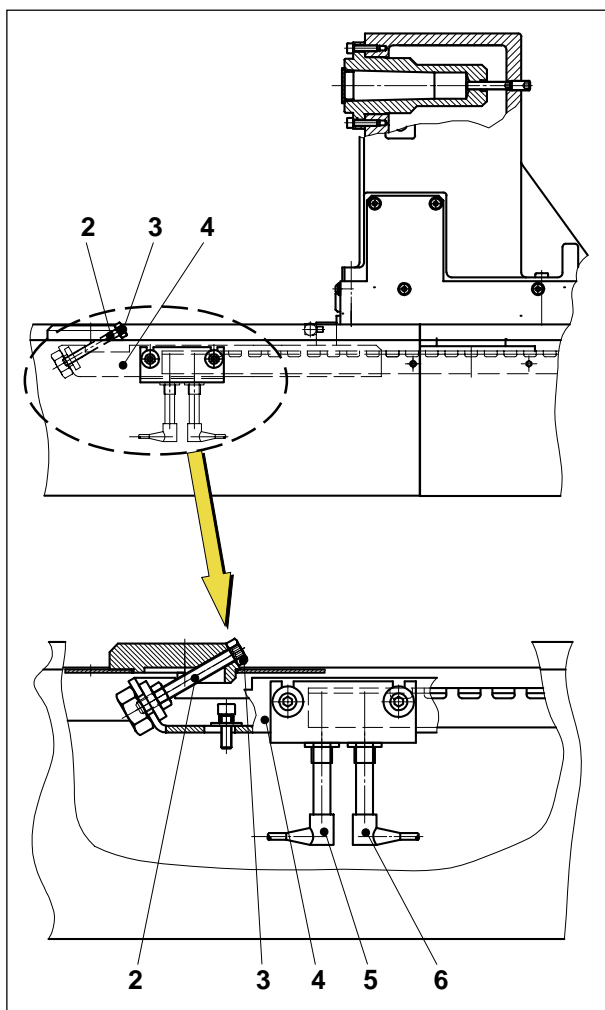
Detail drawing - tailstock, adjustment of the stroke

Automatic Tailstock

Programming

M20   Sleeve backward (rear final position)
M21   Sleeve forward (first to be done manually)
          The tailstock moves to the clamping position approached last in JOG
M20 H...   Sleeve backward (not less than H11)
M21 H...   Sleeve forward
          The sleeve moves to the Z-position indicated under H

The tailstock moves until 20 mm before final position in rapid motion, then with reduced feed. If the sleeve passes over the determined positioned (JOG or H..), there is an alarm.

Adjustment of the tailstock stroke monitoring

The position of the tailstock is monitored by the proximity detectors (5) and (6).
Setting the monitoring is carried out by displacing the connecting link (4).

- Set operating mode "JOG".

- Call up parameter "D0060" via the softkeys "PMC ➠ PMCPRM ➠ DATA ➠ G.DATA in the system menu.

- On the screen the parameter "D0060" is displayed with a value (1), e.g. "0011".

- Loosen counter nut (3).

- Set setting screw (2) flush with counter nut (3) by means of Allan key.

- Insert workpiece and approach tailstock against the workpiece.
  (Setting the tailstock contact pressure see under "hydraulics" in chapter "B Description").

- Turn setting screw (2) out of the tailstock until the value (1) of the parameter "D0060" on the screen shows more by the number 1 (e.g. change of the value from "0011" to "0012").

- Turn setting screw back again, until the value (1) counts back again by "1" (from "0012" to "0011").

- Turn screw (2) still another rotation into tailstock.

- Secure setting screw (2) with the counter nut (3) in this position.

- Exit from the parameter display with [RESET].

EMCO

Spindle Brake

Programming

M10   Main spindle brake on
M11   Main spindle brake off

Manual operation

JOG mode

Brake on: AUX ON [AUX] + Spindle Stop [⊡▽]

Brake off: Reset [╱╱]

Note
The brake force is set at works (hydraulics) and must be altered by a service technician only.

C-axis

For milling surfaces (square, hexagon, etc.) the C-axes and the tool slide must be moved in a certain relation to each other (=hobbing).

With the "polar coordinate-interpolation" such surfaces can be programmed easily.

Switching on the C-axis

after M5 with M52
G0 C0..360.xxx   (angle in degree)

Deselection of the C-axis

M53 or M3, M4, M5, M10

Note
Once in a program the C-axis has to be referenced after M52 with the command G28 C...

Operation JOG of the C-axes

-C   +C

By pressing the respective "C"-key on the operating panel of the control, the C-axis can be moved manually in JOG operation.
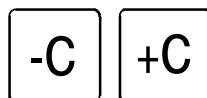
Workpiece Collection Device (Option)

The swivelling operation can only be carried out via CNC program. Swivelling in and out is possible only with closed door.

Programming

With feed-hold function

M 23   Collecting tray backward
M 24   Collecting tray forward

Without feed-hold function

M 230   Collecting tray backward
M 240   Collecting tray forward below spindle
M 241   Collecting tray forward into waiting position

Tool Turret

Manual operation

JOG mode, the tool turret swivels for one position at each key press.

Swivel free at active alarm:

Press AUX-ON and tool turret key simultanously.

Influencing the Tool Turret in AUTO-MATIC and JOG Mode

Aborting the swivel sequence

When aborting with EMERGENCY OFF or RESET, the tool turret looses its reference point.

Remedy:
Unlock EMERGENCY OFF, RESET, AUX ON, chip guard door open-close, Traverse tool turret with AUX ON and X, Z key out of the danger area, referencing with tool turret key.

EMCO

Macro Variables

The control FANUC-21i offers calculating variables in order to create custom macros.
For a detailed information see Description - Chapter 15).

Calling up Variables

The variables will be shown in the function "OFFSET/SETTING ➟ ▷ [+] ➟ MACRO:

#1 - #33: local variables
cleared with Power OFF

#100 - #149: Common variables
Cleard with Power OFF

#500 - #531: Common variables
Hold on with Power OFF

#1000 and above: System variables
To read / write NC-data

Workpiece Counter

Parts Count
(corresponds to system variable #3901)

Shows the actual workpiece number. It is also displayed in the main screens at the right upper side.

Parts Required
(corresponds to system variable #3902)

Shows the nominal workpiece number.

Parts Total

This value can only be set by parameter 6712. It will be incremented with "1" by M2 or M30.

The Fanuc parts counter is located in the function

SETTING/OFFSET ➟ SETTING ➟ [PAGE ▼] ➟ [PAGE ▼].

Settings can only be made in the MDI-mode.

Function

- If a nominal workpiece number is entered in the variable "Parts Required", "Parts Count" will count from 0 up to the nominal workpiece number.
  If the nominal workpiece number is worked off, message "2011 - Required Number of Pieces" will appear.

- If "Parts Required" and "Parts Count" will be set to 0, both variables counts from 0 upwards.

Programming

The parts counter can be programmed by using the macro variables, programming a block with e.g. Nxx #500=#500+1.
The number of machined parts will be displayed in "OFFSET/SETTING no. 500 of MACRO sreen.

Example
250 pieces are to be machined.

- Set "Parts Required = 250".
  The workpiece counter counts from 0 upwards to 250 and emits the message "2011 - Required Number of Pieces".

- Set "Parts Required = 0".
  Set "Parts Count = 0".
  The workpiece counter counts from 0 upwards to 250 and emits no message.

> **Note**
> See also "Fanuc-Control Description" - Chapter 11.4.9.

Power On Time
(corresponds to system variable #3001)

Displays the total time which the machine is switched ON.

Operating Time

Indicates total running time, excluding stop time and feed hold.

Cutting Time
(corresponds to system variable #3002)

Displays the total time during cutting with a programmed feedrate (time in active Cycle START).

Machining Time

The timers are located below the workpiece counter, together with the current date and day time.

Free Purpose

Can be used, for example, as a total timer for coolant flow.

Cycle Time

Indicates the running time of one operation (program), excluding feed hold and stop time.

Date and Time
(#3011 corresp. to date, #3012 corresp. to time)

Displays actually set date and time

Output signal	M-function		Input signal
	on	**off**	
Y 7.0	M81	M80	X 20.4
Y 7.1	M83	M82	X 20.5
Y 7.2	M85	M84	X 20.6
Y 7.3	M87	M86	X 20.7

Survey

Free Programmable Outputs

You have the possibility to set several functions for the freeprogrammable outputs with machine datas.

- The output won't be resetted with M30/reset, but only with the M-function.

- The output will be resetted after a time, adjusted with machine datas.

- By setting the output a feed hold will be triggered in this channel, in which the M-function wa pragrammed, until the belonging input will be "one".

- By setting the output a read lockout will be triggered in this channel, in which the M-function wa pragrammed, until the belonging input will be "one".

Standard Setting

- The output will be resetted with the Reset-key or with M30
  (or with the M-function)

- Feed hold and read lockout won't be triggerd.

For other settings please contact EMCO-Service Department.

EMCO

Survey Program Input, Administration

Program input occurs in EDIT mode (⌖⟩).

Listing of the most frequent edit functions:

List programs

- Press the key PROG (eventually 2x).
- The screen shows all stored programs.
- Scroll list: PAGE▼ and ▲PAGE.

Call program

- List programs (PROG).
- Enter program number e.g.O1234.
- Press softkey O-SRH.
- When the entered program is in memory, it will be caleed up, otherwise alarm 71 is shown.

Create new program

- List programs (PROG).
- Enter free program number e.g. O5678.
- Press the key sequence INSERT, EOB and INSERT.
- The new program is opened and you can enter the program blocks.

Rename program

- Call program.
- Put cursor on program number (e.g. O1234), enter new program number and press key ALTER.

Copy program

- Call the program to be copied.
- Press softkey OPRT and the right menu extension key ▷.
- Press softkeys EX-EDT, COPY, ALL.
- Enter new program number (without O) and press the key INPUT.
- Press the softkey EXEC.

Delete program

- List programs (PROG).
- Enter program number e.g.O1234 and press the key DELETE.
- The entered program will be deleted.
- Delete all programs: O-9999 DELETE.

Program block input

- Enter NC commands (e.g. N90G0X9Z-6).
- Take over the block with EOB INSERT.
- Delete backward while input: CAN
- Entering a block number at the beginning of a block is not necessary.

Insert block

- Put the cursor on the block end (EOB sign ";") of that block, that should be before the new block.
- Enter the new block.

Delete block

- Put the cursor on the beginning of the block to be deleted and press the keys EOB and DELETE.

Insert word

- Put the cursor on that word that should be before the new word.
- Enter new word (address and value, e.g. X0) and press key INSERT.

Find word

- Enter word (e.g. X50) or address of the word to be found (e.g.: X).
- Press cursor key ↓.

Alter word

- Put cursor on the word to be altered.
- Enter new word and press the key ALTER.

Delete word

- Put cursor on the word to be deleted.
- Press the key DELETE.

G-Commands - Survey

The Fanuc control uses for the G commands the allocation groups A, B, C; that means one command has different numbers in different groups.
The Fanuc control is programmed with the commands of allocation group C, but the Fanuc manuals describe the allocation group A.

Use the following table to find the description of the G commands in the Fanuc manuals.

Example
G71 Inch input can be found in the Fanuc manuals as G21.

Code Grp. C	Meaning	Code Grp. A	Code Grp. C	Meaning	Code Grp. A
G00	Rapid motion	G00	G54	Selection workpiece-coordinate system 1	G54
G01	Linear interpolation	G01	G55	Selection workpiece-coordinate system 2	G55
G02	Circular interpolation/clockwise	G02	G56	Selection workpiece-coordinate system 3	G56
G03	Circular interpolation/counter-clockwise	G03	G57	Selection workpiece-coordinate system 4	G57
G04	Dwell time	G04	G58	Selection workpiece-coordinate system 5	G58
G05[*]	Machining in quick cycle	G05	G59	Selection workpiece-coordinate system 6	G59
G07[*]	Interpolation with fictitious axis	G07	G65	Macro call	G65
G07.1 (G107)	Cylindric interpolation	G07.1 (G107)	G66	Modal macro call	G66
			G67	Modal macro call end	G67
G10	Data setting	G10	G68[*]	Axis mirroring for double turret heads on or tare cut mode	G68
G11	End data setting	G11			
G12.1 G112	Mode "Polar coordinate interpolation"	G12.1 (G112)	G69[*]	Axis mirroring for double turret head off or tare cut mode end	G69
G13.1 G113	End mode "Polar coordinate interpolation"	G13.1 (G113)	G70	Input in inch	G20
G17	Selection level XpYp	G17	G71	Input in mm	G21
G18	Selection level ZpXp	G18	G72	Finishing cycle	G70
G19	Selection level YpZp	G19	G73	Contour roughing cycle (ID/OD)	G71
G20	Longitudinal turning cycle (AD/ID)	G90	G74	Face roughing cycle	G72
G21	Thread cutting cycle	G92	G75	Pattern repeating cycle	G73
G22[*]	Memorized traverse limit on	G22	G76	End face cut-in cycle	G74
G23[*]	Memorized traverse limit off	G23	G77	Cut-in cycle	G75
G24	Facing cycle	G94	G78	Multiple threading cycle	G76
G25[*]	Spindle speed monitoring on	G25	G80	Drilling cycle end	G80
G26[*]	Spindle speed monitoring off	G26	G83	Cycle for face drilling	G83
G27	Reference position - return check	G27	G84	Cycle for face tapping	G84
G28	Return to reference point	G28	G85	Reaming cycle axial	G85
G30	Return to 2nd reference point	G30	G87	Side drilling cycle	G87
G31	Cancel remaining path	G31	G88	Side tapping cycle	G88
G33	Thread and oil groove cutting	G32	G89	Reaming cycle radial	G89
G34	Thread cutting with variable lead	G34	G90	Absolute programming	—
G36	Autom. tool correction X	G36	G91	Incremental programming	—
G37	Autom. tool correction Z	G37	G92	Set coordinate system or maximum spindle speed	G50
G40	Deselection cutter radius compensation	G40			
G41	Cutter radius compensation left	G41	G92.1[*]	Workpiece coordinate system	G50.3
G42	Cutter radius compensation right	G42	G94	Feed per minute	G98
G50.2 (G250)	Polygonal turning end	G50.2 (G250)	G95	Feed per rotation	G99
			G96	Constant cutting speed	G96
G51.2 (G251)	Polygonal turning	G51.2 (G251)	G97	Const. cutting speed off	G97
G52	Setting local coordinate system	G52	G98	Return to initial level	—
G53	Setting machine coordinate system traverse movements only in rapid motion	G53	G99	Return to level mit point R	—
				Commands with different group cycles	
			*)	see Fanuc control description	

EMCO

Short description G-Commands

On the following pages you will find a short description of the G-commands of the control Fanuc 21 TB, allocation group C.
This description represents an extract from the programming instructions for the control Fanuc 21 TB and is mainly intended as programming aid.

For detailed information referring to the individual commands see the programming instructions for the control Fanuc 21 TB.
In case of ambiguities and contradictions the specifications in the original Fanuc instructions are valid.

Note:
In these instructions the command classification of the allocation group C is described as used on the Fanuc control at the EMCO machine.
In the descriptive material of Fanuc the allocation group A is described - if you use the Fanuc material you therefore have to recode the commands according to the scheme (see survey G-commands).

G00 Rapid Traverse

Format
N.... G00 X(U)... Z(W)...

The slides are traversed with max. speed to the programmed target point (tool change position, starting point for the following machining procedure).

Notes

• A programmed slide feed F is suppressed during G00.

• The rapid speed is set definitively.

• The feed override switch is limited to 100%.

Example

absolute G90
N50 G00 X40 Z56

incremental G91
N50 G00 U-30 W-30.5
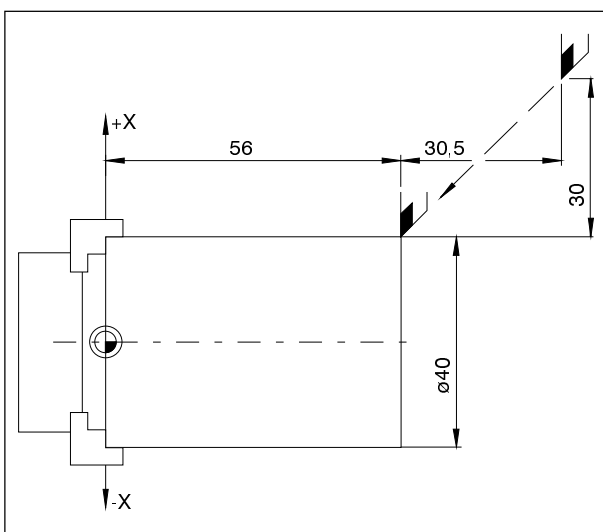
Absolute and incremental values G00

G01 Linear Interpolation (Feed)

Format

N... G01 X(U)... Z(W).... F....

Linear slide movements (face, longitudinal, taper turning) at the programmed feedrate.

Example

absolute G90

N.. G95
.....
N20 G01 X40 Z20.1 F0.1

incremental G91

N.. G95 F0.1
.....
N20 G01 X20 W-25.9
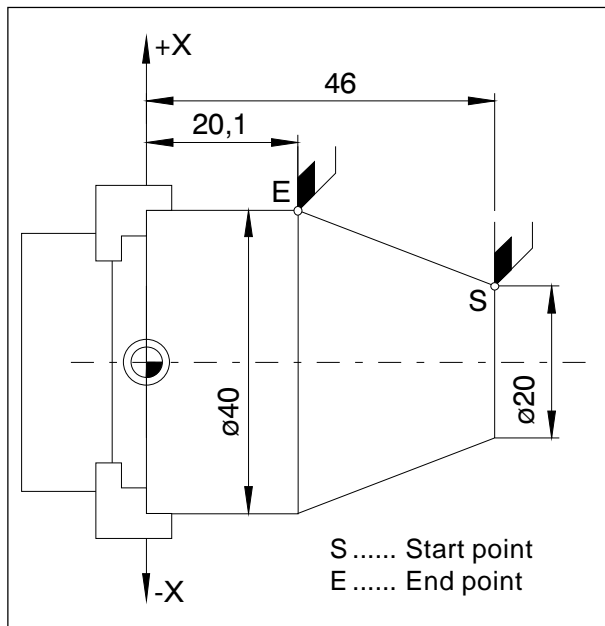
Absolute and incremental measures for G00

S ...... Start point
E ...... End point

Insertion of Chamfers and Radii

Example

```
....
N 95    G 01    X 26    Z 53
N 100   G 01    X 26    Z 27    R 6
N 105   G 01    X 86    Z 27    C 3
N 110   G 01    X 86    Z 0
....
```

Notes

- Chamfers and radi can be inserted between two G00/G01 movements only.

- The movement, which is programmed in the second block, has to start at point b (drawing). With incremental programming the distance from point b has to be programmed.

- With single block mode the tool stops first at point c and then at point d.

- If the movements in one of the blocks are so short, that there is with inserting a chamfer or radius no intersection point, alarm no. 055 occurs.

Insertion of chamfers and radii

EMCO
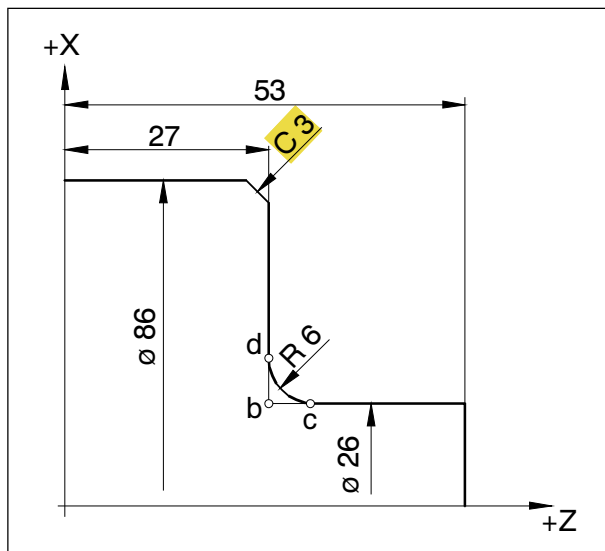
Direct Drawing Input

	Commands	Tool movements
1	$X_2...$ $(Z_2...)$ **A...** **Note:** **Bold printed commands can be executed only with the option comfort programming.**	
2	**$A_1...$** **$X_3...$ $Z_3...$ $A_2...$**	
3	$X_2...$ $Z_2...$ $R...$ $X_3...$ $Z_3...$ or **$A_1...$ R...** **$X_3...$ $Z_3...$ $A_2...$**	
4	$X_2...$ $Z_2...$ $,C...$ $X_3...$ $Z_3...$ or **$A_1...$,C...** **$X_3...$ $Z_3...$ $A_2...$**	
5	$X_2...$ $Z_2...$ $R_1...$ $X_3...$ $Z_3...$ $R_2...$ $X_4...$ $Z_4...$ or **$A_1...$ $R_1...$** **$X_3...$ $Z_3...$ $A...$ $R_2...$** **$X_4...$ $Z_4...$**	

	Commands	Tool movements
6	$X_1...$ $Z_1...$ $,C_1...$ $X_3...$ $Z_3...$ $,C_2...$ $X_4...$ $Z_4...$ or **$A_1...$ $,C_1...$** **$X_3...$ $Z_3...$ $A_2...$ $,C_2...$** **$X_4...$ $Z_4...$**	
7	$X_2...$ $Z_2...$ $R_1...$ $X_3...$ $Z_3...$ $,C_2...$ $X_4...$ $Z_4...$ or **$A_1...$ $R_1...$** **$X_3...$ $Z_3...$ $A_2...$ $,C_2...$** **$X_4...$ $Z_4...$**	
8	$X_2...$ $Z_2...$ $,C_1...$ $X_3...$ $Z_3...$ $R_2...$ $X_4...$ $Z_4...$ or **$A_1...$ $,C_1...$** **$X_3...$ $Z_3...$ $A_2...$ $R_2...$** **$X_4...$ $Z_4...$**	

Note:

- Missing intersection point coordinates need not to be calculated.
  In programs angles (A), chamfers (C) and radi (R) can be programmed directly.
  The following block after a block in which C or R was specified, has to be a block with G01.
  To command a chamfer is possible only with the comma-character "**,C**". Otherwise alarm for illegal use of C-axis will appear.

- The input of angles (A) is possible with the option comfort programming only.

- The following G commands must not be used for the blocks with chamfer or radius.

  G-Codes in group 00:
  G05, G7.1, G10, G11,
  G27, G28, G30, G31,
  G92, G92.1,
  G52, G53,
  G72, G73, G74, G75, G76, G77, G78

  G-Codes in group 01:
  G02, G03, G20, G21 and G24

  They must not be used between the blocks with chamfer or radius, which define the succession numbers.

EMCO

Rotational direction and parameter of an arc

G02 Circular Interpolation Clockwise

G03 Circular Interpolation Counterclockwise

Format
N... G02 X(U)... Z(W)... I... K... F...
or
N... G02 X(U)... Z(W)... R... F...

X,Z .... End point of the arc
U,W, I,K .... Incremental circle parameters
(Distance from start point to centre of arc, I is related to X, K to Z)
R .... Radius of arc

The tool will be traversed to the target point along the defined arc with the programmed feed F.

Note

- Programming the value 0 for I and K can be omitted.
- With the input of R only an arc < 180° can be done. R is alway entered positive.

G04 Dwell

Format
N... G04 X(U)...   [sec]
or
N... G04 P...   [msec]

The tool movement will be stopped at the last reached position for a dwell defined by X,U or P - sharp edges - transitions, cleaning cut-in ground, precise stop

Note
- With address P no decimal point is allowed
- The dwell time starts at the moment when the tool movement speed is zero.
- t max. = 2000 sec, t min. = 0,1 sec
- input resolution 100 msec (0,1 sec)

Examples
N75 G04 X2.5 (U2.5)   (dwell time= 2.5sec)
N95 G04 P1000   (dwell time = 1 sec = 1000 msec)

G10 Data Setting

With the command G10 control data can be overwritten, parameters can be programmed, tool data can be written etc.

In user practice G10 is mainly advisable to program the workpiece zero point.

Attention
- The format G10 P0 X(U).. Z(W).. and/or G92 X(U).. Z(W).. is placed above all other zero point shifts (from G52 to G59)!
- The shift is not displayed on the screen!
- Attention with programs which were programmed for former control types (Fanuc 0-T, Fanuc 21-T).

  Remedy
  First call up G10 P0 X0 Z0 (reset of the zero point shift in G92) and input the following two blocks:
  G10 L2 P1 X(U).. Z/W).. ;
  G54 ;

- G92 is placed above G54 to G59 and EXT! Therfore the shifts of G54 to G59 and EXT will be added to the shift of G92!

Zero point shift G92

Format

N.. G10 P0 X(U).. Z(W)..
P0.....selection zero point shift G92

Input/Display (MDI mode)

OFFSET/SETTING ➙ "+" (2x) ➙ "W-SHIFT"

Zero point shift with workpiece coordinate systems G54 to G59

In general with this control it is advisable to use one of six preset workpiece coordinate systems G54 to G59 when setting a workpiece-zero point shift, since here all overwritten values can be controlled and reproduced on the screen at any time. (The description of the commands G54 to G59 is carried out later in this chapter.)

Format

N.. G10 L2 P1 X(U).. Z(W)
(overwrites workpiece coordinate system G54)

N.. G54
(calls up workpiece coordinate system G54 in the program)

G10 L2 .... overwriting the entered workpiece coordinate system.
P0 .... selection of the external workpiece coordinate system.
P1 (...bis P6) .. selection of the workpiece coordiante system G54 (...to G59).

Input/Display (MDI mode)

OFFSET/SETTING ➙ "WORK"
(see descriptions G54 to G59)

Notes:
- The call-up of the selected workpiece coordinate system in the program must be carried out in the next block.
- By calling up the external workpiece coordinate system the basis of all coordinate systems called up in the following is shifted by the dimension entered there.
- The selected workpiece coordinate system can be overwritten or replaced by another one within a program as often as you want by means of the format mentioned above.
- The workpiece coordinate systems overwritten in this way can be controlled after reading in under the function SETTING / OFFSET under picture WORK.

EMCO

Longitudinal turning cycle without taper

Longitudinal turning cycle with negative taper R
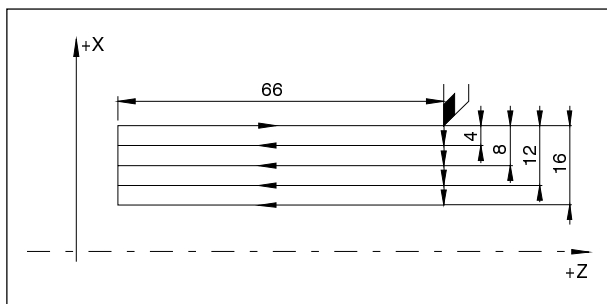
Example: G20 Longitudinal turning cycle

G20 Longitudinal Turning Cycle
(OD/ID Machining Cycle)

Format

N... G20 X(U)... Z(W)... F...   (straight)
or
N... G20 X(U)... Z(W)... R... F...   (taper)

X(U), Z(W) ..... Absolute (incremental) coordinates
of the contour point K

R [mm] .... Incremental taper dimension in X
with direction (+/-)

Notes

• This cycle is modal and will be deselected by a
G command of the same group.

• For following blocks only the altered coordinates
have to be programmed (see example).

• A negative taper parameter (-R) defines the
taper as shown in the drawing.

N100 G91
.....
N110 G20 U-4 W-66 F0.18
N115 U-8
N120 U-12
N125 U-16
N130 G00 .....
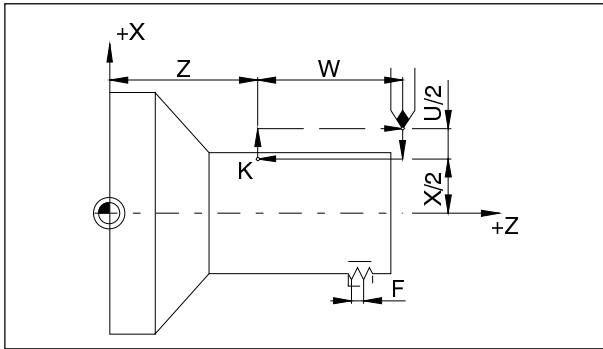
Straight thread cutting cycle
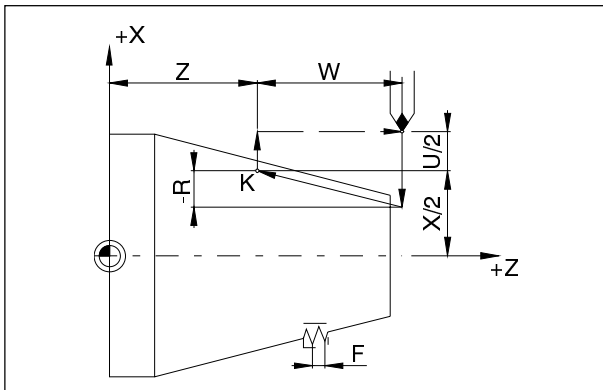
Tapered thread cutting cycle
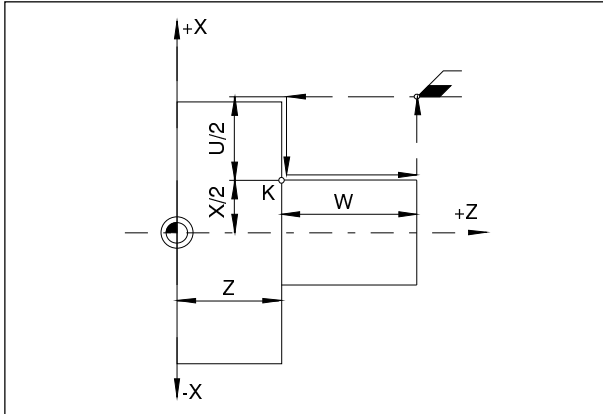
Face turning cycle without taper
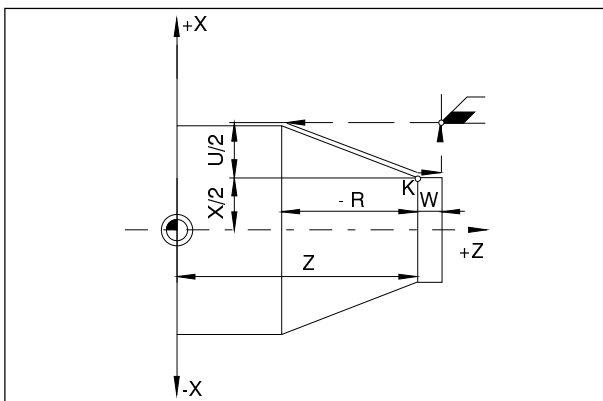
Face turning cycle with negative taper R

G21 Thread Cutting Cycle

Format

N... G21 X(U)... Z(W)... F... .... (straight)
or
N... G21 X(U)... Z(W)... R... F... .... (taper)

F .... Thread pitch [mm]

R [mm] .... Incremental taper dimension in X
with direction (+/-)

Notes

• This cycle is modal and will be deselected by a G command of the same group.

• For following blocks only the altered coordinates have to be programmed (see example).

• A negative taper parameter (-R) defines the taper as shown in the drawing.

G24 Face Turning Cycle

Format

N... G24 X(U)... Z(W)... F... .... (straight)
or
N... G24 X(U)... Z(W)... R... F... .... (taper)

R .... Incremental value of the taper in Z axis

Notes

• This cycle is modal and will be deselected by a G command of the same group (G0, G1, G2).

• For following blocks only the altered coordinates have to be programmed (see example).

A negative taper parameter (-R) defines the taper as shown in the drawing.

EMCO

G28 Reference Point Return
(approach reference point)

Format

N... G28 X(U)... Z(W)... C(H)...

X,Z,C ..... absolute coordinates of the inter-
mediate position
U,W,H ... incremental coordinates of the inter-
mediate position

The G28-instruction is used to approach the
reference point via an intermediate position (X(U),
Z(W)).
First the withdrawal is carried out to X(U) and/or
Z(W), subsequently the reference point is ap-
proached. Both movement sequences are carried
out with G00!

Note:
After the first selection of the C-axis (with M52)
the command

G28 C0

must be carried out to reference the C-axis.

G33 Thread Cutting

Format

N... G33 X(U)... Z(W)... F...

F .... Thread pitch [mm]

Straight, tapered an scroll threads can be cut.
Because of no automatic return to the start point,
the multiple threading cycle G78 will be preferred.
Machining routines like knurling are also possible.

Note

- With taper threading the thread pitch has to be
defined with the higher value in X or Z axis.
- Continuous thread cutting is possible (multiple
threads)
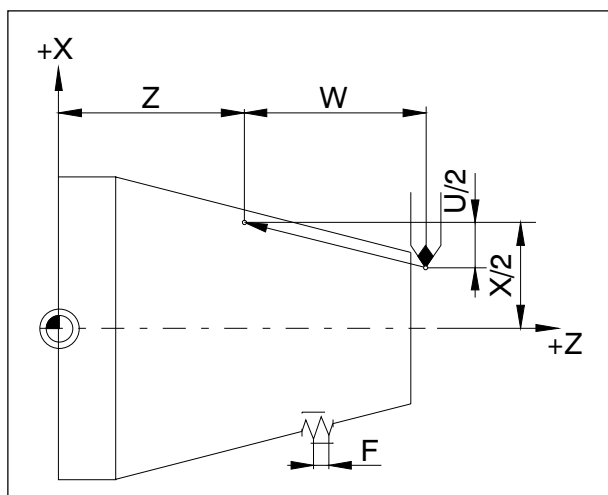
Measures for thread cutting
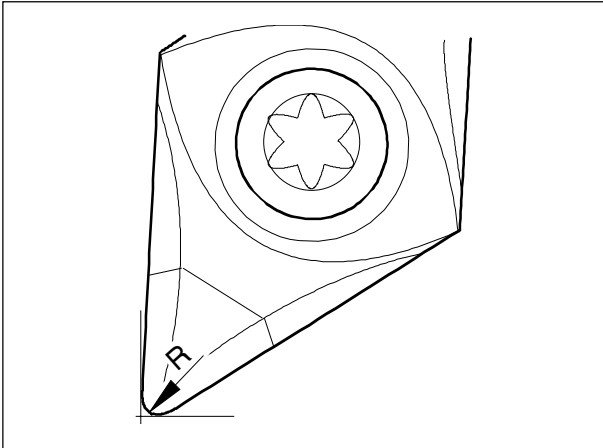
Tool tip radius and theoretical cutter tip
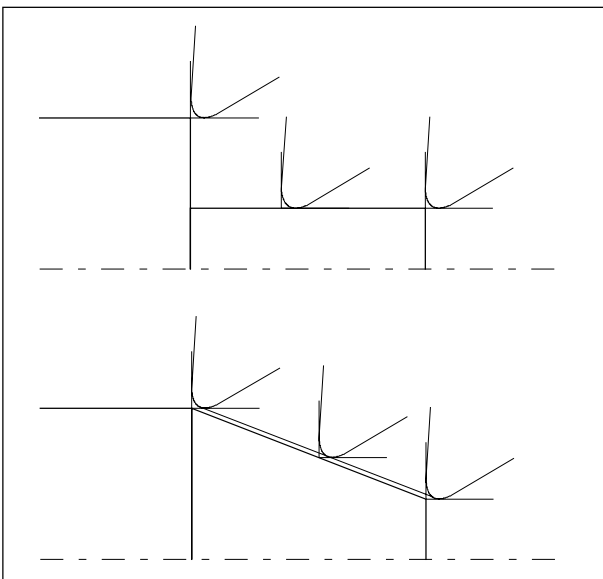
Movements parallel to the axes and oblique
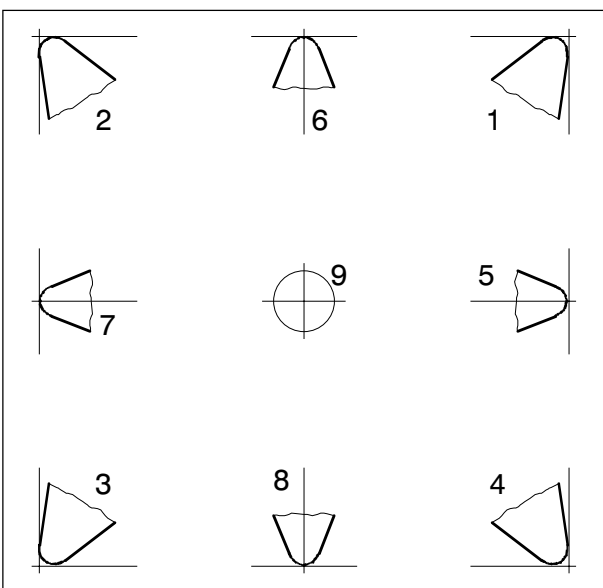
Cutter position

Cutter Radius Compensation

During tool measurement the tool tip is measured only at two points (touching the X and Z axes).

The tool offset therefore only describes a theoretical cutter tip.

This point is traversed on the workpiece in the programmed paths.

With movements in the axis directions (longitudinal and face turning) the points on the tool tip touching the axes are used.

No dimensional errors are therefore produced on the workpiece.

With simultaneous movements in both axis directions (tapers, radii) the position of the theoretical cutter point no longer coincides with the point on the tool tip actually cutting.

Dimensional errors occur on the workpiece.

Maximum dimensional error without cutter radius compensation with 45° movements:
Tool tip radius 0,4 mm ≙ 0,16 mm path distance ≙ 0,24 mm distance in X and Z.

If cutter radius compensation is used, these dimensional errors are automatically calculated and compensated by the control.

For the cutter radius compensation you must enter the cutter radius R and the cutter position T when entering the tool data.

The cutter position is indicated by a number (see draft).

To decide the cutter position look at the tool as it is clamped on the machine.

EMCO

Tool pathes with selection / cancellation of the cutter radius compensation
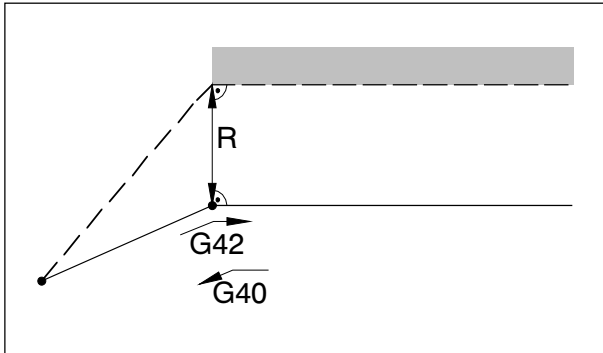
Frontal approach or leaving of an edge point
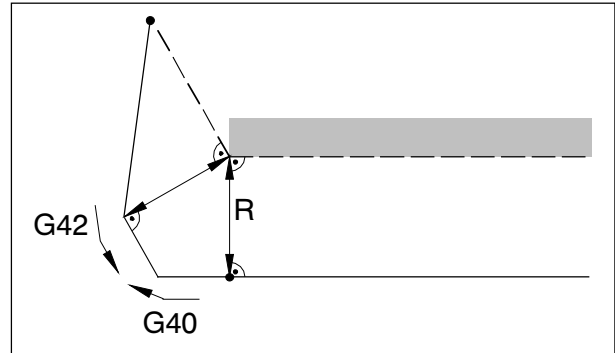
Approach or leaving an edge point at side behind

— — — programmed tool path
———— real traversed tool path

With arcs always the tangent of the end or start point of the arc will be approached.

The approaching path to the contour and the leaving path from the contour must be larger than the tool tip radius R, otherwise program interruption with alarm.

If contour elements are smaller than the tool tip radius R, contour violations could happen. The software computes three blocks forward to recognize this contour violations and interrupt the program with an alarm.

Approach or leaving an edge point behind

Tool pathes with program run with active cutter radius compensation
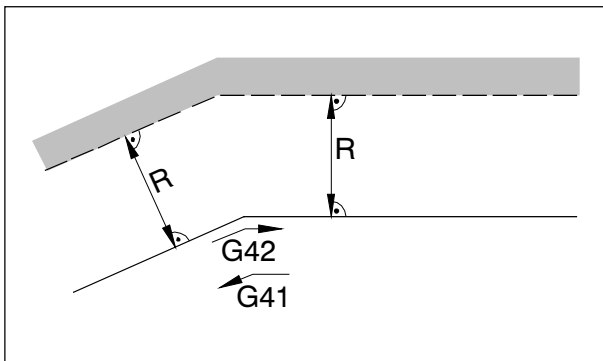
Tool path at an inner edge
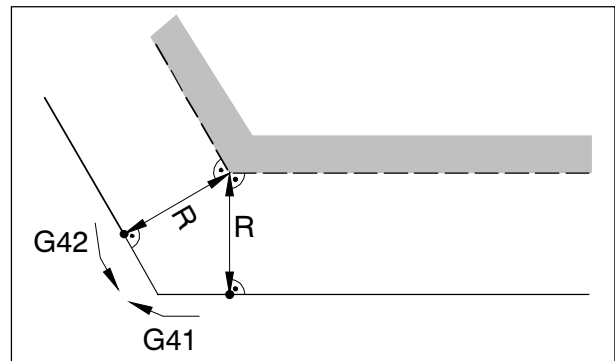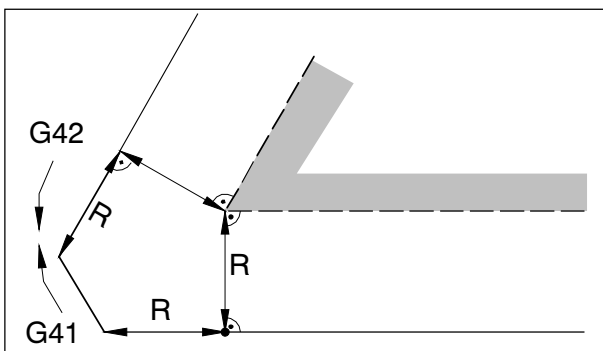
Tool path at an outer edge > 90°

— — — programmed tool path
———— real traversed tool path

With arcs always the tangent of the end or start point of the arc will be approached.

If contour elements are smaller than the tool tip radius R, contour violations could happen. The software computes three blocks forward to recognize this contour violations and interrupt the program with an alarm.

Tool path at an outer edge < 90°

G40 Cancel Cutter Radius Compensation

The radius compensation will be cancelled with G40. Cancellation is only permitted in combination with a linear traversing command (G00, G01) G00 or G01 can be programmed in the same block or as the first traversing movement after cancellation.

G41 Cutter Radius Compensation Left

If the programmed tool path (viewed in the direction of machining) is on the left of the material to be machined, the radius compensation is to be selected with G41.

Notes
- No direct change between G41 and G42 - cancel with G40 previously.
- Cutter radius R and cutter position T must be defined.
- Selection is only permitted in conjunction with G00 or G01.
- Change of tool correction is not possible with active cutter radius compensation.
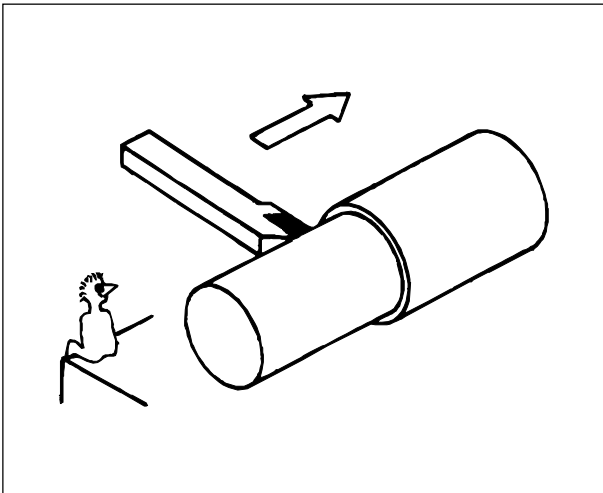
Definition G41 cutter radius compensation left

G42 Cutter Radius Compensation Right

If the programmed tool path (viewed in the direction of machining) is on the right of the material to be machined, the radius compensation is to be selected with G42.

Notes see G41!
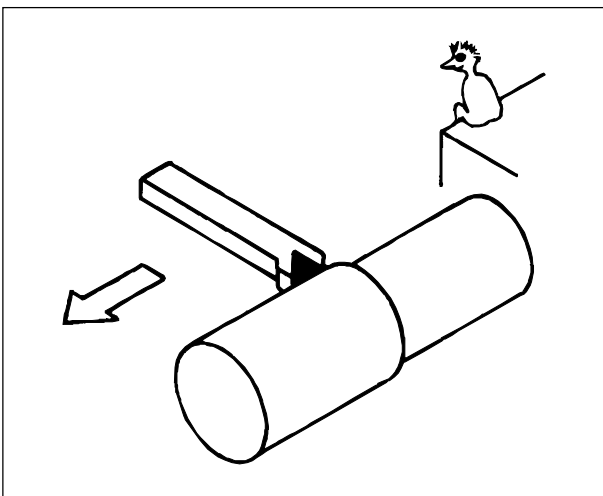
Definition G42 cutter radius compensation right

EMCO

G52 Local Coordinate

During programming in a workpiece coordinate system (G54 to G59) a "subsidiary coordinate system" can be set up within the selected workpiece coordinate system to facilitate the programming of certain functions.
Such an additive coordiante system is called a local coordinate system.

Format
N.. G52 X(U).. Z(W).. C(H).. ;
(Setting up a local coordinate system)

N.. G52 X(U)0 Z(W)0 C(H)0 ;
(Deleting a local coordinate system)

The workpiece zero point set in the selected workpiece coordinate system is shifted by the positive and negative dimensions indicated in the local coordinate system.
After deleting the local coordinate system the zero point is reset to the workpiece zero point originally set in the selected workpiece coordinate system.
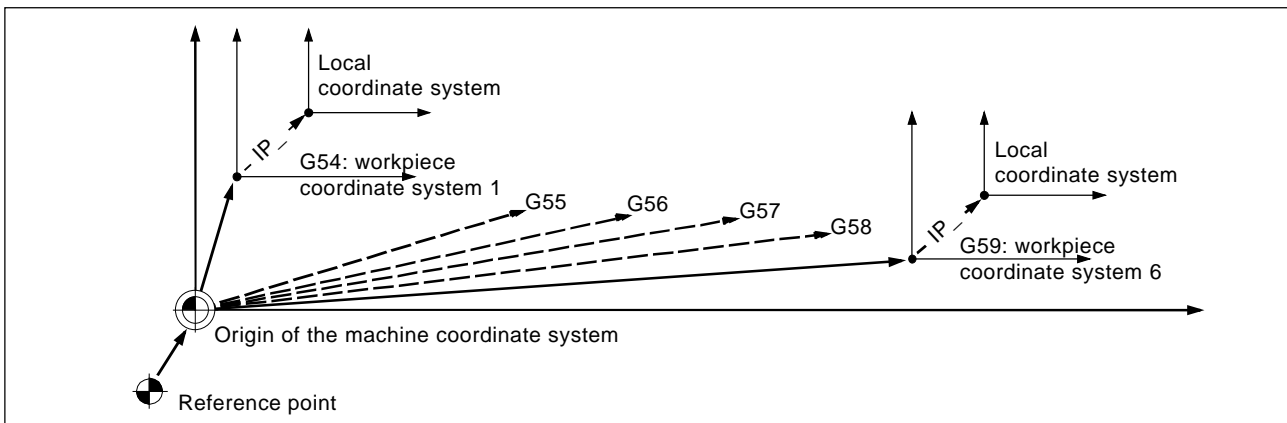
Note
- During the set-up of local coordinate systems, workpiece and machine coordinate systems remain unchanged.
- Traverse commands immediately after the selection or deselction of the local coordinate system must be absolute dimension commands.

Attention:
By pressing the RESET key the existing local coordinate system is deleted and during the subsequent block search it is overread.
Also if coordinates for any axis of the local coordinate system are set newly, the existing local coordinate system is deleted.

G53 Selection of the Machine Coordinate System

A coordinate system with its origin on the machine zero point is called machine coordinate system. After switching on the machine the machine coordinate system must be set by reference point approach prior to indicating the command G53.
G53 is a G-code effective once for the selection of the machince coordinate system. Commands are thus only valid in the block with G53 and are exclusively traversed in rapid motion!

Format
N.. G53 X.. Z..

Notes
- With command G53 cutting radius correction and tool correction are switched off.
- G53 must be indicated in absolute dimension. With incremental dimension values the command G53 is ignored.

G54 to G59 Selection of a Workpiece Coordinate System

A coordinate system used for machining work-pieces is called workpiece coordinate system. It is determined by a machining program and is to be set prior to machining start. A workpiece coordinate system already set can be altered by shifting the point of origin.
Six (specified via G54 to G59) preset workpiece coordinate systems and an external workpiece coordinate system are available:

EXT workpiece coordinate system 0
G54 workpiece coordinate system 1
G55 workpiece coordinate system 2
G56 workpiece coordinate system 3
G57 workpiece coordinate system 4
G58 workpiece coordinate system 5
G59 workpiece coordinate system 6

EMCO

All workpiece coordinate systems specified via G54 to G59 are shifted altogether via the EXT workpiece coordinate system.

EXOFS: ....external workpiece zero point value
ZOFS1 → ZOFS6 .....workpiece zero point correction value
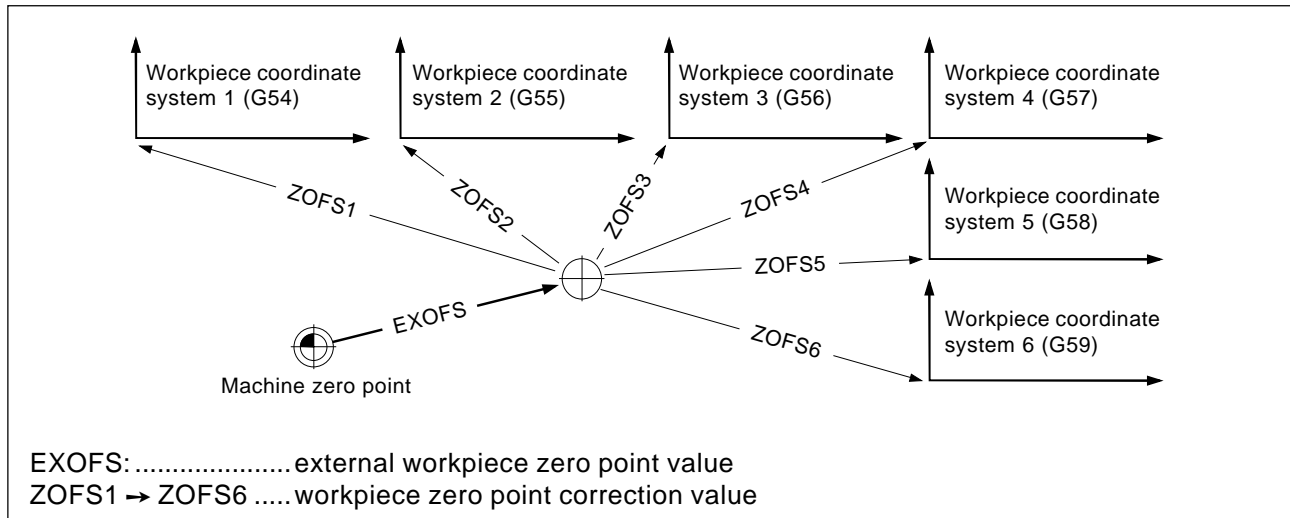
There are two possibilities to set the workpiece coordinate systems:
(1) Input via the manual input keyboard (operating mode MDI ➟ function OFFSET / SETTING ➟ softkey WORK).
(2) Programming via G10: separate shift of the individual workpiece coordinate systems including the external work coordinate system (see chapter G10).

Notes
• The workpiece coordinate systems 1 to 6 are set after reference point approach.
• The coordinate system G54 is selected during switch-on.

G70 Measuring in Inches

Format
N5 G70

By programming G70 the following values will be converted to inches:
• Feedrate F [mm/min, inch/min, mm/rev, inch/rev]
• Offset values (zero point, geom., wear, ...) [mm, inch]
• Movement pathes [mm, inch]
• Display of actual position [mm, inch]
• Speed [m/min, feet/min]

Notes
• For clearness G70 should be defined in the first block of the program
• The measuring system which was programmed last will be active - also after main power off/on.
• To get back to the origin measuring system it is best to use the MDI mode (e.g. MDI G70 Cycle Start)

Remark:
It is also possible to exchange the control from metric-system to inch-system in the "SETTING screen" ➠ "SETTING", by inserting "1" as the "INPUT UNIT".
All the coordinate systems in the control screen will be converted immediately, including all of the tool offsets.
The programs themselves will not be converted!
Before running the machine with the converted measuring system:
Switch machin OFF and ON, to perform a reference point return.

G71 Metrical Measuring

Format
N5 G71

See G70!

G72 Finishing Cycle

Format
N... G72 P... Q...

P... .... Block number of the first block for the program of finishing shape.
Q .... Block number of the last block for the program of finishing shape.

After rough cutting by G73, G74, G75 the G72 command permits finishing.
The programmed shape between P and Q which was also used for rough cutting, will be repeated without cutting depth division and without finishing offset.

Notes
• F, S and T functions specified between P and Q are only effective for G72. They are not effective for G73, G74 and G75!
• The finishing cycle G72 must only be programmed after the cycles G73, G74 and G75.

EMCO

G73 Contour Roughing Cycle
(Stock removal in turning)

Format
N... G73 U_1... R...
N... G73 P... Q... U_2+/-... W+/-... F... S... T...
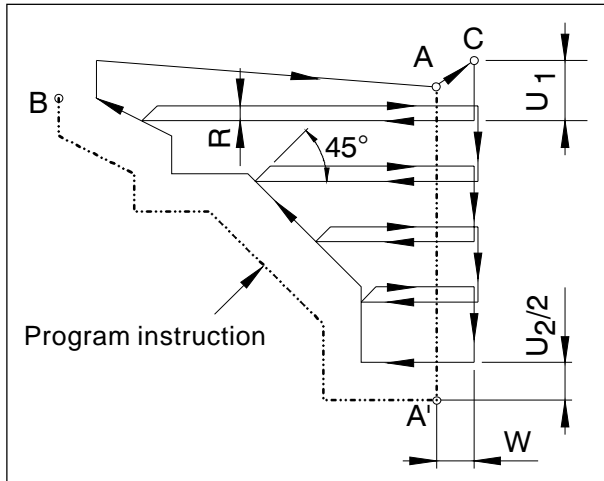
first block	U_1 [mm]	depth of cut, incremental, without sign, in the drawing shown as U_1
	R [mm]	retract height
second block	P...........	block number of the first block for the programmed shape
	Q	block number of the last block for the programmed shape
	U_2 [mm]	distance and direction of finishing offset in X direction (diameter or radius designation), in the drawing shown as $U_2/2$
	W [mm]	Distance and direction of finishing offset in Z direction, incremental, without sign
	F, S, T	Feed, speed, tool

Before machining the tool is at point C. Between the block numbers P and Q a contour (A to A' to B) will be programmed, it will be machined with the corresponding cutting depth division onto the defined finishing offset (2. block, in the drawing $U_2/2$).

Turning cycle contour

Contour without undercut
The first contour block from A to A' must contain a G0 or G1 movement in X direction only.

Contour with undercut
The first contour block from A to A' must contain a G0 or G1 movement in X and Z direction.
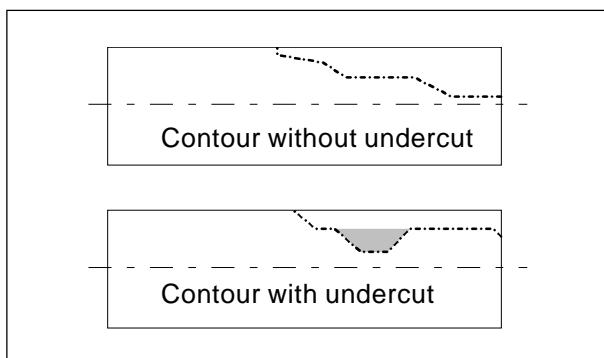
Notes
• F, S and T functions between P and Q are ignored.
• The point C (tool position before the cycle) must be out of the contour.
• The first movement from A to A' must be G00 or G01, is permitted in X only (G00 X...) and must be programmed in absolute coordinates.
• Between P and Q no subroutine call is permitted.
• By using G73 as an OD Roughing Cycle: Coordinate X_C has to be greater than X_B.
• By using G73 as an ID Roughing Cycle: Coordinate X_C has to be less than X_B.

Contour without undercut

Contour with undercut

Undercut
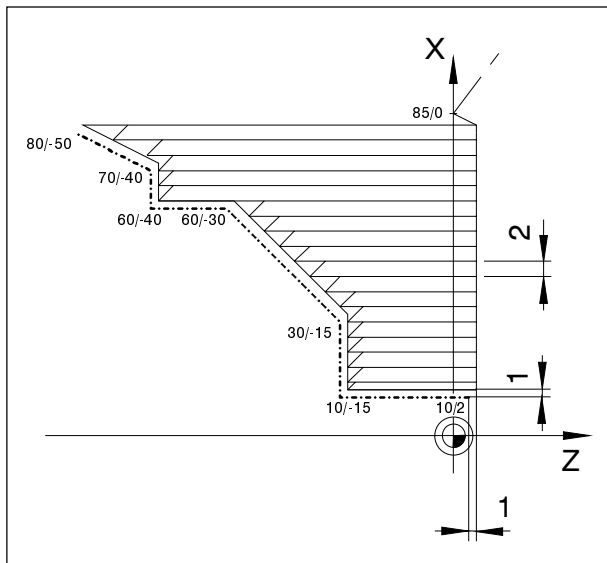
Example contour turning

**Example contour turning cycle with G73:
Machining the contour shown beside.**

Program:
O2000
N10   G95 G1   F0.5
N11   G0   X85   Z20
N12   T0101
N20   M3   S3000
N30   G00 X85   Z2
             (Start point for cycle)
N40   G73 U2   R2
N50   G73 P60   Q120 U2   W1
             (Contour turning cycle)
N60   G0   X10
N70   G1   Z-15
N80   X30
N90   X60 Z-30
N100 Z-40
N110 X70
N120 X80 Z-50
N130 G0   X85   Z20
N140 S3000   F0.6 T0202
             (Select finishing tool)
N150 G0   X85   Z2
             (Start point for finishing)
N160 G72 P60   Q120   (Finishing cycle)
N170 M30

EMCO

G74 Face Roughing Cycle
(Stock removal in facing)
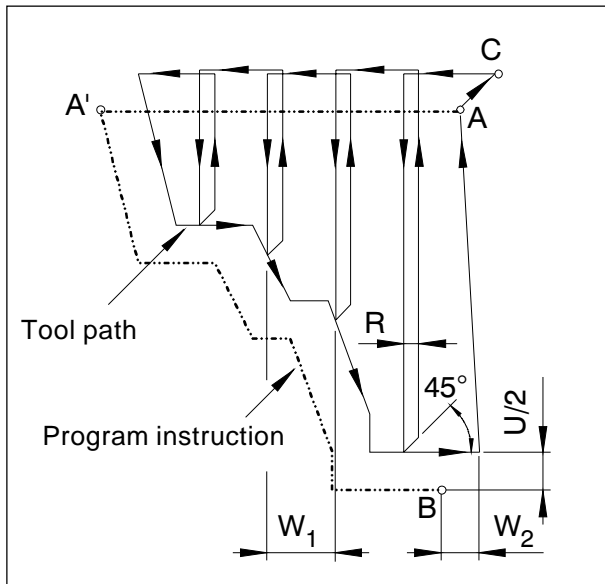
Face Roughing cycle contour

Format

N... G74 W$_1$... R...
N... G74 P... Q... U+/-... W$_2$+/-... F... S... T...

first block	W$_1$ [mm]	depth of cut in Z, incremental, without sign, in the drawing shown as W$_1$
	R [mm]	retract height
second block	P..........	block number of the first block for the programmed shape
	Q	block number of the last block for the programmed shape
	U [mm]	distance and direction of finishing offset in X direction (diameter or radius designation), in the drawing shown as U/2
	W$_2$ [mm]	Distance and direction of finishing offset in Z direction, incremental, without sign, in the drawing shown as W$_2$
	F, S, T	Feed, speed, tool

Before machining the tool is at point C. Between the block numbers P and Q a contour (A to A' to B) will be programmed, it will be machined with the corresponding cutting depth division onto the defined finishing offset (2. block, in the drawing W$_2$).

Notes
• F, S and T functions between P and Q are ignored.
• The point C (tool position before the cycle) must be out of the contour.
• The contour between A' and B has to be programmed decreasing, that means the diameter has to decrease.
• The first movement from A to A' must be G00 or G01, is permitted in Z only (G00 X...) and must be programmed in absolute coordinates.
• Between P and Q no subroutine call is permitted.
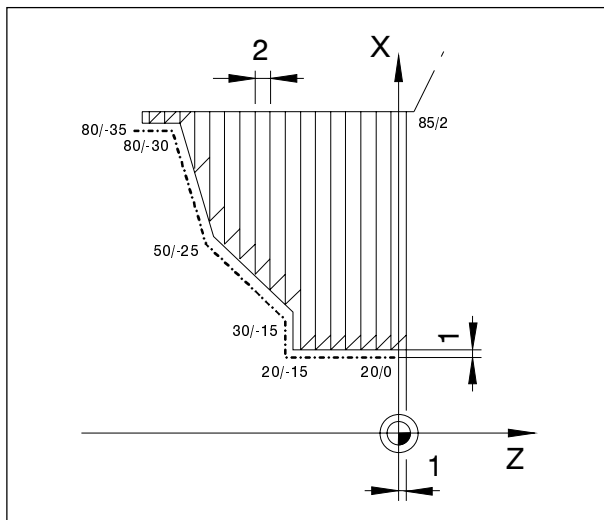
Example facing cycle

Example Facing cycle with G74:
Machining the contour shown beside.

Program:
O2001
N10   G95 G1   F0.5
N11   G0   X85   Z20
N12   T0303
N20   M3   S3000
N30   G00 X85   Z2
          (Start point for facing cycle)
N40   G74 W2   R2
N50   G74 P60   Q120 U2   W1
                   (Facing cycle)
N60   G0   Z-35
N70   G01 X80   Z-35
N80   Z-30
N90   X50 Z-25
N100 X30 Z-15
N110 X20
N120 Z0
N130 G0   X85   Z20
N140 S3000   F0.6 T0404
          (Select finishing tool)
N150 G0   X85   Z2
          (Start point for finishing)
N160 G72 P60   Q120   (Finishing cycle)
N170 M30

G75 Pattern Repeating Cycle

Format
N... G75 U_1+/-... W_1+/-... R...
N... G75 P... Q... U_2... W_2... F... S... T...
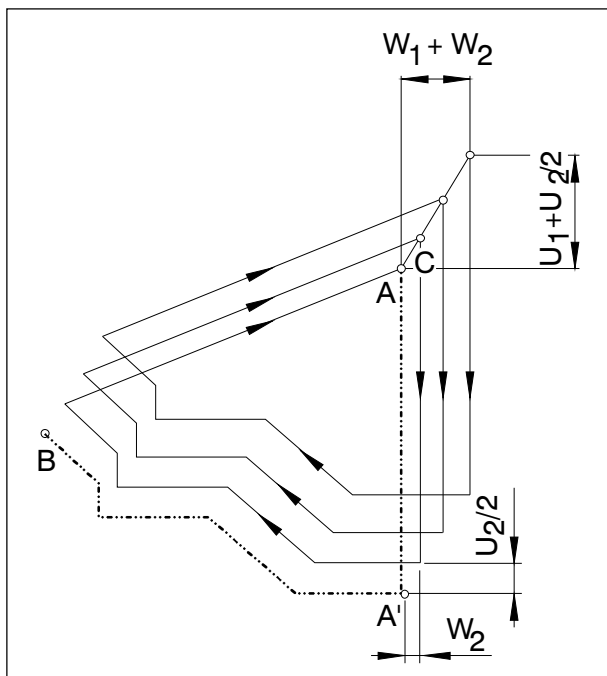
first block	U_1	Start point for the cycle in the X axis (radius designation), in the drawing shown as U_1
	W_1	Start point of the cycle in the Z axis, in the drawing shown as W_2
	R	number of repetitions (equal to cut division)
second block	P	block number of the first block for the programmed shape
	Q	block number of the last block for the programmed shape
	U_2 [mm]	distance and direction of finishing offset in X direction (diameter or radius designation), in the drawing shown as U/2
	W_2 [mm]	Distance and direction of finishing offset in Z direction, incremental, without sign, in the drawing shown as W_2
	F, S, T	Feed, speed, tool

Pattern repeating

The G75 cycle allows machining parallel to the shape of the workpiece, the pattern will be shifted to the finished shape step by step.
Application for semifinished products (forged, cast parts)

Example:

```
O2002
N1    G95  G0    X85   Z0
N5    T0707      M3    S2000       F0.5
N10   G75  U5    W5    R5
N15   G75  P20   Q80   U2    W1
N20   G0   X20
N30   G1   Z-15
N40   X40
N50   X70  Z-30
N60   Z-40
N70   X75  Z-50
N80   X80
N90   M30
```

The contour in N20 (20/0) - N80 (80/-50) will be machined in 5 infeeds.
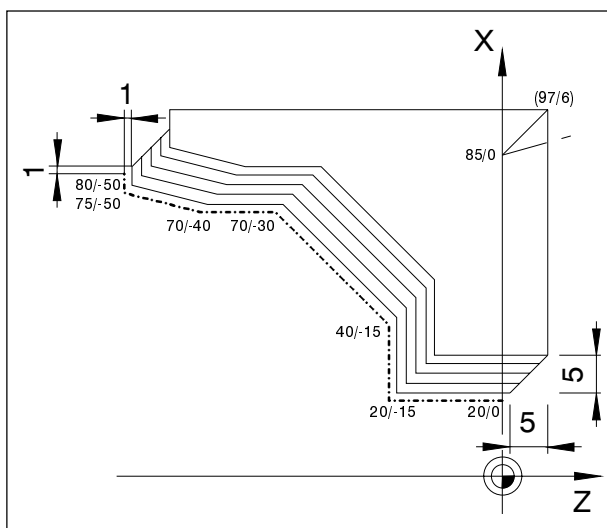
Example Pattern repeating

G76 Face Cut-in Cycle

(Deep Hole Drilling Cycle for end face peck drilling)

Deep hole drilling / Face Cut-in Cycle

Format

N... G76 R...
N... G76 X(U)... Z(W)... P... Q... F...

first block	R [mm]	retraction height for chip breaking (incremental without sign), drawing: R_1
sec. block	X(U), Z(W)	Absolute (incremental) coordinates of the contour edge point K
	or	
	Z(W)	Absolute (incremental) drilling depth
	P [µm]	Incremental feed in X direction (no sign); P < tool width!
	Q [µm]	Cutting depth in Z direction (no sign)
	F	Feed rate

Notes:
- Without addresses X(U) and P G76 can be used as drilling cycle (Move tool to X=0 before!)
- With cut-in cycle the infeed P has to be smaller than tool width B.

Example:
Axial groove, 10mm wide and 10mm deep

T1212 ; (axial cut-in tool 3mm width)
G96 S120 M4 ;
G0 X80 Z1 ;
G76 R0.5 ;
G76 X94 Z-10 P2800 Q4000 F0.06 ;
G0 X150 Z100 ;
M5 ;

EMCO

Cut-in cycle in X

G77 Side Cut-in Cycle (X Axis)

Format
N... G77 R...
N... G77 X(U)... Z(W)... P... Q... F...
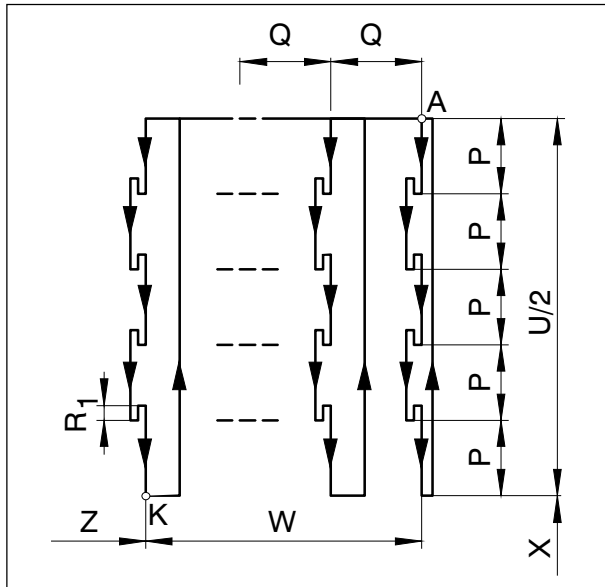
first block	R [mm]	Retraction height for chip breaking, in the drawing shown as R_1
sec. block	X(U), Z(W)	Absolute (incremental) co-ordinates of K
	P [µm]	Cutting depth in X direction (no sign)
	Q [µm]	Incremental infeed in Z direction (no sign)
	F	Feed

Note
- The infeed Q must be smaller than tool width B.
- Tool width will not be taken into consideration with this cycle.

Example:
Radial groove, 10mm wide and 10mm deep

T1111 ; (radial cut-in tool 3mm width)
G96 S120 M4 ;
G0 X102 Z-20 ;
G77 R0.5 ;
G77 X80 Z-27 P4000 Q2800 F0.06 ;
G0 X150 Z100 ;
M5 ;

G78 Multiple Threading Cycle (Thread cutting OD/ID)

Format

N... G78 P_1... Q_1... R_1...
N... G78 X(U)... Z(W)... R_2... P_2... Q_2... F...

first block:

P_1 .... is a 6 digit parameter divided in digit couples:

PXXxxxx → The first two digits of this parameter define the number of finishing cuts

PxxXXxx → The next two digits define the chamfer value P_F (see drawing)

$$P_{xxXXxx} = \frac{P_F \text{ [mm]} \times 10}{F}$$

PxxxxXX → Defines the flank angle of thread in [°].
(allowed: 80, 60, 55, 30, 29, 0)

Q_1 .... Minimum cutting depth [μm] incremental

R_1 .... Finishing offset [mm] incremental

second block   X(U), Z(W) Absolute (incremental) coordinates of the point K
           R_2 [mm] Incremental taper value with sign (R=0 cylindrical thread)
           P_2 [μm] Thread depth (always positive), in the drawing shown as P_2
           Q_2 [μm] Cutting depth of the first cut (radius value) without sign
           F [mm] Thread pitch

Note:
- Negative taper parameter R defines the taper as shown in the drawing.
- By using G78 as an OD Threading Cycle: Coordinate X_S has to be greater than X_K
- By using G78 as an ID Threading Cycle: Coordinate X_S has to be less than X_K

Multiple threading cycle

Formulars for the calculation of the cutting depth for point K in X (metric system):

Thread O.D. (external thread):

> **t = 0,6134 × pitch**

Thread I.D. (internal thread):

> **t = 0,5413 × pitch**
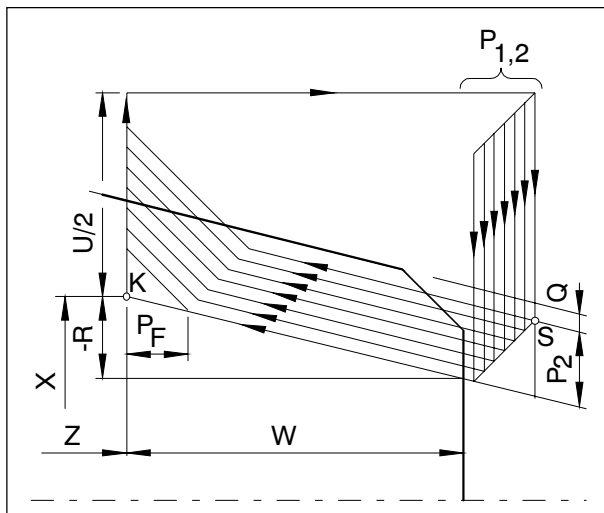
Example:

N5 T0505 (Thread O.D. M60×2) ;
G97 S800 M3 ;
G0 X60.5 Z2 ;
G78 P041060 Q150 R0.1 ;
G78 X57.546 Z-20 P1226 Q150 F2 ;
G0 X150 Z150 M5 ;

EMCO

Drilling cycles with driven tools
Survey of the drilling cycles

G-code	Drilling axis	Hole machining (– direction)	Procedure at the bore bottom	Withdrawal (+direction)	Applications
G80	———	———	———	———	Cycle end
G83	Z-axis	Cutting feed interrupting	Dwell time	Rapid motion	Cycle for face drilling
G84	Z-axis	Cutting feed	Intermission ➝ spindle in counter-clockwise direction	Cutting feed	Cycle for face tapping
G85	Z-axis	Cutting feed	———	Cutting feed	Face reaming cycle
G87	X-axis	Cutting feed interrupting	Dwell time	Rapid motion	Side drilling cycle
G88	X-axis	Cutting feed	Intermission ➝ spindle in counter-clockwise direction	Cutting feed	Cycle for side tapping
G89	X-axis	Cutting feed	Dwell time	Cutting feed	Cycle for side reaming

Withdrawal behaviour G98, G99
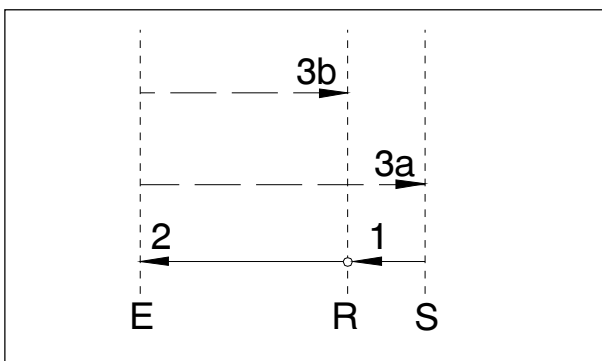
Operating sequence G98, G99

Systematics G98/G99

G98 .... After reaching the drilling depth the tool moves to the starting level

G99 .... After reaching the drilling depth the tool moves to the withdrawal level - defined by R-parameter

If no G98 o G99 is active, the tool returns to the starting level. If G99 (withdrawal on withdrawal level) is programmed, the address R must be defined. With G98, R can be omitted!

R defines the position of the withdrawal level with reference to the last Z-position (initial position for drilling cycle). With a negative value for R, the withdrawal level is below the initial position, with a positive value above the initial position.

Operating sequence

1: The tool traverses with rapid motion from the initial position (S) to the level defined by R (R).

2: Cycle specific drilling up to final depth (E).

3: a: With G98 the withdrawal is carried out up to the starting level (initial position S).
b: with G99 up to the withdrawal level (R).

4: With the NC-parameter "5102#6" can be defined, how the parameter R has to be programmed in dependent upon G90/G91:
inremental distance .... set to 0
absolute distance .... set to 1

G80 Delete Drilling Cycle (G83 to G89)

Format
N... G80

The drilling cycles must be deselected by G80 or another G-code of group 1 (G00, G01, ...), since they are modally effective.
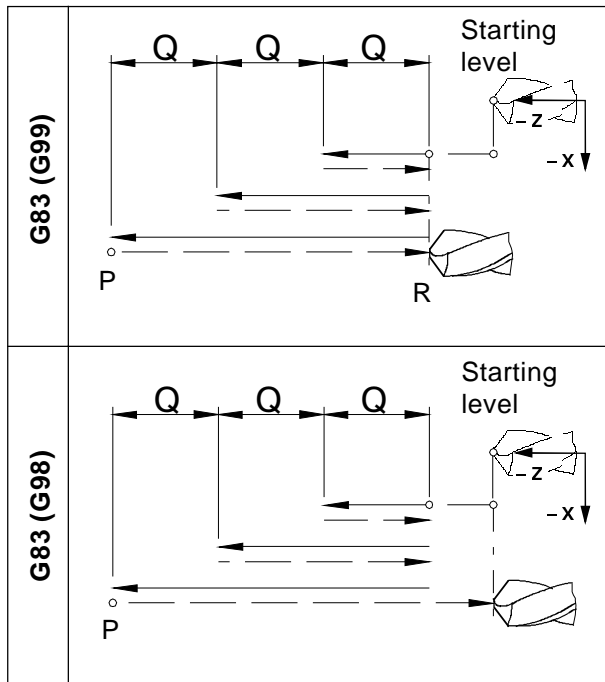
Face drilling cycle G83
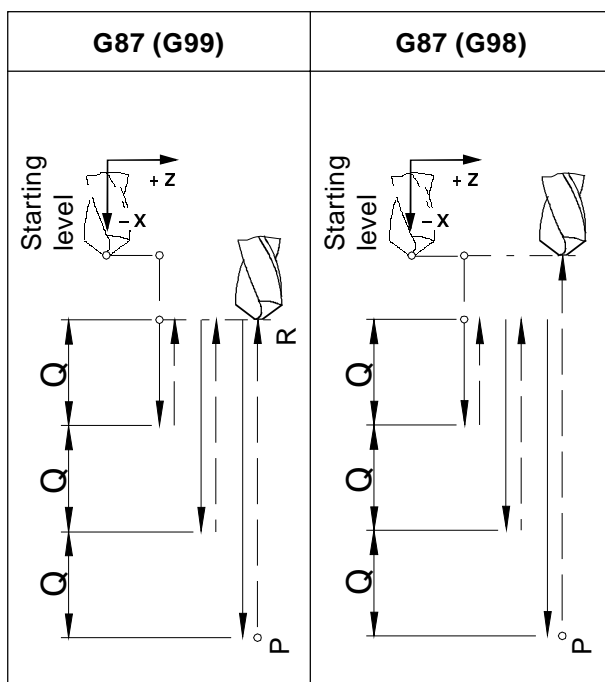
Side drilling cycle G87

G83 Face Drilling/Face Peck Drilling Cycle

G87 Side Drilling/Side Peck Drilling Cycle

Format G83
N... G98(G99) G83 X(U)... C(H)... Z(W)... R...
Q... P... F... (M...) (K...)

Format G87
N... G98(G99) G87 Z(W)... C(H)... X(U)... R...
Q... P... F... (M...) (K...)

G98(G99) .. withdrawal to starting level (withdrawal level)
X(U) .... G83: hole position absolute (incremental) in the X-axis
G87: drilling depth absolute (incremental) in the X-axis
Z(W) .... G83: drilling depth absolute (incremental) in the Z-axis
G87: hole position absolute (incremental) in the Z-axis
C [°] .... drilling position
R [mm] ...... incremental value of the withdrawal level with reference to the starting point in the Z/X-axis (with sign)
Q [µm] .... drilling depth per infeed for peck drilling
P [msec] .... dwell time on the hole bottom:
P1000 = 1 sec
F .... feedrate
(M) .... M-code for C-axis clamping (when parameter "5110" is defined)
(K) .... number of the cycle repetitions (to be used only for incremental programming)

Notes
• Setting of parameter "5101#2":
Set 1: Drill reacts back to R after each step
Set 2: Drill reacts back to specified distance in parameter "5114".
• It is not necessary to program C, X and Z within the cycle for the position of the hole, if in the previous block before the tool has been already traversed to the drilling position. In this case just the drilling depth (Z for G83, X for G87) needs to be commanded.
• Unless Q is specified, a division of cuts is not carried out, i. e. drilling to Z-end point in one movement. Therefore, Q has to be called together with each following drilling position command.

EMCO

Example - G83 drilling cycle / deep-hole drilling cycle in Z-direction (with driven tool axial cycle)

N3 T0303 ;	(drill, driven tool axial)
M52 ;	selection of the main spindle as C-axis.
G28 C0 ;	refence C-axis (necessary only once after first call of M52 in program)
M13 ;	(or M14) selection of driven tool with clockwise tool rotation normally (read note)
G97 S2000 ;	constant spindle speed.
G0 X50 Z10 C30 ;	positioning the tool.

G83 Z-42 R-8 Q6000 F0.5 ;
(bore 1 (Q6000 = 6mm cutting depth per feed during deep-hole drilling).).

C150 Q6000 ;	bore 2.
C270 Q6000 ;	bore 3.
G80 M15 ;	deselection drilling cycle and switch off speed of the tool.
M53 ;	deselection of the main spindle as C-axis.

Note:

The direction of rotation for the drilling tools normally is fixed for right hand tools.
The reason to change the direction of rotation depends on the design of the tool holder, when the gear of the holder reverses it's direction.

Example - G87 drilling cycle / deep-hole drilling cycle in X-direction (with driven tool radial cycle)

N7 T0707	(drill, driven tool radial);
M52 ;	selection of the main spindle as C-axis.
G28 C0 ;	reference. C-axis (necessary only once after first call of M52 in program).
M13 ;	(...or M14)...selection driven tool and sense of rotation of the tool (read note).
G97 S2000 ;	constant spindle speed.
G0 X50 Z-20 C30 ;	positioning the tool.

G87 X18 R-6 Q6000 F0.5 ;
(bore 1 (Q6000 = 6mm cutting depth per feed during deep-hole drilling).)

C150 Q6000 ;	bore 2.
C270 Q6000 ;	bore 3.
G80 M15 ;	deselection drilling cycle and switch off speed of the tool.
M53 ;	deselection of the main spindle as C-axis.

Cycle for face tapping G84
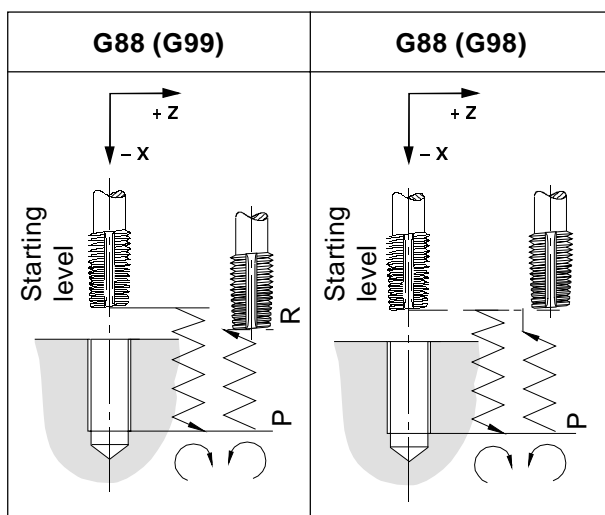
Cycle for side tapping G88

G84 Cycle for face tapping with/ without compensation chuck

G88 Cycle for side tapping with/ without compensation chuck

Tapping without compensation chuck is named "Rigid Tapping".
Face tapping and side tapping (G84 and G88) can be performed in conventional mode or in rigid mode.

Format G84
N... G98(G99) G84 X(U)... C(H)... Z(W)... R... P... F... (M...) (K...)

Format G88
N... G98(G99) G88 Z(W)... C(H)... X(U)... R... P... F... (M...) (K...)

G98(G99) .. withdrawal to starting level (withdrawal level)
X(U) .... G84: hole position absolute (incremental) in the X-axis
G88: drilling depth absolute (incremental) in the X-axis
Z(W) .... G84: drilling depth absolute (incremental) in the Z-axis
G88: hole position absolute (incremental) in the Z-axis
C[°] .... drilling position
R [mm] ...... incremental value of the withdrawal level with reference to the starting point in the Z/X-axis (with sign)
P [msec] .... dwell time at the hole ground: P1000 = 1 sec
F .... feed
(M) .... M-code for C-axis clamping (when parameter "5110" is defined)
(K) .... number of the cycle repetitions (to be used only for incremental programming)

General
• The spindle turns clockwise for right hand taps. At the bottom of the hole the spindle changes its direction to retract the tool counterclockwise.

Note:
In case, that the direction of the tool will even not change its direction with M13/M14 to clockwise, the special invertion command M214 has to be used to reverse the direction of the tool rotation.

EMCO

- During the tapping feedrate and spindle override are without function, and assumed to 100%

- Tapping with comensation chuck:
  In conventional mode, the tool will rotate or stop in synchronization with the motion along the tapping axis (Z for G84 and X for G88).

- Rigid tapping:
  M29 Sxxxx has to be programmed in a block before the cycle.
  The spindle motor is controlled like a servo-motor, so that a faster tapping is possible.

- Mode "Feed per minute":
  The pitch of the thread equals from feed divided by the spindle speed.

- Mode "Feed per rotation":
  The feedrate is equal to the screw lead (=pitch).

Notes:

- The direction of rotation for the drilling tools normally is fixed for right hand tools.
  The reason to change the direction of rotation depends on the design of the tool holder, when the gear of the holder reverses it's direction.

- Do **not** insert a dwell in the cycle for tapping with longitudinal compensation.
  For bigger drills insert the commands M10/M11 (main spindle clamp/unclamp) in order to ensure, that the mainspindle keeps its position.

Example - G84 rigid tapping cycle:
without longitudinal compensation in Z-direction (with driven tool/axial cycle)
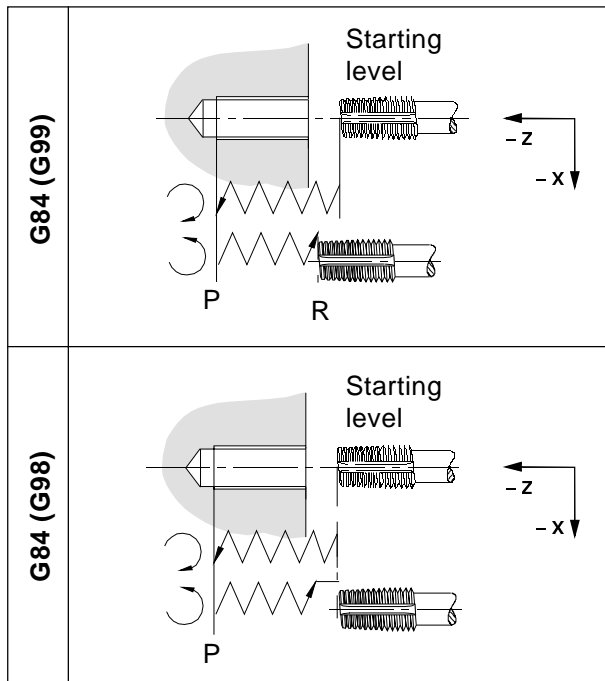
N4 T0404	(tap M6x1, driven tool axial);
M52 ;	selection of the main spindle as C-axis.
G28 C0 ;	reference C-axis (necessary only once after first call of M52 in program).
M214 ;	reverse of rotation direction (necessary only for left hand thread - read note)
M13 ;	selection driven tool
G97 ;	selection of constant spindle speed [rpm]
G0 X50 Z10 C30 ;	positioning of the tool.
M29 S1000 ;	selection tapping without longitudinal compensation and speed for tool.
G84 Z-20 R-6 P500 F1;	thread 1.
C150 ;	thread 2.
C270 ;	thread 3.
G80 M15 ;	deselection drilling cycle and switch off speed of the tool.
M53 ;	deselection of the main spindle as C-axis.

Example - G88 rigid tapping cycle:
without longitudinal compensation in X-direction (with driven tool/radial cycle)

N8 T0808	(tap M6x1, driven tool radial);
M52 ;	selection of the main spindle as C-axis.
G28 C0 ;	reference C-axis (necessary only once after first call of M52 in program).
M214 ;	reverse of rotation direction (necessary only for left hand thread - read note)
M13 ;	selection driven tool
G97 ;	selection of constant spindle speed [rpm]
G0 X80 Z-30 C30 ;	positionining the tool.
M29 S1000 ;	selection tapping without longitudinal compensation and speed for tool.
G88 X50 R-6 P500 F1;	thread 1.
C150 ;	thread 2.
C270 ;	thread 3.
G80 M15 ;	deselection drilling cycle and switch off speed of the tool.
M53 ;	deselection of the main spindle as C-axis.

Example - G84 drilling cycle for tapping with longitudinal compensation in Z-direction (with driven tool/axial cycle)

N1 T0101	(tap M6x1, driven tool axial);
M52 ;	selection of the main spindle as C-axis.
G28 C0 ;	reference C-axis (necessary only once after first call of M52 in program).
M214 ;	reverse of rotation direction (necessary only for left hand thread)
M13 ;	selection driven tool
G97 S1000 ;	selection of constant spindle speed [rpm] and speed for tool
G0 X50 Z10 C30 ;	positioning of the tool.
G84 Z-20 R-6 P0 F1 ;	thread 1.
C150 ;	thread 2.
C270 ;	thread 3.
G80 M15 ;	deselection drilling cycle and switch off speed of the tool.
M53 ;	deselection of the main spindle as C-axis.

Example - G88 drilling cycle for tapping with longitudinal compensation in X-direction (with driven tool/radial cycle)

N2 T0202	(tap M6x1, driven tool radial);
M52 ;	selection of the main spindle as C-axis.
G28 C0 ;	reference C-axis (necessary only once after first call of M52 in program).
M214 ;	reverse of rotation direction (necessary only for left hand thread)
M13 ;	selection driven tool
G97 S1000 ;	selection of constant spindle speed [rpm] and speed for tool
G0 X80 Z-30 C30 ;	positionining the tool.
G88 X50 R-6 P0 F1 ;	thread 1.
C150 ;	thread 2.
C270 ;	thread 3.
G80 M15 ;	deselection drilling cycle and switch off speed of the tool.
M53 ;	deselection of the main spindle as C-axis.

Example - G84 rigid tapping cycle without longitudinal compensation in Z-center line (with main spindle/axial cycle)

N10 T1010	(tap M6x1, fixed tool axial);
M204 ;	for left hand thread only
M3 ;	selection of main spindle
G97 S1000 ;	selection of constant spindle speed [rpm] and speed for tool
G0 X0 Z10 ;	positioning of the tool.
M29 S1000 ;	select rigid mode
G84 Z-20 R-6 P0 F1 ;	thread 1.
C150 ;	thread 2.
C270 ;	thread 3.
G80 M5 ;	deselection drilling cycle

EMCO

G85 Cycle for Face Reaming

G89 Cycle for Side Reaming
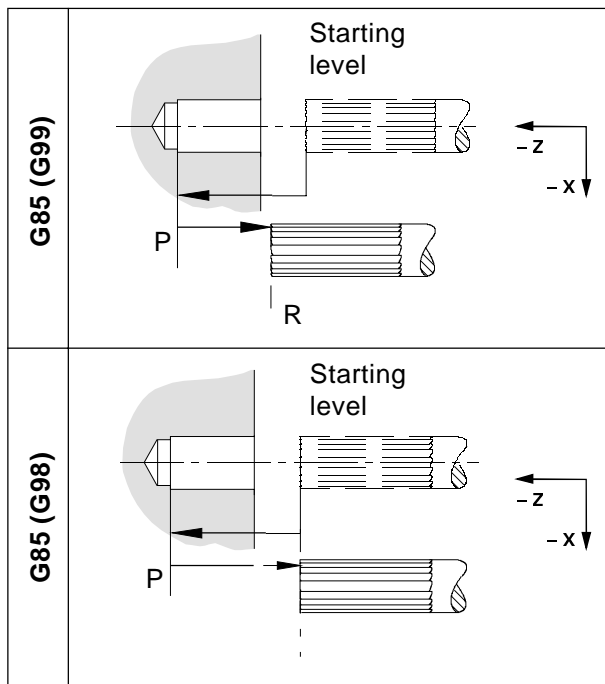
Reaming cycle G85
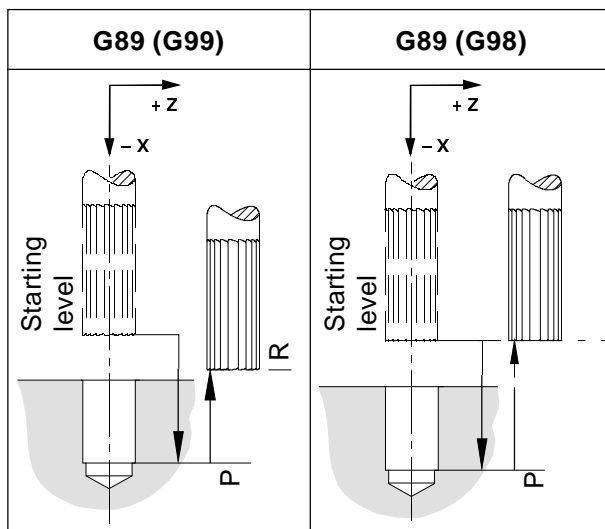
Cycle for side reaming G89

Format G85

N... G98(G99) G85 X(U)... C(H)... Z(W)... R... P... F... (K...) (M...)

Format G89

N... G98(G99) G89 Z(W)... C(H)... X(U)... R... P... F... (K...) (M...)

G98(G99) .. withdrawal to starting level (withdrawal level)

X(U) .... G85: hole position absolute (incremental) in the X-axis
G89: drilling depth absolute (incremental) in the X-axis

Z(W) .... G85: drilling depth absolute (incremental) in the Z-axis
G89: hole position absolute (incremental) in the Z-axis

C[°] .... drilling position

R [mm] ...... incremental value of the withdrawal level with reference to the starting point in the Z/X-axis (with sign)

P [msec] .... dwell time at the hole bottom: P1000 = 1 sec

F .... feed

(K) .... number of cycle repetitions (only for incremental programming)

(M) .... M-code for axis clamping (when parameter "5110" is defined)

Notes

• If G99 (withdrawal to withdrawal level) is programmed, the address R must also be defined. With G98 R can be omitted!

• It is not necessary to program C, X and Z within the cycle for the position of the hole, if in the previous block before the tool has been already traversed to the drilling position. In this case just the drilling depth needs to be commanded.

• The withdrawal to the start point is carried out with the double feed speed, which has been programmed in block G85/G89.

• A division of the cut is not possible by indicating the Q-parameter.

Example - G85 drilling cycle / reaming cycle in Z-direction (with driven tool/axial cycle)

N5 T0505	(reamer, driven tool axial);
M52 ;	selection of the main spindle as C-axis.
G28 C0 ;	reference. C-axis (necessary only once after first call of M52 in program).
M13 ;	(...or M14)...selection driven tool with clockwise tool rotation normally (read note)
G97 S2000 ;	constant spindle speed.
G0 X50 Z10 C30 ;	positioning the tool.
G85 Z-32 R-8 P500 F0.5 ; bore 1.	
C150 ;	bore 2.
C270 ;	bore 3.
G80 M15 ;	deselection of the drilling cycle and switch off speed of the tool.
M53 ;	deselection of the main spindle as C-axis

Note:

The direction of rotation for the drilling tools normally is fixed for right hand tools.
The reason to change the direction of rotation depends on the design of the tool holder, when the gear of the holder reverses it's direction.

Example - G89 Drilling cycle / reaming cycle in X-direction (with driven tool/radial cycle)

N9 T0909	(reamer, driven tool radial);
M52 ;	selection of the main spindle as C-axis
G28 C0 ;	reference. C-axis (necessary only once after first call of M52 in program).
M13 ;	(...or M14)...selection driven tool with clockwise tool rotation normally (read note)
G97 S2000 ;	constant speed of the tool.
G0 X50 Z-40 C30 ;	positioning the tool.
G89 X28 R-6 P500 F0.5 ; bore 1.	
C150 ;	bore 2.
C270 ;	bore 3.
G80 M15 ;	deselection of the drilling cycle and switch off speed of the tool.
M53 ;	deselection of the main spindle as C-axis

EMCO

G12.1/G13.1
Polar Coordinate Interpolation

The polar coordinate interpolation is adequate for machining the end face of a turned part or grinding the camshaft on lathes.

It converts a command programmed in the Cartesian coordinate system into the movement of a linear axis X (tool movement) and a rotating axis C (workpiece rotation) for the path control.

The rotary axis C serves as axis address for the second (virtual) axis.
This axis is set immediately after programming G12.1 at coordinate C0.
Here an angle of 0 is supposed as tool position.

Level of the polar coordinate interpolation

+ C
Rotary axis (virtual axis)

Linear axis

+ X

Origin of the workpiece coordinate system

Format
N.. G12.1 ;   starts the operating mode and enables polar coordinate interpolation
         :   geometry program (based on Cartesian coordinates)
N.. G13.1 ;   terminates the operating mode polar coordinate interpolation.

G12.1 chooses a level (G17) in which the polar coordinate interpolation is carried out.
The level G18 used by G12.1 before programming is deleted.
It is reestablished by means of the command G13.1 (polar coordinate interpolation end).

After switching on the machine or with a system RESET the condition "polar coordinate interpolation" is also cancelled, (G13.1) selected and the level defined by means of G18 is used.

Notes:

• Also with diameter programming for the linear axis (X-axis) radius programming is used for the rotary axis (C-axis).

• Cutter position 0 must be assigned to the tool in the offset data.
However, the miller radius must be entered.

• In G12.1- mode the coordinate system must not be altered.

• G12.1 and/or G13.1 must be programmed in the mode "cutter radius compensation off" (G40) and cannot be started or terminated within "cutter radius compensation on" (G41 or G42).

• G12.1 and G13.1 are to be programmed in separate blocks.
In a block between G12.1 and G13.1 an interrupted program cannot be brought to a new start.

• The arc radius with circular interpolation (G2 oder G3) can be programmed by means of an R-command and/or via I- and J-coordinates.

• In the geometry program between G12.1 and G13.1 no rapid motion (G0) must be programmed (see adjoining scheme).

G-codes which may be programmed in the mode "polar coordinate interpolation:

G-Code	Use
G01	Linear interpolation
G02, G03	Circular interpolation
G04	Intermission
G40, G41, G42	Cutter radius compensation (polar coordinate interpolation is applied on the tool path after the tool compensation)
G65, G66, G67	User macro command
G98, G99	Feed per minute, feed per rotation

point	X	C
S	34,64	10
1	34,64	0
2	17,32	-15
3	-17,32	-15
4	-34,64	0
5	-17,32	15
6	17,32	15
7	34,64	0
E	34,64	-10

Example 1 - polar coordinate interpolation

X- axis with diameter programming and C-axis with radius programming.
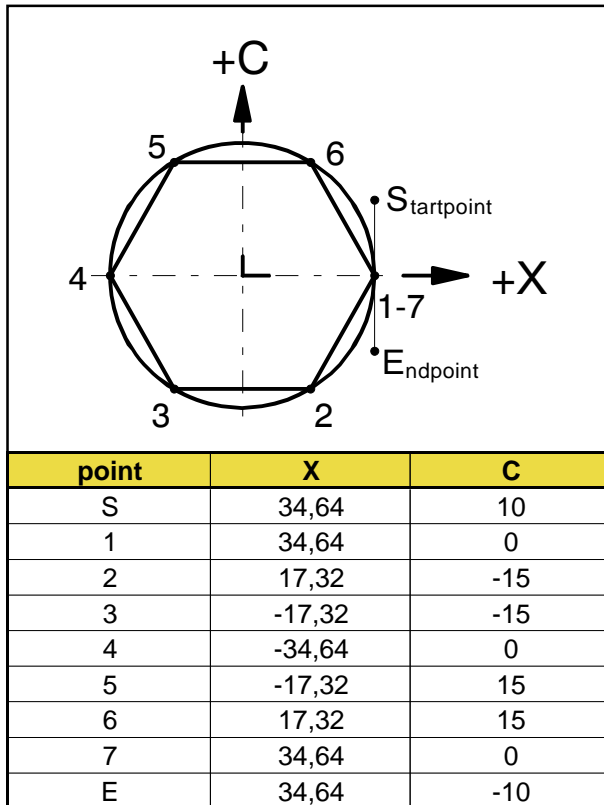
N1 T0101 (end mill Dm 10);
  driven tool axial / miller radius 5.0, cutter radius position 0.
M13 ;   (...or M14)...selection driven tool and direction of the tool rotation.
G97 S1000   constant speed of the tool.
M52 ;   selection of the main spindle as C-axis
G28 C0 ;   reference. C-axis (necessary only once after first call of M52 in program).
G52 C.. ;   poss. shift of angle of C-axis.
G40 G0 X60 C0 Z-6 ;
  Positioning the tool in rapid motion (poss. deselection cutter radius compensation).
G12.1 ;   start of the polar coordinate interpolation.
G42 G1 X34.64 C10 F0.1 ;
  to point S in feed and selection cutter radius compensation.
C0 ;   to point 1.
X17.32 C-15 ;   to point 2.
X-17.32 ;   to point 3.
X-34.64 C0 ;   to point 4.
X-17.32 C15 ;   to point 5.
X17.32 ;   to point 6.
X34.64 C0 ;   to point 7.
C-10 ;   to point E.
G40 X60 Z10 ;   move away from the part in the feed and deselection cutter radius compensation.
G13.1   end of the polar coordinate interpolation.
G52 C0 ;   poss. reset of offset of the angle in the C-axis.
M53 ;   poss. deselection of the main spindle as C-axis.
G0 X150 Z150 M15 ;
  Release in rapid motion and speed driven tool off.

EMCO

Example 2 - polar coordinate interpolation with cutter radius compensation

X-axis with diameter and C-axis with radius programming.
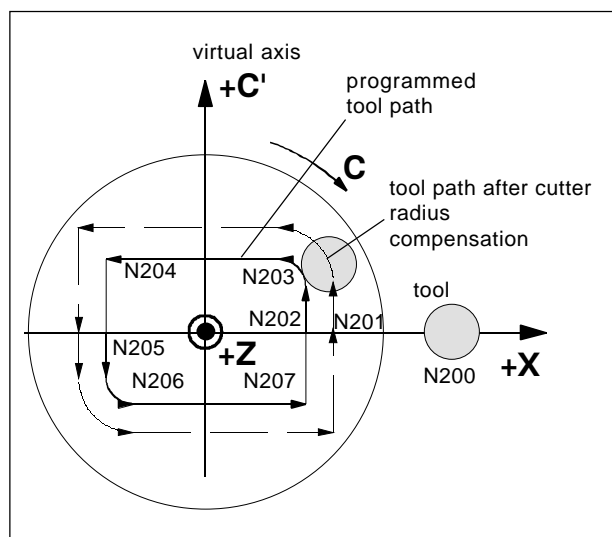
N1 T1212;	(end mill Dm10, axial driven tool, cutter radius 5.0, cutter radius position 0)
M13 ;	(or M14) rotation direction
G97 S1000 ;	constant spindle speed
M52 ;	selection on C-axis
G00 X120.0 C0 Z... ;	positioning at the starting position
G12.1 ;	start of the polar coordinate interpolation

```
G42 G01 X40.0 F... ;      ⎫
C10.0                     ⎪  geometry
G03 X20.0 C20.0 R10.0 ;   ⎪  program
G01 X-40.0 ;              ⎪  (based on
C-10.0 ;                  ⎬  Cartesian
G03 X-20.0 C-20.0 I10.0 J0 ; ⎪ coordinates in
G01 X40.0 ;              ⎪  X-C'plane)
C0 ;                      ⎪
G40 X120.0 ;              ⎭
```

G13.1 ;	polar coordinate interpolation end
Z... ;	
X... C... ;	
M30 ;	

Polar coordinate interpolation with cutter radius compensation

G7.1 Cylindrical interpolation

This function enables the development of a cylinder surface in programming.
In this way e.g. programs for cyclindrical cam machining on lathes can be created.

The traverse amount of the rotary axis C programmed by indication of the angle is converted in the control into the distance of a fictitious linear axis along the external surface of the cylinder.
Thus, it is possible that linear and circular interpolations on this area can be carried out with another axis.

Format

G1 G19 G90 W0 H0 ;   (or G1 G19 G91 Z0 C0 ;) determine centre of the reference level (PRM1022#C=6).
G7.1 C.. ;   (or G7.1 H.. ;) activates the mode cylinder interpolation with indication of the cylinder diameter (in [mm] for the calculation of the path feed rate.
    :   Geometry program.
G7.1 C0 ;   (or G7.1 H0 ;) terminates the mode cylinder interpolation.
G18 ;   reset to the original plane

Calculating of Y[mm]- into C[°]-coordinates for path programming

$$C_p = \frac{Y_p[mm] \cdot 360[°]}{2\pi \cdot R_{cyl}[mm]}$$

C_p [°] .... distance to move in C-axis
Y_p [°] .... drawing distance to be calculated
R_{cyl} [mm] .... radius of the cylinder surface

With G19 the level is determined in which the rotary axis C is preset in parallel to the Y-axis. The block structure for the geometry program is then as follows:
G1 Z.. [in mm] C.. [in °] ; linear interpolation.
G2 (G3) Z.. [in mm] C.. [in °] R.. [in mm] ; circular interpolation.

Notes:

- The reference point of the cylinder must be entered incrementally, since otherwise it would be approached by the tool!

- In the offset data cutter position 0 must be allocated to the tool. However, the miller radius must be entered.

- In mode G7.1 the coordinate system must not be changed.

- G7.1 C.. and/or G13.1 C0 must be programmed in the mode "cutter radius compensation off" (G40) and cannot be started or terminated within "cutter radius compensation on" (G41 or G42).

- G7.1 C.. and G7.1 C0 must be programmed in separate blocks.

- In a block between G7.1 C.. and G7.1 C0 an interrupted program cannot be restarted.

- The arc radius with circular interpolation (G2 or G3) must be programmed via an R-command and must not be programmed in degree and/or via K and J-coordinates.

- In the geometry program between G7.1 C.. and G7.1 C0 no rapid motion (G0) and/or positioning procedures causing rapid motion movements (G28) or drilling cycles (G83 to G89) must be programmed.

- The feed entered in the mode cylindric interpolation is to be considered as traverse speed on the unrolled cylinder area.

EMCO

Example 1 - cylinder interpolation

X-axis with diameter programming and C-axis with angle programming.
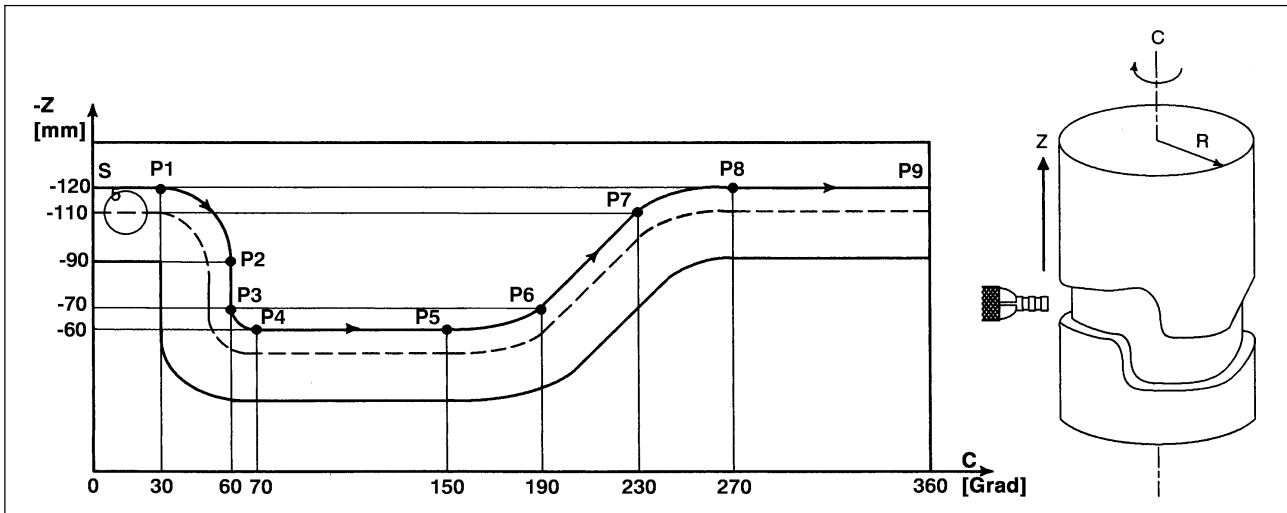
N2 T0202 (end mill Dm 12);	driven tool radial / cutter radius 6.0, cutter radius position 0.
M13	(...or M14)...selection driven tool and sense or rotation of the tool.
G97 S1000	constant speed of the tool.
M52 ;	selection of the main spindle as C-axis.
G28 C0 ;	reference. C-axis (necessary only once after after first call of M52).
G52 C.. ;	poss. shift of the angle of the C-axis.
G40 G0 X60 C0 Z-100 ;	positioning the tool in rapid motion (poss. deselection cutter radius compensation).
G19 W0 H0 ;	(or G19 G91 Z0 C0).. determine incrementally centre point of the reference level.
G7.1 C57.299;	start of the cylinder interpolation with cylinder diameter notation in [mm].
G42 G1 G94 Z-120 F350 ;	selection cutter radius compensation outside the workpiece on point S.
X48 F100 ;	feed in X.
C30 F250 ;	to point 1.
G2 Z-90 C60 R30 F250 ;	to point 2.
G1 Z-70 ;	to point 3.
G3 Z-60 C70 R10 ;	to point 4.
G1 C150 ;	to point 5.
G3 Z-70 C190 R75 ;	to point 6.
G1 Z-110 C230 ;	to point 7.
G2 Z-120 C270 R75 ;	to point 8.
G1 C360 ;	to point 9.
X60 F350 ;	move away in X.
G40 Z-100 ;	deselection cutter radius compensation outside the workpiece.
G7.1 C0 ;	end of the cylinder interpolation by deselection of the cylinder radius.
G95 ;	deselection G94 (feed in mm/min).
G52 C0 ;	poss. reset of the shift of the angle in the C-axis.
M53 ;	poss. deselection of the main spindle as C-axis.
G0 X150 Z150 M15 ;	release in rapid motion and speed driven tool off.
G18 ;	reset to original plane

G51.2 / G50.2 Polygonal Turning

During polygonal turning, workpiece and tool are turned in a certain relation to each other in order to produce workpiece contours.

By changing the transmission ratio between tool and workpiece or changing the number of cutting tools the part contour can be altered.

The advantage compared with polar coordinate interpolation is to be found in the shorter machining time, however, the part contour is not exactly polygonal.

Polygonal turning is mainly used for the production of e.g. collar head screws and hexagonal screws.

Format:

G51.2 P... Q... ;

P ...... integer number 1 to 9
transmission number of the main spindle

Q ..... integer number 1 to 9
transmission number of the axis of the driven tool (=Y-axis)
Negative sign for Q is negative sense of rotation of the tool.

Q-... Negative sign for Q means negative direction of the tool rotation.

G250;Spindle synchronisation OFF

Example 1:
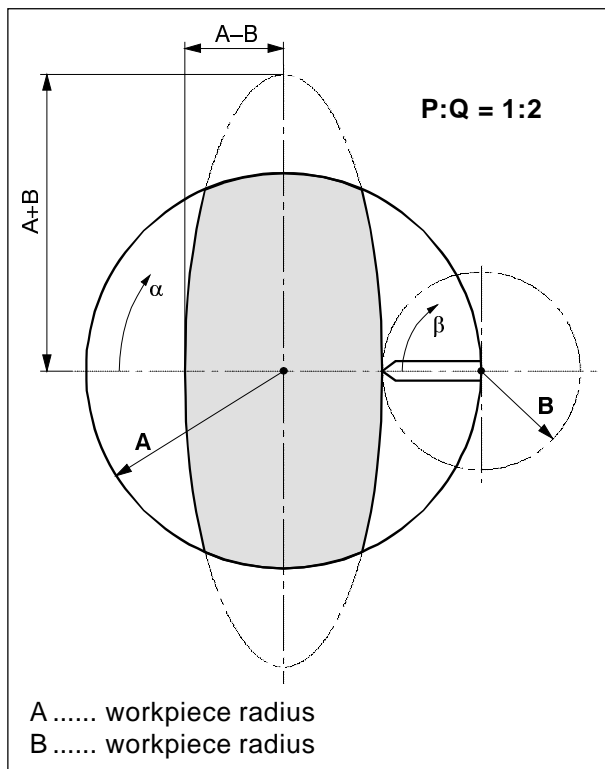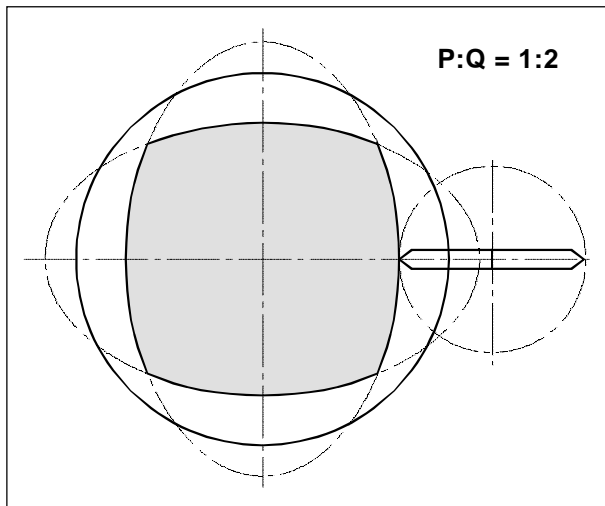• transmission ratio P:Q = 1:2
• single-edged tool

G51.2 P1 Q2 ;   (or Q-2)

The use of a tool with a cutter signifies that the cutter tip describes an ellipse, with "A+B" as long and "A–B" as short side.
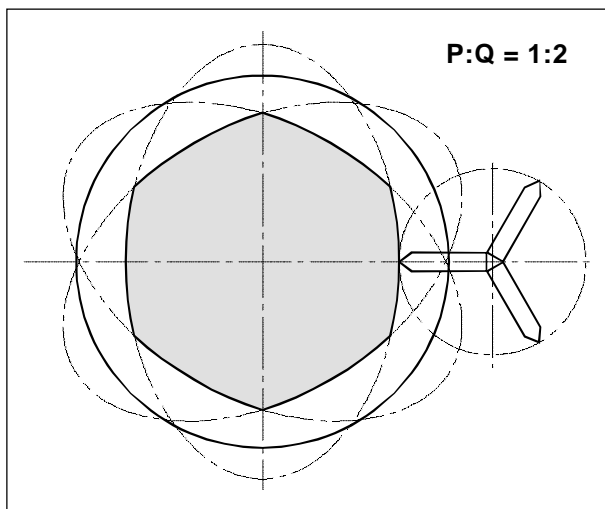You receive the marked area in the adjoining picture as workpiece contour.

P:Q = 1:2

A ...... workpiece radius
B ...... workpiece radius

Polygonal turning with single-edged tool

EMCO

Polygonal turning with double-edged tool

Polygonal turning with three-edged tool

- In the position display of the Y-axis during the movement of the Y-axis the machine coordinate values change from 0 to the parameter setting (traverse amount per rotation).
  Absolute and/or relative coordinate values are not updated.

- Absolute position recognition cannot be applied to the Y-axis.

- Manual continuous feed or handwheel feed are not possible if the Y-axis works in synchronous operation.

- The Y-axis in synchronous operation does not belong to the number of the simultanously controlled axes.

Example 2:
- Transmission ratio P:Q = 1:2
- double-edged tool (offset 180°)

G51.2 P1 Q2

A rectangle results as workpiece contour (see adjoining picture).

Example 3:
- transmission ratio P:Q = 1:2
- three-edged tool (offset 120°)

G51.2 P1 Q2

A hexagon results as workpiece contour (see adjoining picture).

Notes:
- For thread-cutting an error occurs at the starting point during synchronous operation.
  During thread-cutting the synchronization is to be switched off with G50.2.

- The following signals are effetive and/or not effective for the axis of the driven tools (Y-axis) in synchronous operation:
  valid signals :
  - machine look
  - servo off
  invalid signals:
  - feed stop
  - lock
  - oversteering
  - test run (in block G51.2 the rotation signal is not waited for during the test run).

- The Y-axis cannot be approached with a traverse command "Y...", it can only be indicated with the ratio to the main spindle speed.

- To return to the reference point the command "**G28 V0**" must be entered.
  In case of the input of G50.2 the Y-axis would stop in an undefined position. This could cause problems in case that the same contour is to be machined with two tools (roughing → finishing).

  By programming "G28 V0" first the tool and spindle are brought into the same position prior to further machining similar to the first machining procedure (the tool starts turning as soon as the spindle receives the "one rotation signal" of the position encoder).

- The Y-axis for the control of the tool rotation during polygonal turning is the 4th axis.

Example for Polygonal Turning:
Hexagon with key size 32mm

...	
G0X110Z80M5	
N11T909	driven 3-edge tool, dia 100 mm
G97S1000M4	Main spindle ON, counterclockwise
M214	reverse of rotation direction of driven tool (necessary only for left hand thread)
G0X32Z5	Positioning of the tool
G51.2P1Q2	(or Q-2) transmission ratio P:Q = 1:2
G1Z-5F.1	machining
G0 X100 Z20	Releasing position
G50.2	Synchronization OFF
M5	Spindle stop
G0X150Z20	Tool changing position
...	

EMCO

G90 Absolute Programming

Format

N... G90

The addresses have to be programmed as following:

X .... Diameter
U+/- .... Incremental in diameter (except some cycles)
Z+/- .... Absolute (referred to the workpiece zero point)
W+/- ... Incremental (actual) traverse distance

Notes
- Direct switchover from block to block between G90 and G91 is permitted.
- G90 and G91 may also be programmed with some other G functions
  (N... G90 G00 X... Z...).

G91 Incremental Programming

Format

N... G91

The addresses have to be programmed as following:

X,U ..... Diameter
Z,W .... Incremental (actual) traverse distance with sign

Notes see G90.

⚠

Attention
- The format G10 P0 X(U).. Z(W).. and/or G92 X(U).. Z(W).. is placed above all other zero point shifts (from G52 to G59)!
- The shift is not displayed on the screen!
- Attention with programs which were programmed for former control types (Fanuc 0-T, Fanuc 21-T).

Remedy
First call up G10 P0 X0 Z0 (reset of the zero point shift in G92) and input the following two blocks:
G10 L2 P1 X(U)... Z(W)... ;
G54 ;

G92 Adjustment for maximum Spindle Speed

Format

N... G92 S...   (Spindle speed limit)

With the command G92 the max. spindle speed can be declared (rpm) for the constant cutting speed (G96).

G92.1 Coordinate System Setting

See also to "Attention" on this side!

Format

N... G92 X... Z...   (Set coordinate system)
oder
N... G92 U... W...   (Shift coordinate system)

Example
You want to shift your workpiece zero from the right to the left side of the workpiece
Diameter of workpiece = 30 mm
Length of workpiece = 100 mm

Program

N... G90 .... Programming absolute
… .... Workpiece zero point right
… .... Right side of contour is finished
N180 G00 X35........ Retract
N185 Z-100 .... Movement distance = workpiece length
N190 G92 X35 Z0 . New zero point on the left side
… .... Workpiece zero point left
… .... Machine left side
N305 G00 X35........ Retract
N310 Z100 .... Movement distance = workpiece length
N315 G92 X35 Z0 . Workpiece zero point left again
… .... etc.

Zero offset with G92 is modal and it is not cancelled through M30 or RESET!
In this way, do not forget to reset the zero offset G92 before program is finished.
When zero offset will be inserted incrementally, the values U and W will be added to the last valid zero offset.

G94 Feed Rate in Minutes

The entry of the command G94 means that all values programmed under "F" (feed) are in mm/min.

G95 Feed Rate in Revolutions

The entry of the command G95 means that all commands programmed under "F" are in mm/revolution.

G96 Constant Cutting Speed

Unit: m/min
The control continually computes the spindle speed corresponding to the respective diameter.

G97 Constant Rotational Speed

Unit: rev/min

EMCO

M-Commands - Survey

Code	Meaning	Code	Meaning
M00	Programmed stop	M80	Reset freely programmable exit 1
M01	Selective stop	M81	Set freely programmable exit 1
M02	Program end	M82	Reset freely programmable exit 2
M03	Main spindle ON in clockwise direction	M83	Set freely programmable exit 2
M04	Main spindle ON in counter-clockwise direction	M84	Reset freely programmable exit 3
M05	Main spindle stop	M85	Set freely programmable exit 3
M08	Coolant ON	M86	Reset freely programmable exit 4
M09	Coolant OFF	M87	Set freely programmable exit 4
M10	Spindle clamping ON	M90	Manual chuck
M11	Spindle clamping OFF	M91	Tensile clamping device
M13	Driven tool - spindle ON in clockwise direction	M92	Pressure clamping device
M14	Driven tool - spindle ON in counter-clockw. dir.	M93	Cancel final position monitoring (no alarm if part clamped and final position reached)
M15	Driven tool - spindle stop		
M20	Tailstock backward	M94	Bar feed, bar feed magazine active
M21	Tailstock forward	M95	Bar feed, bar feed magazine not active
M23	Collecting tray backward	M96	Auto Cycle Start with closed door ON
M24	Collecting tray forward (swivel under spindle)	M97	Auto Cycle Start with closed door OFF
M25	Open clamping device	M98	Call-up subroutine
M26	Close clamping device	M99	Subroutine end
M29	Tapping without longitudinal compensation for driven tools	M103	Main spindle ON in clockwise direction with waiting time, until spindle speed is reached
M30	Program end	M104	Main spindle ON in counter-clockwise direction with waiting time, until spindle speed is reached
M39	AWZ positioning ON		
M40	AWZ positioning OFF	M108	SINIS minimal coolant ON (option)
M48	hydr. tool probe swivel IN in working area	M109	SINIS minimal coolant OFF
M49	hydr. tool probe swivel OUT	M113	Driven tool - spindle ON in clockwise direction with waiting time, until spindle speed is reached
M50	Deselection direction logic		
M52	Selection C-axis	M114	Driven tool - spindle ON in counter clockw. dir. with waiting time, until spindle speed is reached
M53	Deselection C-axis (also with M10)		
M57	Spindle oscillation ON (S....frequency)	M190	automatic changing of screen OFF
M58	Spindle oscillation OFF	M191	autom. changing of screen ON (into alarm mode)
M60	Tool probe calibration	M204	Reverse of rotation direction for main spindle ON (M3 = M4, M4 = M3)
M61	Start of tool measuring		
M66	Bar feed / bar loading magazine feed ON with feed stop	M205	Reverse of rotation direction, main spindle OFF (M3 = M3, M4 = M4)
M67	Bar feed / bar loading magazine feed ON without feed stop	M214	Reverse of rotation direction for driven tool ON (M13 = M14, M14 = M13)
M68	Bar feed / bar loading magazine feed OFF	M215	Reverse of rotation direction for driven tool OFF (M13 = M13, M14 = M14)
M69	Start bar change (without feed stop / read-in release)		
		M230	Collecting tray backward (without feed hold)
M70	Precise stop fine	M240	Collect. tray forward below spindel (without feedhold)
M71	Flushing pump for spindle ON	M241	Collect. tray forward to wating pos.l (without feedhold)
M72	Flushing pump for spindle OFF		

Attention:

If you switch from the main spindle over to the driven tool spindle or vice-versa, the command M3, M4, M13, M14 must be programmed in a separate line.

M-code and S-code must not be programmed in one line, the S-code must be programmed in the next line.

M0 Programmed Stop Unconditional

These command effects a stop in the execution of the part program.
Main spindle, feed and coolant will be switched off.
The chip protection door can be opened without triggering an alarm.

With "NC START" ⬦ the program run can be continued. After that the main drive will be switched on with all values which were valid before.

M1 Programmed Stop Conditional

M01 works like M00, but only if the key [OPT. STOP] was pressed.

With "NC START" ⬦ the program run can be continued. After that the main drive will be switched on with all values which were valid before.

M2 Main Program End

M02 works like M30.

M3 Main Spindle ON Clockwise

The spindle is switched on provided that a spindle speed or cutting speed has been programmed, the chip protection door is closed and a workpiece correctly clamped.
M03 must be used for all right-hand cutting or overhead clamped tools.

M4 Main Spindle ON Counterclockwise

The same conditions as described under M03 apply here.
M04 must be used for all left-hand cutting or normal clamped tools.

M5 Main Spindle Off

The main drive is braked electrically.
At the program end the main spindle is automatically switched off.

M8 Coolant ON

The coolant will be switched on.

M9 Coolant OFF

The coolant will be switched off.

M10 Spindle clamping ON

Spindle clamping is activated

M11 Spindle clamping OFF

Spindle clamping is switched off

M13 Driven Tool ON Clockwise

The driven tool is switched on provided that a spindle speed or cutting speed has been programmed, the chip protection door is closed and a workpiece correctly clamped.

M14 Driven Tool ON Counterclockwise

The same conditions as described under M13 apply here, but counterclockwise rotation direction.

M15 Driven Tool Off

The tool drive is braked electrically.
At the program end the driven tool is automatically switched off.

M20 Tailstock BACK

M21 Tailstock FORWARD

- The main spindle must be stopped for traversing the tailstock
- The main spindle can not be switched on and NC Start will be ignored, as long as the tailstock is in an undefined position (not in front or back end position)
- If the tailstock is in an intermediate position after power on, it traverses to the back end position automatically after pressing the AUX ON key.
- After pressing the tailstock key the screen shows the message "Tailstock in intermediate position" as long as the tailstock reaches a defined position (back end position or clamping position). The tailstock can be positioned in tip mode.

EMCO

M23 Workpiece Collection Device BACK

The collecting tray flaps back and leaves the working room.

M24 Workpiece Collection Device FORWARD

The collecting tray flaps forward to catch the workpiece.

The swivel sequence can occur only via CNC program. Swivel in and swivel out is possible only with closed door.

The supervision records only the swivel movement but not the needed time. That means: The execution time for the commands "workpiece tray back" and "workpiece tray forward" must be considered with programming (G4 - dwell time)

M25 OPEN Clamping Device

M26 CLOSE Clamping Device

Notes

- If you change the clamping device, check the clamping direction and adapt the machine configuration with M90-92.
- Open and close the clamping device is possible only with main spindle stop.
- The program run will be continued only after the clamping device is opened or closed completely.
- Opening the clamping device (manual or with M25) is completed only after the pressure switch and the end position proximity switch have acknowledged..

Manual operating

[clamping device key icon] Clamping device key

M29 Selection Rigid Tapping (without longitudinal compensation)

Is programmed in connection with S-code immediately before the cal of cycle G84 or G88

M30 Program END

With M30 all drives are switched off and the control is returned to the start of the program. Moreover, the counter level is increased by 1.

M39 Driven Tool Positioning ON
M40 Driven Tool Positioning OFF

The positioning of the driven tool will be activated or deselected.

M50 Direction Logic CANCEL of Tool Change

Effective blockwise.
The following tool change will be done clockwise (not necessarely on the shortest way).

M52 Selection Main Spindle as C-axis
M53 Deselection Main Spindle as C-axis

The main spindle is defined as C-axis.
Deselection can also be carried out by means of M10 (spindle clamping).

M70 Precise STOP Fine

If M70 is programmed in a traversing block, the following traverse movement will be started only after the axes have reached the target point (short axis stop).

M80-87 Freely Programmable Exits

see earlier in this chapter

Machine Configuration M90-M95

The M functions M90 - M95 describe the configuration of the machine (type of clamping device, bar feed / loader active, not active).

The control requires these data to control and supervise the corresponding functions correctly.

Procedure

- Set the key switch "data protection" on position 1.
- For the corresponding change at the machine (e.g. other clamping device) enter the respective M function in the MDI mode.
- The set machine configuation is kept, until it is deselected by an opposite M function (also after switching off the machine).

M90 Manual Chuck

(no clamping supervision)

M91 PULL Clamping Device

Clamping device that closes when the connecting tube pulls, e.g.:

EMCOTURN 325:
- Pneumatic jaw chuck, that clamps from outside to inside
- Pneumatic collet chuck

EMCOTURN 345:
- Hydraulic jaw chuck

M92 PUSH Clamping Device

Clamping device that closes when the connecting tube pulls, e.g.:

EMCOTURN 325:
- Pneumatic jaw chuck, that clamps (a tubular workpiece) from inside to outside

EMCOTURN 345:
- Hydraulic collet chuck

M93 Clamping Limit Monitoring CANCEL

No clamping supervision

M94 Bar Feed / Bar Loader Active

M95 Bar Feed / Bar Loader not Active

M96 Automatic Cycle-Start when Door Closed

(Option)

The machining program starts as soon as the chip guard door is closed.
The option "automatic cycle start when door closed" can also be used without automatic door. For safety reasons this function must be activated again (M96) after every machine off-on.
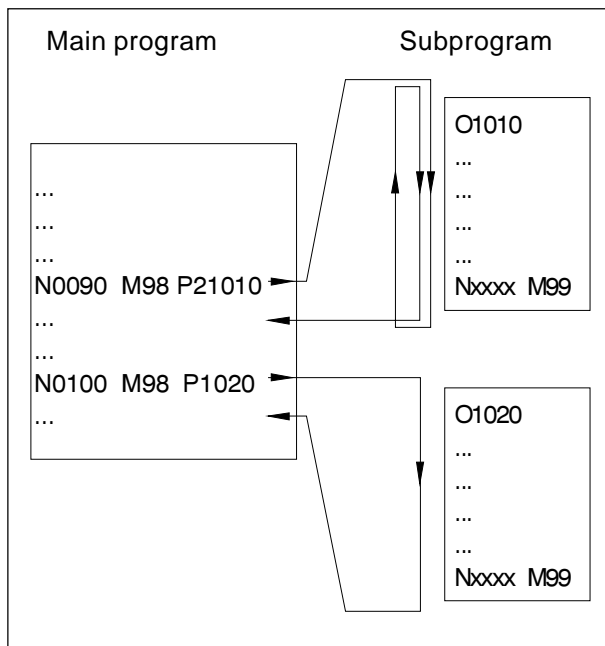
M97 CANCEL M96

EMCO

M98 Subprogram Call

Format
N... M98 P...

P .... The first four digits from the right deter-
mine the subprogram number, the other
digits the number of repetitions.

Notes

- M98 can be designated in the same block as the movement command (e.g. G01 X25 M98 P25001).

- When the count of repetitions is not specified, the subprogram is called once (M98 P5001).

- When the programmed subroutine does not exist an alarm occurs.

- A two loop subprogram call can be executed.
  (N... M98 P25001 ;   the program will be called two times)

M99 Subprogram End,
Jump Instruction

Format
N... M99 P...

M99 in the main program
Without jumping address:
Jump to the program start.
With jumping address Pxxxx:
Jump on block no. xxxx

M99 in the subprogram
Without jumping address:
Jump to the calling up program, on the next block after the calling up block (see drawing).
With jumping address Pxxxx:
Jump to the calling up program on block no. xxxx

Note
- M99 must be the last command in the sub-program.
- The jump-back occurs automatically into the next following block in the main program.
- M99 cannot be used internal the main program as a jump instruction.
  To perform a jump instruction in the main program it is possible to use the command GOTO
  For example:
  To jump from the actual position in program to block number N12, program the command GOTO12 in a seperate block.

Subroutine call from a main program

M103 Main Spindle ON clockwise with Waiting Time

Same code as M3, but:
The following blocks only then will be executed, if the spindlespeed has been reached.

M104 Main Spindle ON counter clockwise with Waiting Time

Same code as M4, but:
The following blocks only then will be executed, if the spindlespeed has been reached.

M113 Driven Tool ON clockwise with Waiting Time

Same code as M13, but:
The following blocks only then will be executed, if the spindlespeed of driven tool has been reached.

M114 Driven Tool ON counter clockwise with Waiting Time

Same code as M14, but:
The following blocks only then will be executed, if the spindlespeed of driven tool has been reached.

M204 Reverse of Rotation Direction of the Main Spindle ON

This command reverses the commands M3 and M4: M3=M4, M4=M3.

M205 Reverse of Rotation Direction of the Main Spindle OFF

This command delets the reverse the commands M3 and M4: M3=M3, M4=M4.

M214 Reverse of Rotation Direction of Driven Tool ON

M214 reverses the commands M13 and M14 (driven tool on): M13=M14, M14=M13.

M215 Reverse of Rotation Direction of Driven Tool OFF

M215 delets the reverse the commands M13 and M14: M13=M13, M14=M14.

EMCO

Bar Feed / Bar Loader Interface

General

- The signals from the machine to the loader are free programmable in the part program via M functions.

- The signals from the loader to the machine can be called free in the part program and can be used for jumps (branches) or waiting loops.

- The part program will always be started automatically by the machine, also at bar end.

M Functions

M57   Main spindle swing on
M58   Main spindle swing off
M66   Bar feed / Bar loader FEED ON with feed stop/input release
M67   Bar feed / Bar loader FEED ON without feed stop/input release
M68   Bar feed / Bar loader FEED OFF
M69   Start bar change without feed stop/input release
M71   Turn Spindle and clean clamping device ON
M72   Turn Spindle and clean clamping device OFF

Control signal (MACRO variable)

Bar end .... #1000
Loader has pushed forward .... #1001
Start after bar change .... #1002

By means of these signals (MACRO variable) you can branch in the part program and call or skip corresponding program parts or program a waiting loop.

Activate / Deactivate

M94   Activate bar feed / bar loader
M95   Deactivate bar feed / bar loader

Program Example for Operating a Bar Feed

Used functions

M67 Bar feed / Bar loader FEED ON without feed stop/input release
(aktive until M68 FEED OFF)

M68 Bar feed / Bar loader FEED OFF

N20 IF [#1000EQ0] GOTO 10
Query whether signal "bar end" is active.
The MACRO variable #1000 is compared with zero (EQ0).
When signal "bar end" is active (#1000 = 0) the program jumps to block number 10.
This creates an infinite loop (waiting loop), as long as "bar end" is active.

Program

%		
:5000		
M94	Activate bar feed / bar loader	
G0 X20 Z100 T101	Place bar stop	
M25	Open clamping device	
M67	Bar feed ON	Regular bar feed
G1 G94 F1000 Z150	Accompany bar to stop position	
M26	Close clamping device	
M68	Bar feed OFF	
N10	Block number for jump address	Infinite loop at bar
N20 IF [#1000EQ0] GOTO 10	Query for bar end	end
G0 X50 Z180	Approach free position	
T202		
G0 X30 M3 S3000		
G1 G95 F0.3 Z150		
....		Part machining
....		
....		
G0 X50 Z180		
M30		
%		

Note

This programming example was created for the EMCOTURN 325.
On other machines a larger traversing area can be programmed.

Alternative

When the bar feed / bar loader gives the signal "loader has pushed forward", you need not to program the function "bar feed OFF" (M68).

Changes in program:
• M66 Bar feed / Bar loader FEED ON with feed stop/input release instead of M67
• Cross out M68 (bar feed OFF).

EMCO

Program Example for Operating the Bar Loader EMCO LM800

Note
This programming example was created for the EMCOTURN 345.
Mind the machining area on other machines.

```
%
:2000                        (example for loaderprogram with MKE LM 800)

G28U0W0
G92S5000
G10P0Z-200
T0202                        (bar stop)
G0X100Z50
X0Z30
Z-15                         (pick up position)
M25                          (clamping device opening)
M66                          (feed in ON)
G1G94F6000Z0.4               (feed out position 0.4 for facing)
G4X0.5                       (delay time)
IF[#1000EQ1]GOTO10           (check end of bar:
                             if "yes", control will read next line, if "not" jump to line number N10)
G1G94F8000Z250               (position to trop the rest pice)
M69                          (start bar change)
G0Z0.4                       (position for the new bar)
N20
IF[#1002EQ0]GOTO20           (check barfeed feeded forward)
M66                          (feed in ON)
N10 M26                      (clamping device closing)
M66                          (check barfeed feeded backward)
G0Z30
X150Z120                     (index position)
G95
T1212
......................       (standard program to machine the part)
M30                          (end of program)
```

Information:

Keep-relay: K1 = 1xxx1xxx
aktivate lader: M94

Program Example for Operating the Bar Loader Kupa LM

Used functions

M66   Bar feed / Bar loader FEED ON with feed stop/input release

M69   Start bar change

IF [#1000EQ1] GOTO 66
Query whether signal "bar end" is not active.
The MACRO variable #1000 is compared with one (EQ1).
When signal "bar end" is not active (#1000 = 1) the program jumps to block number 66 and a normal bar feed will be done.
When signal "bar end" is active, no jump is done and the rest piece will be ejected and the bar change will be proceeded.

> **Note**
> This programming example was created for the EMCOTURN 345.
> Mind the machining area on other machines.

Subprogram for bar loader

% :1000 (KUPALOADER SPF)

M94	Activate bar loader
T0404	Place bar stop
G0 X0 Z-20	Stop position
M25	Open clamping device
IF [#1000 EQ1] GOTO 66	Jump over bar change
G0 X0 Z150	Approach free position
M24	Swivel in workpiece tray
G4 X3	Dwell
M66	Bar feed - eject rest piece
M23	Swivel out workpiece tray
G4 X1	Dwell
G0 X0 Z20	Stop position
M69	Bar change
N66 M66	Bar feed until bar stop
G1 G94 F2000 Z.5	Bar stop pushes bar to clamping position
M26	Close clamping device
G4 X.5	Dwell
G0 X120 Z100	Approach tool change position
(M66)	Some KUPA-types (software versions) need this command, when an alarm occurs remove this command out of the program.
M99	Subprogram end

Bar change (applies to block from G0 X0 Z150 to M69)

Bar feed (applies to block from N66 M66 to (M66))

Main program

% :1234 (MAINPROGRAM)

G28 U0 W0	Approach reference point
G92 S4000	Spindle speed limit
G10 P0 Z-200	Zero offset
M98 P1000	Subprogram call

Bearbeitung

N300 M30	Program end

EMCO

Program Example for Operating a Bar Loader

(not for Kupa LM and EMCO LM800 loaders)

Used functions

M67 Bar feed / Bar loader FEED ON without feed stop/input release
(aktive until M68 FEED OFF)

M68 Bar feed / Bar loader FEED OFF

M69 Start bar change

IF [#1000EQ1] GOTO 10
Query whether signal "bar end" is active.
The MACRO variable #1000 is compared with one (EQ1).
When signal "bar end" is not active (#1000 = 1) the program jumps to block number 10 - the bar change will be skipped.

IF [#1002EQ0] GOTO 100
Query whether signal "start after bar change" is active.
The MACRO variable #1002 is compared with zero (EQ0).
When signal "start after bar change" is not active (#1002 = 0) the program jumps to block number 100.
This creates an infinite loop (waiting loop), until "start after bar change" is active.

Alternative to the following program

When the bar feed / bar loader gives the signal "loader has pushed forward", you need not to program the function "bar feed OFF" (M68).

Changes in program:
• M66 Bar feed / Bar loader FEED ON with feed stop/input release instead of M67
• Cross out M68 (bar feed OFF).

Note
This programming example was created for the EMCOTURN 325.
Mind the machining area on other machines.

Subroutine for bar loader

```
%
:1000 (KUPA-BAR LOADER-CYCLE-METR)
```

M94	Activate bar loader	

T202	Swivel in bar stop	
G0 Z10		
X0 Z0	Position bar stop	Normal bar feed
M25	Open clamping device	
M66	Bar feed on	
G1 G94 F1000 Z0	Bar forward	
IF [#1000EQ1] GOTO 10	Inquiry bar end	

M25	Open clamping device	
G0 Z100	Releasing the bar stop	
M67	Bar feed on	Residual
G1 G94 F1000 Z100	Residual workpiece ejection	workpiece ejection
G4 X2	Dwell time 2 seconds	

M25	Open clamping device	
T202	Bar stop	
G0 X0 Z3	Position bar stop at beginning of bar	Bar change
M69	Start bar change	

N100	Block no. for jump address	Waiting loop at
IF [1002EQ0] GOTO 100	Inquiry start after bar change	bar change end

T202	Bar stop	
G0 X0 Z3	Position bar stop	Bar feed after bar
M66	Bar feed on	change
G1 G94 F1000 Z3	Bar forward	
M26	Close clamping device	

G0 X100 Z100		
T1212	Rough facing device	
G96 S120 M4 G95 F.15		First part
G0 Z10		machining
X38 Z0		
G1 X-1 F.15 M8		
G0 X150 Z10 M5 M9		
GOTO 30	Jump to program end	

N10 M26	Close clamping device	
G0 X100 Z80	Bar stop backward	Normal part
G95	Selection feed in mm/r	machining

N30 M99	Subroutine end	
%		

EMCO

Reversing Play Compensation

Only for EMCOTURN 325.

The control compensates the reversing play of the slides.
After every readjustment of the slideways the reversing play must be measured and entered into the machine parameters. See "F Maintenance and Readjustment works".

To alter the parameter first set write enable:

- Set the key switch "data protection" on position 1.

- Select MDI mode (⬛▶).

- Press the key [OFFSET SETING].

- Press the softkey SETING

- Move the cursor on PARAMETER WRITE.

- Enter: 1 [INPUT].

- The screen shows alarm 100.

- Cancel alarm 100 with [CAN] [RESET].

- Now the machine parameters can be altered.

Input of the reversing play

- Press the key [SYSTEM].

- Press the softkey PARAM.

- Enter: 1851 and softkey NO.SRH.

- The scren shows: 1851 BACKLASH X, Z

- Enter the measured values for the reversing play in X and Z.

> **Notes:**
> To avoid accidentally altering reset the write enable parameter back to 0 (is done automatically when switching off the machine).
>
> **For Emcoturn 345-II:**
> The reversing play for the driven tool has to be adjusted by setting a compensation value in the parameter "5321" (\#1):
> 1 revolution = 4096 increments

Chip conveyor

Programming:

While the programming is running, the chip conveyor is switched on in certain intervals, runs for the period set in C8 and is switched off again.

- The intermission time between the running periods is programmed in the counter C 12 (ex works: 200 s).
- Only the time in which the program is active is calculated.
- If the chip conveyor is switched on (LED above conveyor key is illuminating), it runs according to the running time C8 even if the program is terminated.

Manual operation:

Switch on chip conveyor:
Forward: press key less than 1 second, forward run for time C8 (LED is illuminating).
Backward: press key more than 1 second, reverse run as long as key is pressed (LED is flashing slowly), in the subsequent forward run for time C8.

Interrupt chip conveyor:
For emptying etc. press simultaneously AUX ON and chip conveyor key (LED is flashing quickly). Switch on again with chip conveyor key.

Note
To avoid contaminations of the emulsion, the chips are to be removed from the working area prior to operation end.
Thus, a manual after-running of the chip conveyor is recommended.

C08 = Run Time [sec]
C12 = Intermission Time [sec]

Adjustment of run time/intermission time of the chip conveyor

Program interruption to open the door

Sequence:

- Set feed override switch to 0%.

- Press key NC Stop.

- Door can be opened (consent key)..

- Close door

- Press key NC Start.

- Set override switch to 100 %.

EMCO

Tool Data Measuring with Scratching

- Clamp a worpiece with measured diameter and length and return the slides of the machine to reference point.
- Start spindle in MDI mode (M03/M04 S ....)
- Swivel in the desired tool.

X correction

- Scratch with the tool on the diameter of the workpiece (B).

- Press the key [OFFSET SETTING] and the softkey GEOM.

- Select tool station number of the respective tool with cursor keys [↑], [↓].

- Press the softkey OPRT.

- Enter the workpiece diameter e.g. 47.

- Press the softkey MEASUR.

- The X value will be taken over into the tool data register.

Z correction

- Scratch with the tool on the face of the workpiece (A).

- Press the key [OFFSET SETTING] and the softkey GEOM.

- Select tool station number of the respective tool with cursor keys [↑], [↓].

- Press the softkey OPRT.

- Enter the length L (workpiece length + chuck length - see drawing), e.g. [Z_B] 72.

- Press the softkey MEASUR.

- The Z value will be taken over into the tool data register.

Repeat this sequence for every required tool.

Dimensions for scratching method:
A   Scratching on face
B   Scratching on circumference
D   Work piece diameter
L   Work piece length + chuck length

Tool Data Measuring with the Optical Preset Device

- Mount optical preset device
- Clamp gauge with toolholder in tool turret disk.
- MANUAL mode, traverse gauge into the reticule of the optical preset device (at open door in setup mode with consent key).
- Press key `POS` and softkey REL.
- Press keys `X_A` and `0`.
- Softkey PRESET drücken (X-Wert wird gelöscht).
- Press the keys `Z_B` and `0`.
- Press the softkey PRESET (Z value will be deleted).
- Set mode selection switch to INC 1000 and traverse in W to position -20 mm (= length of the gauge).
- Set W value to 0 again (`Z_B`, `0`, PRESET).
- Swivel in tool and traverse it into the reticule.
- Press the key `OFFSET SETTING`.
- Press the softkey OPRT.
- Select tool station number of the respective tool with cursor keys `↑`, `↓`.

X correction

- Set cursor to the X-value of the selected offset-number.
- Press the key `X_A` and the softkey INP C.
- X value is taken over into the tool data memory.

Z correction

- Set cursor to the Z-value of the selected offset-number.
- Press the key `Z_B` and the softkey INP C.
- Z value is taken over into the tool data memory.
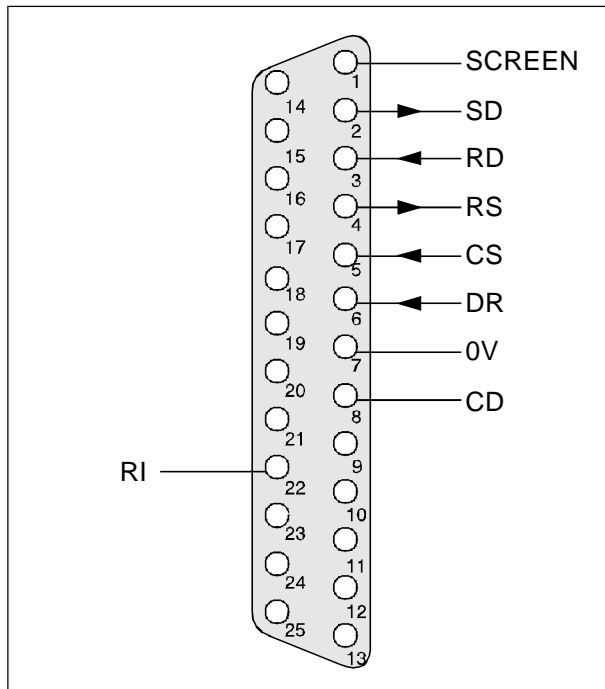
EMCO

Serial Data Interface

You can send data (programs, tool data, machine data) from the control to the PC (e.g. for archive) or also create programs at a PC and send it to the control.
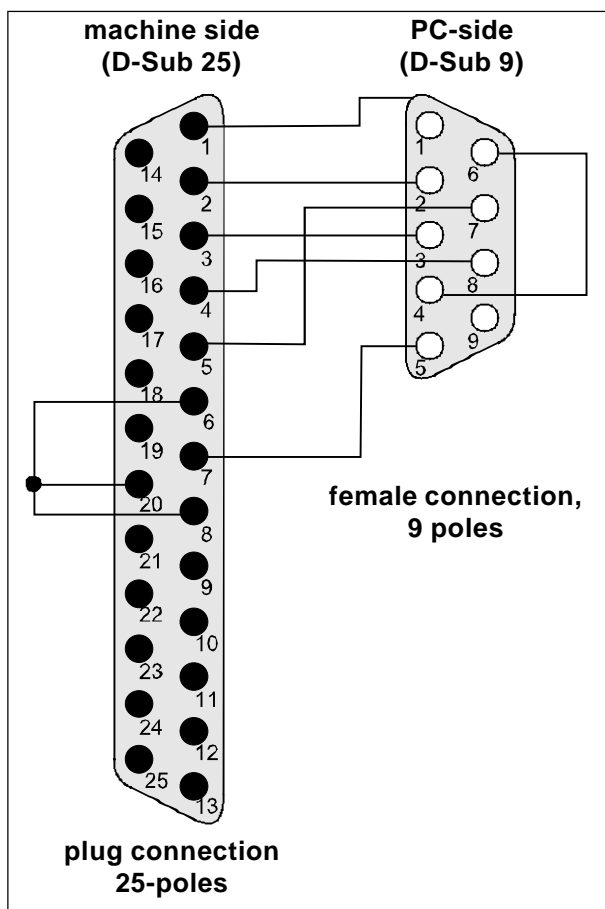
Signal names

SCREEN	Schielding
SD	Transmit data
RD	Receive data
RS	Request to send
CS	Clear to send
DR	Data send ready
0V	Ground
CD	Data terminal ready
RI	Ring indicator

Serial interface - occupation

Transmission cable

To transmit data between control and Personal Computer you need a cable with the pin assignement as shown beside.

Settings for data transmission

Standard values of the control:
- 9600 baud
- 2 Stop bits
- 7 Data bits
- Parity even

Please set your PC equivalent to the control.

Cable connection machine-PC

Transmission of NC-programs

- EDIT mode
- Press key SYSTEM
- 3 x press the right menu extension key ▷
- Press the following softkeys:
  ALL IO, PROGR, OPRT, PUNCH, EXEC.

Data will be transmitted. The transmission is executed successfully, when no error message appears.