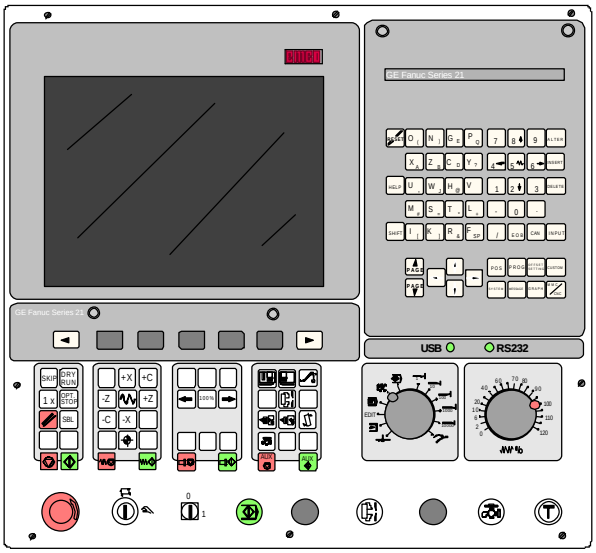
EMCO WinNC GE Series Fanuc 21 TB

Software description/ Software version from 13.76



Software description
EMCO WinNC Fanuc 21 TB

Ref.No. EN 1902 Edition C2003-7

EMCO Maier Ges.m.b.H.

P.O. Box 131

A-5400 Hallein-Taxach/Austria

Phone ++43-(0)62 45-891-0

Fax ++43-(0)62 45-869 65

Internet: www.emco.at

E-Mail: service@emco.co.at

innovative machine tools
industrial training systems

Preface

The EMCO WinNC GE SERIES FANUC 21TB Turning Software is part of the EMCO training concept on PC-basis.

This concept aims at learning the operation and programming of a certain machine control on the PC.

The milling machines of the EMCO PC TURN und CONCEPT TURN series can be directly controlled via PC by means of the EMCO WinNC for the EMCO TURN.

The operation is rendered very easy by the use of a digitizer or the control keyboard with TFT flat panel display (optional accessory), and it is didactically especially valuable since it remains very close to the original control.

This manual does not include the whole functionality of the control software GE SERIES FANUC 21TB Turning, however emphasis was laid on the simple and clear illustration of the most important functions so as to achieve a most comprehensive learning success.

In case any questions or proposals for improving this manual should arise, please contact us directly:

EMCO MAIER Gesellschaft m. b. H.
Department for technical documentation
A-5400 Hallein, Austria

All rights reserved, reproduction only by authorization of Messrs. EMCO MAIER
© EMCO MAIER Gesellschaft m.b.H., Hallein 2003

Contents

A: Key Description

Control Keyboard, Digitizer Overlay A1
Key Functions A1
Data Input Keys A2
Function Keys A2
Machine Control Keys A4
PC Keyboard A6

B: Basics

Reference Points of the EMCO Lathes B1
Zero Offset B2
The Coordinate System B2
Coordinate System for Absolute Value Programming B2
Coordinate System for Incremental Value Programming . B2
Input of the Zero Offset B3
Tool Data Measuring B4
Tool Data Measuring with the Optical Presetting Device B5
Tool Data Measuring with Scratching B6

C: Operating Sequences

Survey Operating Modes C1
Approach the Reference point C2
Input of the Gear Position C3
Setting of Language and Workpiece Directory C3
Program Input C4
Call Up a Program C4
Input of a block C4
Search a Word C4
Insert a Word C4
Alter a Word C4
Delete a Word C4
Insert a Block C4
Delete a Block C4
Data Input - Output C5
Delete a Program C5
Delete All Programs C5
Adjusting the Serial Interface C5
Program Output C6
Program Input C6
Tool Offset Output C6
Tool Offset Input C6
Print Programs C6
Program Run C7
Start of a Part Program C7
Displays while Program Run C7
Block Search C7
Program Influence C7
Program interruption C7
Display of the Software Versions C7
Part Counter and Piece Time C8
Graphic Simulation C9

D: Programming

Program Structure D1
Used Addresses D1
Survey of G Commands for Command Definition A, B, C D2
Survey of G Commands for Command Definition C D2
M- Commands D3
Description of G Commands D4
G00 Positioning (Rapid Traverse) D4
G01 Linear Interpolation (Feed) D4
Insertion of Chamfers and Radii D5
Direct Drawing Input D6
G02 Circular Interpolation Clockwise D8
G03 Circular Interpolation Counterclockwise D8
G04 Dwell D8
G7.1 Cylindrical Interpolation D9
Example - Cylindrical Interpolation D10
G10 Data Setting D11
Notes: D12
G12.1/G13.1 Polar Coordinate Interpolation D12
G-codes which may be programmed in the mode "polar coordinate interpolation: D12
Example - Polar Coordinate Interpolation D13
G17-G19 Plane Selection D14
G20 Longitudinal Turning Cycle D15
G21 Thread Cutting Cycle D16
G24 Face Turning Cycle D17
G28 Return to Reference Point D17
G33 Thread Cutting D18
Cutter Radius Compensation D19
Tool pathes with selection / cancellation of the cutter radius compensation D20
Tool pathes with program run with active cutter radius compensation D20
G40 Cancel Cutter Radius Compensation D21
G41 Cutter Radius Compensation Left D21
G42 Cutter Radius Compensation Right D21
G70 Measuring in Inches D22
G71 Metrical Measuring D22
G72 Finishing Cycle D23
G73 Contour turning cycle D24
G74 Facing cycle D26
G75 Pattern Repeating D28
G76 Deep hole drilling /Face Cut-in Cycle D29
G77 Cut-in Cycle (X Axis) D30
G78 Multiple Threading Cycle D31
Systematic G98/G99 ..... D32
G80 Cancel Cycles D33
G83 Drilling Cycle D33
G84 Tapping Cycle D34
Deep-hole drilling, G83 and tapping, G84 at the main spindle with stationary tools D35
G85 Reaming Cycle D36
G90 Absolute Programming D37
G91 Incremental Programming D37
G92 Spindle Speed Limit D37
G92 Coordinate System Setting D37
G94 Feed Rate in Minutes D38
G95 Feed Rate in Revolutions D38
G96 Constant Cutting Speed D38
G97 Constant Rotational Speed D38

Description of M Commands D39
M00 Programmed Stop Unconditional D39
M01 Programmed Stop Conditional D39
M02 Main Program End D39
M03 Main Spindle ON Clockwise D39
M04 Main Spindle ON Counterclockwise D39
M05 Main Spindle Off D39
M08 Coolant ON D40
M09 Coolant OFF D40
M20 Tailstock BACK D40
M21 Tailstock FORWARD D40
M25 Open Clamping Device D40
M26 Close Clamping Device D40
M30 Program End D40
M71 Puff Blowing ON D40
M72 Puff Blowing OFF D40
M98 Subprogram Call D41
M99 Subprogram End, Jump Instruction D41
Application of the C-axis D43
Note D43
Axial working with driven tools D44
Deep-hole drilling axial with driven tools, G83 ..... D44
Tapping axial with driven tool, G84 ..... D45
Deep-hole drilling, G83 and tapping,
G84 axial with driven tool D46
Radial working with driven tools D47
Deep-hole drilling radial with driven tool, G77 ..... D47
Tapping radial with driven tool, G33 ..... D48
Deep-hole drilling, G77 and tapping,
G33 radial with driven tool D49

G: Flexible NC programming

Variables and arithmetic parameters G1
Calculating with variables G1
Control structures G2
Relational operators G2

H: Alarms and Messages

Input Device Alarms 3000 - 3999 ..... H2
Machine Alarms 6000 - 7999 ..... H3
Axis Controller Alarms 8000 - 9999 ..... H11

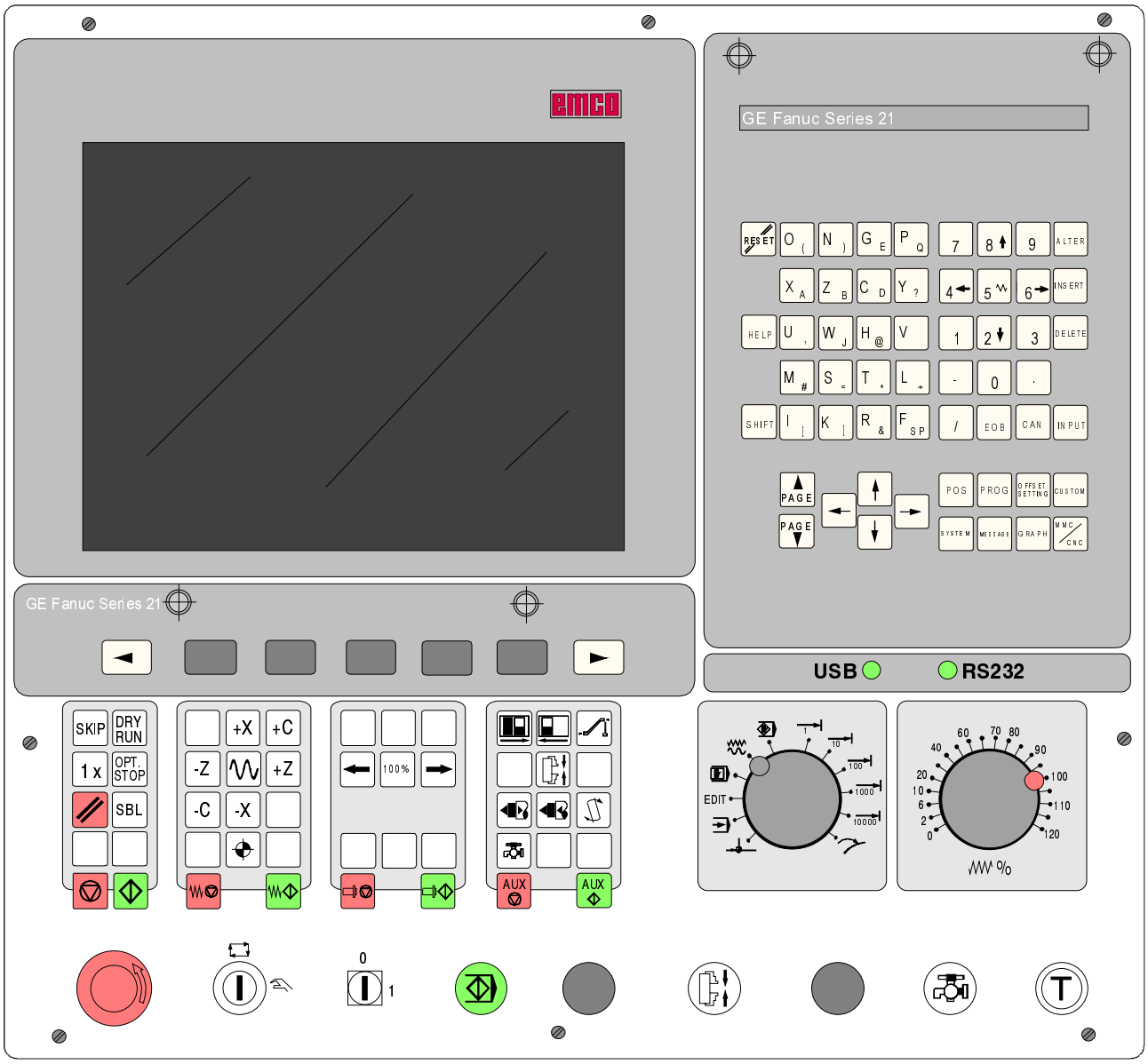
I: Control Alarms

Control Alarms I1

Starting Information

see attachment

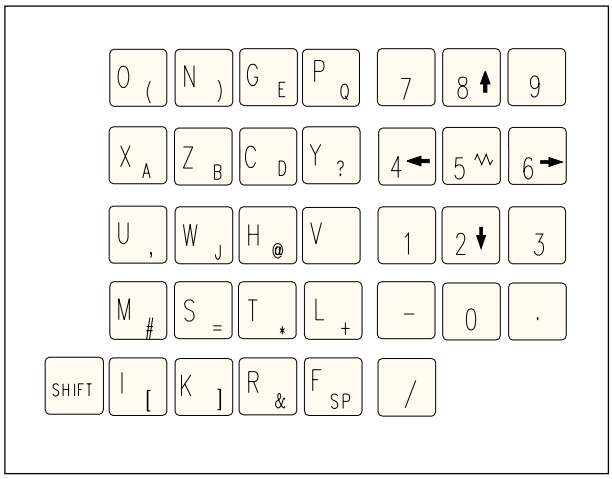
A: Key Description
Control Keyboard, Digitizer Overlay



Key Functions

RESET ..... Cancel an alarm, reset the CNC (e.g. interrupt a program), etc.
HELP ..... Helping menue
CURSOR ..... Search function, line up/down
PAGE ..... Page up/down
ALTER ..... Alter word (replace)
INSERT ..... Insert word, create new program
DELETE ..... Delete (program, block, word)
EOB ..... **E**nd **O**f **B**lock

CAN ..... Delete input
INPUT ..... Word input, data input
POS ..... Indicates the current position
PROG ..... Program functions
OFSET SETTING . Setting and display of offset values, tool and wear data, variables
SYSTEM ..... Setting and display of parameter and display of diagnostic data
MESSAGES ..... Alarm and message display
GRAPH ..... Graphic display

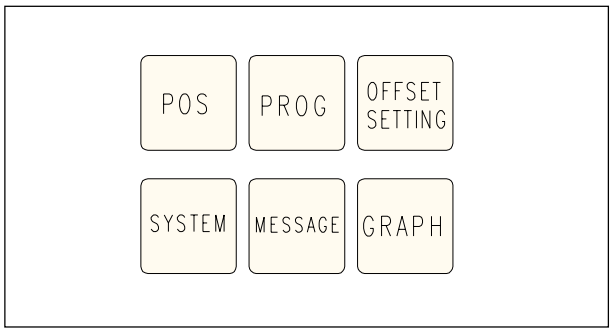


Data input keys

Data Input Keys

Note for the Data Input Keys

Each data input key runs several functions (numbers, address character(s)). Repeated pressing of the key switches to the next function automatically.



Function keys

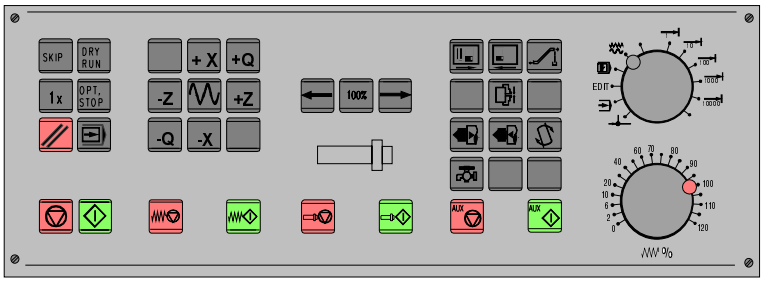
Function Keys

Note for Function Keys

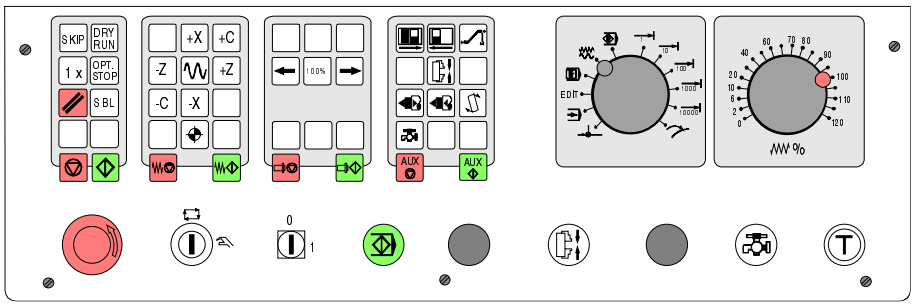
With the PC keyboard the function keys can be displayed as softkeys by pressing the key F12.

Machine Control Keys

The machine control keys are in the lower block of the control keyboard resp. the digitizer overlay.
Depending on the used machine and the used accessories not all functions may be active.



Machine control keyboard of the EMCO control keyboard



Machine control keyboard of the EMCO PC- Turn Series

SKIP (skip blocks will not be executed)

DRY RUN (test run of programs)

OPT STOP (program stop at M01)

RESET

Single block machining

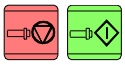
Program stop / program start

Manual axis movement

Approaching the reference point in all axes

Feed stop / feed start

Spindle override lower / 100% / higher



Spindel stop / spindle start; spindle start in JOG and INC1...INC10000 mode:

Clockwise: perss key short, Counterclockwise: press min. 1 sec.

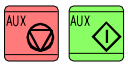
Open / close door

Close / open clamping device

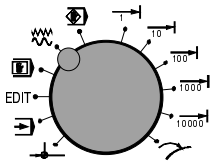
Tailstock back / forward

Swivel tool holder

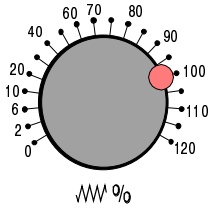
Coolant / puff blowing on / off



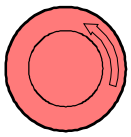
AUX OFF / AUX ON (auxiliary drives off / on)



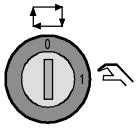
Mode selector



Feed / rapid feed override switch



EMERGENCY OFF (Unlock: pull out button)



Key switch for special operations (siehe Maschinenbeschreibung)

Additional NC start key

Additional key clamping device

Consent key

No function

PC Keyboard

$4 = 4 ⇧ $4 = $ Strg $4 = ⚙ Alt $4 = INC 1 000

Some alarms will be acknowledged with the key ESC.

By pressing the key F1 the modes (MEM, EDIT, MDI,...) will be displayed in the softkey line.

The assignement of the accessory functions is described int the chapter "Accessory Functions".

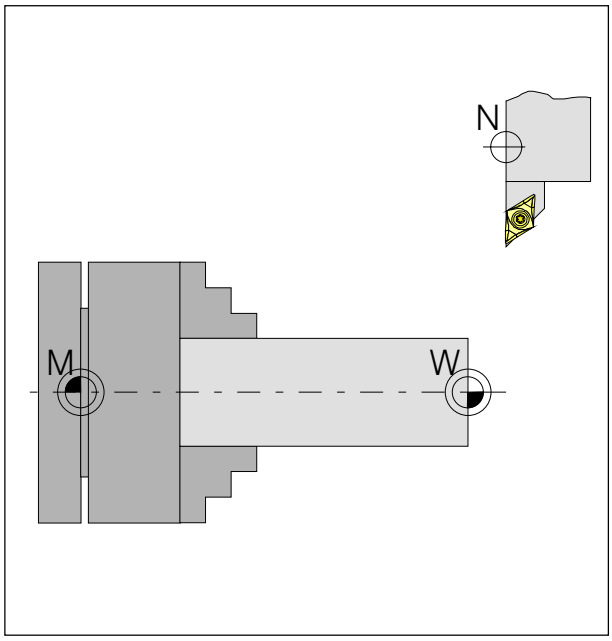
The meaning of the key combination ctrl 2 depends on the machine:
EMCO PC MILL 50/55: Puff blowing ON/OFF
EMCO PC MILL 100/125/155: coolant ON/OFF

* With F12 the function keys POS, PROG, OFFSET SETTING, SYSTEM, MESSAGES and GRAPH will be displayed in the softkey line.

The machine functions in the numeric key block are active only with active NUM lock.

B: Basics

Reference Points of the EMCO Lathes



Reference points in the working area

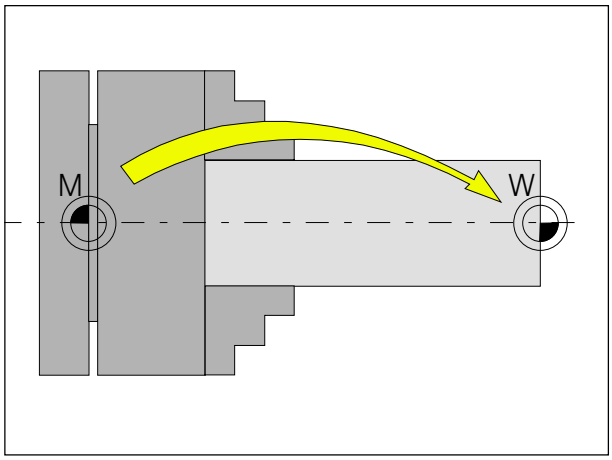
M = Machine zero point
An unchangeable reference point established by the machine manufacturer.
Proceeding from this point the entire machine is measured.
At the same time "M" is the origin of the coordinate system.

R = Reference point
A position in the machine working area which is determined exactly by limit switches. The slide positions are reported to the control by the slides approaching the "R".
Required also after every power failure.

N = Tool mount reference point
Starting point for the measurement of the tools. "N" lies at a suitable point on the tool holder system and is established by the machine manufacturer.

W = Workpiece zero point
Starting point for the dimensions in the part program. Can be freely established by the programmer and moved as desired within the part program.

Zero Offset



Zero offset from machine zero point M to workpiece zero point W

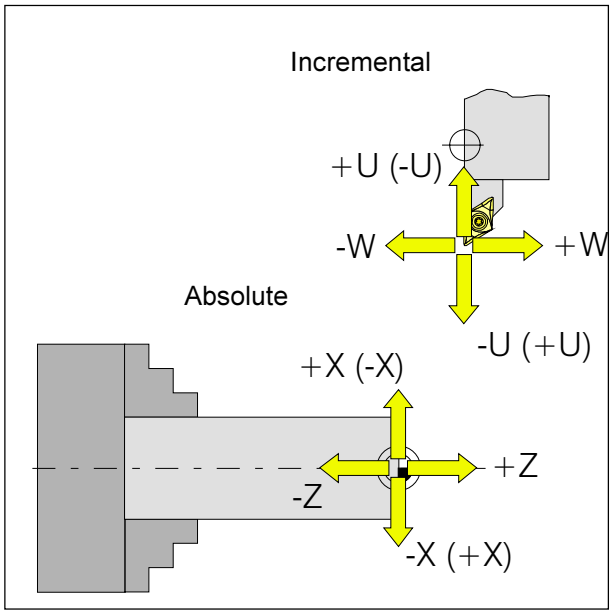
With EMCO lathes the machine zero "M" lies on the rotating axis and on the end face of the spindle flange. This position is unsuitable as a starting point for dimensioning. With the so-called zero offset the coordinate system can be moved to a suitable point in the working area of the machine.

The offset register offers one adjustable zero offset.

When you define a value in the offset register, this value will be considered with program start and the coordinate zero point will be shifted from the machine zero M to the workpiece zero W.

The workpiece zero point can be shifted within a program with "**G92 - Coordinate system setting**" in any number. At work often be done this with G10 -Data Setting.
More informations see in the command description.

The Coordinate System

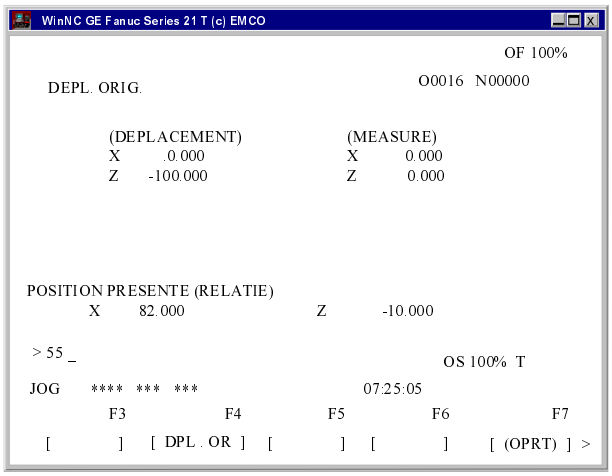


Absolute coordinates refer to a fixed position, incremental coordinates to the tool position.
The bracket values for X, -X, U, -U are valid for the PC TURN 50/55 because the tool is in front of the turning centre on this machine.

The X coordinate lies in the directions of the cross slide, the Z coordinate in the direction of the longitudinal slide.
Coordinate values in minus directions describe movements of the tool system towards the workpiece. Values in plus direction away from the workpiece,

Coordinate System for Absolute Value Programming
The origin of the coordinate system lies at the machine zero "M" or at the workpiece zero "W" following a programmed zero offset.
All target points are described from the origin of the coordinate system by the indication of the respective X and Z distances.
X distances are indicated as the diameter (as dimensioned on the drawing).

Coordinate System for Incremental Value Programming
The origin of the coordinate system lies at the tool mount reference point "N" or at the cutting tip after a tool call-up.
The U coordinate lies in the direction of the cross slide, the W coordinate in the direction of the longitudinal slide. The plus and minus directions are the same as for absolute value programming.
With incremental value programming the actual paths of the tool (from point to point) are described.
X distances are indicated as the diameter.

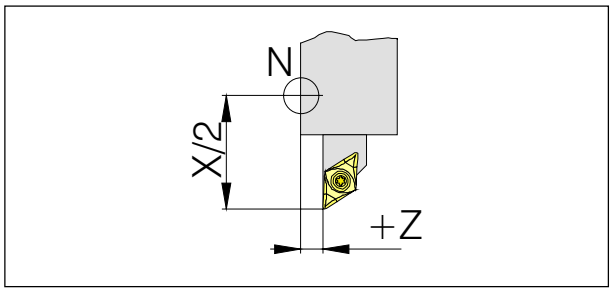


Input pattern for the zero offset

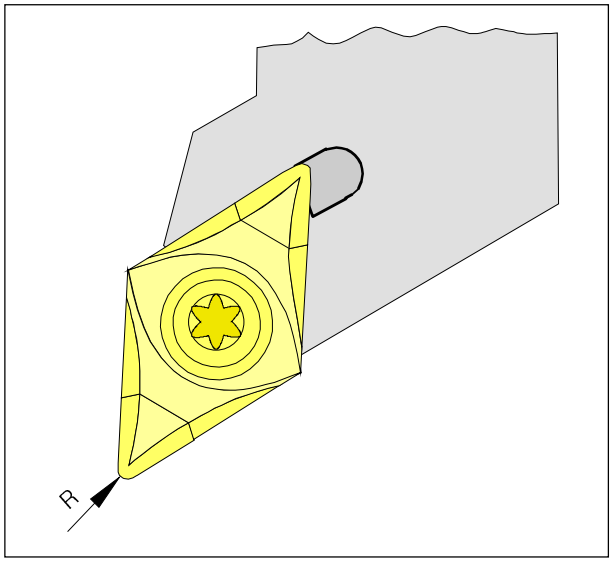
Input of the Zero Offset

- Press the key OFFSET SETTING
- Select the softkey W. SHFT (work shift)
- The input pattern beside appears
- Below (SHIFT VALUE) X, Z you can enter the offset **from the workpiece zero point to the machine zero point (neg. sign)**.
- Enter the offset (e.g.: Z-30.5) and press the key INSERT
- This offset is always active (without separate call-up).

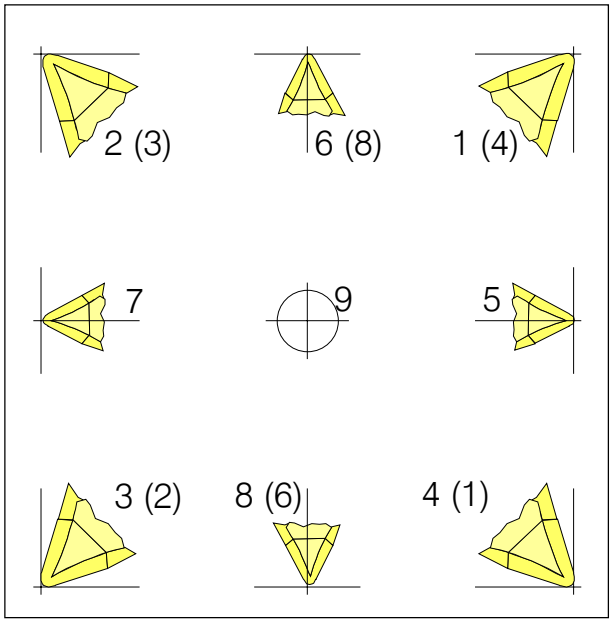
Note:
With this offset normally the coordinate zero will be shifted from the spindle flange to the stop face of the clamping device.
The work piece length (zero shift to the right work piece face) will be considered in the program with G92.



Length correction



Radius of the cutter tip R



Cutter position T
Look at the tool like it is clamped at the machine to determine the cutter position. For machines on which the tool is below (in front of) the turning centre (e.g. PC TURN 50/55) use the values in brackets because of the opposite +X direction.

Tool Data Measuring

Aim of the tool data measuring:
The CNC should use the tool tip for positioning, not the tool mount reference point.

Every tool which is used for machining has to be measured. The distances in both axis directions between tool tip and tool mount reference point "N" are to be measured.

In the so-called tool register the measured length corrections, the cutter radius and the cutter position can be stored.
(standard = 16)

The correction number can be any register number, but has to be considered with tool call in program.

Example
The length corrections of a tool in the tool turret station 4 have been stored as correction number 4.

Tool call in program: **T0404**

The first two numbers of the T word mark the position in the tool turret, the two last numbers mark the correction number belonging to it.

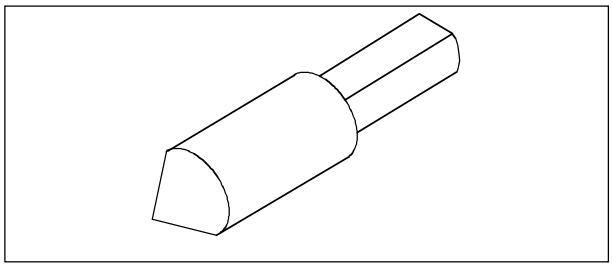
The length corrections can be measured half-automatically, **cutter radius** and **cutter position** have to be inserted manually.

Inserting cutter radius and cutter position is only necessary for using cutter radius compensation with this tool.

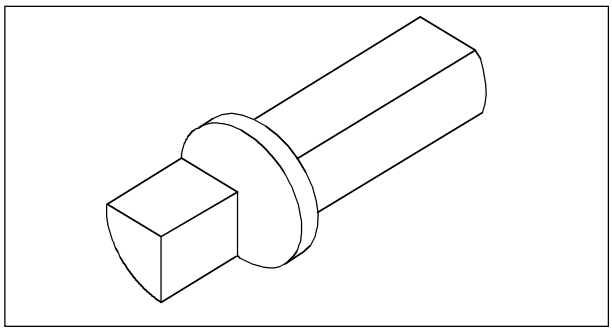
Tool data measuring occurs for
X in diameter
Z absolute from point "N"
R radius of the cutter tip
T cutter position

With "**offset wear**" occurs the correction of not exact measured tool data or of worn tools after several machining runs. The inserted length corrections will be added to or subtracted from the geometry of the tool incrementally.

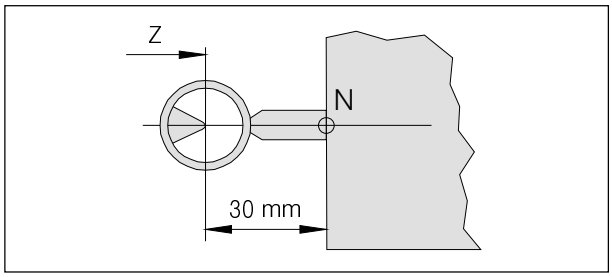
X+/- .... incremental in diameter to the value of the geometry
Z+/- ..... incremental to the value of the geometry
R+/- .... incremental to the value of the geometry



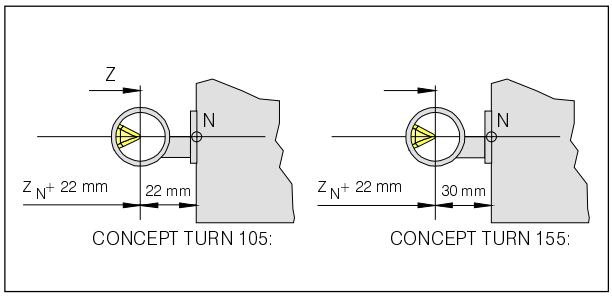
Reference tool Concept TURN 50/55



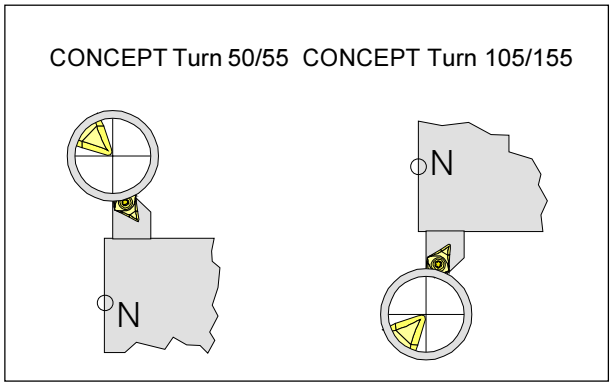
Reference tool Concept TURN 105/155



Reference tool measuring Concept Turn 50/55



Reference tool measuring Concept Turn 105/155



Traverse into the graticule with the tool

Tool Data Measuring with the Optical Presetting 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 key X A and softkey PRESET (X value will be deleted).
- Press the key Z B and softkey PRESET (Z value will be deleted).
- Set mode selection switch to INC 1000 and traverse in Z the length of the gauge (Z-) (Concept Turn 50/55/155: -30, Concept Turn 105: -22)
- Press the key Z B and softkey PRESET (Z value will be deleted).
- 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 ↓ ↑.

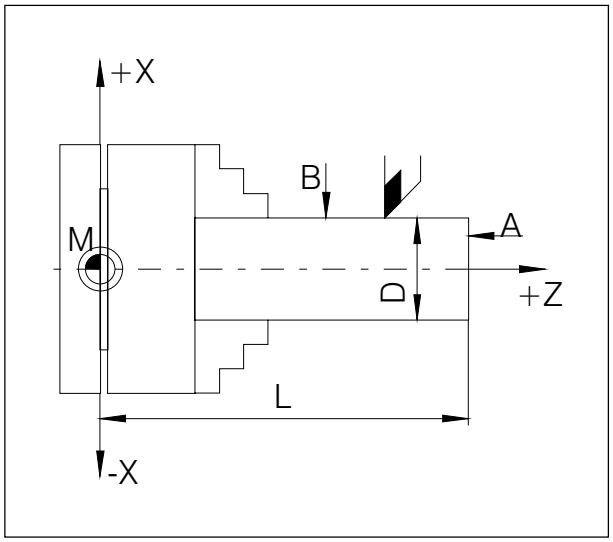
X correction

- Press the key X A and the softkey INP C.
- X value is taken over into the tool data memory.

Z correction

- Press the key Z B and the softkey INP C.
- Z value is taken over into the tool data memory.

Tool Data Measuring with Scratching



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

- Clamp a worpiece with measured diameter and length
- 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. X A 47.
- Press the softkey MEASUR.
- The X value will be taken over into the tool data register.

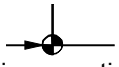
Z correction

- 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.

C: Operating Sequences

Survey Operating Modes

REF 

In this operating mode the reference point will be approached.

With reaching the reference point the actual position display is set to the value of the reference point coordinates.
By that the control acknowledges the position of the slides in the working area.

With the following situations the reference point has to be approached:

- After switching on the machine
- After mains interruption
- After alarm "Approach reference point" or "Ref. point not reached"
- After collisions or if the slides stucked because of overload

MEM

For working off a part program the control calls up block after block and interprets them.

The interpretation considers all correction which are called up by the program.

The so-handled blocks will be worked off one by one

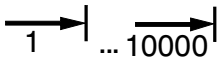
EDIT
In the EDIT mode you can enter part programs and transmit data.

MDI

In the MDI mode you can switch on the spindle and swivel the tool holder.
The control works off the entered block and deletes the intermediate store for new inputs..

JOG

With the KONV keys the slides can be traversed manually.

I1 ... I1000 

In this operation mode the slides can be traversed for the desired increment (1...1000 in µm/10^{-4} inch) by means of the JOG keys **-X** **+ X** **-Z** **+Z**.

The selected increment (1, 10, 100, ...) must be larger than the machine resolution (lowest possible traverse movement), otherwise no movement occurs.

REPOS

Repositioning, approach back to the contour in JOG mode.

Teach In

Making programs in dialogue with the machine in MDA mode.

Approach the Reference point

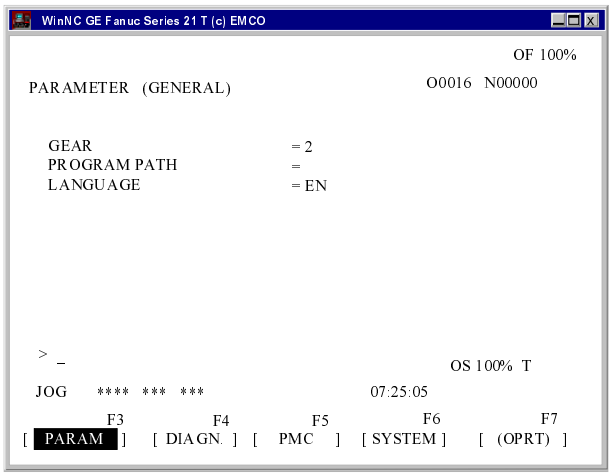
By approaching the reference point the control will be synchronized to the machine.

- Change into REF mode.
- Actuate fist the direction keys -X or +X, then -Z or +Z to approach the reference point in the respective direction.
- With the REF ALL key both axes will be approached automatically (PC keyboard).

Danger of collisions

Mind for obstacles in the working area (clamping devices, clamped work pieces, etc.).

After reaching the reference point its position will be displayed as actual position. Now the machine is synchronized to the control.



Input of the Gear Position

(only with EMCO PC Turn 55)
For that the machine runs the correct spindle speed, the selected gear (belt) position of the machine has to be entered in EMCO WinNC.

- Press the key SYSTEM.
- Press the key PAGE multiple, until the setting page (PARAMETER GENERAL) will be displayed.
- Move the cursor on the input field GEAR and enter the corresponding gear position.
  1 gear position 1 120 - 2000 U/min
  2 gear position 2 280 - 4000 U/min

Setting of Language and Workpiece Directory

- Press the key SYSTEM.
- Press the key PAGE multiple, until the setting page (PARAMETER GENERAL) will be displayed.

Workpiece Directory

In the workpiece directory the CNC programs created by the operator will be stored.

The workpiece directory is a subdirectory of the directory which was determined with installation.

Enter in the input field PROGRAM PATH the name of the workpiece directory with the PC keyboard, max. 8 characters, no drives or pathes. Not existing directories will be created.

Active Language

Selection from installed languages, the selected language will be activates with restart of the software. Enter the language sign in the input field LANGUAGE

- DT for German
- EN for English
- FR for French
- SP for Spanish

Program Input

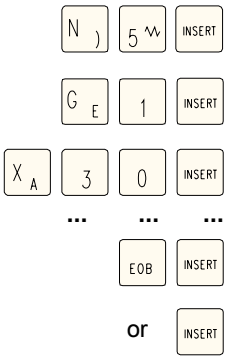
Part programs and subprograms can be entered in the EDIT mode.

Call Up a Program

- Change into EDIT mode
- Press the key PROG
- With the softkey DIR the existing programs will be displayed.
- Enter program number O...
  Its don´t be allowed to use the program numbers from 9500 because there are reserved for internal aims.
- New program: Press the key INSERT
- Existing program: Press the softkey O SRH.

Input of a block

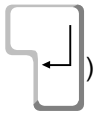
Example:



Block number (not necessary)

1. word

2. word

EOB - End of block (on PC keyboard also )

Note:
With the parameter SEQUENCE NO (PARAMETER MANUELL) you can determine whether block numbering should occur automatically (1 = yes, 0 = no).

Search a Word

Enter the address of the word to be searched (e.g.: X) and press the softkey SRH ↓.

Insert a Word

Move the cursor before the word, that should be before the inserted word, enter the new word (address and value) and press the key INSERT.

Alter a Word

Move the cursor before the word that should be altered, enter the word and press the key ALTER.

Delete a Word

Move the cursor before the word, that should be deleted and press the key DELETE.

Insert a Block

Move the cursor before the EOB sign ";" in that block which should be before the inserted block and enter the block to be inserted.

Delete a Block

Enter block number (if no block number exists: N0) and press the key DELETE

Delete a Program

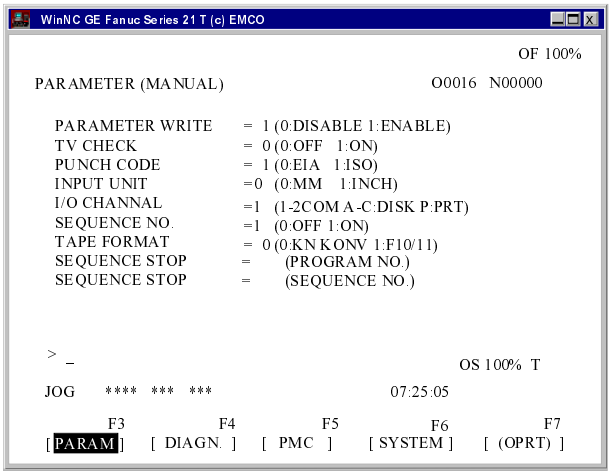
EDIT mode
Enter the program number (e.g.: O22) and press the key DELETE.

Delete All Programs

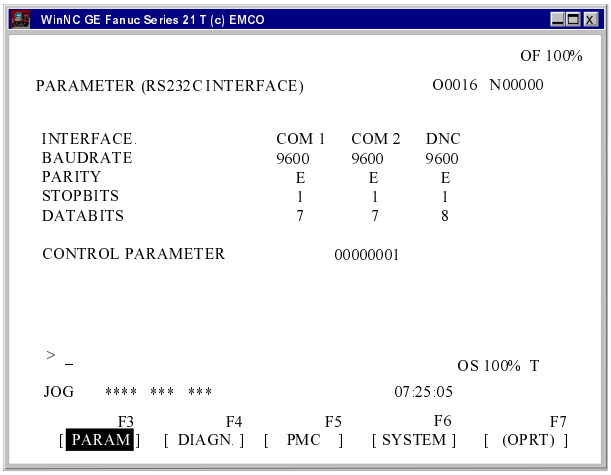
EDIT mode
Enter the program number O 0-9999 and press the key DELETE.

Data Input - Output

- Press the key SYSTEM.
  The screen shows (PARAMETER MANUAL).
- Below "I/O Channal" you can enter a serial interface (1 or 2) or a drive (A, B or C).
  1 serial interface COM1
  2 serial interface COM2
  A disk drive A
  B disk drive B
  C hard disk drive C, workpiece directory (Established with installation or in (PARAMETER GENERAL)), or any path (adjustment with Win Config).
  P Printer.



Selection of the input/output interface



Adjusting the serial interface

NOTE

When you use an interface expansion card (e.g. for COM 3 and COM 4), take care that for every interface a separate interrupt is used (e.g.: COM1 - IRQ4, COM2 - IRQ3, COM3 - IRQ11, COM4 - IRQ10).

Adjusting the Serial Interface

- Press the key SYSTEM.
- Press the key PAGE ▼ or PAGE ▲, until (PARAMETER RS232C INTERFACE) is displayed.

Settings:
Baudrate 110, 150, 300, 600, 1200, 2400, 4800, 9600
Parity E, O, N
Stopbits 1, 2
Datenbits 7, 8
Data transmission from / to original control in ISO-Code only.
Standard adjustment:
7 Datenbits, Parity even (=E), 1 Stopbit, 9600 boad

Control parameter:
Bit 0: 1...Transmission will be cancelled with ETX (End of Text) code
0...Transmission will be cancelled with RESET
Bit 7: 1...Overwrite part program without message
0...Message, if a program already exists
ETX code: % (25H)

Program Output

- EDIT mode
- Enter the receiver in (PARAMETER MANUAL) below "I/O".
- Press the key PROG.
- Press the softkey OPRT.
- Press the key F11.
- Press the soktkey PUNCH
- Enter the program number to be send (e.g. O22).
- When you enter e.g. O5-15, all programs with the numbers 5 to inclusive 15 will be printed.
  When you enter the program numbers 0-9999 all programs will be put out.
- Press softkey EXEC

Program Input

- EDIT mode
- Enter the receiver in (PARAMETER MANUAL) below "I/O".
- Press the key PROG.
- Press the softkey OPRT
- Press key F11.
- Press softkey READ
- With input from disk or hard disk you have to enter a program number.
  Enter the program number when you want to read in one program (e.g.: O22).
  When you enter e.g. O5-15, all programs with the numbers 5 to inclusive 15 will be transmitted.
  When you enter O-9999 as program number, all programs will be transmitted.
- Press the softkey EXEC.

Tool Offset Output

- EDIT mode
- Enter the receiver in (PARAMETER MANUAL) below "I/O".
- Press the key OFFSET SETTING
- Press the softkey OPRT.
- Press the key F11
- Pres the softkey PUNCH
- Press the softkey EXEC

Tool Offset Input

- EDIT mode
- Enter the receiver in (PARAMETER MANUAL) below "I/O".
- Press the key OFFSET SETTING.
- Press the softkey OPRT.
- Press the key F11
- Press the softkey READ
- Press the softkey EXEC

Print Programs

- The printer (standard printer in Windows) must be connected and must be in ON LINE status.
- EDIT mode
- Enter P (Printer) as receiver in (PARAMETER MANUAL) below "I/O".
- Press the key PROG.
- Press the softkey OPRT.
- Press the key F11.
- Press the softkey PUNCH.
- Enter the program to be printed (e.g. O22) when you want to print one program.
  When you enter e.g. O5-15, all programs with the numbers 5 to inclusive 15 will be printed.
  When you enter the program number O-9999 all programs will be printed.
- Press the softkey EXEC.

Program Run

Start of a Part Program

Before starting a program the control and the machine must be ready for running the program.

- Select the EDIT mode.
- Press the key PROG
- Enter the desired part program number (e.g.: O79).
- Press the key ↑.
- Change to MEM mode.
- Press the key ◇.

Displays while Program Run

While program run different values can be shown.

- Press the softkey PRGRM (basic status). While program run the actual program block will be displayed.
- Press the softkey CHECK. While program run the actual program block, the actual positions, active G and M commands and speed, feed and tool will be displayed.
- Press the softkey CURRNT. While the program run the aktiv G commands will be displayed.
- Press the key POS. The positions will be shown enlarged at the screen.

Block Search

With this function you can start a program at any block.
While block search the same calculations will be proceeded as with normal program run but the slides do not move.

- EDIT mode
- Select the program to be machined.
- Move the cursor with the keys ↑ and ↑ on that block, with which machining should start.
- Change to MEM mode.
- Start the program with the key ◇.

Program Influence

DRY RUN
DRY RUN is used for testing programs. The main spindle will not be switched on and all movements occur in rapid feed.
If DRY RUN is active, DRY will be displayed in the first line on the screen.

SKIP
With SKIP all program blocks which are marked with a "/" (e.g.: /N0120 G00 X...) will not be proceeded and the program will be continued with the next block without a "/" sign.
If SKIP is active, SKP will be displayed in the first line on the screen.

Program interruption

Single block mode
After every program block the program will be stopped.
Continue the program with the key ◇.
If the program block is aktivated SBL will be displayed in the first line on the screen.

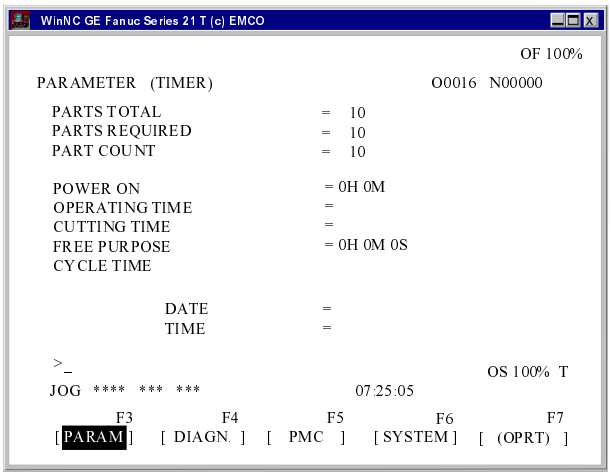
M00
After M00 (programmed stop) in the program the program will be stopped. Continue the program with the key ◇.

M01
If OPT. STOP is active, (display OPT in the first line of the screen) M01 works like M00, otherwise M01 has no effect.

Display of the Software Versions

- Press the key SYSTEM
- Select softkey SYSTEM

The software version of the control system and the eventually connected axcontroller, PLC, working status,... will be displayed.



Display of part counter and piece time

Part Counter and Piece Time

Below the position display the part counter and the piece time are displayed.

The part counter shows the number of program runs. Each M30 (or M02) increases the part counter for 1.

RUN TIME shows the complete running time of all program runs.

CYCLE TIME shows the running time of the actual program and will be reset to 0 with every program start.

Part Counter Reset

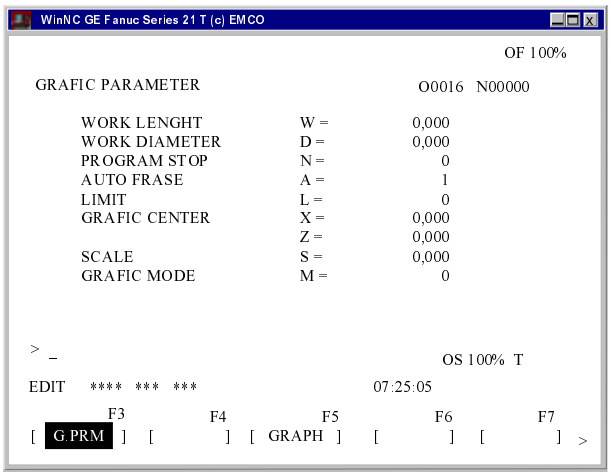
- Press softkey POS.
- Press softkey OPRT
- Select between PTSPRE (reset part counter to 0) or RUNPRE (reset run time to 0).

Preset of the Part Counter

The part counter can be preset in (PARAMETER TIMER).
Therefore move the curor on the desired value and enter the new value.

PARTS TOTAL:
Each M30 increases this number by 1. Every program run of every program will be counted (= number of all program runs).

PARTS REQUIRED:
Preset part number. When this number is reached the program will be stopped and message 7043 PIECE COUNT REACHED will be displayed.
After that the program can be started only after resetting the part counter or increasing the preset part number.



Input pattern for the graphic simulation

Graphic Simulation

NC-programs can be simulated graphically.

Press the key GRAPH.

The screen shows the input pattern for graphic simulation.
The simulation area is a rectangular window, which is determined by the right upper and left lower edge.

Inputs:

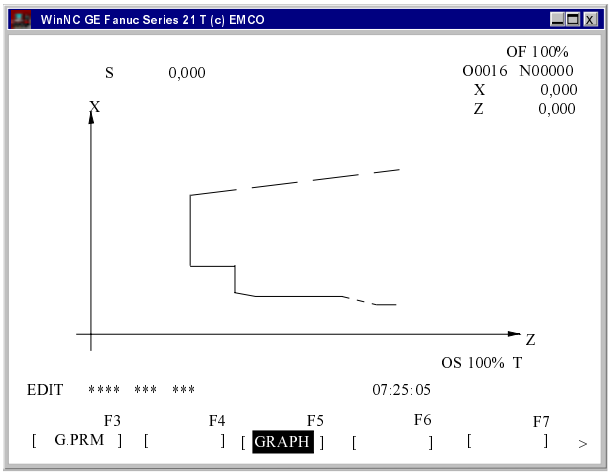
WORK LENGHT	W
WORK DIAMETER	D

Enter here the right upper (X,Z) edge of the simulation area.

GRAPHIK-CENTER	X, Z

Enter here the left lower (X,Z) edge for the simulation area.

After pressing the key ► the softkey 3DVIEW will be shown..
Win 3D View is an option and not included in the basic version of the software.



Simulation window

With the softkey GRAPH you will get into the simulation window.

With the key G. PRM you will go back to the input pattern for graphic simulation.

With the softkey NC-START the graphic simulation starts.

With the softkey NC-STOP the graphic simulation stops.

With the softkey RESET the graphic simulation will be aborted.

Movements in rapid traverse will be displayed as dashed lines, movements in working traverse will be displayed as full lines.

D: Programming

Program Structure

CNC programming for machine tools according to DIN 66025 is used.
The CNC program is a sequence of program blocks which are stored in the control.
With machining of workpieces these blocks will be read and checked by the computer in the programmed order.
The corresponding control signals will be sent to the machine.

The CNC program consists of:
- program number
- CNC blocks
- words
- addresses
- number combinations (for axis addresses partly with sign)

Used Addresses

O ..... program number 1 to 9499 for part programs and subroutines
N..... block number 1 to 9999
G ..... path function
X, Z.... position data in absolute values (X also dwell time)
U, W .. position data in incremental values (U also dwell time)
R..... radius, taper value, cycle parameter
C..... chamfer
I, K..... circle parameter
F ..... feed rate, thread pitch
S ..... spindle speed, cutting speed
T ..... tool call (tool correction)
M ..... miscellaneous function
P ..... dwell, subprogram call, cycle parameter
Q ..... cycle parameter
; ..... block end

Survey of G Commands for Command Definition A, B, C

With software installation you can select the command definition A, B or C.
The difference between the versions is only the code for a command, but not the function of the command (see table).
In this manual only the command definition C is described (european standard).
If you use the command definition A or B, note the codes in the command description.

Gr.		Command			Function
		A	B	C	
0	+	G04	G04	G04	Dwell
	+	G07.1	G07.1	G07.1	Cylindrical Interpolaton
	+	G10	G10	G10	Data setting
	+	G11	G11	G11	Data setting Off
	+	G28	G28	G28	Return to reference point
	+	G70	G70	G72	Finishing cycle
	+	G71	G71	G73	Stock removal in turning
	+	G72	G72	G74	Stock removal in facing
	+	G73	G73	G75	Pattern repeating
	+	G74	G74	G76	Deep hold drilling, cut-in cycle in Z
	+	G75	G75	G77	Cut in cycle in X
	+	G76	G76	G78	Multiple threading cycle
	+	G50	G92	G92	Coord.syst.set., Spindle speed limit
1	•	G00	G00	G00	Positioning (rapid traverse)
		G01	G01	G01	Linear interpolation clockwise
		G02	G02	G02	Circular interpolation clockwise
		G03	G03	G03	Circular interp. counterclockwise
		G90	G77	G20	Longitudinal turning cycle
		G92	G78	G21	Thread cutting cycle
		G94	G79	G24	Face turning cycle
		G32	G33	G33	Thread cutting
2		G96	G96	G96	Constant cutting speed
	•	G97	G97	G97	Direct spindle speed programming
3	•	-	G90	G90	Absolute programming
		-	G91	G91	Inkremental programming
5		G98	G94	G94	Feed per minute
	•	G99	G95	G95	Feed per revolution
6		G20	G20	G70	Inch data input
		G21	G21	G71	Metric data input
7	•	G40	G40	G40	Cancel cutter radius compensation
		G41	G41	G41	Cutter radius compensation left
		G42	G42	G42	Cutter compensation right
10	•	G80	G80	G80	Cancel cycles
		G83	G83	G83	Drilling cycle
		G84	G84	G84	Tapping cycle
		G85	G85	G85	Reaming cycle
11	•	-	G98	G98	Return to initial plane
		-	G99	G99	Return to withdrawal plane
16		G17	G17	G17	Plane selection XY
		G18	G18	G18	Plane selection ZX
		G19	G19	G19	Plane selection YZ
21		G12.1	G12.1	G12.1	Polar Coordinate Interpolation ON
		G13.1	G13.1	G13.1	Polar Coordinate Interpolation OFF

• ..... Initial status
+ ..... Blockwise effective

With version A the commands of group 3 and 11 does not exist. Incremental programming occurs with version A always with U and W, Retraction movements occur always to the initial plane.

Survey of G Commands for Command Definition C

G00• ..... Positioning (rapid traverse)
G01 ..... Linear interpolation (feed)
G02 ..... Circular interpolation clockwise
G03 ..... Circular interpolation counterclockwise
G04+ ..... Dwell
G7.1 ..... Cylindrical Interpolation
G10 ..... Data setting
G11 ..... Data setting Off
G12.1 ..... Polar Coordinate Interpolation ON
G13.1 ..... Polar Coordinate Interpolation OFF
G17 ..... Plane selection XY
G18 ..... Plane selection ZX
G19 ..... Plane selection YZ
G20 ..... Longitudinal turning cycle
G21 ..... Thread cutting cycle
G24 ..... Face turning cycle
G28+ ..... Return to reference point
G33 ..... Thread cutting
G40 ..... Cancel cutter radius compensation
G41 ..... Cutter radius compensation left
G42 ..... Cutter radius compensation right
G70 ..... Inch data input
G71 ..... Metric data input
G72+ ..... Finishing cycle
G73+ ..... Stock removal in turning
G74+ ..... Stock removal in facing
G75+ ..... Pattern repeating
G76+ ..... Deep hole drilling, cut-in cycle in Z
G77+ ..... Cut-in cycle in X
G78+ ..... Multiple threading cycle
G80 ..... Cancel cycles (G83 up to G85)
G83 ..... Drilling cycle
G84 ..... Tapping cycle
G85 ..... Reaming cycle
G90• ..... Absolute programming
G91 ..... Incremental programming
G92+ ..... Coordinate system setting, spindle speed limit
G94 ..... Feed per minute
G95• ..... Feed per revolution
G96 ..... Constant cutting speed
G97• ..... Direct spindle speed programming
G98• ..... Return to initial plane
G99 ..... Return to withdrawal plane

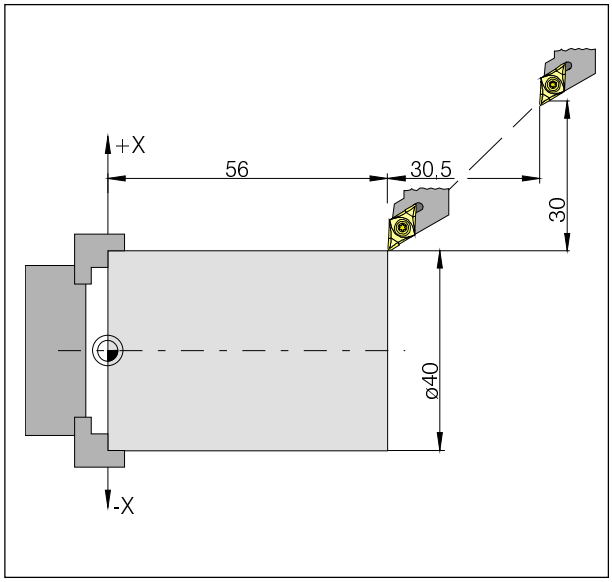
• ..... Initial status
+ ..... Blockwise effective

M- Commands

COMMAND	MEANING
M0	Programmed stop
M1	Progammed stop, conditional
M2	Program end
M3	Spindle ON clockwise
M4	Spindle ON counterclockwise
M5	Spindle OFF
M8	Coolant ON
M9	Coolant OFF
M13	Driven tools ON clockwise
M14	Driven tools ON counterclockwise
M15	DRIVEN TOOLS OFF
M20	Quill FORWARD
M21	Quill BACKWARD
M23	Collecting tray BACKWARD
M24	Collecting tray FORWARD
M25	Clamping device OPEN
M26	Clamping device CLOSE
M30	Main program end
M32	Program end for loading operation
M52	Round axis operation (C-axis ON)
M53	Spindle operation (C-axis OFF)
M57	Spindle oscillate ON
M58	Spindle oscillate OFF
M67	Bar feed / loading magazine feed ON
M68	Bar feed / loading magazine feed OFF
M69	Bar change
M71	Blow-out ON
M72	Blow-out OFF
M90	Manual clamping chuck
M91	Tensile clamping device
M92	Pressure clamping device
M93	Final position monitoring OFF
M94	Bar feed / loading magazine activate
M95	Bar feed / loading magazine deactivate
M98	Subroutine call
M99	Subroutine end, jump command

Description of G Commands

G00 Positioning (Rapid Traverse)



Absolute and incremental measures for G00

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

The slides are traversed at maximum speed to the programmed target point.
Incremental and absolute commands can be used at the same time.

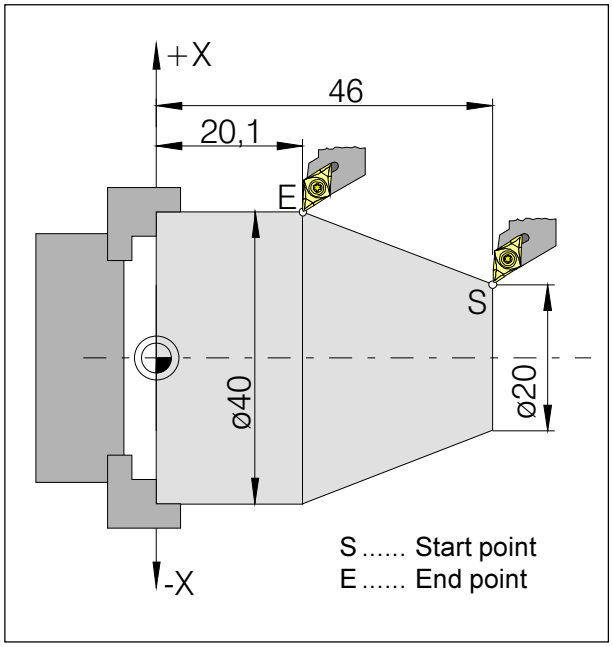
Note
- A programmed feed F will be suppressed while G00.
- The maximum feed is defined by the producer of the machine.
- The feed override switch is limited to 100%.

Example

absolute G90
N50 G00 X40 Z56

incremental G91
N50 G00 U-30 W-30.5

G01 Linear Interpolation (Feed)



Absolute and incremental measures for G00

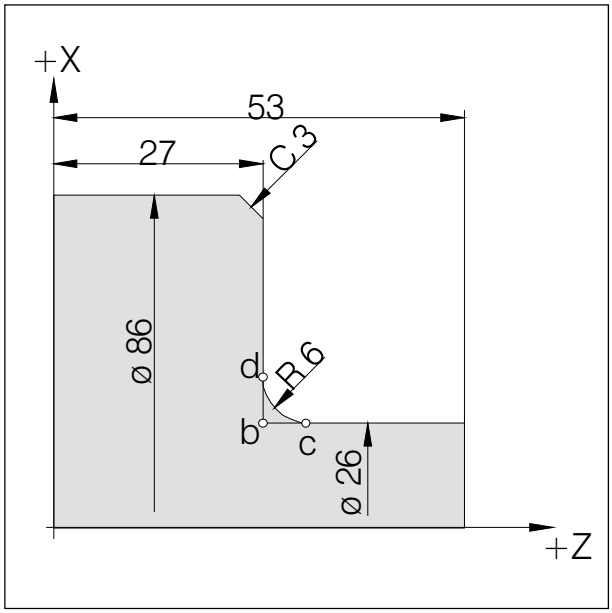
Format
N... G01 X(U)... Z(W).... F....

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



Insertion of chamfers and radii

Insertion of Chamfers and Radii

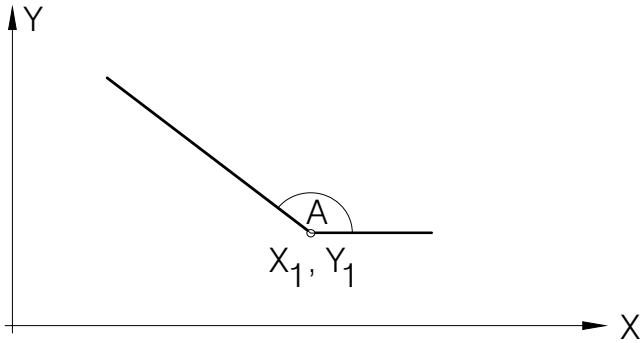
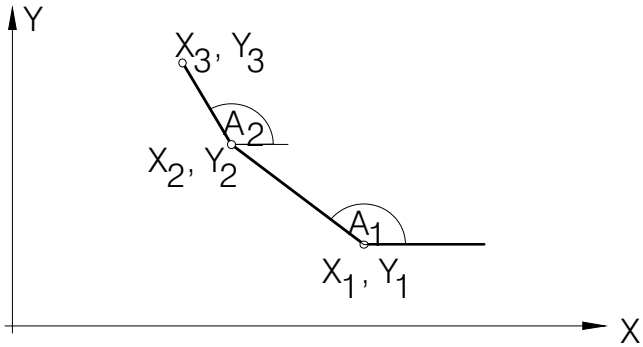
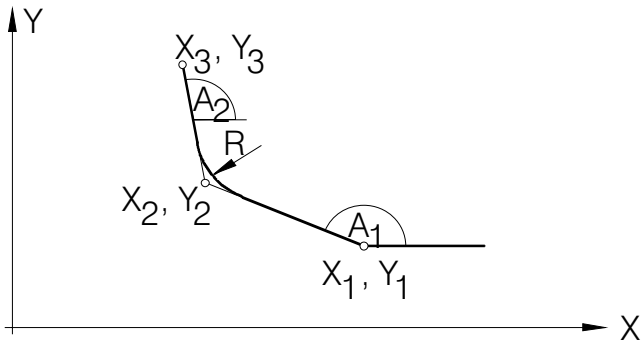
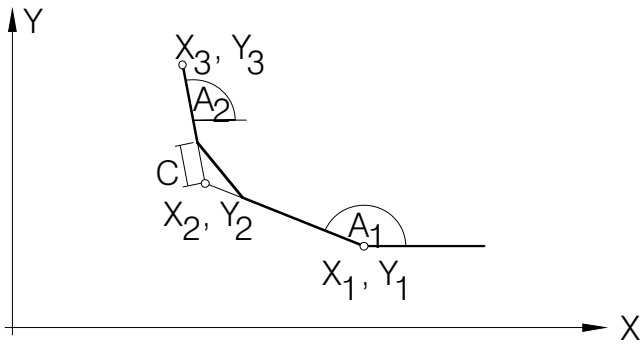
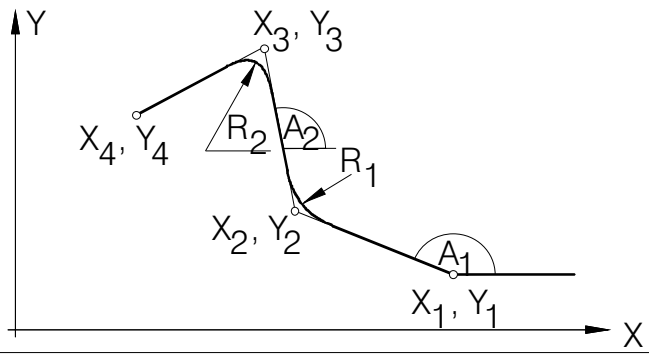
Example

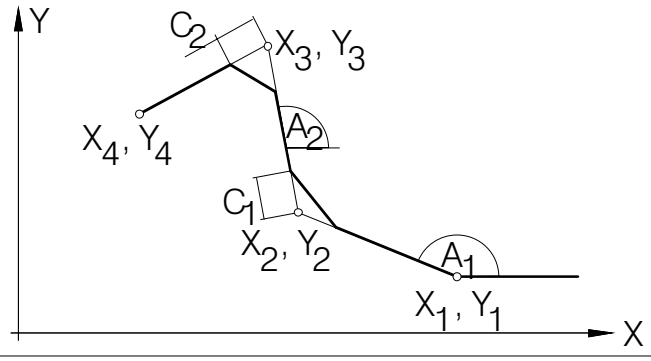
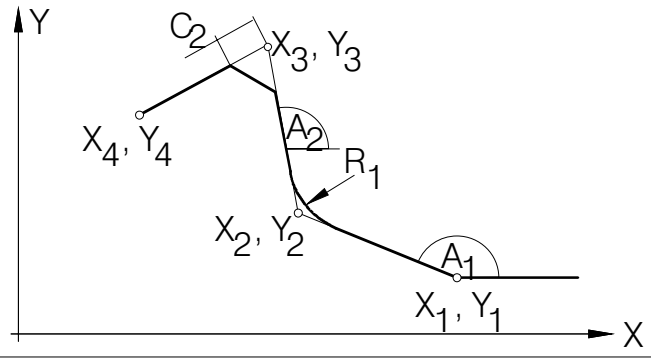
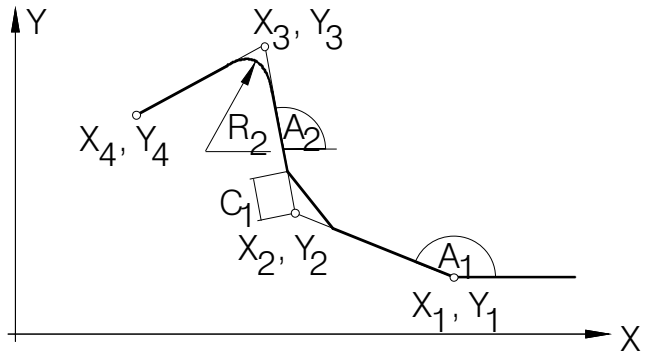
```
....
N 95    G 01    X 26    Z 53
N 100   G 01    X 26    Z 27    R 6
N 105   G 01    X 86    Z 27    C 3
N 110   G 01    X 86    Z 0
....
```

Notes

- Chamfers and radii 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.

Direct Drawing Input

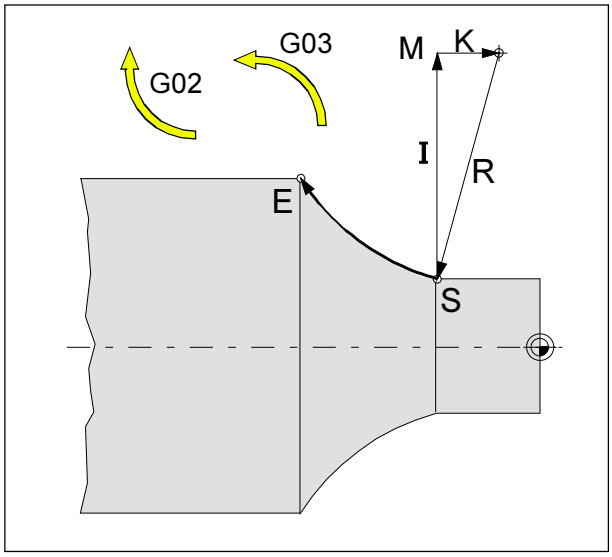
	commands	tool movements
1	***X_2... (Z_2...) A...*** ***Note:*** ***Bright printed commands are only used with the option luxery programing.***	
2	***A_1...*** ***X_3... Z_3... A_2...***	
3	X_2... Z_2... R... X_3... Z_3... oder ***A_1... R...*** ***X_3... Z_3... A_2...***	
4	X_2... Z_2... C... X_3... Z_3... oder ***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... oder ***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...	

Missing intersection point coordinates need not to be calculated.
In programs angles (A), chamfers (C) and radii (R) can be programmed directly.

Note
The following G commands must not be used for the blocks with chamfer or radius. They must not be used between the blocks with chamfer or radius, which define the succession numbers.
- G-Codes (except G04) in group 00
- G02, G03, G20, G21 and G24 in group 01

The input of angels (A) are only possible with the option luxery programing.



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.
- Input of R with a positive sign effects an arc <180°, a negative sign effects an arc >180°.

G04 Dwell

Format

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

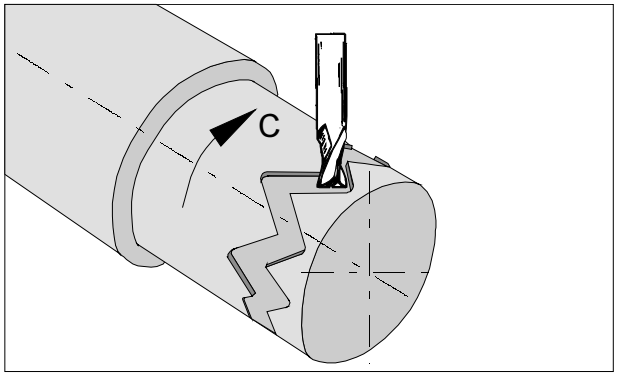
The tool movement will be stopped at the last reached position for a dwell defined by X,U or P.

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 (dwell time= 2.5sec)
N95 G04 P1000 (dwell time = 1 sec = 1000 msec)



G7.1 Cylindrical Interpolation

Format:
N... G7.1 C...
N... G7.1 C0

G7.1 C...	Starts the cylinder interpolation. The C- value describes the radius of the the blank part.
G7.1 C0	End of cylinder interpolation

The tool tip position 0 must be programmed for all tools that will be used for the cylindrical interpolation.

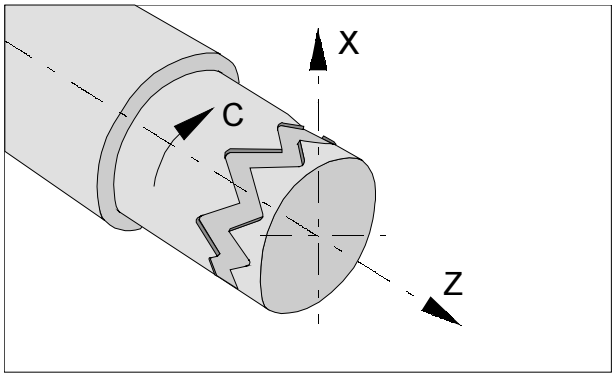
This function enables the development of a cylinder surface in programming.
In this way e.g. programs for cyclindrical cam machining on lathes can be created.

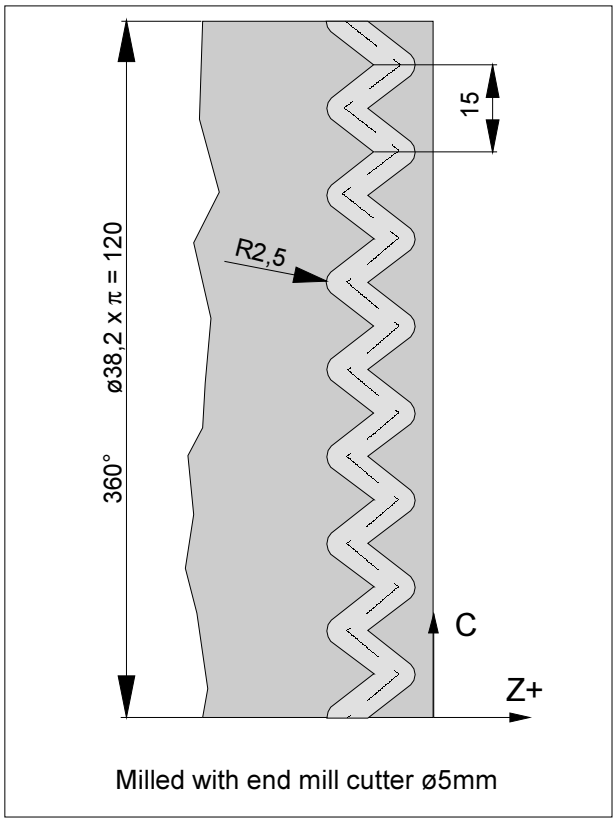
The traverse amount of the rotary axis C programmed by indication of the angle is converted in the control into the distance of a fictitious linear axis along the external surface of the cylinder.
Thus, it is possible that linear and circular interpolations on this area can be carried out with another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

With G19 the level is determined in which the rotary axis C is preset in parallel to the Y-axis.

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.





Milled with end mill cutter ø5mm

Example - Cylindrical Interpolation

X axis with diametrical programming and C axis with angular programming.

```
O0002 (Cylindrical Interpol.)
N15 T0505
N25 M13          Sense of rotation for driven tools
                 (be equivalent to M3)
N30 G97 S2000
N32 M52          Positioning of the spindle
N35 G7.1 C19.1   Start of the interpolation /
                 blank part radius
N37  G94  F200
N40 G0 X45 Z-5
N45 G1 X35 C0 Z-5
N50 G1 Z-15 C22.5
N55 Z-5 C45
N60 Z-15 C67.5
N65 Z-5 C90
N70 Z-15 C112.5
N75 Z-5 C135
N80 Z-15 C157.5
N85 Z-5 C180
N90 Z-15 C202.5
N95 Z-5 C225
N100 Z-15 C247.5
N105 Z-5 C270
N110 Z-15 C292.5
N115 Z-5 C315
N120 Z-15 C337.5
N125 Z-5 C360
N130 X45
N135 G7.1 C0     End of interpolation
N140 M53         Ende des roundaxis
                 operation
N145 G0 X80 Z100 M15
N150 M30
```

G10 Data Setting

The command G10 allows to overwrite control data, programming parameters, writing tool data etc...
G10 is frequently used to program the workpiece zero point.

Example:
Zero point offset

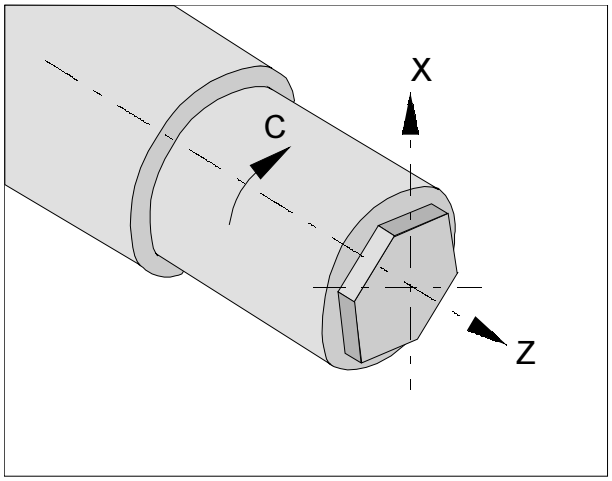
Format
N... G10 P...X...Z...R...Q...;
oder
N... G10 P...U...W...C...Q...;

P: wear offset number

0	Traverse value for the workpiece-coordinate system
1-64	Tool tear correction value The Comand value is the offset number
10000+(1-64)	tool geometry offset number (1-64)

X... Offset number in the X- axis (asolute)
Z... Offset number in the Z- axis (absolute)
U... Offset number in the X- axis (inkremental)
W... Offset number in the Z- axis (inkremental)
R... tool nose radius offset value (absolute)
R... tool nose radius offset value (inkrementel)
Q... imaginary tool nose number

With G10 P0 the workpiece zero point become overwrite.
For this reason the work piece lenght etc can be taken into consideration.



Notes:

- Tool data
  In the offset-setting for the milling tool the following is entered under geometry:
  X -20
  Z (tool length in Z)
  R (miller radius)
  T 0 (type 0)
- Selection G12.1 and deselection G13.1 must be programmed in mode G40. I.e. the miller radius compensation is only programmed after switching on the polar coordinate interpolation.
- With active polar coordinate interpolation no movement can be traversed in rapid motion in G0.
- After switching on G12.1 a sufficently large approach movement in the X-axis must be programmed prior to the first movement with G42/ G41(see program example).
- Also with diameter programming for the linear axis (X-axis) radius programming is used for the rotary axis (C-axis).
- In G12.1- mode the coordinate system must not be altered.
- 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.

G12.1/G13.1 Polar Coordinate Interpolation

Format:
N... G12.1
N... G13.1

G12.1 Starts the polar coordinate interpolation

G13.1 Terminates the polar coordinate interpolation

The polar coordinate interpolation is adequate for machining the end face of a turned part.

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.

With this function the system changes to the G17-plane (X-Y). Any contours can then be milled at the front side with axial milling tools.
The X-axis is continued to be programmed with Ø-values. The fictitious Y-axis is under 90° counterclockwise to the X-axis and is programmed with the address "C“ in the radius.

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) and the level defined by means of G18 is used.

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

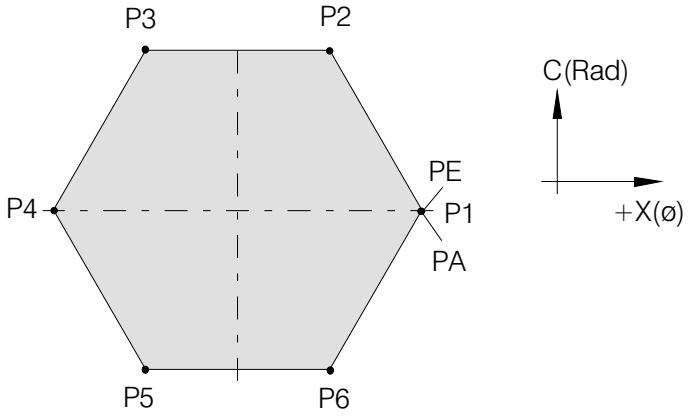
G-Code	Use
G01	Linear interpolation
G02, G03	Circular interpolation
G04	Intermission
G40, G41, G42	Cutter radius compensation (polar coordinate interpolation is applied on the tool path after the tool compensation)
G65, G66, G67	User macro command
G98, G99	Feed per minute, feed per rotation

Example: Milling hexagonal SW17

SW = 17

Point	Formula C	C	X	Formula X
P1	0	0	19,63	(SW / SIN60°)
P2	SW*0,5	8,5	9,81	TAN30° *SW
P3	SW*0,5	8,5	-9,81	TAN30° *SW*-1
P4	0	0	-19,63	`-1* (SW / SIN60°)
P5	SW*0,5	-8,5	-9,81	TAN30° *SW*-1
P6	SW*0,5	-8,5	9,81	TAN30° *SW

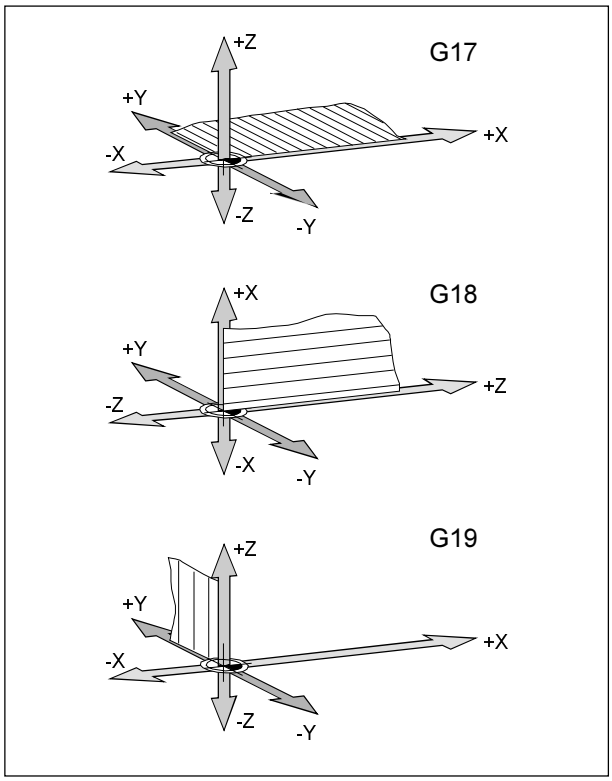
Side length l = 9,81



Example

```
O0104 ( BSP SW 17 MILLING )
N5 G10 P0 Z-72
G40 G90 G95
G92 S4000
T0000 G0 X150 Z150
N10 T1212 ( EXTERNAL TURNING TOOL)
N15 G96 S350 M4 F0.18
N20 G0 X26 Z0
N25 G1 X-0.8
N30 G0 X15 Z1
N35 G42 G1 Z0.05 F0.12
N40 X19.6 C1
N45 Z-5
N50 X24 C1
N51 Z-12
N55 X26
N60 G40 G0 X50 Z80 M5
N65 T0707
N66 M52
N70 M13
N75 G97 S2000
N85 G28 G0 C0
N90 G0 X50 Z2
N95 G12.1
G1 X50 C-10 F0.3
G1 Z-5
N100 G42 G1 X19.63 C-10 F0.2 ( PA )
N105 G1 C0 ( P1 )
N110 X9.81 C8.5 ( P2 )
N115 X-9.81 ( P3 )
N120 X-19.63 C0 ( P4 )
N125 X-9.81 C-8.5 ( P5 )
N130 X9.81 ( P6 )
N135 X19.63 C0 ( P1=P7 )
N140 C5 ( PE )
N145 G40 G1 X45 C0 F0.4
N150 G13.1
N155 G0 X80 Z20 M15
N160 M53

T0101 (  CUT-OFF TOOL)
G97 S2000 M4 F0.08
G0 X27 Z5
Z-10
G1 X22
G0 X26
W1
G1 X24.1
Z-10 A225
X8
G97 S1200
M24
G1 X-1 F0.06
M23
G0 X26 W1
X50 Z50 M5
N165 M30
```



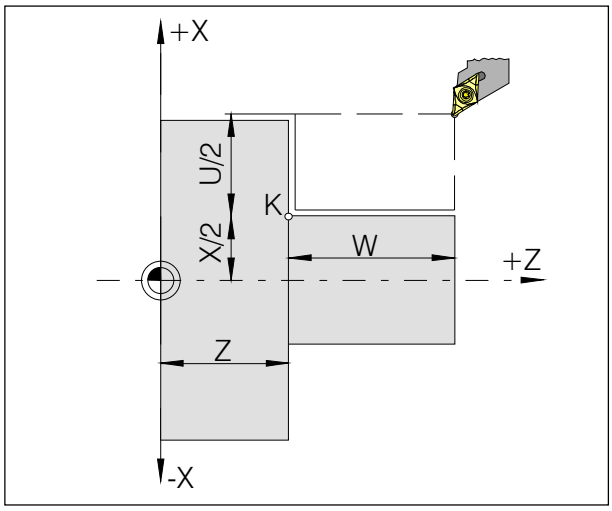
Definition of the main planes

G17-G19 Plane Selection

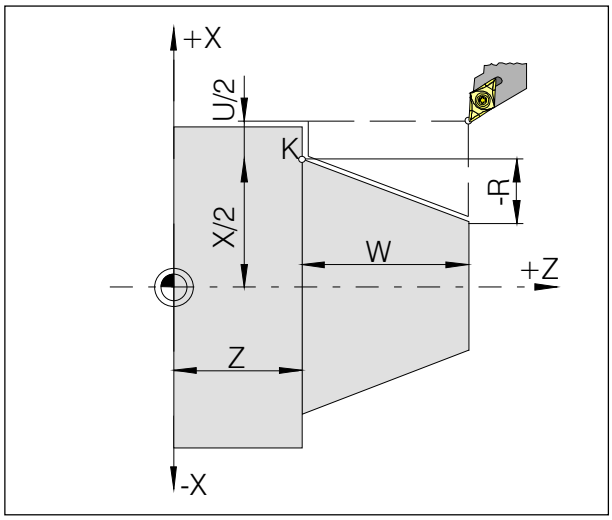
Format
N... G17/G18/G19

With G17 to G19 the plane will be defined, in which circular interpolation and polar coordinate interpolation can be proceeded and in which the cutter radius compensation will be calculated.
In the vertical axis to the active plane the tool length compensation will be proceeded.

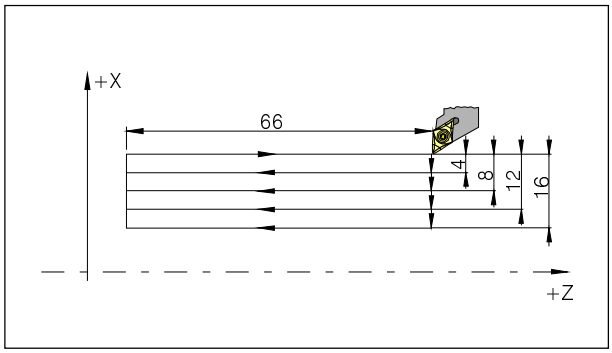
G17 XY-Plane
G18 ZX-Plane
G19 YZ-Plane



Longitudinal turning cycle without taper R



Longitudinal turning cycle with negative taper R



Example: G20 Longitudinal Turning Cycle

G20 Longitudinal Turning Cycle

Format

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

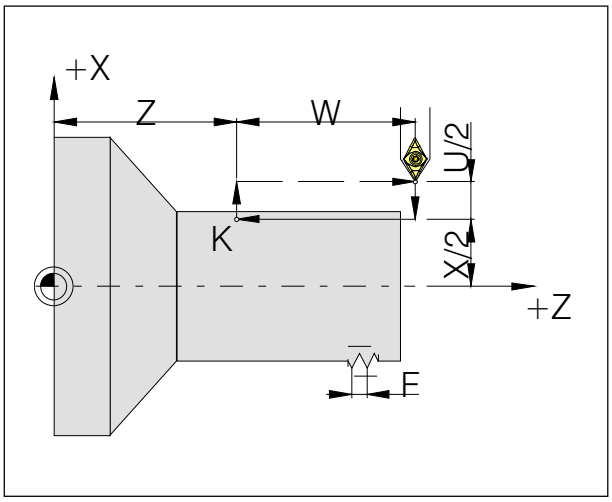
X(U), Z(W)..... Absolute (incremental) coordinates of the contour point K

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

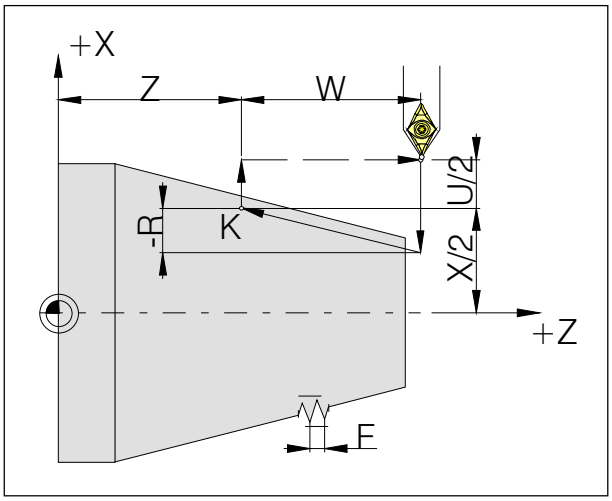
Notes

- 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

G21 Thread Cutting Cycle

Format

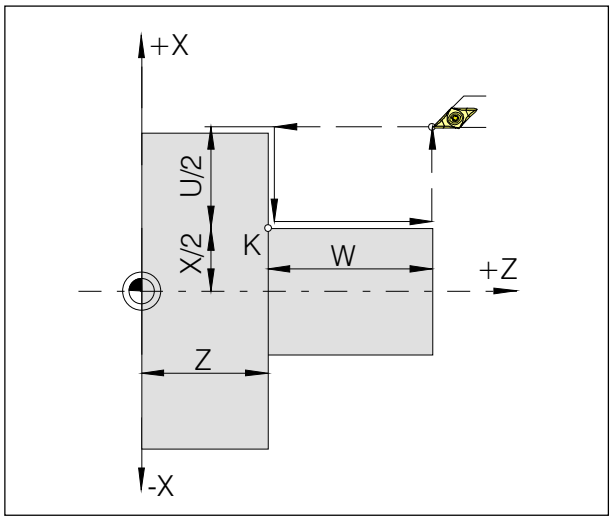
N... G21 X(U)... Z(W)... F... (straight)
or
N... G21 X(U)... Z(W)... R... F... (taper)

F ..... Thread pitch [mm]

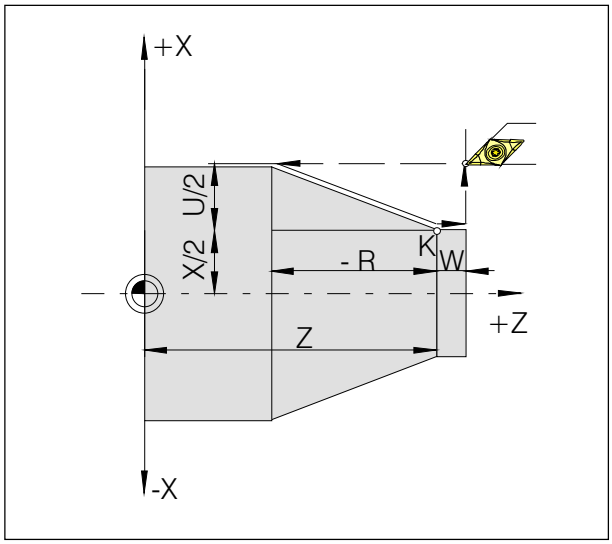
R [mm] ..... Incremental taper dimension in X with direction (+/-)

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.



Face turning cycle without taper R



Face turning cycle with negative taper R

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.
- 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.

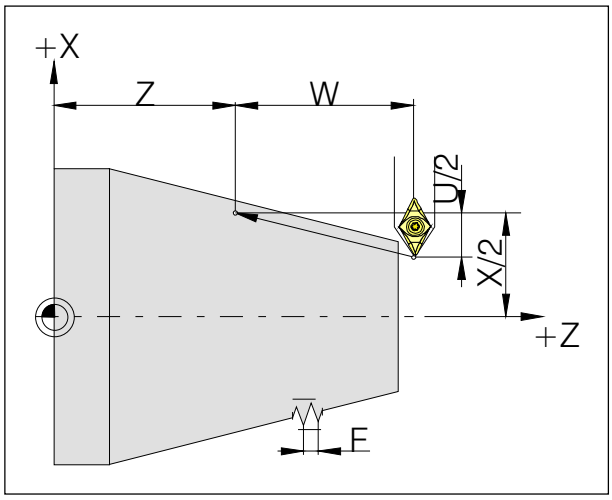
G28 Return to Reference Point

Format

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

X,Z..... Absolute intermediate coordinates
U,W ... Incremental intermediate coordinates

The G28 command is used to approach the reference point via an intermediate position (X(U), Z(W)). First the movement to X(U) and Z(W), then movement to the reference point. Both movements with G00!



Measures for thread cutting

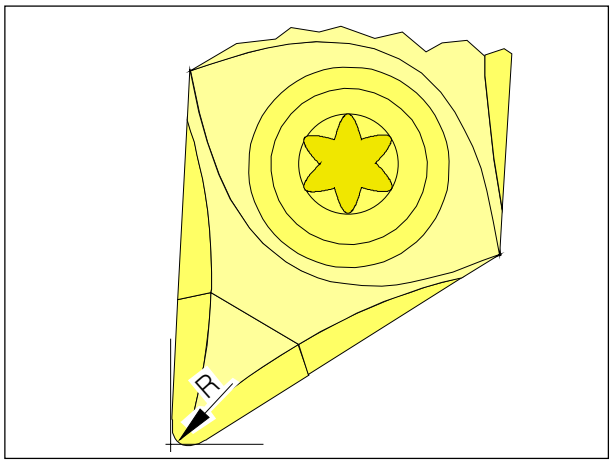
G33 Thread Cutting

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

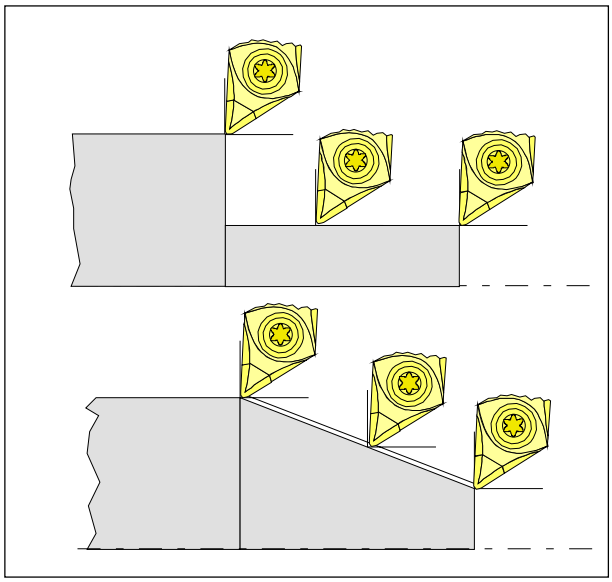
F ..... Thread pitch [mm]

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.

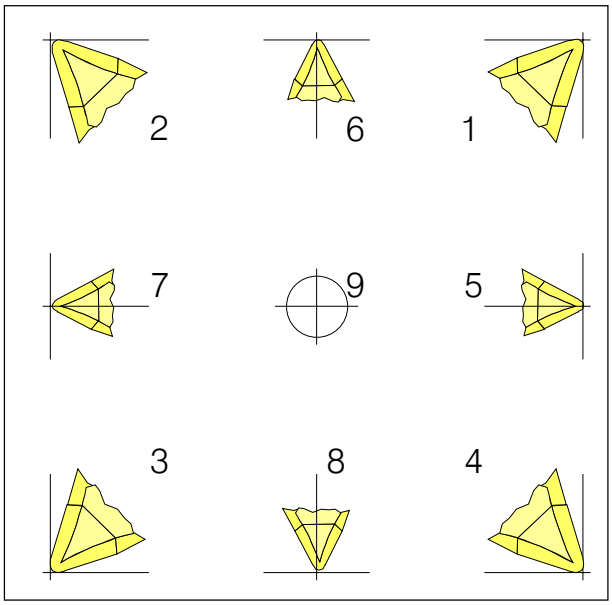
Note
- With taper threading the thread pitch has to be defined with the higher value in X or Z axis.
- Continuous thread cutting is possible (multiple threads)



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.

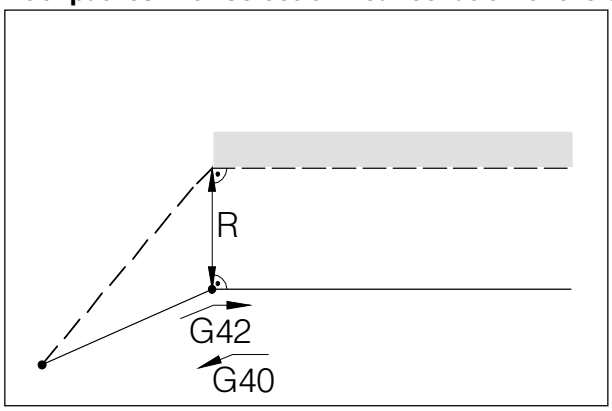
If cutter radius compensation is used, these dimensional errors are automatically calculated and compensated by the control.

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

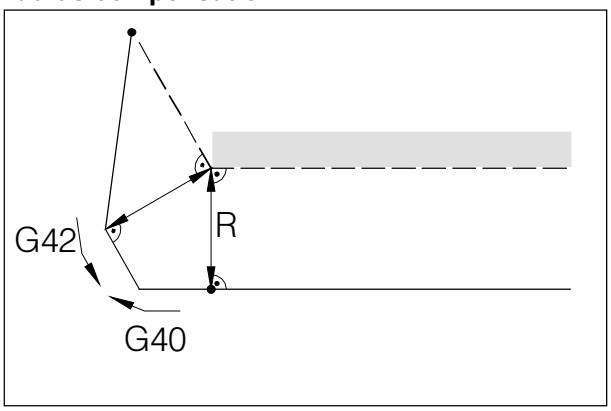
The cutter position is indicated by a number (see draft).

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

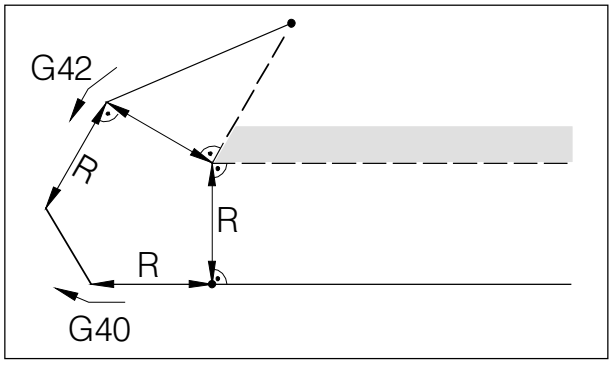
Tool pathes with selection / cancellation of the cutter radius compensation



Frontal approach or leaving of an edge point



Approach or leaving an edge point at side behind



Approach or leaving an edge point behind

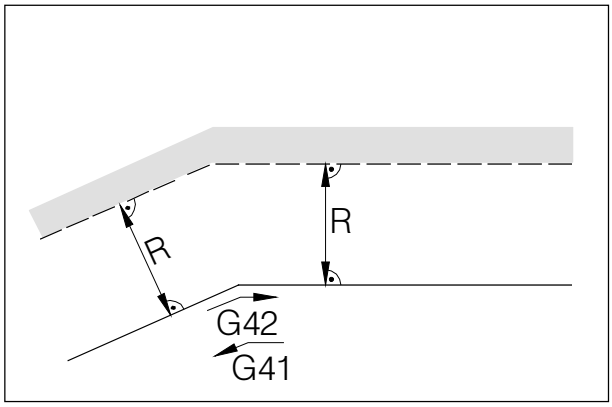
— — — programmed tool path
——— real traversed tool path

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

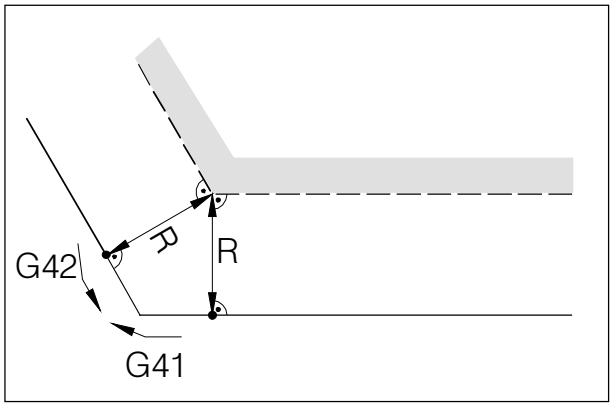
The approaching path to the contour and the leaving path from the contour must be larger than the tool tip radius R, otherwise program interruption with alarm.

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.

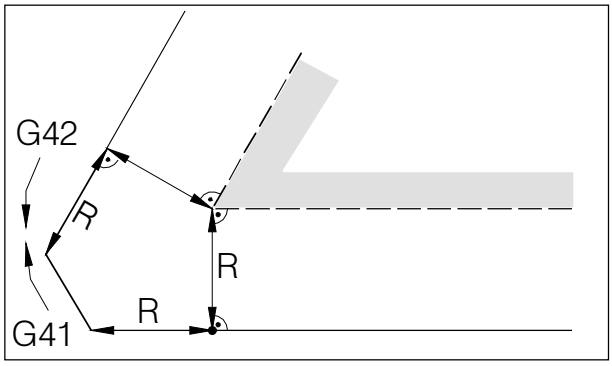
Tool pathes with program run with active cutter radius compensation



Tool path at an inner edge



Tool path at an outer edge > 90°



Tool path at an outer edge < 90°

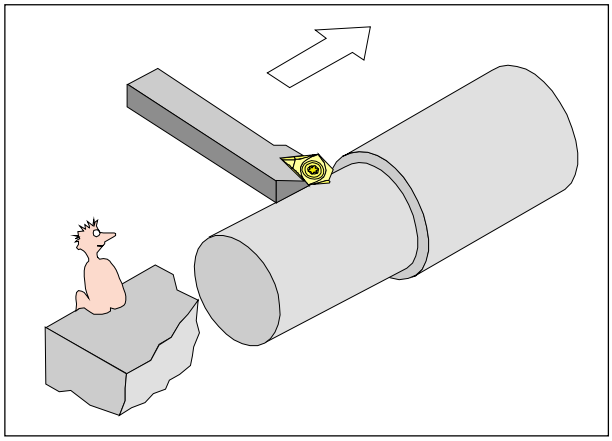
— — — programmed tool path
——— real traversed tool path

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.

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)
G40 can be programmed in the same block as G00 or G01 or in the preceding block.



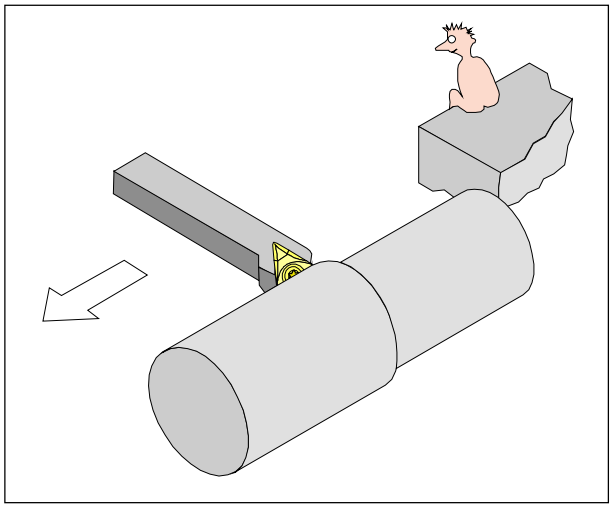
Definition G41 cutter radius compensation left

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 G42 cutter radius compensation right

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!

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)

G71 Metrical Measuring

Format
N5 G71

See G70!

G72 Finishing Cycle

Format

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

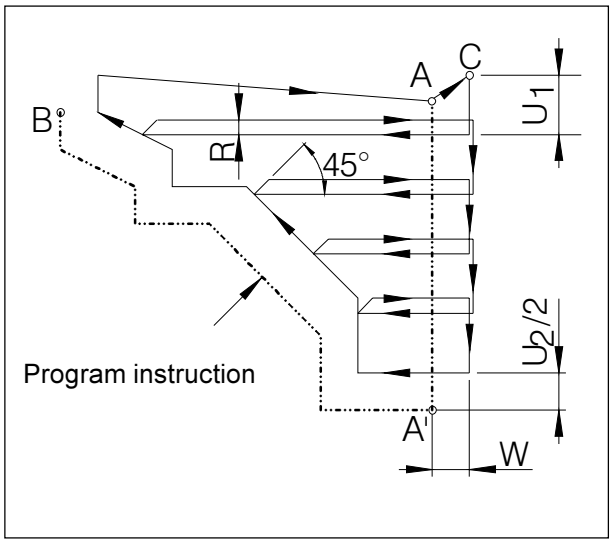
P... ..... Block number of the first block for the program of finishing shape.

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.
- Befor starting the finishing cycle G72 the tool must stood on suitable starting point
- Between P and Q no block advance is permitted.



Turning cycle contour

G73 Contour turning cycle

Format

N... G73 U_1... R...

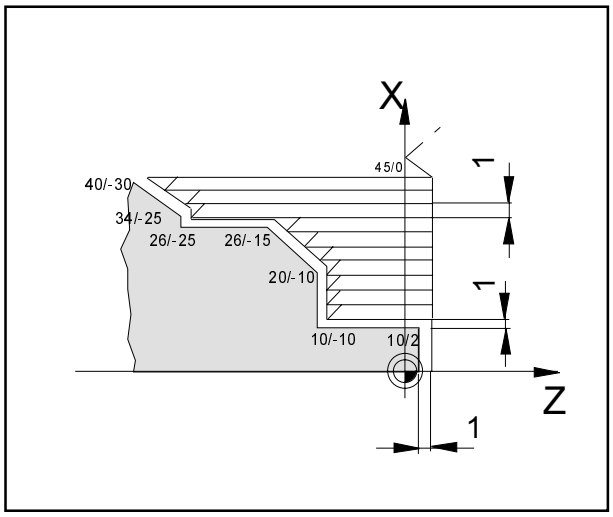
N... G73 P... Q... U_2+/-... W+/-... F... S... T...

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).

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.
- Between P and Q no block advance is permitted.

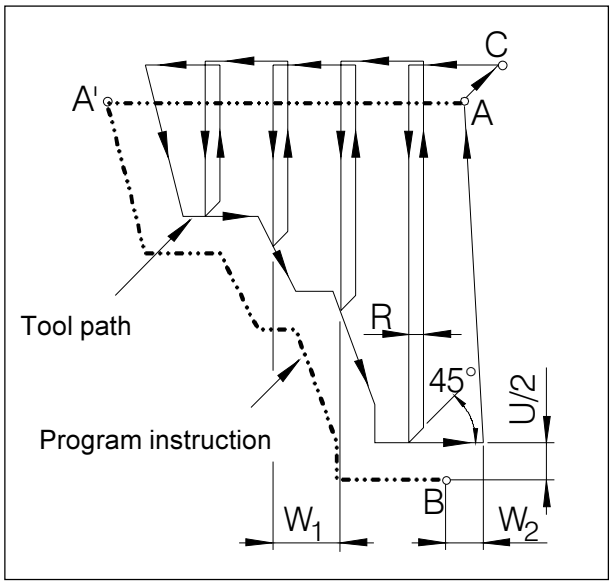


Example contour turning

Example G73 Contour turning cycle:
Machining the contour shown beside.

Program:

```
O2000
N10  G95  G1     F0.5
N11  G0   X45    Z20
N12  T0202
N20  M3   S3000
N30  G00  X45    Z2
          (Start point for cycle)
N40  G73  U2     R2
N50  G73  P60    Q120 U1    W1
          (Contour turning cycle)
N60  G0   X10
N70  G1   Z-10   (from N60 to N120 description
N80  X20         of the contour)
N90  X26  Z-15
N100 Z-25
N110 X34
N120 X40  Z-30
N130 G0   X45    Z20
N140 S3000       F0.6  T0404
          (Select finishing tool)
N150 G0   X45    Z2
          (Start point for finishing)
N160 G72  P60    Q120       (Finishing cycle)
N170 M30
```



Facing cycle contour

G74 Facing cycle

Format

N... G74 W_1... R...

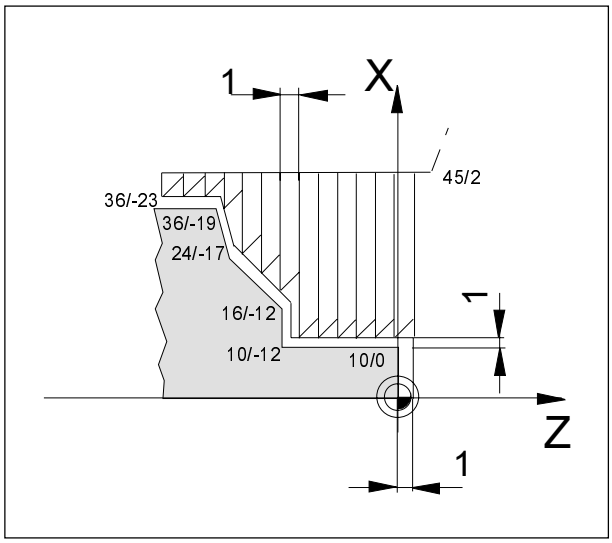
N... G74 P... Q... U+/-... W_2+/-... F... S... T...

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.
- Between P and Q no block advance is permitted.

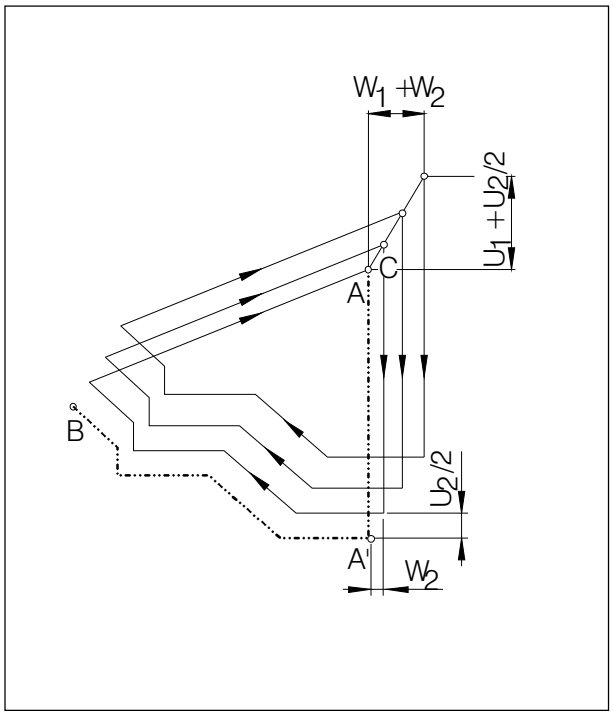


Example facing cycle

Example G74 Facing cycle:

Programm:

```
O2001
N10  G95  G1    F0.5
N11  G0   X45   Z20
N12  T0202
N20  M3   S3000
N30  G00  X45   Z2
          (Start point for facing cycle)
N40  G74  W2    R2
N50  G74  P60   Q120 U1    W1
                (Facing cycle)
N60  G0   Z-23
N70  G01  X36   Z-23  (from N60 to N12
N80  Z-19              description of
N90  X24  Z-17         the contour)
N100 X16  Z-12
N110 X10
N120 Z0
N130 G0   X45   Z20
N140 S3000      F0.6  T0404
                (Select finishing tool)
N150 G0   X45   Z2
                (Start point for finishing)
N160 G72  P60   Q120        (Finishing cycle)
N170 M30
```



Pattern repeating

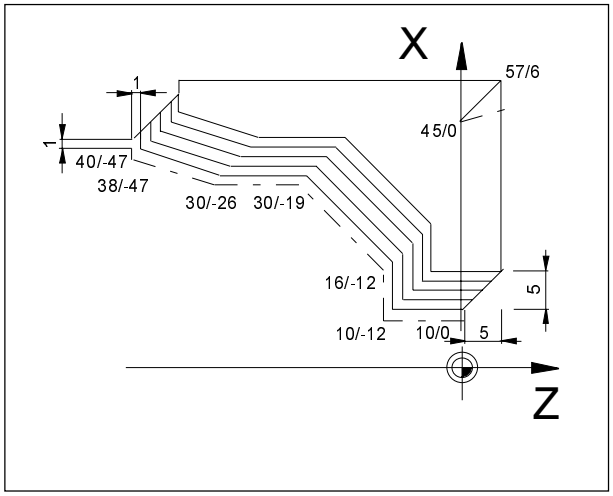
G75 Pattern Repeating

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 (incremental with sign)
	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)
	W_2 [mm]	Distance and direction of finishing offset in Z direction (incremental with sign)
	F, S, T	Feed, speed, tool

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)



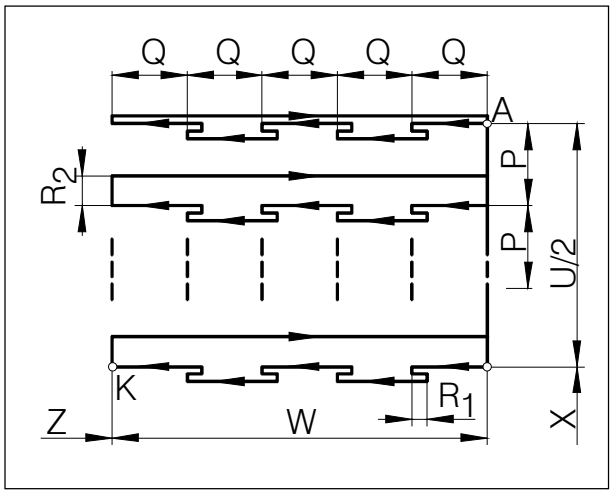
Example Pattern repeating

Example:

O2002

N1	G95	G0	X45	Z0	
N5	M3	S2000	F0.5	T0202	
N10	G75	U5	W5	R5	
N15	G75	P20	Q80	U2	W1
N20	G0	X10			
N30	G1	Z-12	(from N20 to N70		
N40	X16		description of		
N50	X30	Z-19	the contour)		
N60	Z-26				
N70	X38	Z-37			
N80	X40				
N90	M30				

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



Deep hole drilling / Face Cut-in Cycle

G76 Deep hole drilling / Face Cut-in Cycle

Format

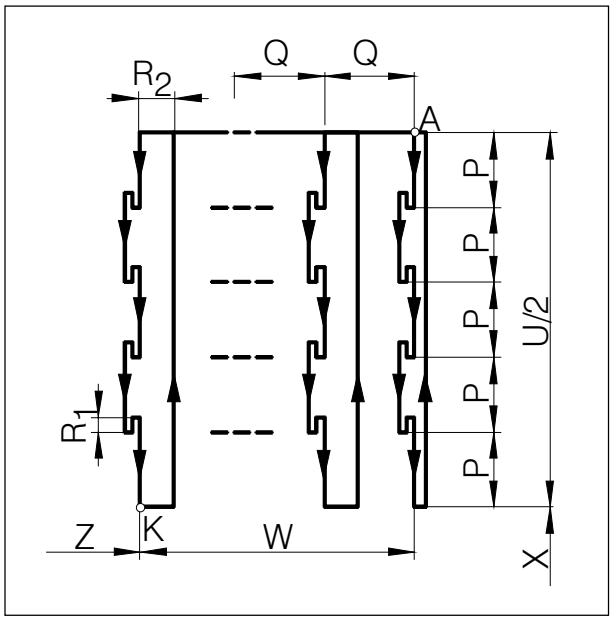
N... G76 R...

N... G76 X(U)... Z(W)... P... Q... R_2... F...

first block	R_1 [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)
	R_2	Undercut at Z end point
	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.
- With the first cut no undercut will be executed at the Z end point.
- The undercut must be a positive value.



Cut-in cycle in X

G77 Cut-in Cycle (X Axis)

Format

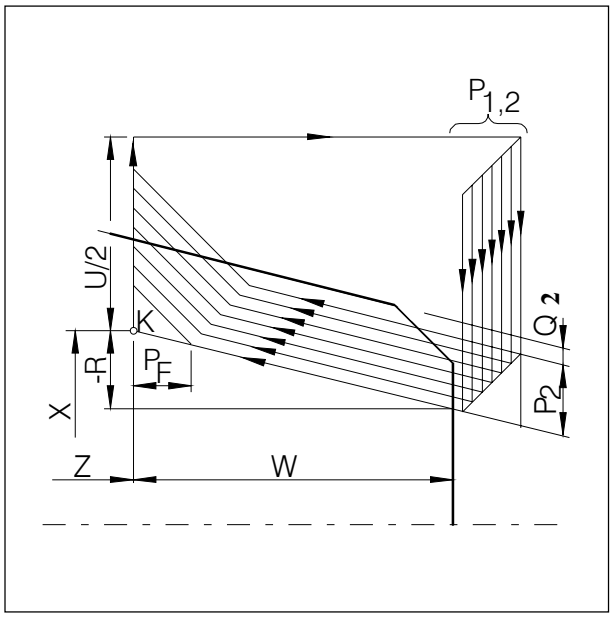
N... G77 R_1...

N... G77 X(U)... Z(W)... P... Q... R_2... F...

first block	R_1 [mm]	Retraction height for chip breaking, in the drawing shown as R_1
sec. block	X(U), Z(W)	Absolute (incremental) coordinates of K
	P [µm]	Cutting depth in X direction (no sign)
	Q [µm]	Incremental infeed in Z direction (no sign)
	R	Undercut at X end point, in the drawing shown as R_2
	F	Feed

Note

- The infeed Q must be smaller than tool width B.
- Tool width will not be taken into consideration with this cycle.
- With the first cut no undercut will be executed.
- The undercut must be a positive value.



Multiple threading cycle

G78 Multiple Threading Cycle

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)

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

PxxxxXX → Defines the flank angle of thread in [°] (allowed: 0,29,30,56,60,80)

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)

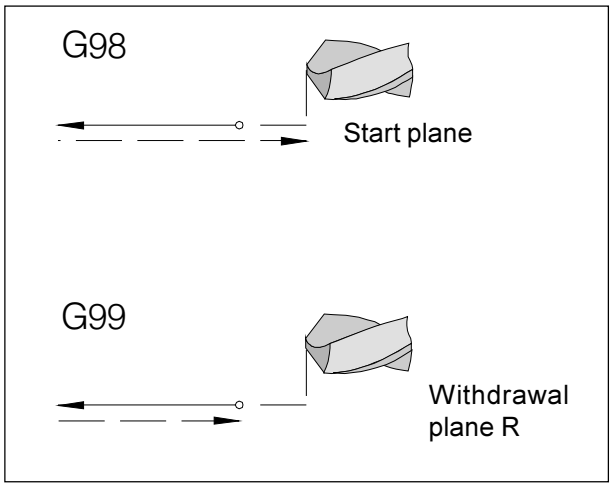
P_2 [µm] Thread depth (always positive), in the drawing shown as P_2

Q_2 [µm] Cutting depth of the first cut (radius value) without sign

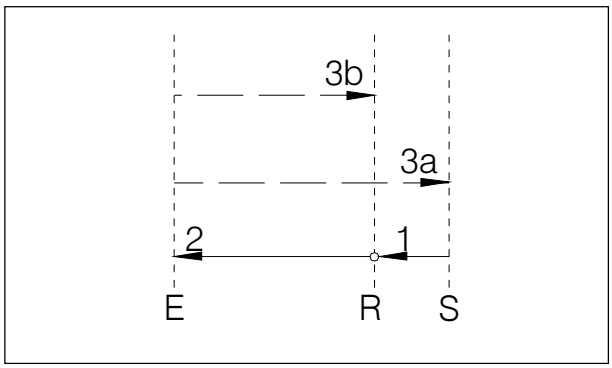
F [mm] Thread pitch

Note

Negative taper parameter R defines the taper as shown in the drawing.



Retraction characteristics G98, G99



Sequence of movements G98, G99

Systematic G98/G99

G98 .... After reaching the drilling depth the tool retracts to the starting plane
G99 .... After reaching the drilling depth the tool retracts to the withdrawal plane - defined by the R parameter

If no G98 or G99 is active, the tool retracts to the starting plane. If G99 (retraction to withdrawal plane) is programmed, The address R must be defined. With G98 R need not to be programmed.

R defines the height of the withdrawal plane related to the last Z position (start position of the drilling cycle). With a negative value for R the withdrawal plane will be below the start position, with a positive value the withdrawal plane will be over the start position

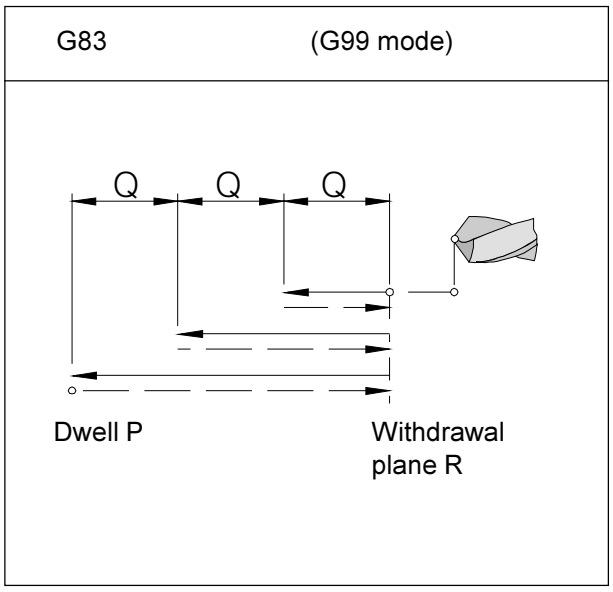
Sequence of movements

1: The tool traverses with rapid speed from the start position (S) to the plane defined by R (R).
2: Cycle-specific drill machining down to end deptht (E).
3: The withdrawal occurs a: with G98 to the start plane (S) and b: with G99 to the withdrawal plane.

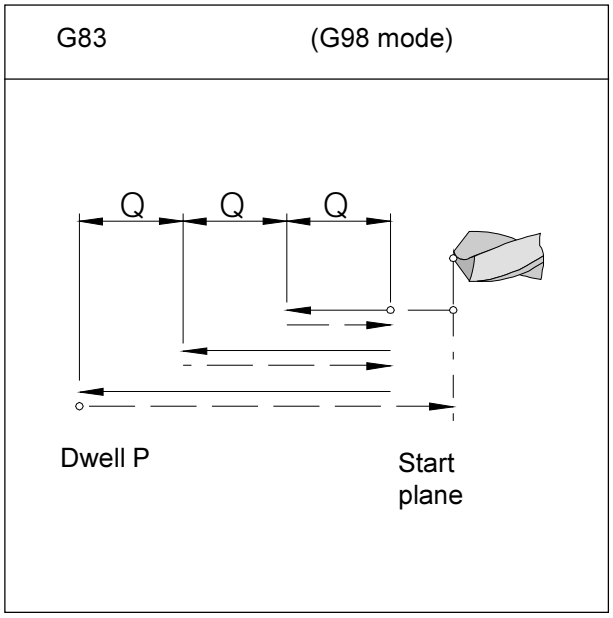
G80 Cancel Cycles (G83 - G85)

Format
N... G80

The drilling cycles are modal. They have to be cancelled by G80 or another group 1 command (G00, G01, ...).



Drilling cycle with retraction to withdrawal plane



Drilling cycle with retraction to start plane

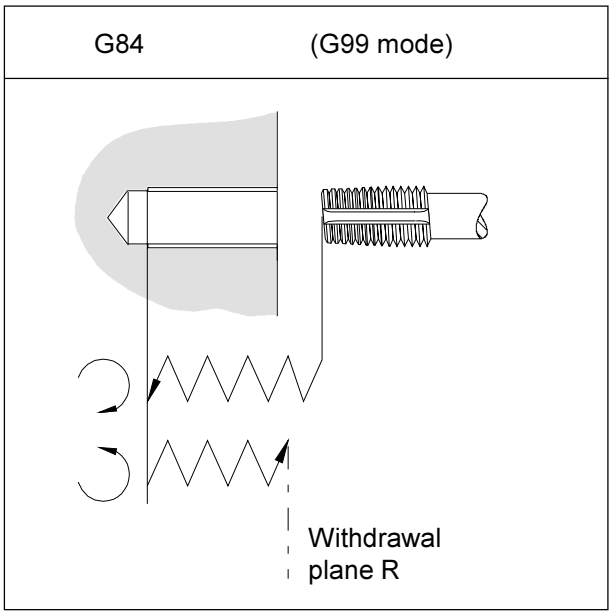
G83 Drilling Cycle

Format
N... G98(G99) G83 X0 Z(W)... (R...) Q... P... F... M...

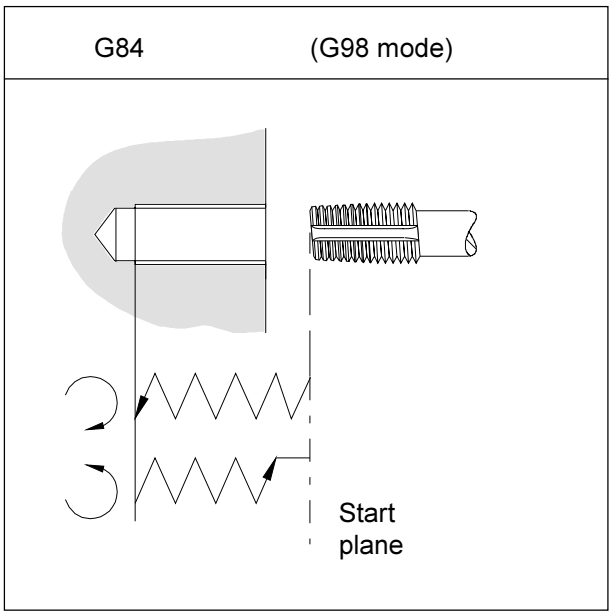
G98(G99) .. Return to starting plane (withdrawal plane)
X0 ..... Hole position in X (always zero)
Z(W) ..... Absolute (incremental) drilling depth
R [mm] ..... Incremental value for the withdrawal plane related to the start point in Z (with sign)
Q [µm] ..... Drilling depth per cut
P [msec] .... Dwell at the hole bottom
P1000 = 1 sec
F ..... Feed rate
M ..... Spindle direction (M03 or M04)
K ..... Number of cycle repetitions

Notes

- If G99 (withdrawal to the withdrawal plane) is programmed, also the address R must be defined. With G98 R can be omitted!
- It is not necessary to program X0 if in the previous block the tool has already been traversed to the turning centre (N... G00 X0 Z...)
  If X0 is programmed, in the previous block only the start position for the Z-axis must be approached (N.. G00 Z3)
- Unless Q is specified, a division of cuts is not carried out, i.e. drilling to the Z-end point in one movement.



Tapping cycle with retraction to the withdrawal plane



Tapping cycle with retraction to the start plane

G84 Tapping Cycle

Format

N... G98(G99) G84 X0 Z(W)... (R...) F... M...

F ..... Thread pitch
X0 ..... Hole position in X (always zero)
Z(W) ..... Absolute (incremental) drilling depth
R [mm] ..... Incremental value for the withdrawal plane related to the start point in Z (with sign)
P [msec] .... Dwell at the hole bottom
P1000 = 1 sec
F ..... Feed rate
M ..... Spindle direction (M03 or M04)

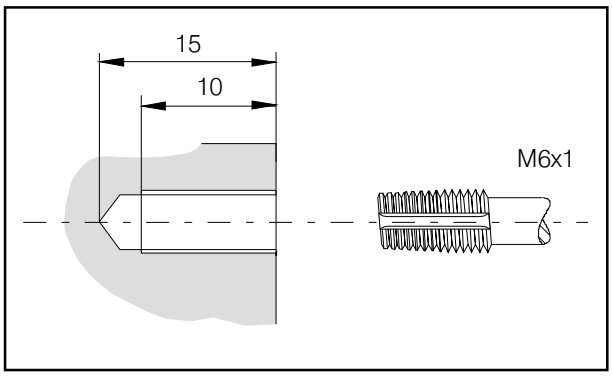
Notes

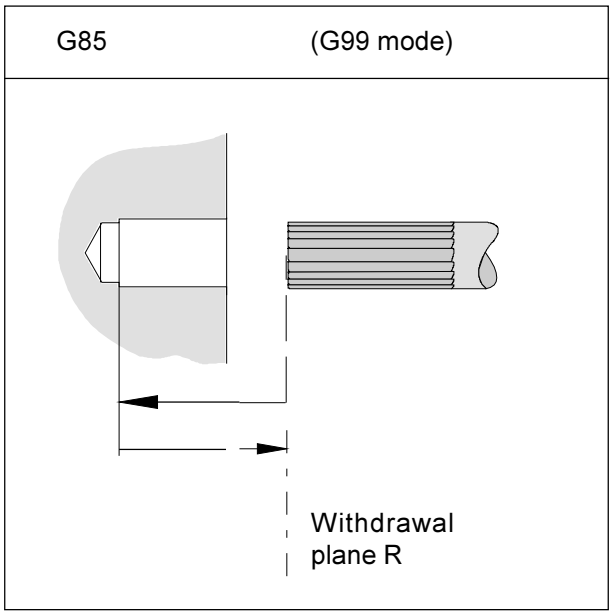
- If G99 (withdrawal to withdrawal plane) is prorammed, the address R must also be defined. With G98, R can be omitted!
- It is not necessary to program X0, if in the previous block the tool has already been traversed to the turning centre (N... G00 X0 Z...)
  If X0 is programmed, in the previous block only the starting position for the Z-axis must be approached (N.. G00 Z3).
- The tapping cycle is started with the respective M-function (M03 or M04). At the target point the sense of rotation of the spindle is reserved automatically for returning. If the starting position is reached again, the system switches to the original sense of rotation.

Deep-hole drilling, G83 and tapping, G84 at the main spindle with stationary tools

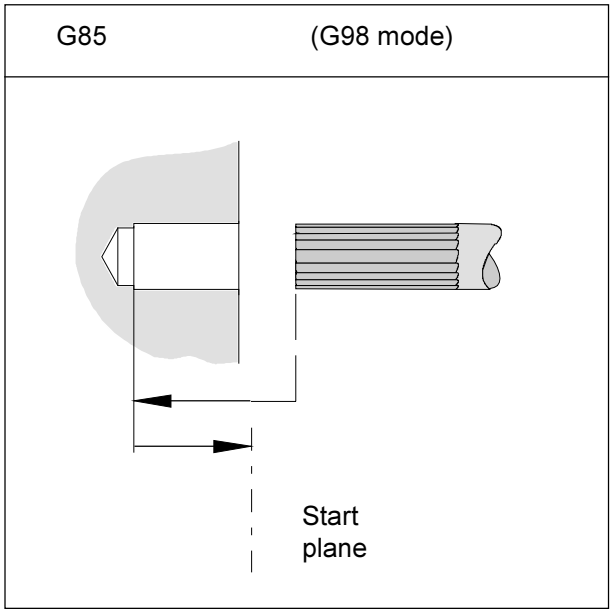
Example:

```
G10 P0 Z-100
T0000 G0 X100 Z150
G90 G40 G95
T0505 ( driller diam.5 )
G97 S2000 M3
G0 X0 Z2
G83 Z-15 Q5000 F0.15
G0 Z50
T0707 ( AWZAXIALGEWB. )
N90 G97 S300
G0 X0 Z5
G84 Z-10 F1 M3
G0 Z20 M5
M30
```





Reaming cycle with retraction to the withdrawal plane



Reaming cycle with retraction to the start plane

G85 Reaming Cycle

Format
N... G98(G99) G85 X0 Z(W)... (R...) P... F... M...

X0 ..... Hole position in X (always zero)
Z(W) ..... Absolute (incremental) drilling depth
R [mm] ..... Incremental value for the withdrawal plane related to the start point in Z (with sign)
P [msec] .... Dwell at the hole bottom
P1000 = 1 sec
F ..... Feed rate
M ..... Spindle direction (M03 or M04)

Notes
- Parameter R must be specified when G99 is programmed. You can leave R when G98 is active.
- It is possible to leave address X0 at G83 if you have programmed the starting position for execution of the cycle in both axis (N... G00 X0 Z...) at the block before.
  Otherwise just the starting position in Z axis has to be approached (N.. G00 Z3) at the block before.

Retraction to the start point is executed with double the programmed feed.
A cut division by defining Q is not possible.

G90 Absolute Programming

Format
N... G90

The addresses have to be programmed as following:
X ..... Diameter
U+/- .... Incremental in diameter (expect 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.

G92 Spindle Speed Limit

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

The max. spindle speed (U/min) for constant cutting speed (G96) can be established with the command G92.

G92 Coordinate System Setting

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.

Format

N... G94 F...

G95 Feed Rate in Revolutions

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

Format

N... G95 F...

G96 Constant Cutting Speed

Unit: m/min

The control continually computes the spindle speed corresponding to the respective diameter.

Format

N... G96 S...

G97 Constant Rotational Speed

Unit: rev/min

Format

N... G97 S...

Description of M Commands

M Commands are switching or addidional functions. The M commands can stand alone in a program block or together with other commands. Commands of the same group cancel each other, that means the M command programmed last cancels the previously programmed M command of the same group.

Note
The following pages describe the standard M commands. Wheter these M commands are executable depends on the type of the machine and the used accessories.

M00 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.

M01 Programmed Stop Conditional

M01 works like M00, but only if the function PROGRAMMED STOP YES was switched on by softkey in the menu PROGRAM CONTROL.

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.

M02 Main Program End

M02 works like M30.

M03 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, if machining occurs behind the turning centre.

M04 Main Spindle ON Counterclockwise

The same conditions as described under M03 apply here.
M03 must be used for all left-hand cutting or normal clamped tools, if machining occurs behind the turning centre.

M05 Main Spindle Off

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

M08 Coolant ON

only for EMCO PC Turn 120/125/155.
The coolant will be switched on.

M09 Coolant OFF

only for EMCO PC Turn 120/125/155.
The coolant will be switched off.

M20 Tailstock BACK

only for accessory automatical tailstock.
The tailstock moves back.
See "Accessory Functions - Automatic Tailstock"

M21 Tailstock FORWARD

only for accessory automatical tailstock only on the PC TURN 120/125/155.
The tailstock moves forward.
See "Accessory Functions - Automatic Tailstock"

M25 Open Clamping Device

only for accessory automatical clamping device only on the PC TURN 120/125/155.
The clamping device opens.
See "Accessory Functions - Automatic Clamping Device"

M26 Close Clamping Device

only for accessory automatical clamping device only on the PC TURN 120/125/155.
The clamping device closes.
See "Accessory Functions - Automatic Clamping Device"

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.

M71 Puff Blowing ON

only for accessory puff blowing device
The puff blowing device will be switched on.

M72 Puff Blowing OFF

only for accessory puff blowing device
The puff blowing device will be switched off.

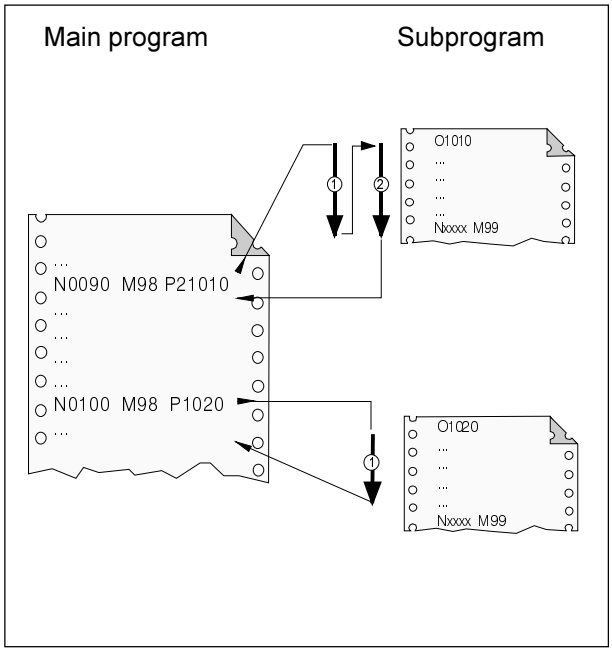
M98 Subprogram Call

Format
N... M98 P...

P ..... The first four digits from the right determine 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.



Subroutine call from a main program

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 subprogram.
- The jump-back occurs automatically into the next following block in the main program.

Application of the C-axis

COMMAND	MEANING
M13	DRIVEN TOOL ON clockwise
M14	DRIVEN TOOL ON counterclockw.
M15	DRIVEN TOOL OFF
M52	Round axis operation (C-axis ON)
M53	Spindle operation (C-axis OFF)

Note

The original FANUC 21T control does not support an operation with the C-axis (round axis) and with driven tools.

To be able to use this function still on the respective EMCO machines (EMCO TURN 325/II, PC TURN 155, CONCEPT TURN 155), EMCO WinNC FANUC 21T has been expanded by that scope of commands of FANUC 21i which supports the C-axis with driven tools.

For that purpose the adjoining M-functions have been taken over.

For milling with polar coordinate interpolation the functions G12.1/G13.1 have been taken over. The cycles G83 and G84 have been modified correspondingly for the operation with driven tools. Programming of G83 and G84 is identical with the one in the original FANUC 21T control.

Axial working with driven tools

Note:
When using driven tools (EMCO Turn 325/II, PC Turn 155, Concept Turn 155) an offset of X-20mm must be entered in the tool data for the driven tool in X-direction . This offset results from the different positions of driven tools versus rigid tools.

Deep-hole drilling axial with driven tools, G83

Format

N... G83 Z-15 Q.... F...

G83 ..... call drilling cycle
Z-15 ..... final drilling depth absolute (here 15)
Q [µm] ..... infeed depth to withdrawal
F ..... drilling feed

Prior to the call of the drilling cycle the tool in the X- and C-axis must be positioned on drilling centre and in the Z-axis at a safety distance. After the cycle end the tool is positioned in rapid motion to the last position before cycle call (safety distance). The number of revolutions and the rotational direction must be programmed prior to the cycle call.

General notes

- X must always be programmed even if in the previous block the tool has already been traversed to the turning centre (N... G00 X Z...)
  If X is progammed, in the previous block only the start position for the Z-axis must be approached (N.. G00 Z3)
- If Q is not specified, no cut division is carried out, i.e. drilling till Z-endpoint in one movement.

Tapping axial with driven tool, G84

Format
N... G84 Z-10 F... M...

G84 ..... call drilling cycle
Z-10 ..... tapping depth absolute (here 10)
F [mm] ..... thread pitch
M ..... rotational direction of spindle (M13 or M14)

Note:
When using driven tools (EMCO Turn 325/II, PC Turn 155, Concept Turn 155) an offset of X-20mm must be entered in the tool data for the driven tool in X-direction . This offset results from the different positions of driven tools versus rigid tools.

Prior to the call of the drilling cycle the tool in the X- and C-axis must be positioned on drilling centre and in the Z-axis on safety distance.

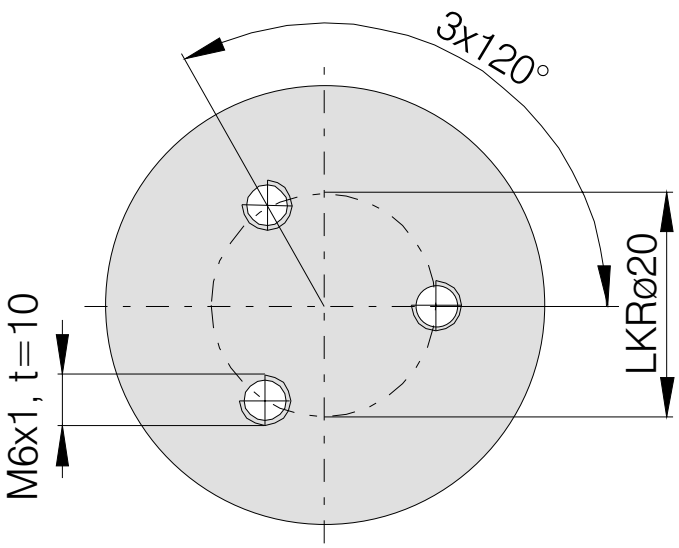
General notes

- X must always be programmed even if in the previous block the tool has already been traversed to the turning centre (N... G00 X Z...)
  If X is progammed, in the previous block only the start position for the Z-axis must be approached (N.. G00 Z3)
- The tapping cycle is started with the respective M-function (M13 or M14). At the target point the sense of rotation of the spindle is automatically reversed for returning. When the starting position is reached again, the system switches automatically to the original sense of rotation.
- Tapping with driven tool (M13, M14) only with axial tappings, which are outside of the turning centre. For that purpose also the C-axis must previously be activated and correspondingly positioned (M52).

Deep-hole drilling, G83 and tapping, G84 axial with driven tool

Example:

```
( LKR DMR.20 )
( M6 10mm tief )
( 3x 120GRAD )
G10 P0 Z-100
T0000 G0 X100 Z150
G90 G40 G95
T0505
M52
G28 G0 C0
M13
G97 S2000
G0 X20 Z2
G83 Z-15 Q5000 F0.15
G0 C120
G83 Z-15
G0 C240
G83 Z-15
G0 Z20 M15
T0707 ( AXIAL TAPPING )
N90 G97 S300
N95 G0 X20 Z5
G0 C0
G84 Z-10 F1 M13
G0 C120
G84 Z-10 F1 M13
G0 C240
G84 Z-10 F1 M13
G0 Z20 M15
M53
M30
```



Radial working with driven tools

Note:
When using driven tools (EMCO Turn 325/II, PC Turn 155, Concept Turn 155) an offset of X-20mm must be entered in the tool data for the driven tool in X-direction . This offset results from the different positions of driven tools versus rigid tools.

Deep-hole drilling radial with driven tool, G77

Format
N... G77 R1
N... G77 X-4 P... F...

G77 ..... call drilling cycle
R1 [mm] .... withdrawal (here 1)
X-4 ..... final drilling depth (here 4)
P [µm] ..... infeed to withdrawal
F ..... drilling feed

Prior to the call of the drilling cycle the tool in the Z- and C-axis must be positioned on drilling centre and in the X-axis at a safety distance. After the cycle end the tool is positioned in rapid motion to the last position before cycle call (safety distance). The number of revolutions and the rotational direction must be programmed prior to the cycle call.

General notes

- If P is not specified, no cut division is carried out i.e. drilling till Z-endpoint in one movement.

Tapping radial with driven tool, G33

Format
N... G33 X2 F... M13
N... G33 X24 F... M14

G33 ..... tapping
X2 [mm] ..... thread depth absolute (here 2)
X24 [mm] ... starting point
F [mm] ..... thread pitch drilling/withdrawal
M13 ..... rotational direction of spindle drilling
M14 ..... rotational direction of spindle withdrawal

Prior to the call of the drilling cycle the tool in the Z- and C-axis must be positioned on drilling centre and in the X-axis at safety distance.

General notes

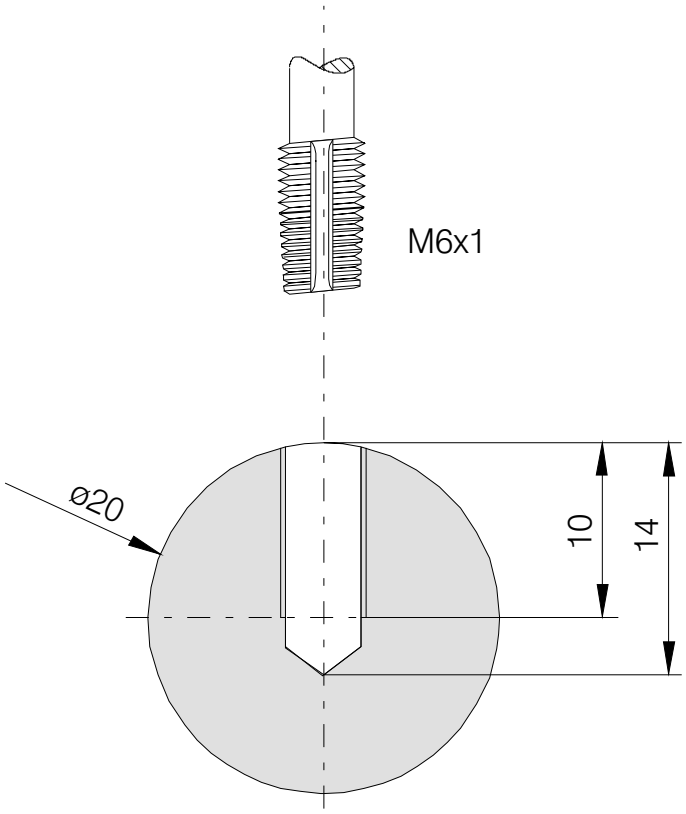
- The tapping cycle is started with the respective M-function (M13 or M14). At the target point the sense of rotation of the spindle is automatically reversed for returning. When the starting position is reached again, the system switches automatically to the original sense of rotation.

Note:
When using driven tools (EMCO Turn 325/II, PC Turn 155, Concept Turn 155) an offset of X-20mm must be entered in the tool data for the driven tool in X-direction . This offset results from the different positions of driven tools versus rigid tools.

Deep-hole drilling, G77 and tapping, G33 radial with driven tool

Example

```
( M6 10mm deep)
G10 P0 Z-100
T0000 G0 X100 Z150
G90 G40 G95
T0909
M52
G28 G0 C0
M13
G97 S2000
G0 X24 Z-10
G77 R1
G77 X-8 P5000 F0.15
G0 C120
G0 X80 M15
T0909
N90 G97 S300
N95 G0 X26 Z-10
G0 C0
G33 X0 F1 M13
G33 F1 X24 M14
G0 X80 Z20 M15
M53
M30
```



G: Flexible NC programming

Variable number	Variable type	Function
#0	Always zero system variable	This variable has always the value zero. Not changeable
#1-33	Local variable	At disposal for calculations in the program
#100-149	Global variables	At disposal for calculations in the program
#500-531	System variable	At disposal for calculations in the program
#1000	System variable	Loading magazine: bar end reached
#1001	System variable	Loading magazine: loader has advanced
#1002	System variable	Loading magazine: first part after bar change
#3901	System variable	Nominal piece number
#3901	System variable	Actual piece number

Variables and arithmetic parameters

By using variables instead of fixed values, a program can be configured more flexibly. Thus, you can react to signals, such as e.g. measuring values, or the same program can be used for different geometries by using variables as nominal value.
Together with variable calculation and program jumps you get the possibility to create a highly-flexible program archive and thus save programming time.

Local and global variables can be read and written. All other variables can only be read.

Local variables can only be used in that macro in which they are defined.
Global variables can be used in every macro irrespective of the macro in which they were defined.

Calculating with variables

With the four basic arithmetic operations the usual mathematic notation is valid.

Function	Example
=	#1=2
+	#1=#2+#3
-	#1=#2-#3
*	#1=#2*#3
/	#1=#2/#3

The term at the operator's right can contain constants and/or variables combined by functions.

Each variable can be replaced again by an arithemetic term in square brackets or by a constant.

Example

#1=#[#2]

During the calculation the limitation is valid that the execution of the calculation is carried out from left to right without observance of the calculation rule point before line.

Example

#1=#2*3+#5/2

Control structures

In programs the control sequence can be changed by IF and GOTO instructions. Three types of branchings are possible:

- IF[<condition>] THEN
- IF[<condition>] GOTO <n>
- GOTO <destination>

IF[<Condition>] THEN

After IF a provisory term must be indicated. If the provisory term applies, a determined macro instruction is carried out. Only one macro instruction can be carried out.

Example

With equal values of #1 and #2 the value 5 is allocated to #3.

IF [#1 EQ #2] THEN#3=5

IF[<Condition>] GOTO <n>

After IF a provisory term must be indicated. If the provisory term applies, the branching is carried out to block number n. Otherwise the subsequent block is carried out.

Example

If the value of the variable #1 is greater than 10, the branching is carried out to block number N4. Otherwise the subsequent block is carried out.

IF [#1 GT 10] GOTO 4

GOTO <n>

The jump command GOTO can also be programmed without condition. A variable or constant can be used as a branch destination. With a variable the number can be replaced again by a calculation term in square brackets.

Example

Jump to block number 3

GOTO 3

Example

Jump to variable #6

GOTO#6

Relational operators

Relational operators consist of two letters and are used to determine, in comparison with two values, if these are equal or if one value is greater and/or less than the other.

Operator	Meaning
EQ	Equal (=)
NE	Unequal (≠)
GT	Greater than (>)
GE	Greater than or equal (=)
LT	Less than (<)
LE	Less than or equal (=)

The expressions to be compared can be variable n or constants. A variable can be replaced again by a calculation term in square brackets.

Example

IF[#12 EQ 1] GOTO10

Comprising macro programming examples:

IF[#1000 EQ 1] GOTO10

IF[#[10]] NE #0] GOTO#[#1]

IF[1 EQ 1] THEN#2 =5

IF[#[#4+#[#2/2]] GT #20] THEN#[#10]] =#1*5+#7

H: Alarms and Messages

Machine Alarms 6000 - 7999

These alarms will be triggered by the machines.
There are different alarms for the different machines.
The alarms 6000 - 6999 normally must be confirmed with RESET. The alarms 7000 - 7999 are messages which normally will disappear when the releasing situation is finished.

PC MILL 50 / 55 / 100 / 105 / 125 / 155
Concept MILL 55 / 105 / 155

6000: EMERGENCY OFF
The EMERGENCY OFF key was pressed. Remove the endangering situation and restart machine and software.

6001: PLC-CYCLE TIME EXCEEDING
Contact EMCO Service.

6002: PLC - NO PROGRAM CHARGED
Contact EMCO Service.

6003: PLC - NO DATA UNIT
Contact EMCO Service.

6004: PLC - RAM MEMORY FAILURE
Contact EMCO Service.

6005: OVERHEAT BRAKEMODUL
Main drive was braked too often, large changes of speed within a short time. E4.2 active

6006: OVERLOAD BRAKE RESISTOR
see 6005

6007: SAFETY CIRCUIT FAULT
Axis and main drive contactor with machine switched off not disabled. Contactor got stuck or contact error. E4.7 was not active during switch-on.

6009: SAFETY CIRCUIT FAULT
Defective step motor system.
A running CNC program will be interrupted, the auxiliary drives will be stopped, the reference position will be lost.
Contact EMCO Service.

6010: DRIVE X-AXIS NOT READY
The step motor board is defective or too hot, a fuse or cabling is defective.
A running program will be stopped, the auxiliary drives will be switched off, the reference position will be lost.
Check fuses or contact EMCO service.

6011: DRIVE Y-AXIS NOT READY
see alarm 6010.

6012: DRIVE Z-AXIS NOT READY
see alarm 6010.

6013: MAIN DRIVE NOT READY
Main drive power supply defective, main drive too hot, fuse defective.
A running program will be stopped, the auxilliary drives will be switched off.
Check fuses or contact EMCO Service.

6014: NO MAIN SPINDLE SPEED
This will be released, when the spindle speed is lower than 20 rpm because of overload.
Alter cutting data (feed, infeed, spindle speed).
The CNC program will be aborted, the auxilliary drives will be stopped.

6019: VICE TIME EXCEED
The electric vice has not reached a stop position within 30 seconds.
The control or the clamping device board are defective, the vice is stuck. Adjust the proximity switches of the stop position.

6020: VICE FAILURE
When the electric vice is closed, the signal "clamping device clamped" of the clamping device board has failed.
The control, the clamping device board or the wiring are defective.

6022: CLAMPING DEVICE BOARD DEFECTIVE
The signal "clamping device clamped" is constantly released, although no command has been given.
Replace the board.

6024: MACHINE DOOR OPEN
The door was opened while a machine movement.
The program will be aborted.

6027: DOOR LIMIT SWITCH DEFECTIVE
The limit switch of the automatic door is displaced, defective, wrong cabled.
Contact EMCO service.

6028: DOOR TIMEOUT
The automatic door stucks, the pressured air supply is insufficient, the limit switch is displaced.
Check door, pressured air supply, limit switch or contact EMCO service.

6030: NO PART CLAMPED
No workpiece inserted, vice cheek displaced, control cam displaced, hardware defective.
Adjust or contact EMCO service.

6040: TOOL TURRET INDEX FAILURE
After WZW procedure drum pressed down by Z-axis. Spindle position wrong or mechanical defect. E4.3=0 in lower state

6041: TOOL CHANGE TIMEOUT
Tool drum stucks (collision?), main drive not ready, fuse defective, hardware defective.
A running CNC program will be stopped.
Check for collisions, check fuses or contact EMCO service.

6043-6046: TOOL DISK POSITION FAULT
Position error of main drive, error of position supervising (inductive proximity switch defective or disadjusted, drum allowance), fuse defective, hardware defective.
The Z axis could have been slipped out of the toothing while the machine was switched off.
A running CNC program will be stopped.
Contact EMCO service.

6047: TOOL DISK UNLOCKED
Tool drum turned out of locked position, inductive proximity switch defective or disadjusted, fuse defective, hardware defective.
A running CNC program will be interrupted.
Contact EMCO service.
When the tool drum is turned out of locked position (no defect), act as following:
Turn the drum into locking position manually
Change into MANUAL (JOG) mode.
Turn the key switch. Traverse the Z slide upwards, until the alarm disappears.

6048: DIVIDING TIME EXCEEDED
Dividing head stucks, insufficient pressured air supply, hardware defective.
Check for collision, check pressured air supply or contact EMCO service.

6049: INTERLOCKING TIME EXCEEDED
see alarm 6048

6050: M25 AT RUNNING MAIN SPINDLE
Cause: Programming mistake in NC program.
A running program will be aborted.
The auxilliary drives will be switched off.
Remedy: Correct NC program

6064: DOOR AUTOMATIC NOT READY
Cause: pressure failure automatic door
automatic door stucks mechanically
limit switch for open end position defective
security print circuits defect
cabling defective
fuses defective
A running program will be aborted.
The auxilliary drives will be switched off.
Remedy: service automatic door

6069: CLAMPING FOR TANI NOT OPEN
When opening the clamping pressure switch does not fall within 400ms. Pressure switch defective or mechanical problem. E22.3

6070: PRESSURE SWITCH FOR TANI MISSING
When closing the clamping pressure switch does not respond. No compressed air or mechanical problem. E22.3

6071: DIVIDING DEVICE NOT READY
Servo Ready Signal from frequency converter missing. Excess temperature drive TANI or frequency converter not ready for operation.

6072: VICE NOT READY
Attempt to start the spindle with an open vice or without clamped workpiece.
Vice stucks mechanically, insufficient compressed air supply, compressed air switch defective, fuse defective, hardware defective.
Check the fuses or contact EMCO service.

6073: DIVIDING DEVICE NOT READY
Cause: locking switch defective
cabling defective
fuses defective
A running program will be aborted.
The auxilliary drives will be switched off.
Remedy: service automatic dividing device
lock the dividing device

6074: DIVIDING TIME EXCEEDED
Cause: dividing device stucks mechanically
locking switch defective
cabling defective
fuses defective
insufficient compressed-air supply.
A running program will be aborted.
The auxilliary drives will be switched off.
Remedy: Check for collision, check the compressed-air supply or contact the EMCO service.

6075: M27 AT RUNNING MAIN SPINDLE
Cause: Programming mistake in NC program.
A running program will be aborted.
The auxilliary drives will be switched off.
Remedy: Correct NC program

7000: INVALID TOOL NUMBER PROGRAMMED
The tool position was programmed larger than 10.
The CNC program will be stopped.
Interrupt program with RESET and correct the program.

7001: NO M6 PROGRAMMED
For an automatic tool change you also have to program a M6 after the T word.

7007: FEED STOP!
The axes have been stopped by the robotics interface (robotics entry FEEDHOLD).

7016: SWITCH ON AUXILIARY DRIVES
The auxiliary drives are off. Press the AUX ON key for at least 0.5 sec. (to avoid accidentally switching on) to switch on the auxiliary drives.

7017: REFERENCE MACHINE
Approach the reference point.
When the reference point is not active, manual movements are possible only with key switch at position "setting operation".

7018: TURN KEY SWITCH
With NC-Start the key switch was in position "setting operation".
NC-Start is locked.
Turn the key switch in the position "automatic" to run a program.

7020: SPECIAL OPERATION MODE ACTIVE
Special operation mode: The machine door is opened, the auxiliary drives are switched on, the key switch is in position "setting operation" and the consent key is pressed.
Manual traversing the axes is possible with open door. Swivelling the tool turret is not possible with open door. Running a CNC program is possible only with standing spindle (DRYRUN) and SINGLE block operation.
For safety: If the consent key is pressed for more than 40 sec. the function of this key is interrupted, the consent key must be released and pressed again.

7021: INITIALIZE TOOL TURRET
The tool turret operating was interrupted.
No traversing operation is possible.
Press tool turret key in JOG operation. Message occurs after alarm 6040.

7022: INITIALIZE TOOL TURRET !
see 7021

7023: WAITING TIME MAIN DRIVE!
The LENZE frequency converter has to be separated from the mains supply for at least 20 seconds before you are allowed to switch it on again. This message will appear when the door is quickly openend/ closed (under 20 seconds).

7038: LUBRICATION SYSTEM FAULT
The pressure switch is defective or gagged.
NC-Start is locked. This can be reset only by switching off and on the machine.
Contact EMCO service.

7039: LUBRICATION SYSTEM FAULT
Not enough lubricant, the pressure switch is defective.
NC-Start is locked.
Check the lubricant and lubricate manually or contact EMCO service.

7040: MACHINE DOOR OPEN
The main drive can not be switched on and NC-Start can not be activated (except special operation mode)
Close the machine to run a program.

7042: INITIALIZE MACHINE DOOR
Every movement and NC-Start are locked.
Open and close the machine door to initialize the safety circuits.

7043: PIECE COUNT REACHED
A predetermined number of program runs was reached. NC-Start is locked. Reset the counter to continue.

7050: NO PART CLAMPED
After switching on or after an the vice is neither at the open position nor at the closed position.
NC-Start is locked.
Traverse the vice manually on a valid end position.

7051: DIVIDING HEAD NOT LOCKED!
Either the dividing head is in an undefined position after the machine has been switched on, or the locking signal after a dividing process is missing.
Initiate the dividing process, check, respectively adjust the proximity switch for locking.

7054: VICE OPEN
Cause: the workpiece is not clamped
When switching on the main spindle with M3/M4 alarm 6072 (vice not ready) will be released.
Remedy: Clamp

7055: OPEN TOOL CLAMPING SYSTEM
A tool is clamped in the main spindle and the control does not recognize the corresponding T number.
Eject the tool from the main spindle when the door is open by means of the PC keys "Strg" and " 1 ".

7056: SETTING DATA INCORRECT
An invalid tool number is stored in the setting data.
Delete the setting data in the machine directory xxxxx.pls.

7057: TOOLHOLDER OCCUPIED
The clamped tool cannot be positioned in the tool turret since the position is occupied.
Eject the tool from the main spindle when the door is open by means of the PC keys "Strg" and " 1 ".

7058: RETRACTING THE AXES
The position of the tool turret arm cannot be clearly defined during the tool change.
Open the machine door, push the tool turret magazine backwards to the stop. Move the milling head in the JOG mode upwards to the Z reference switch and then traverse the reference point.

7270: OFFSET COMPENSATION ACTIVE !
Only with PC-MILL 105
Offset compensation activated by the following operation sequence.
- Reference point not active
- Machine in reference mode
- Key switch in manual operation
- Press STRG (or CTRL) and simultaneously 4

This must be carried out if prior to the tool change procedure spindle positioning is not completed (tolerance window too large)

7271: COMPENSATION FINISHED,DATA SAVED !
see 7270

PC TURN 50 / 55 / 105 / 120 / 125 / 155
Concept TURN 55 / 105 / 155

6000: EMERGENCY OFF
The EMERGENCY OFF key was pressed.
The reference position will be lost, the auxiliary drives will be switched off.
Remove the endangering situation and restart machine and software.

6001: PLC-CYCLE TIME EXCEEDING
The auxiliary drives will be switched off.
Contact EMCO Service.

6002: PLC - NO PROGRAM CHARGED
The auxiliary drives will be switched off.
Contact EMCO Service.

6003: PLC - NO DATA UNIT
The auxiliary drives will be switched off.
Contact EMCO Service.

6004: PLC - RAM MEMORY FAILURE
The auxiliary drives will be switched off.
Contact EMCO Service.

6008: MISSING CAN SUBSCRIBER
The SPS-CAN board is not identified by the control.
Check the interface cable and the power supply of the CAN board.

6009: SAFETY CIRCUIT FAULT
Defective step motor system.
A running CNC program will be interrupted, the auxiliary drives will be stopped, the reference position will be lost.
Contact EMCO Service.

6010: DRIVE X-AXIS NOT READY
The step motor board is defective or too hot, a fuse is defective, over- or undervoltage from mains.
A running program will be stopped, the auxiliary drives will be switched off, the reference position will be lost.
Check fuses or contact EMCO service.

6012: DRIVE Z-AXIS NOT READY
see 6010.

6013: MAIN DRIVE NOT READY
Main drive power supply defective or main drive too hot, fuse defective, over- or undervoltage from mains.
A running program will be stopped, the auxilliary drives will be switched off.
Check fuses or contact EMCO Service.

6014: NO MAIN SPINDLE SPEED
This alarm will be released, when the spindle speed is lower than 20 rpm because of overload.
Alter cutting data (feed, infeed, spindle speed).
The CNC program will be aborted, the auxiliary drives will be switched off.

6015: NO DRIVEN TOOL SPINDLE SPEED
see 6014.

6016: AUTOMATIC TOOL TURRET SIGNAL COUPLED MISSING

6017: AUTOMATIC TOOL TURRET SIGNAL UNCOUPLED MISSING
In the tool turret that can be coupled, the position of the coupling and uncoupling magnet is monitored by means of two proximity switches. It has to be made sure that the coupling is in the rear stop position so that the tool turret can get to the next tool position. Equally, during operation with driven tools the coupling has to be safe in the front stop position.
Check and adjust the cables, the magnet and the stop position proximity switches.

6021: COLLET TIME OUT
During closing of the clamping device the pressure switch has not reacted within one second.

6022: CLAMPING DEVICE BOARD DEFECTIVE
The signal "clamping device clamped" is constantly released, even though no command has been given. Replace the board.

6023: COLLET PRESSURE MONITORING
The pressure switch turns off when the clamping device is closed (compressed air failure for more than 500ms).

6024: MACHINE DOOR OPEN
The door was opened while a machine movement. The program will be aborted.

6025: GEARBOX COVER NOT CLOSED
The gearbox cover was opened while a machine movement. A running CNC program will be aborted.
Close the cover to continue.

6027: DOOR LIMIT SWITCH DEFECTIVE
The limit switch of the automatic door is displaced, defective, wrong cabled.
Contact EMCO service.

6028: DOOR TIMEOUT
The automatic door stucks, the pressured air supply is insufficient, the limit switch is displaced. Check door, pressured air supply, limit switch or contact EMCO service.

6029: TAILSTOCK QUILL TIME EXCEED
The tailstock quill does not reach a final position within 10 seconds.
Adjust the control and the stop position proximity switches, or the tailstock quill is stuck.

6030: NO PART CLAMPED
No workpiece inserted, vice cheek displaced, control cam displaced, hardware defective.
Adjust or contact EMCO service.

6031: QUILL FAILURE

6032: TOOL CHANGE TIMEOUT
see alarm 6041.

6033: TOOL TURRET SYNC ERROR
Hardware defective.
Contact EMCO service.

6037: CHUCK TIMEOUT
The pressure switch does not react within one second when the clamping device is closed.

6039: CHUCK PRESSURE FAILURE
The pressure switch turns off when the clamping device is closed (compressed air failure for more than 500ms).

6040: TOOL TURRET INDEX FAILURE
The tool turret is in no locked position, tool turret sensor board defective, cabling defective, fuse defective.
A running CNC program will be stopped.
Swivel the tool turret with the tool turret key, check fuses or contact EMCO service.

6041: TOOL CHANGE TIMEOUT
Tool drum stucks (collision?), fuse defective, hardware defective.
A running CNC program will be stopped.
Check for collisions, check fuses or contact EMCO service.

6042: TOOL TURRET OVERHEAT
Tool turret motor too hot.
With the tool turret a max. of 14 swivel procedures a minute may be carried out.

6043: TOOL CHANGE TIMEOUT
Tool drum stucks (collision?), fuse defective, hardware defective.
A running CNC program will be stopped.
Check for collisions, check fuses or contact EMCO service.

6045: TOOL TURRET SYNC MISSING
Hardware defective.
Contact EMCO service.

6046: TOOL TURRET ENCODER FAULT
Fuse defective, hardware defective.
Check fuses or contact EMCO service.

6048: CHUCK NOT READY
Attempt to start the spindle with open chuck or without clamped workpiece.
Chuck stucks mechanically, insufficient pressured air supply, fuse defective, hardware defective.
Check fuses or contact EMCO service.

6049: COLLET NOT READY
see 6048

6050: M25 DURING SPINDLE ROTATION
With M25 the main spindle must stand still (consider run-out time, evtl. program a dwell)

6055: NO PART CLAMPED
This alarm occurs when with rotating spindle the clamping device or the tailstock reach the end position. The workpiece has been pushed out of the chuck or has been pushed into the chuck by the tailstock.
Check clamping device settings, clamping forces, alter cutting data.

6056: QUILL NOT READY
Attempt to start the spindle or to move an axis or to swivel the tool turret with undefined tailstock position.
Tailstock is locked mechanically (collision), insufficient pressured air supply, fuse defective, magnetic switch defective.
Check for collisions, check fuses or contact EMCO service.

6057: M20/M21 DURING SPINDLE ROTATION
With M20/M21 the main spindle must stand still (consider run-out time, evtl. program a dwell)

6058: M25/M26 DURING QUILL FORWARD
To actuate the clamping device in an NC program with M25 or M26 the tailstock must be in back end position.

6059: C-AXIS SWING IN TIMEOUT
C-axis does not swivel in within 4 seconds.
Reason: not sufficient air pressure, and/or mechanics stuck.

6060: C-AXIS INDEX FAILURE
When swivelling in the C-axis the limit switch does not respond.
Check pneumatics, mechanics and limit switch.

6064: AUTOMATIC DOOR NOT READY
Door stucks mechanically (collision), insufficient pressured air supply, limit switch defective, fuse defective.
Check for collisions, check fuses or contact EMCO service.

6065: LOADER MAGAZINE FAILURE
Loader not ready.
Check if the loader is switched on, correctly connected and ready for operation and/or disable loader (WinConfig).

6066: CLAMPING DEVICE FAILURE
No compressed air at the clamping device
Check pneumatics and position of the clamping device proximity detectors.

6067: NO COMPRESSED AIR
Turn the compressed air on, check the setting of the pressure switch.

7000: INVALID TOOL NUMBER PROGRAMMED
The tool position was programmed larger than 8.
The CNC program will be stopped.
Interrupt program with RESET and correct the program.

7007: FEED HOLD
In the robotic mode a HIGH signal is at input E3.7.
Feed Stop is active until a low signal is at E3.7.

7016: SWITCH ON AUXILIARY DRIVES
The auxiliary drives are off. Press the AUX ON key for at least 0.5 sec. (to avoid accidentally switching on) to switch on the auxiliary drives (also a lubricating pulse will be released).

7017: REFERENCE MACHINE
Approach the reference point.
When the reference point is not active, manual movements are possible only with key switch at position "setting operation".

7018: TURN KEY SWITCH
With NC-Start the key switch was in position "setting operation".
NC-Start is locked.
Turn the key switch in the position "automatic" to run a program.

7019: PNEUMATIC LUBRICATION MONITORING!
Refill pneumatic oil

7020: SPECIAL OPERATION MODE ACTIVE
Special operation mode: The machine door is opened, the auxiliary drives are switched on, the key switch is in position "setting operation" and the consent key is pressed.
Manual traversing the axes is possible with open door. Swivelling the tool turret is possible with open door. Running a CNC program is possible only with standing spindle (DRYRUN) and SINGLE block operation.
For safety: If the consent key is pressed for more than 40 sec. the function of this key is interrupted, the consent key must be released and pressed again.

7021: TOOL TURRET NOT LOCKED
The tool turret operating was interrupted.
NC start and spindle start are locked. Press the tool turret key in the RESET status of the control.

7022: COLLECTION DEVICE MONITORING
Time exceed of the swivelling movement.
Check the pneumatics, respectively whether the mechanical system is jammed (possibly a workpiece is jammed).

7023: ADJUST PRESSURE SWITCH !
During opening and closing of the clamping device the pressure switch has to turn off and on once.
Adjust the pressure switch. This alarm does not exist any more for versions starting with PLC 3.10.

7024: ADJUST CLAMPING DEVICE PROXIMITY SWITCH !
When the clamping device is open and the position stop control is active, the respective proximity switch has to feed back that the clamping device is "Open".
Check and adjust the clamping device proximity switch, check the cables.

7025 WAITING TIME MAIN DRIVE !
The LENZE frequency converter has to be separated from the mains supply for at least 20 seconds before you are allowed to switch it on again. This message will appear when the door is quickly openend/ closed (under 20 seconds).

7038: LUBRICATION SYSTEM FAULT
The pressure switch is defective or gagged.
NC-Start is locked. This alarm can be reset only by switching off and on the machine.
Contact EMCO service.

7039: LUBRICATION SYSTEM FAULT
Not enough lubricant, the pressure switch is defective.
NC-Start is locked.
Check the lubricant and lubricate manually or contact EMCO service.

7040: MACHINE DOOR OPEN
The main drive can not be switched on and NC-Start can not be activated (except special operation mode)
Close the machine to run a program.

7041: GEARBOX COVER OPEN
The main spindle cannot be switched on and NC start cannot be activated.
Close the gearbox cover in order to start a CNC program.

7042: INITIALIZE MACHINE DOOR
Every movement and NC-Start are locked.
Open and close the machine door to initialize the safety circuits.

7043: PIECE COUNT REACHED
A predetermined number of program runs was reached. NC-Start is locked. Reset the counter to continue.

7048: CHUCK OPEN
This message shows that the chuck is open. It will disappear if a workpiece will be clamped.

7049: CHUCK - NO PART CLAMPED
No part is clamped, the spindle can not be switched on.

7050: COLLET OPEN
This message shows that the collet is open. It will disappear if a workpiece will be clamped.

7051: COLLET - NO PART CLAMPED
No part is clamped, the spindle can not be switched on.

7052: QUILL IN UNDEFINED POSITION
The tailstock is in no defined position.
All axis movements, the spindle and the tool turret are locked.
Drive the tailstock in back end position or clamp a workpiece with the tailstock.

7053: QUILL - NO PART CLAMPED
The tailstock reached the front end position. Traverse the tailstock back to the back end position to continue.

7054: NO PART CLAMPED
No part clamped, switch-on of the spindle is locked.

7055: CLAMPING DEVICE OPEN
This message indicates that the clamping device is not in clamping state. It disappears as soon as a part is clamped.

AC95 / ACC ALARMS

Axis Controller Alarms

8000 Fatal Error AC

8100 Fatal init error AC
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8101 Fatal init error AC
see 8101.

8102 Fatal init error AC
see 8101.

8103 Fatal init error AC
see 8101.

8104 Fatal system error AC
see 8101.

8105 Fatal init error AC
see 8101.

8106 No PC-COM card found
Cause: PC-COM board can not be accessed (ev. not mounted).
Remedy: Mount board, adjust other address with jumper

8107 PC-COM card not working
see 8106.

8108 Fatal error on PC-COM card
see 8106.

8109 Fatal error on PC-COM card
see 8106.

8110 PC-COM init message missing
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8111 Wrong configuration of PC-COM
see 8110.

8113 Invalid data (pccom.hex)
see 8110.

8114 Programming error on PC-COM
see 8110.

8115 PC-COM packet acknowledge missing
see 8110.

8116 PC-COM startup error
see 8110.

8117 Fatal init data error (pccom.hex)
see 8110.

8118 Fatal init error AC
see 8110, ev. insufficient RAM memory

8119 PC interrupt no. not valid
Cause: The PC interrupt number can not be used.
Remedy: Find out free interrupt number in the Windows95 system control (allowed: 5,7,10, 11, 12, 3, 4 und 5) and enter this number in WinConfig.

8120 PC interrupt no. unmaskable
see 8119

8121 Invalid command to PC-COM
Cause: Internal error or defective cable
Remedy: Check cables (screw it); Restart software or reinstall when necessary, report to EMCO, if repeatable.

8122 Internal AC mailbox overrun
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8123 Open error on record file
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8124 Write error on record file
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8125 Invalid memory for record buffer
Cause: Insufficient RAM, record time exceeding.
Remedy: Restart software, ev. remove drivers etc. to gain more RAM, reduce record time.

8126 AC Interpolation overrun
Cause: Ev. insufficient computer performance.
Remedy: Set a longer interrupt time in WinConfig. This may result in poorer path accuracy.

8127 Insufficient memory
Cause: Insufficient RAM
Remedy: Close other programs, restart software, ev. remove drivers etc. to gain more RAM.

8128 Invalid message to AC
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8129 Invalid MSD data - axisconfig.
see 8128.

8130 Internal init error AC
see 8128.

8130 Internal init error AC
see 8128.

8132 Axis accessed by multiple channels
see 8128.

8133 Insufficient NC block memory AC
see 8128.

8134 Too much center points programmed
see 8128.

8135 No centerpoint programmed
see 8128.

8136 Circle radius too small
see 8128.

8137 Invalid for Helix specified
Cause: Wrong axis for helix. The combination of linear and circular axes does not match.
Remedy: Program correction.

8140 Maschine (ACIF) not responding
Cause: Machine off or not connected.
Remedy: Switch on machine or connect.

8141 Internal PC-COM error
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8142 ACIF Program error
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8143 ACIF packet acknowledge missing
see 8142.

8144 ACIF startup error
see 8142.

8145 Fatal init data error (acif.hex)
see 8142.

8146 Multiple request for axis
see 8142.

8147 Invalid PC-COM state (DPRAM)
see 8142.

8148 Invalid PC-COM command (CNo)
see 8142.

8149 Invalid PC-COM command (Len)
see 8142.

8150 Fatal ACIF error
see 8142.

8151 AC Init Error (missing RPG file)
see 8142.

8152 AC Init Error (RPG file format)
see 8142.

8153 FPGA program timeout on ACIF
see 8142.

8154 Invalid Command to PC-COM
see 8142.

8155 Invalid FPGA packet acknowledge
see 8142 or hardware error on ACIF board (contact EMCO Service).

8156 Sync within 1.5 revol. not found
see 8142 or Bero hardware error (contact EMCO Service).

8157 Data record done
see 8142.

8158 Bero width too large (referencing)
see 8142 or Bero hardware error (contact EMCO Service).

8159 Function not implemented
Bedeutung: In normal operation this function can not be executed

8160 Axis synchronization lost axis 3..7
Cause: Axis spins or slide is locked, axis synchronisation was lost
Remedy: Approach reference point

8161 X-Axis synchronization lost
Step loss of the step motor. Causes:
- Axis mechanically blocked
- Axis belt defective
- Distance of proximity detector too large (>0,3mm)
  or proximity detector defective
- Step motor defective

8162 Y-Axis synchronization lost
see 8161

8163 Z-Axis synchronization lost
see 8161

8164 Software limit switch max axis 3..7
Cause: Axis is at traverse area end
Remedy: Retract axis

8168 Software limit overtravel axis 3..7
Cause: Axis is at traverse area end
Remedy: Retract axis

8172 Communication error to machine
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.
Check connection PC - machine, eventually eliminate distortion sources.

8173 INC while NC program is running
Remedy: Stop the program with NC stop or with Reset. Traverse the axis.

8174 INC not allowed
Cause: At the moment the axis is in motion.
Remedy: Wait until the axis stops and then traverse the axis.

8175 MSD file could not be opened
Cause: Internal error
Remedy: Restart software oder bei Bedarf neu installieren, report to EMCO, if repeatable.

8176 PLS file could not be opened
see 8175.

8177 PLS file could not be accessed
see 8175.

8178 PLS file could not be written
see 8175.

8179 ACS file could not be opened
see 8175.

8180 ACS file could not be accessed
see 8175.

8181 ACS file could not be written
see 8175.

8183 Gear too high
Cause: The selected gear step is not allowed at the machine.

8184 Invalid interpolaton command

8185 Forbidden MSD data change
see 8175.

8186 MSD file could not be opened
see 8175.

8187 PLC program error
see 8175.

8188 Gear command invalid
see 8175.

8189 Invalid channel assignement
see 8175.

8190 Invalid channel within message
see 8175.

8191 Invalid jog feed unit
Cause: The machine does not support the rotation feed in the JOG operating mode.
Remedy: Order a software update from EMCO.

8192 Invalid axis in command
see 8175.

8193 Fatal PLC error
see 8175.

8194 Thread without length
Cause: The programmed target coordinates are identical to the starting coordinates.
Remedy: Correct the target coordinates.

8195 No thread slope in leading axis
Remedy: Program thread pitch

8196 Too manny axis for thread
Remedy: Program max. 2 axes for thread.

8197 Thread not long enough
Cause: Thread length too short.
With transition from one thread to the other the length of the second thread must be sufficient to produce a correct thread.
Remedy: Longer second thread or replace it by a linear interpolation (G1).

8198 Internal error (to manny threads)
see 8175.

8199 Internal error (thread state)
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8200 Thread without spindle on
Remedy: Switch on spindle

8201 Internal thread error (IPO)
see 8199.

8201 Internal thread error (IPO)
see 8199.

8203 Fatal AC error (0-ptr IPO)
see 8199.

8204 Fatal init error: PLC/IPO running
see 8199.

8205 PLC Runtime exceeded
Cause: Insufficient computer performance

8206 Invalid PLC M-group initialisation
see 8199.

8207 Invalid PLC machine data
see 8199.

8208 Invalid application message
see 8199.

8212 Rotation axis not allowed
see 8199.

8213 Circle and rotation axis can't be interpolated

8214 Thread and rotation axis cant't be interpolated

8215 Invalid state
see 8199.

8216 No rotation axis for rotation axis switch
see 8199.

8217 Axis type not valid!
Cause: Switching during the rotary axis operating mode when the spindle is running.
Remedy: Stop the spindle and switch over to the rotary axis operating mode.

8218 Referencing round axis without selected round axis!
see 8199.

8219 Thread not allowed without spindle encoder!
Cause: Thread cutting, respectively tapping is only possible with spindles with encoders.

8220 Buffer length exceeded in PC send message!
see 8199.

8221 Spindle release although axis is no spindle!
see 8199.

8222 New master spindle is not valid
Cause: The indicated master spindle is not valid when switching over to the master spindle.
Remedy: Correct the spindle number.

8224 Invalid stop mode
see 8199.

8225 Invalid parameter for BC_MOVE_TO_IO!
Cause: The machine is not configurated for touch probes. A traversing movement with rotary axis is not allowed during touch probe operating mode.
Remedy: Remove the rotary axis movement from the traversing movement.

8226 Rotary axis switch not valid (MSD data)!
Cause: The indicated spindle does not have a rotary axis.

8228 Rotary axis switch not allowed while axis move!
Cause: The rotary axis has moved during switching over to the spindle operating mode.
Remedy: Stop the rotary axis before switching.

8229 Spindle on not allowed while rotary axis is active!

8230 Program start not allowed due to active spindle rotation axis!

8231 Axis configuration (MSD) for TRANSMIT not valid!
Cause: Transmit is not possible at this machine.

8232 Axis configuration (MSD) for TRACYL not valid!
Cause: Tracyl is not possible at this machine.

8233 Axis not available while TRANSMIT/ TRACYL is active!
Cause: Programming of the rotary axis is not allowed during Transmit/ Tracyl.

8234 Axis control grant removed by PLC while axis interpolates!
Cause: Internal error
Remedy: Delete error with reset and inform EMCO.

8235 Interpolation invalid while axis control grant is off by PLC!
see 8234.

8236 TRANSMIT/TRACYL activated while axis or spindle moves!
see 8234.

8237 Motion through pole in TRANSMIT!
Cause: It is not allowed to move through the coordinates X0 Y0 inTransmit.
Remedy: Alter the traversing movement.

8238 Speed limit in TRANSMIT exceeded!
Cause: The traversing movement gets too close to the coordinates X0 Y0. In order to observe the programmed feed rate, the maximum speed of the rotary axis would have to be exceeded.
Remedy: Reduce the feed rate. Set the value of the C-axis feed limitation in WinConfig, machine data settings / general machine data/ to 0.2. Thus, the feed rate will be automatically reduced near the coordinates X0 Y0.

8239 DAU exceeded 10V limit!
Cause: Internal error
Remedy: Start the software again or install it anew. Report the error to EMCO.

8240 Function not valid during active transformation (TRANSMIT/TRACYL)!
Cause: The Jog and INC operating mode are not possible during Transmit in X/C and during Tracyl in the rotary axis.

8241 TRANSMIT not enabled (MSD)!
Cause: Transmit is not possible at this machine.

8242 TRACYL not enabled (MSD)!
Cause: Tracyl is not possible at this machine.

8243 Round axis invalid during active transformation!
Cause: It is not allowed to program the rotary axis during Transmit/Tracyl.

8245 TRACYL radius = 0!
Cause: When selecting Tracyl, a radius of 0 was used.
Remedy: Correct the radius.

8246 Offset alignment not valid for this state!
see 8239.

8247 Offset alignment: MSD file write protected!

8248 Cyclic supervision failed!
Cause: The communication with the machine keyboard is interrupted.
Remedy: Start the software again or install it anew. Report the error to EMCO.

8249 Axis motion check alarm!
see 8239

8250 Spindle must be rotation axis !
see 8239

8251 Lead for G331/G332 missing !
Cause: The threading pitch is missing or the starting coordinates are identical to the target coordinates.
Remedy: Program the threading pitch.
Correct the target coordinates.

8252 Multiple or no linear axis programmed for G331/G332 !
Remedy: Program exactly one linear axis.

8253 Speed value for G331/G332 and G96 missing !
Cause: No cutting speed has been programmed.
Remedy: Program the cutting speed.

8254 Value for thread starting point offset not valid!
Cause: The thread starting point offset is not within the range of 0 to 360°.
Remedy: Correct the thread starting point offset.

8255 Reference point not in valid software limits!

Cause: The reference point has been defined outside the software limit switches.

Remedy: Correct the reference points in WinConfig.

8256 Spindle speed too low while executing G331/G332!

Cause: During tapping the spindle speed has decreased. Perhaps the incorrect threading pitch was used or the core drilling is not correct.

Remedy: Correct the threading pitch. Adapt the diameter to the core drilling.

8257 Real Time Module not active or PCI card not found!

Cause: ACC could not be started correctly or the PCI card in the ACC was not recognized.

Remedy: Report the error to EMCO.

8258 Error allocating Linux data!

see 8239.

8259 Current thread in sequence not valid!

Cause: One block of a thread in sequence has been programmed without thread G33.

Remedy: Correct the program.

8261 Missing thread in sequence !

Cause: A successive thread has not been programmed for a thread in sequence, the number has to be in accordance with the SETTHREADCOUNT () that has been defined before.

Remedy: Correct the number of threads in the thread in sequence and add a thread.

8262 Reference marks are not close enough !

Cause: The settings of the linear scale have been changed or the linear scale is defective.

Remedy: Correct the settings. Contact EMCO.

8263 Reference marks are too close together!

see 8262.

22000 Gear change not allowed

Cause: Gear step change when the spindle is active.

Remedy: Stop the spindle and carry out a gear step change.

22270 Feed too high (thread)

Cause: Thread pitch too large / missing, Feed for thread reaches 80% of rapid feed

Remedy: Program correction, lower pitch or lower spindle speed for thread

I: Control Alarms

Control Alarms

These alarm s can occur only with operating and programming the control functions or with running CNC programs.

1 RS232 parity error !
Cause: Data transmission error parity error, wrong RS 232 setting in external device
Remedy: Check data cables, set serial interface of the external device

2 RS232 transmission error !
Cause: Data transmission error character overflow Data transmission error invalid data frame
Remedy: Check data cables, set serial interface of the external device

10 Nxxxx Invalid G-code
Remedy: Program correction

11 ORDxx Feed wrong/missing
Cause: Attempt to start with feed = 0, also with G95/96, if S = 0 or M5
Remedy: Program correction

21 Nxxxx Circle: Wrong plane selected
Cause: The wrong plane (G17, 18, 19) is active for a circle
Remedy: Program correction

30 Nxxxx Invalid tool offset number
Cause: The lower 2 digits of the T number are to great
Remedy: Program correction

33 Nxxxx CRC can't be determined
Cause: Too much blocks without new position programmed, invalid contour element, programmed circle radius smaller than cutter radius, contour element to short.
Remedy: Program correction

34 Nxxxx Error on deactivating CRC
Remedy: Program correction

37 Nxxxx Plane change while CRC act.
Cause: Change of plane not permitted with active cutter radius compensation
Remedy: Program correction

41 Nxxxx Contour violation CRC
Cause: Invalid contour element, programmed circle radius smaller than cutter radius, contour element to short, contour violation with full circle.
Remedy: Program correction

51 Nxxxx Wrong chamfer/radius value
Cause: The contour elements between a chamfer / radius should be inserted are too short.
Remedy: Program correction

52 Nxxxx Invalid contour draft
Cause: From the programmed parameters no valid contour draft would result
Remedy: Program correction

53 Nxxxx Wrong parameter structure
Cause: From the programmed parameters no valid contour draft would result, wrong parameter programmed
Remedy: Program correction

56 Nxxxx Wrong angle value
Cause: With the programmed angle no intersection point would result
Remedy: Program correction

57 Nxxxx Error in contour draft
Cause: Invalid parameters programmed.
Remedy: Program correction

58 Nxxxx Contour draft not determinable
Cause: Too much blocks without new position programmed, program end while contour draft
Remedy: Program correction

60 Nxxxx Block number not found
Cause: Jump target not found
Remedy: Program correction

62 Nxxxx General cycle error
Cause: Call-up counter of subprogram call invalid, feed<=0, thread pitch missing/<=0, cutting depth missing/<=0/invalid, retraction height to small, block address P/Q missing, declaration pattern repetition missing/invalid, infeed for next cut missing/invalid, undercut at cycle ground <0, cycle end point missing/invalid, thread end point missing/invalid;
Remedy: Program correction

63 Nxxxx Wrong Cycle call
Cause: P/Q missing, wrong address
Remedy: Program correction

70 Insufficient memory
Cause: The PC has not enough memory
Remedy: Close all other Windows applications, remove resident programs from memory, restart the PC

71 Program not found
Cause: NC program not found
With program start no program was selected
Remedy: Correct call-up or create program, select program

73 File already exists !
Remedy: Select other file name.

77 Insufficient RAM for subroutine
Cause: Subprograms interlocked too deep
Remedy: Program correction

83 Nxxxx Circle not in active plane
Cause: Circle is not in active plane for CRC
Remedy: Program correction

142 Wrong simulation area
Cause: Wrong scale factor (e.g. 0) programmed
Remedy: Program correction

142 Invalid scale factor
Cause: No or an invalid simulation area was entered
Remedy: Enter correct simulation area

315 ORDxx Rotatory checking X
Cause: The step motor has fallen out of pace
Remedy: Reduce infeed and feed, check slides for smooth running, approach reference point

325 ORDxx Rotatory checking Y
see alarm 315

335 ORDxx Rotatory checking Z
see alarm 315

500 ORDxx Target point exceeds work.area
Cause: Target point, circle target point or circle out of working area limitation
Remedy: Program correction

501 ORDxx Target point exceeds SW limit
Cause: Target point, circle target point or circle out of working area limitation
Remedy: Program correction

510 ORDxx Software-limit switch X
Cause: Software limit switch in X exceeded (JOG)
Remedy: Traverse back manually

520 ORDxx Software-limit switch Y
see 510

530 ORDxx Software-limit switch Z
see 510

2501 ORDxx Synchronisation-error AC
Remedy: RESET, report to EMCO if reproducible

2502 ORDxx Synchronisation-error AC
see 2501

2503 ORDxx Synchronisation-error AC
see 2501

2504 ORDxx No memory for interpreter
Cause: Too less RAM memory, continueing the program is not possible
Remedy: Close all Windows application, close WinNC,remove resident programs from AUTOEXEC.BAT and CONFIG.SYS, restart the PC

2505 ORDxx No memory for interpreter
see 2504

2506 ORDxx Too less RAM
see 2504

2507 ORDxx Reference point not active
Remedy: Approach reference point

2508 ORDxx Internal error NC core
Remedy: RESET, report to EMCO if reproducible

2520 ORDxx RS485 device absent
Cause: With program start a RS485 device did not report, while program run a device got defective
AC Axis controller
SPS PLC
MT control keyboard
Remedy: Switch on RS485 device (machine, control keyboard), check cables and plugs, check terminator plug, report to EMCO if reproductible

2521 ORDxx RS485 communication error
Remedy: PC restart, report to EMCO if reproducible

2522 ORDxx RS485 communication error
Remedy: PC restart, report to EMCO if reproducible

2523 ORDxx INIT error on RS485 PC-board
See "Software Installation" , Mistakes with installation of the software

2524 ORDxx Gen.-Failure RS485 PC-board
Remedy: PC restart, report to EMCO if reproducible

2525 ORDxx Transmit error RS485
Cause: Transmission error by poor plug connections, missing terminator, external sources of electromagnetic interference
Remedy: Check the error sources above

2526 ORDxx Transmit error RS485
see 2525

2527 ORDxx Internal error AC
Remedy: Switch machine off/on, report to EMCO if reproducible

2528 ORDxx Operating system error PLC
Remedy: Switch machine off/on, report to EMCO if reproducible

2529 ORDxx External keyboard error
Remedy: The external keyboard always must be switched on after the PC. Restart the software, report to EMCO if reproducible

2540 ORDxx Error saving setting-data
Cause: Hard disk full, wrong path setting, no writing access
Remedy: Check hard disk space, check writing access, reinstallation of the software if reproducible

2545 ORDxx Drive / Device not ready
Remedy: Insert disk, lock drive, check disk drive, ...

2546 ORDxx Checksum error machine-data
Remedy: Restart, report to EMCO if reproductible

2550 ORDxx PLC simulation error
Remedy: Restart, report to EMCO if reproducible

2551 ORDxx PLC simulation error
Remedy: Restart, report to EMCO if reproducible

2562 Read error on CNC program
Cause: Defective program file, DOS read error (disk, hard disk)
Remedy: Solve problem on DOS level, eventually reinstallation of the software

2614 ORDxx Internal error MSD
Remedy: Report to EMCO if reproducible

2650 ORDxx Internal error cycle call up
Cause: Invalid cycle call when a cycle was called with a G command
Remedy: Program correction

2849 Internal error CRC
Remedy: Report to EMCO if reproducible

2904 Helix Z value too large
Cause: The pitch of the helix must not be larger than 45°
Remedy: Program correction